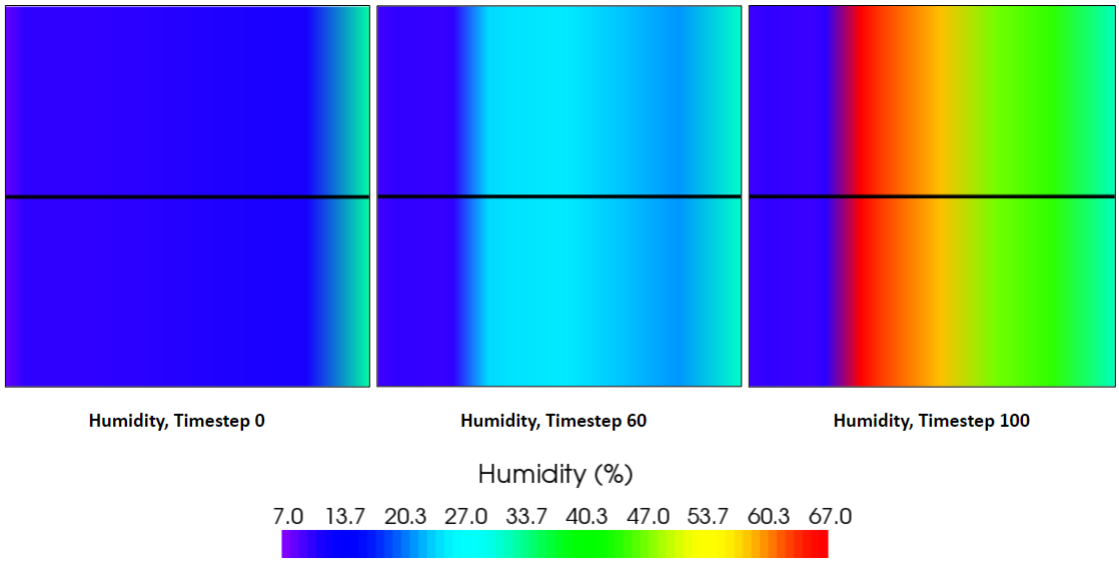
LBL 37371 Rev. 10
November 2026

Program Description

A PC Program

THERM 8.1



for Analyzing Two-Dimensional Heat and Moisture 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

© 1995-2026 Regents of the University of California

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Building Technologies Program, of the U.S. Department of Energy under Contract no. DE-AC02-05CH11231.

DISCLAIMER

This document was prepared as an account of work sponsored by the United States Government. While this document is believed to contain correct information, neither the United States Government nor any agency thereof, nor The Regents of the University of California, nor any of their employees, makes any warranty, express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product process, or service by its trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof, or The Regents of the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, or The Regents of the University of California.

Ernest Orlando Lawrence Berkeley National Laboratory
is an equal opportunity employer.

LBL-37371 Rev. 10

THERM 8.1: Program Description

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

Charlie Curcija, Robin Mitchell, Christian Kohler, Dariush Arasteh
Stephen Czarnecki, Simon Vidanovic, Ling Zhu
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

November 2026

© Regents of the University of California

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Building Technologies Program, of the U.S. Department of Energy under Contract no. DE-AC02-05CH11231.

Contents

1. INTRODUCTION **1-1**

1.1. Overview....1-1
1.2. THERM 8.1 Transient Moisture Model....1-1
1.3. Changes from THERM 7 to THERM 8.1....1-2

2. INSTALLATION **2-1**

2.1. Computer Requirements....2-1
2.2. Installation Procedures....2-1
2.2.1. Before Installing WINDOW....2-1
2.2.2. Installing THERM....2-1

3. QUICK START **3-1**

3.1. Draw Cross Section and Select Materials (for detailed instructions, see Chapter 5, "Drawing Cross-Section Geometry")....3-1
3.2. Drawing Features....3-2
3.3. Materials....3-4
3.4. Insert Glazing System....3-6
3.5. Adding a Custom Spacer....3-8
3.6. Specify Boundary Conditions....3-12
3.7. Run Simulation and View Results....3-13
3.8. Toolbar Reference Table....3-14
3.9. Other Shortcuts....3-16

4. GETTING STARTED **4-1**

4.1. The Main Screen....4-1
4.1.1. Status Bar....4-2
4.2. Main Menu....4-4
4.2.1. Pull-down Lists....4-5
4.2.2. THERM Files....4-6

5. Drawing Cross-Section Geometry **5-1**

5.1. Overview of Drawing a Cross Section....5-1
5.2. Importing a DXF or Bitmap File as an Underlay....5-1
5.2.1. Select Underlay from the File menu....5-2
5.2.2. Select DXF or Bitmap File....5-3
5.2.3. DXF File Conversion....5-8
5.3. Drawing Features....5-10
5.3.1. Sticky Distance....5-10
5.3.2. Drawing Options....5-10

5.3.3. Snap Settings 5-12
5.3.4. Zoom 5-14
5.3.5. Voids and Overlaps 5-15
5.3.6. Other Drawing Features 5-16
5.4. Drawing Using a Mouse 5-17
5.4.1. Suggestions for Drawing Cross Sections 5-19
5.5. Drawing Using the Keyboard (Numeric Cursor Positioning) 5-21
5.6. Editing Polygons 5-23
5.6.1. Selecting a Polygon 5-23
5.6.2. Moving a Polygon 5-24
5.6.3. Deleting a Polygon 5-25
5.6.4. Moving a Point 5-26
5.6.5. Moving a Line 5-26
5.6.6. Adding a Point 5-27
5.6.7. Deleting a Point 5-28
5.7. Flip and Rotate 5-28
5.7.1. Flip 5-29
5.7.2. Rotate 5-30
5.8. Fill Tool 5-32
5.8.1. Using the Fill Tool to Create a New Polygon 5-32
5.9. Eye Dropper Tool 5-35
5.10. Inserting a Glazing System 5-36
5.10.1. Inserting a WINDOW Glazing System 5-37
5.10.2. Condensation Index 5-52
5.10.3. If the WINDOW Glazing System Doesn't Fit 5-56
5.10.4. Nominal Thickness for Glazing Systems 5-57
5.10.5. Inserting More Than One WINDOW Glazing Systems 5-59
5.10.6. Information about the Glazing System 5-59
5.10.7. Multiple Glazing Options 5-65
5.10.8. Editing Keff Value of Glazing System Cavity 5-70
5.11. Assigning Materials After Drawing Polygons 5-73
5.11.1. Use the Material pull-down list to the right of the toolbar: 5-73
5.11.2. Access the Properties for Selected Polygon(s) dialog box: 5-75
5.11.3. Solid Materials 5-77
5.11.4. Frame Cavities 5-78
5.11.5. Linking Material Properties of Polygons 5-80
5.12. Material Library 5-81
5.12.1. Material Library List View 5-82
5.12.2. Material Library Detail View 5-84
5.12.3. Defining New Materials 5-92
5.12.4. Solid Materials Type 5-93
5.12.5. Frame Cavity 5-95
5.12.6. Glazing Cavity 5-97
5.12.7. External Radiation Enclosure 5-97
5.12.8. Shade Material 5-100
5.13. Creating a Component Library 5-102
5.14. Drawing Tips 5-104
5.14.1. Drawing Methods 5-104
5.14.2. DXF Files 5-104
5.14.3. Representing Curves with Line Segments 5-105
5.14.4. DXF Files -- AutoConvert 5-107
5.14.5. Sticky Distance and Snapping 5-109

5.14.6. Drawing Tips....5-109
5.14.7. Drawing with the Keyboard....5-112
5.14.8. Donuts....5-113
5.14.9. Breaking Up Extrusions into Polygons....5-114
5.14.10. Drawing Extrusions from DXF Files -- False Die Mold Method....5-116
5.14.11. Cuttins, Relocating, Flipping and Rotating Discussion....5-118

6. DEFINING BOUNDARY CONDITIONS 6-1

6.1. Overview....6-1
6.1.1. Boundary Condition Definitions....6-1
6.1.2. Boundary Condition Library....6-5
6.2. Assigning Boundary Conditions and U-factor Surface Tags....6-7
6.2.1. Generate Boundary Conditions....6-10
6.2.2. Automatic Check of Geometry Definition....6-11
6.2.3. Review the Boundary Conditions....6-12
6.2.4. Reassign Boundary Conditions....6-15
6.3. Error Detection in THERM....6-22
6.3.1. Finding and Fixing Bad Points....6-23
6.4. Boundary Condition Library....6-30
6.4.1. Boundary Condition Library for Steady-State Model....6-30
6.4.1.2. Boundary Condition Library Detail View....6-32
6.4.2. Boundary Condition Library for Transient Model....6-45
6.4.2.2. Boundary Condition Library Detail View....6-48

7. CALCULATING STEADY-STATE RESULTS 7-1

7.1. Overview of THERM Calculation Methods....7-1
7.1.1. Conrad Simulation Engine....7-1
7.1.2. HygroTherm Simulation Engine....7-1
7.1.3. Mesher....7-1
7.2. Calculations....7-2
7.2.1. Calculation Menu....7-2
7.2.2. Calculating Results for a THERM file....7-5
7.2.3. Calculation Options....7-7
Options / Preferences....7-17
7.3. Feedback and Troubleshooting During the Calculation Process....7-26
7.3.1. Premeshing....7-26
7.3.2. Reading Geometry....7-27
7.3.3. Meshing....7-27
7.3.4. Generating Input....7-28
7.3.5. View Factors (optional)....7-28
7.3.6. Simulating....7-28
7.3.7. Error Estimator (optional)....7-28
7.4. Viewing the results....7-30
7.4.1. Graphic Results....7-30
7.4.2. U-factors....7-46
7.4.3. Value at Cursor....7-53
7.4.4. Average Temperature with Tape Measure....7-55
7.4.5. Report....7-57
7.4.6. THERM File Properties....7-57

8. MODELING FRAME AND EDGE HEAT TRANSFER 8-1

8.1. Overview 8-1
8.2. Cross-Sections to be Modeled 8-1
8.3. Draw the Cross Sections 8-4
8.3.7. Getting Started – Drawings and DXF Files 8-4
8.3.8. File Properties – Therm File Properties tab 8-4
8.3.9. Cross-Section Orientation 8-5
8.3.10. What is Not Modeled 8-6
8.3.11. Deformable Parts 8-7
8.3.12. Frame Cavities 8-8
8.3.13. Slightly Ventilated Exterior Cavities 8-15
8.3.14. Modeling Sloped and Curved Surfaces 8-16
8.3.15. Modeling Sloped Sills 8-17
8.4. Importing Glazing Systems 8-18
8.4.1. Overview 8-18
8.4.2. Inserting Glazing Systems 8-19
8.4.3. Multiple Glazing Options 8-20
8.4.4. Linking Glazing Cavities 8-23
8.4.5. Condensation Calculation 8-26
8.5. Defining Boundary Conditions 8-30
8.5.1. Overview 8-30
8.5.2. Assigning Boundary Conditions and U-factor Surface Tags 8-33
8.5.3. Voids, Overlaps and Bad Points 8-39
8.6. Calculating Results 8-40
8.6.1. Meshing 8-40
8.6.2. Error Estimator 8-40
8.6.3. Default Settings 8-42
8.6.4. Calculations 8-43
8.6.5. Results 8-45
8.6.6. Custom Frame Length 8-46
8.6.7. Importing Results into WINDOW 8-47
8.7. Meeting Rails 8-49
8.7.1. Modeling Meeting Rails 8-49
8.7.2. Steps for Meeting Rail U-factor Calculation 8-50
8.7.3. Steps for Meeting Rail Condensation Calculation 8-55
8.8. Dividers 8-57
8.8.1. Internal Dividers (Suspended Grilles) 8-57
8.8.1.3. Gas Filled Glazing Systems 8-65
8.8.2. Simulated Divided Lites 8-66
8.8.3. True Divided Lites 8-75
8.9. Spacers 8-83
8.9.1. Overview 8-83
8.9.2. Linking Glazing Cavity properties (imported from WINDOW) for Open Spacers 8-84
8.10. Non Continuous Thermal Bridge Elements 8-86
8.10.3. Modeling Steps 8-86
8.10.4. Equations 8-88
8.10.5. Example 1: Bolts in Curtain Wall 8-89
8.10.6. Example 2: Thermally slotted cross-section 8-92
8.10.7. Example 3: Skip-and-debridge: 8-96

8.11. Site Built fenestration products (Curtain Walls, Window Walls and Sloped Glazing) 8-99
8.11.1. Curtain Walls, Window Walls, and Sloped Glazing....8-99
8.11.2. Curtain Wall Modeling Procedure 8-101
8.12. Framed Intermediate Pane 8-113
8.12.1. Overview 8-113
8.12.2. THERM Modeling Details 8-114

9. Transient Moisture and Thermal Simulations 9-1

9.1. Overview....9-1
9.2. File / Properties 9-1
9.3. Material Library 9-12
9.3.1. Material Library List View 9-12
9.3.2. Material Library Detail View 9-15
9.4. Boundary Condition Library....9-25
9.4.1. Boundary Condition Library List View....9-25
9.4.2. Boundary Condition Library Detail View: HygroTherm, Transient ..9-29
9.5. Simulate the Model....9-39
9.6. View the Results....9-42

10. TUTORIAL 10-1

10.1. Tutorial Organization....10-1
10.2. Start THERM 10-2
10.3. Draw the Frame Cross-Section Geometry 10-3
10.3.3. Import Underlay with Autoconvert On 10-3
10.3.4. Tracing a DXF Underlay....10-7
10.3.5. Using a Dimensioned Drawing 10-13
10.4. Import a Glazing System from WINDOW 10-22
10.5. Define Boundary Conditions....10-34
10.6. Run and View the Results....10-38

1. INTRODUCTION

1.1. Overview

THERM is a 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 two-dimensional 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. The material properties are defined for each polygon and the environmental conditions to which the component is exposed are defined by 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.

THERM's results can be used to define the frame elements in the Berkeley Lab WINDOW program's center-of-glass optical and thermal models to determine total window product U-factors and Solar Heat Gain Coefficients.

Program downloads, update information, and Knowledge Base articles about THERM and the other software tools from the Windows and Daylighting Group at LBNL can be found at: https://windows.lbl.gov/software-tools.

If you have questions or problems about using the program, email ThermHelp@lbl.gov or go to the THERM Forum (Google Group) at https://groups.google.com/g/LBNL-THERM where you can post and reply to questions about the program.

1.2. THERM 8.1 Transient Moisture Model

The THERM simulation engine been extended to model time dependent (transient) simulations (a "time domain" has been added to THERM's original 2-D Finite Element Numerical Model (FEM)), and a moisture transfer model has been added.

The transient thermal and moisture simulation engine in THERM (called HygroTherm) allows modeling of thermal bridges and non-homogeneities in building construction without approximation. Such elements are primary pathways and causes for condensation and moisture in building and it is important to model them correctly in a thermal/moisture analysis. By failing to account for the moisture characteristics in the thermal envelope, designers and building can introduce problems that endanger the health and safety of building occupants as well as the durability of the building itself.

The HygroTHERM model in THERM 8.1 will allow building simulation practitioners to accurately model wall, roof, foundation constructions taking into account both the thermal and moisture characteristics of those constructions.

1.3. Changes from THERM 7 to THERM 8.1

The following are the changes made to THERM 8:

- Simulation engine options (File / Properties / Calculation Options)
  - **Conrad:** Steady-state without moisture
  - **HygroTherm:** Transient/moisture
- Material and Boundary Condition Libraries
  - The Libraries are now stored in the XML file format and have a new grid view
- New mesher (Simmetrix) that is able to handle more complex geometry
- Visualization for viewing the moisture results
- New file format
  - The THERM files now have the extension "thmz" which indicates that it is a zip file of all the relevant XML files that are needed for the THERM model represented in the file.

2. INSTALLATION

2.1. Computer Requirements

The following are needed to successfully run THERM.

- Microsoft Windows 10™ or higher.
- If you plan to model glazing systems in THERM, you need to install the same version of WINDOW as the THERM version you are installing. If you do not anticipate needing to include a glazing system in your modeling, this is not required.

2.2. Installation Procedures

2.2.1. Before Installing WINDOW

You **must be Administrator** on your machine in order to successfully install THERM.

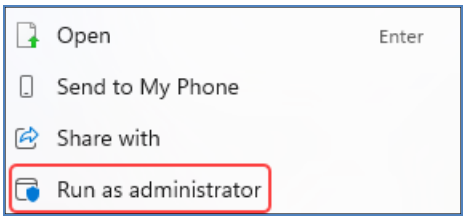


Figure 2-1. Right click on the install file and select Run as Administrator if needed.

2.2.2. Installing THERM

Download the installation file from the https://windows.lbl.gov/therm-software-downloads website.

Select the section for the THERM version you are interested in installing. *It is possible to have both THERM 7 and THERM 8.1 installed on your computer at the same time, if that is desired*.

Click on the installation file, which will cause it to be downloaded to your computer.

Click on the downloaded installation file and the program will walk through the installation steps.

You will be asked to agree to the license. If you do not agree, the program will not install the program.

If you have previous versions of THERM 8.1 installed, the program will make copies of the databases created with that version.

The program will be installed by default in this directory:

C:\Program Files (x86)\LBNL\

The working directory, where the program database is installed, is:

C:\Users\Public\LBNL

Depending on the operating system version, an icon for THERM will be added to the desktop, and will also be available under **LBNL Software**.

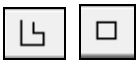
3. QUICK START

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")

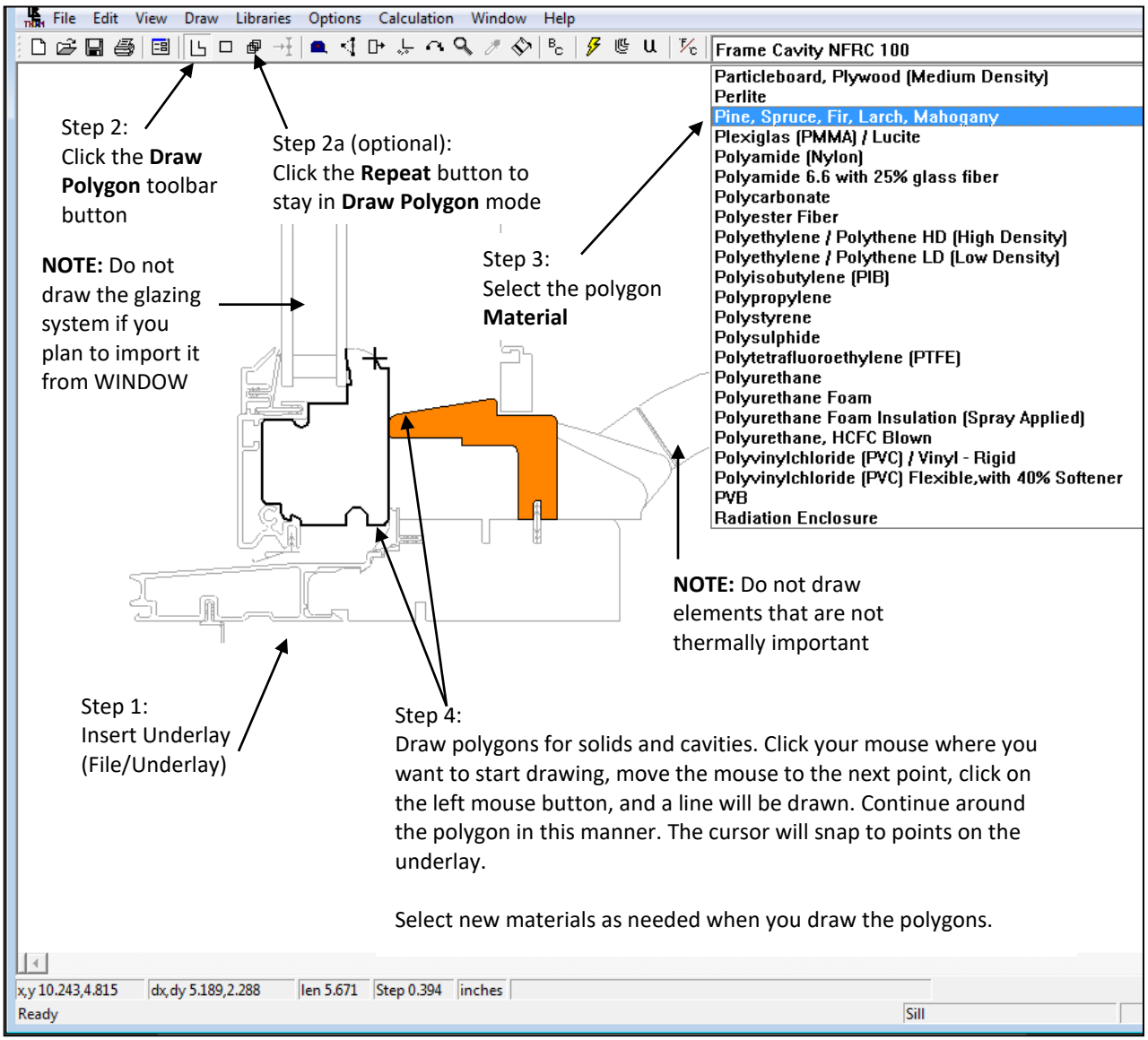
1. If tracing an underlay, bring in a DXF or bitmap file (using **File/Underlay**); to draw the cross section geometry yourself, go to the next step.

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**.


File Edit View Draw Libraries Options Calculation Window Help
Frame Cavity NFRC 100
Particleboard, Plywood (Medium Density)
Perlite
Pine, Spruce, Fir, Larch, Mahogany
Plexiglas (PMMA) / Lucite
Polyamide (Nylon)
Polyamide 6.6 with 25% glass fiber
Polycarbonate
Polyester Fiber
Polyethylene / Polythene HD (High Density)
Polyethylene / Polythene LD (Low Density)
Polyisobutylene (PIB)
Polypropylene
Polystyrene
Polysulphide
Polytetrafluoroethylene (PTFE)
Polyurethane
Polyurethane Foam
Polyurethane Foam Insulation (Spray Applied)
Polyurethane, HCFC Blown
Polyvinylchloride (PVC) / Vinyl - Rigid
Polyvinylchloride (PVC) Flexible,with 40% Softener
PVB
Radiation Enclosure
Step 2: Click the Draw Polygon toolbar button
Step 2a (optional): Click the Repeat button to stay in Draw Polygon mode
NOTE: Do not draw the glazing system if you plan to import it from WINDOW
Step 3: Select the polygon Material
NOTE: Do not draw elements that are not thermally important
Step 1: Insert Underlay (File/Underlay)
Step 4: Draw polygons for solids and cavities. Click your mouse where you want to start drawing, move the mouse to the next point, click on the left mouse button, and a line will be drawn. Continue around the polygon in this manner. The cursor will snap to points on the underlay.
Select new materials as needed when you draw the polygons.
x,y 10.243,4.815
dx,dy 5.189,2.288
len 5.671
Step 0.394
inches
Ready
Sill

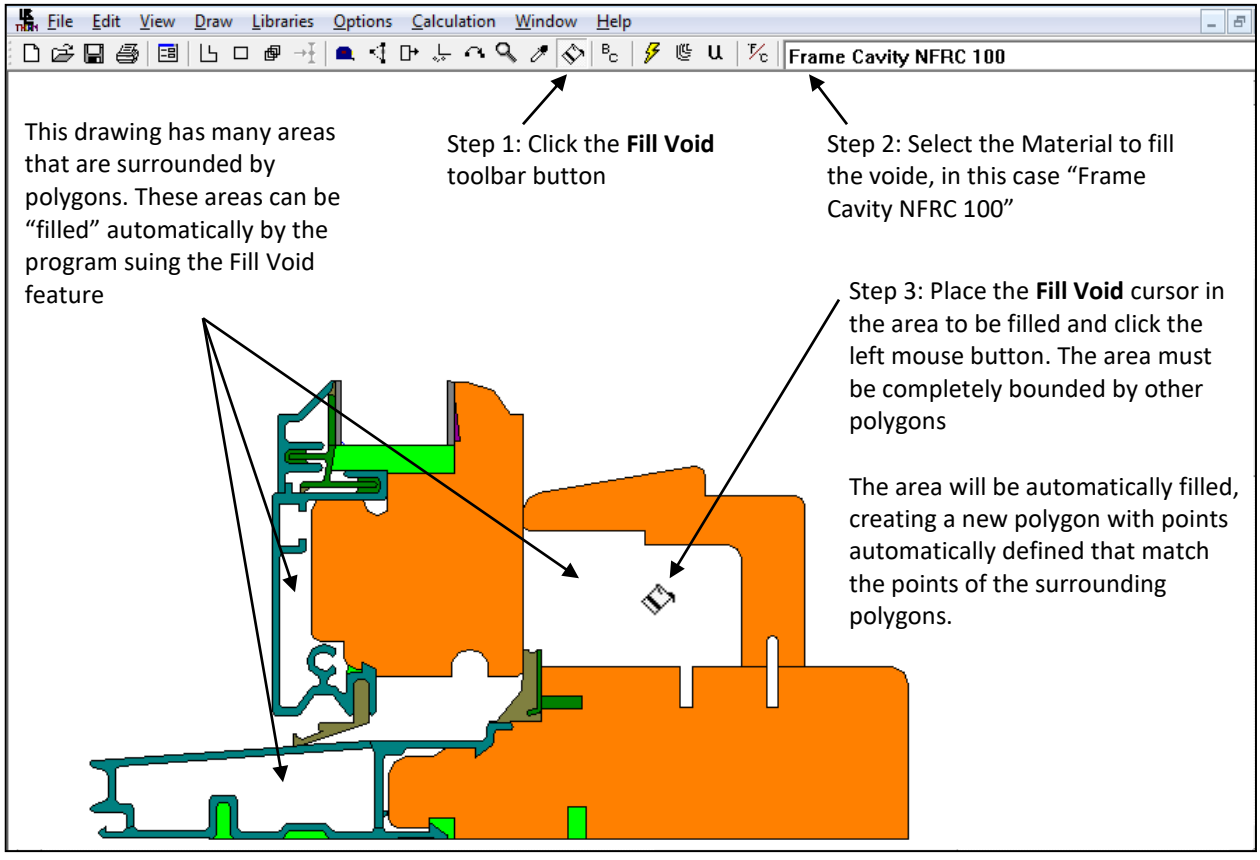
3.2. Drawing Features

There are several 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.

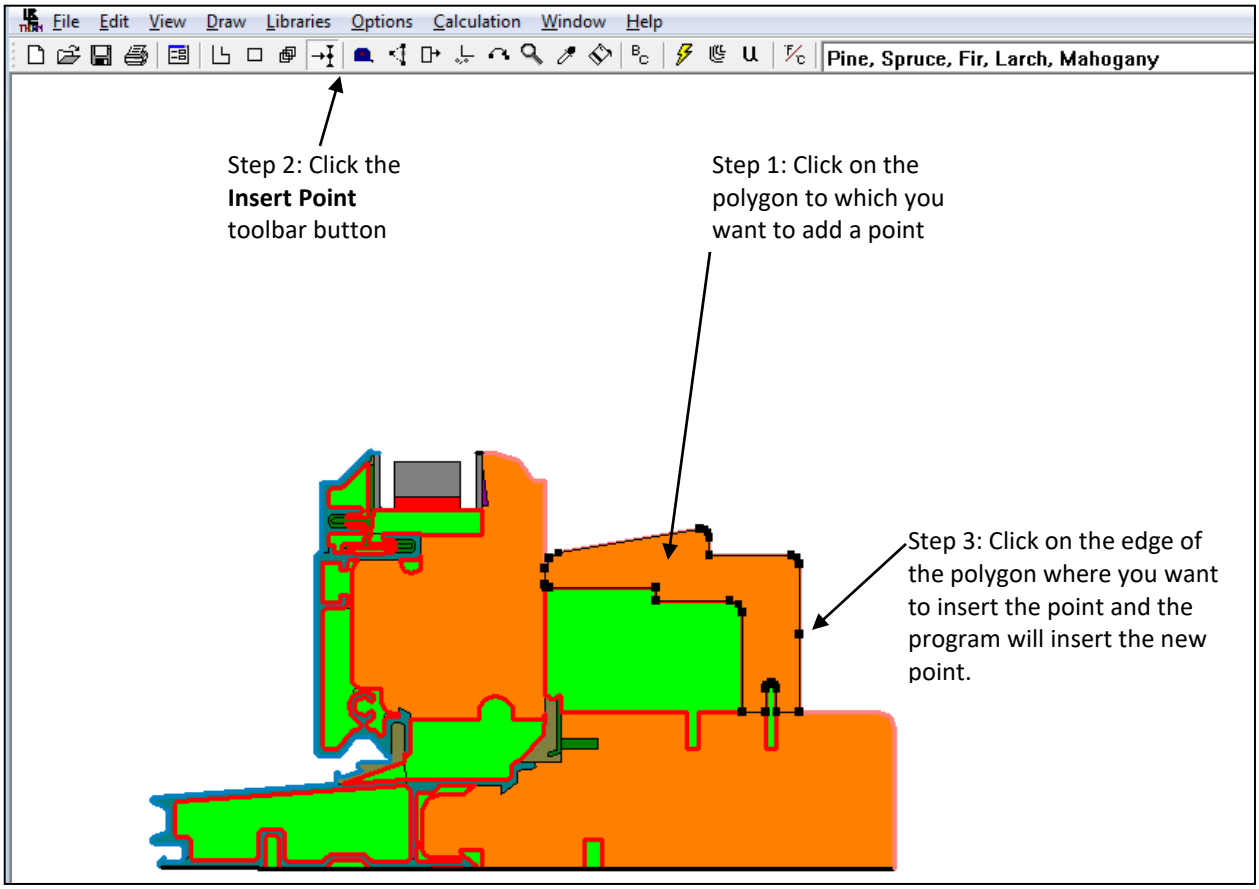
1. **Fill Void feature:** An area of the drawing that is completely surrounded by polygons can be automatically "filled" by THERM, using the **Fill Void** toolbar button.

   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 calculate the model, 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.

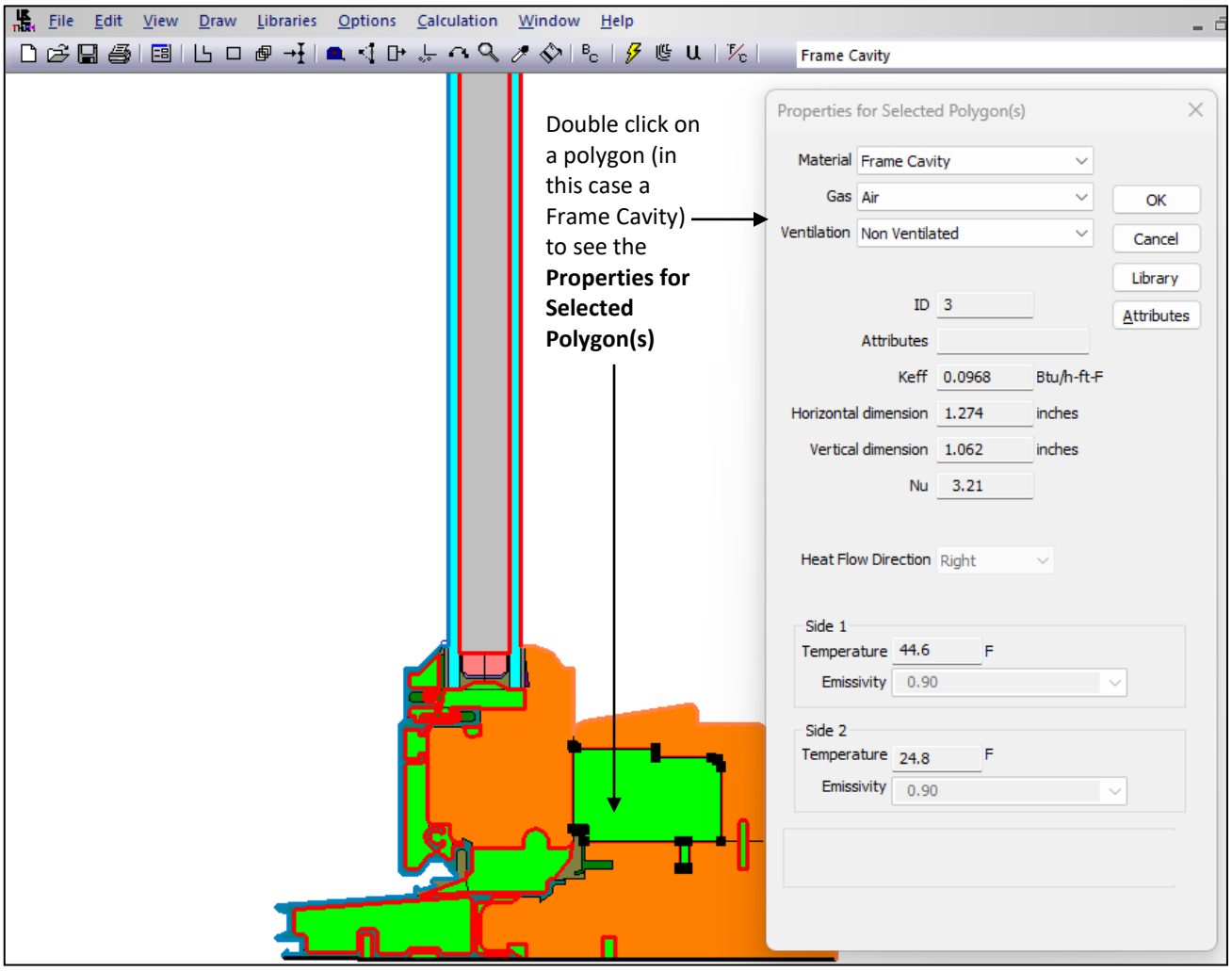


3.3. Materials

(See *Section 5.11, Assigning Materials After Drawing Polygons* and *Section 5.12.3, Defining New Materials*)

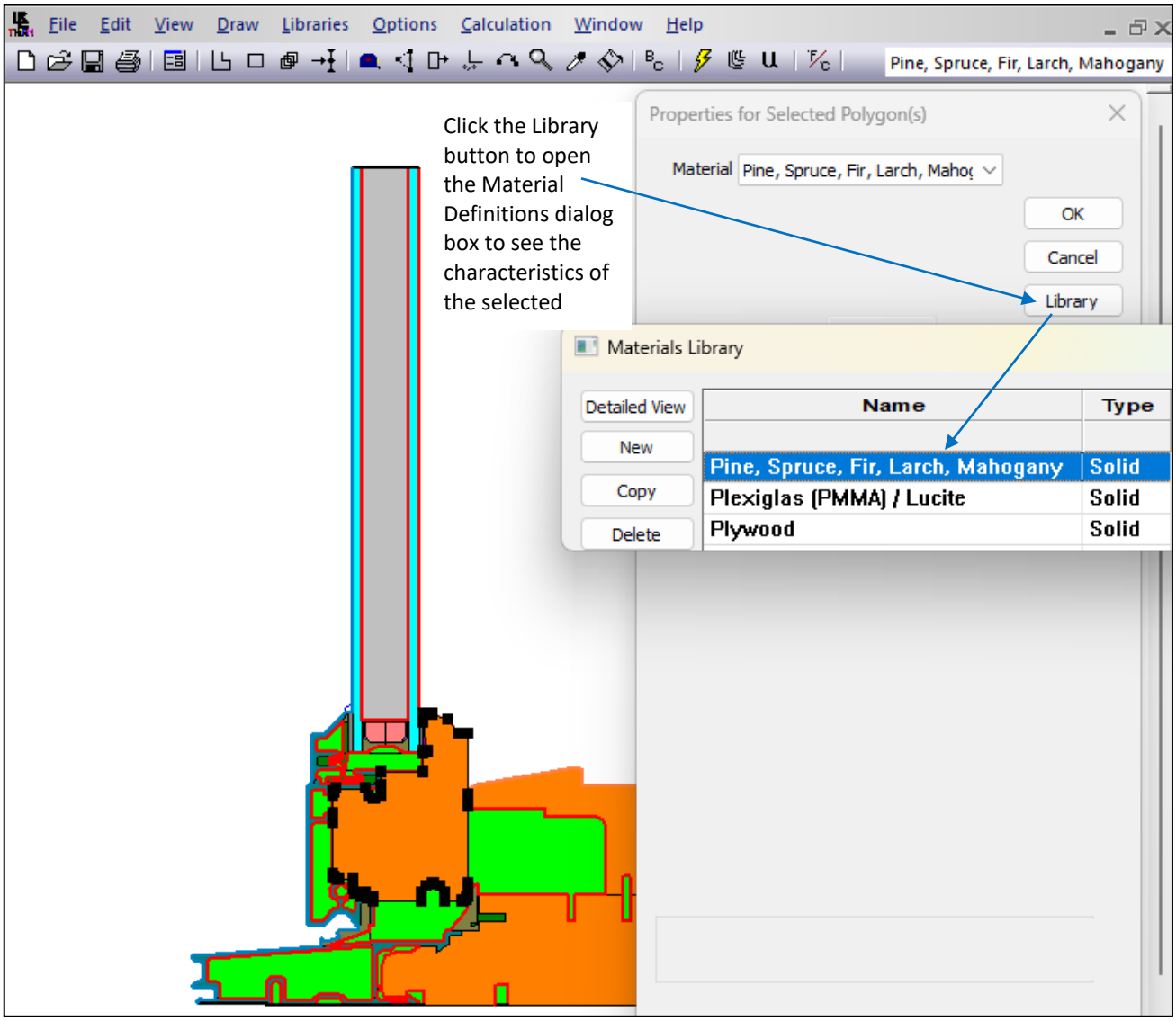
Materials can be specified as you are drawing; they can also be changed after polygons have been created.

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.

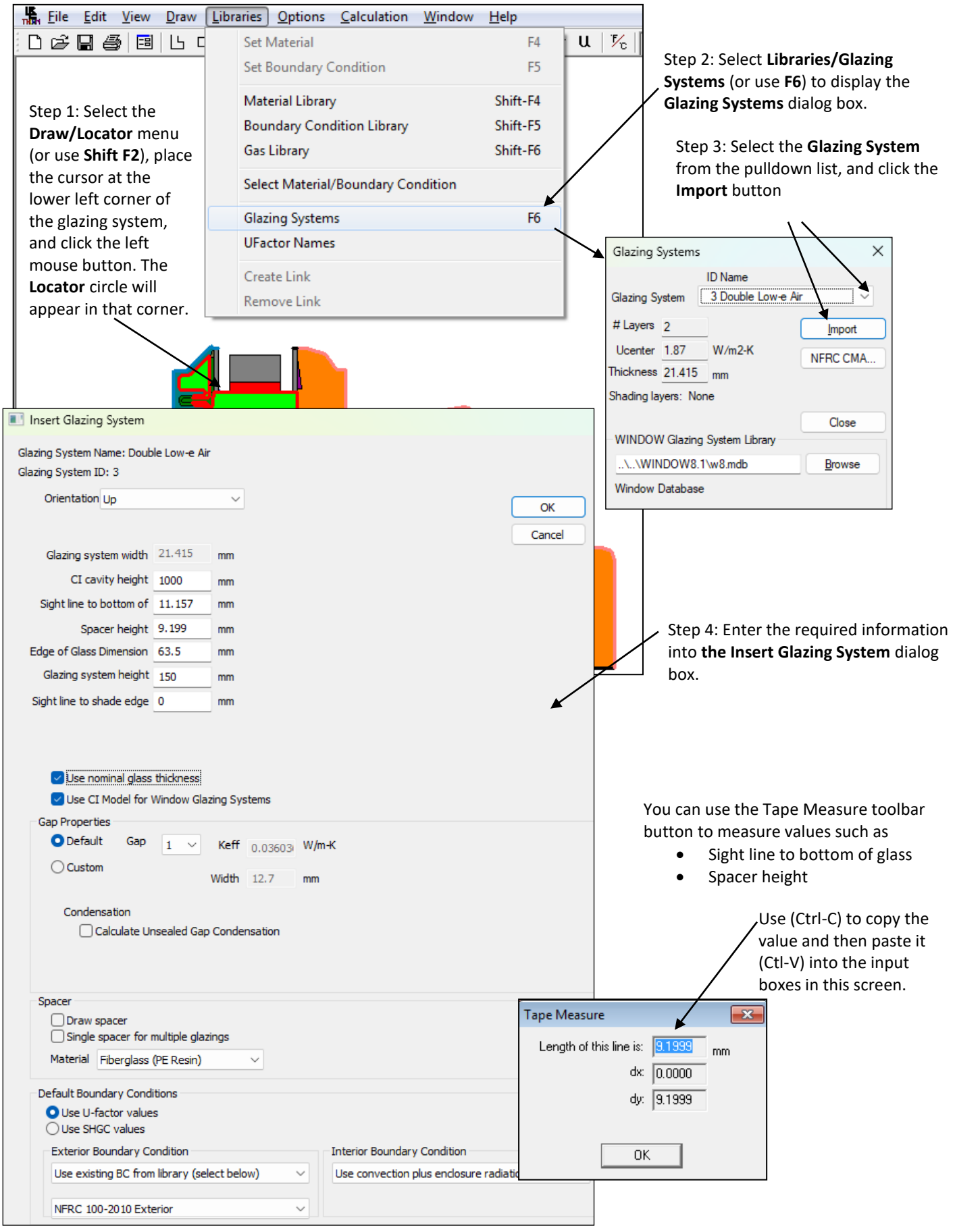
3. The THERM material library contains predefined materials that cannot be edited; however, from the **Material Definitions** dialog box you can create new material library entries, using the **New** button.



3.4. Insert Glazing System

(See *Chapter 5, Drawing Cross-Section Geometry*)

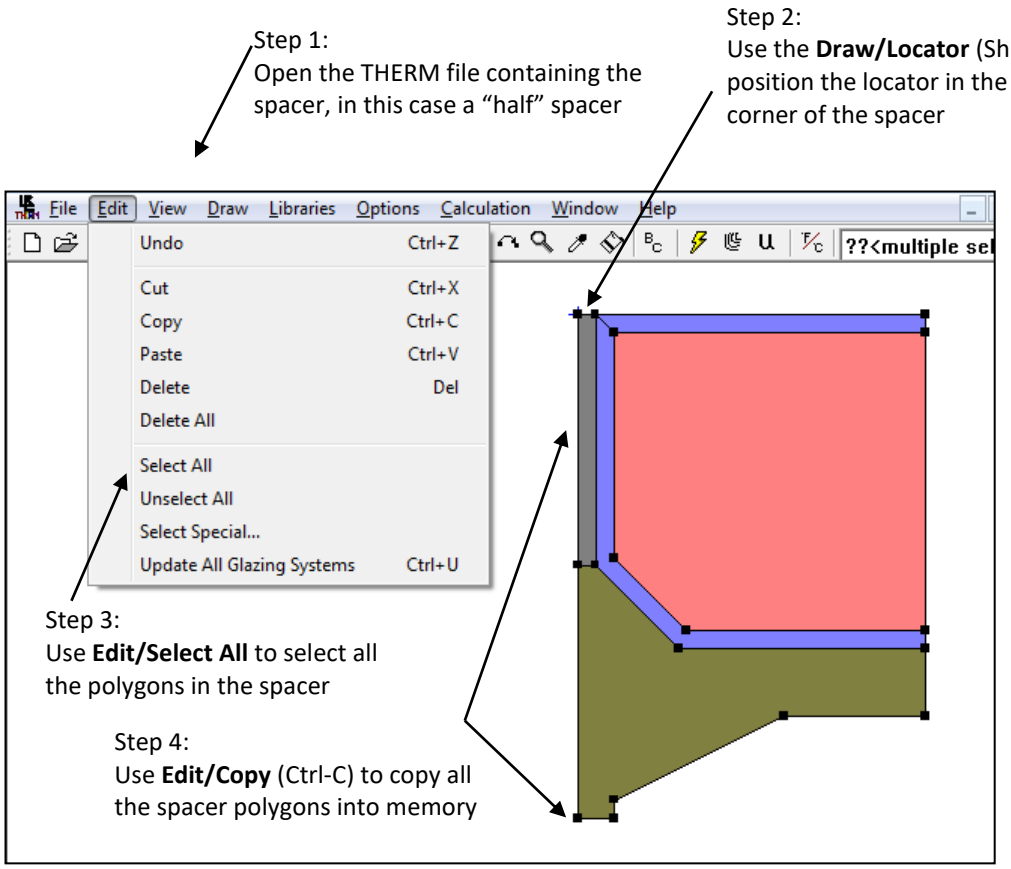
1. On the left-hand side of the glazing unit, draw a rectangle to represent the sealant between the glazing and the frame.
2. Use the **Locator**, accessed from **Draw/Locator**, to define the lower left corner of the glazing system to be inserted.
3. Select the **Libraries/Glazing System** menu choice to access the **Glazing Systems** dialog box, and choose a glazing system previously created in WINDOW 8.1.
4. From the **Glazing System** dialog box, use the pull-down list to select a glazing system from the WINDOW 8.1 Glazing System Library; click on the **Import** button. You can use the **Browse** button to change to a WINDOW database with another name or 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.

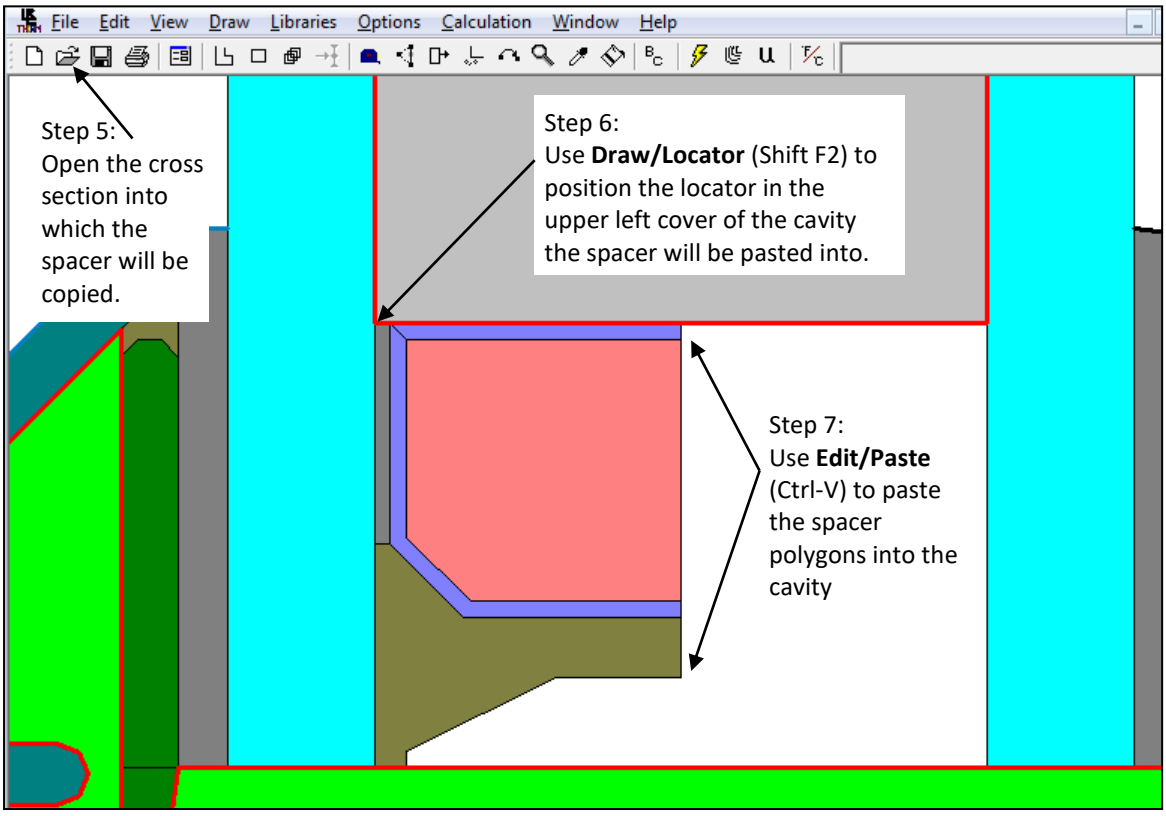

File Edit View Draw Libraries Options Calculation Window Help
Set Material F4
Set Boundary Condition F5
Material Library Shift-F4
Boundary Condition Library Shift-F5
Gas Library Shift-F6
Select Material/Boundary Condition
Glazing Systems F6
UFactor Names
Create Link
Remove Link
Step 1: Select the **Draw/Locator** menu (or use **Shift F2**), place the cursor at the lower left corner of the glazing system, and click the left mouse button. The **Locator** circle will appear in that corner.
Step 2: Select **Libraries/Glazing Systems** (or use **F6**) to display the **Glazing Systems** dialog box.
Step 3: Select the **Glazing System** from the pulldown list, and click the **Import** button
Glazing Systems
ID Name
Glazing System 3 Double Low-e Air
Layers 2
Ucenter 1.87 W/m2-K
Thickness 21.415 mm
Shading layers: None
Import
NFRC CMA...
Close
WINDOW Glazing System Library
..\..\WINDOW8.1\w8.mdb
Browse
Window Database
Insert Glazing System
Glazing System Name: Double Low-e Air
Glazing System ID: 3
Orientation Up
OK
Cancel
Glazing system width 21.415 mm
CI cavity height 1000 mm
Sight line to bottom of 11.157 mm
Spacer height 9.199 mm
Edge of Glass Dimension 63.5 mm
Glazing system height 150 mm
Sight line to shade edge 0 mm
Use nominal glass thickness
Use CI Model for Window Glazing Systems
Gap Properties
Default Gap 1 Keff 0.03603 W/m-K
Custom
Width 12.7 mm
Condensation
Calculate Unsealed Gap Condensation
Spacer
Draw spacer
Single spacer for multiple glazings
Material Fiberglass (PE Resin)
Default Boundary Conditions
Use U-factor values
Use SHGC values
Exterior Boundary Condition
Use existing BC from library (select below)
NFRC 100-2010 Exterior
Interior Boundary Condition
Use convection plus enclosure radiatio
Step 4: Enter the required information into **the Insert Glazing System** dialog box.
You can use the Tape Measure toolbar button to measure values such as
• Sight line to bottom of glass
• Spacer height
Use (Ctrl-C) to copy the value and then paste it (Ctl-V) into the input boxes in this screen.
Tape Measure
Length of this line is: 9.1999 mm
dx: 0.0000
dy: 9.1999
OK

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.

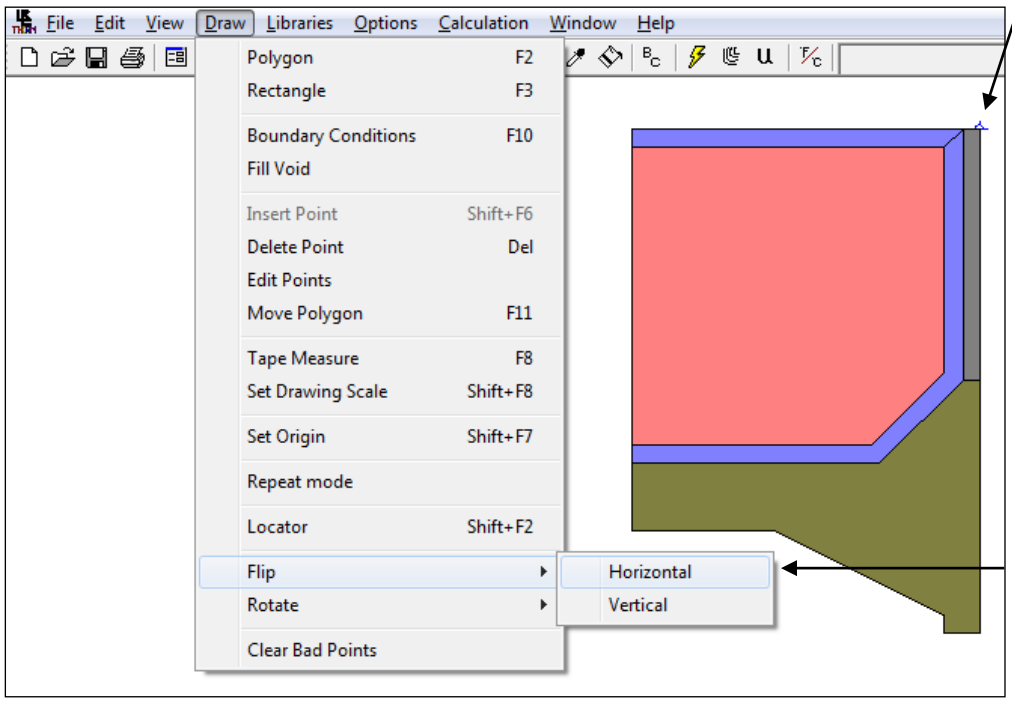
1. Open the THERM file containing the spacer. In the example shown, the file contains half of a spacer.
2. Use the **Draw/Locator** menu choice and click your left mouse button on the upper left corner of the spacer.
3. Use the **Edit/Select All** menu choice to select all the polygons in the spacer.
4. Use the **Edit/Copy** menu choice or **Ctrl-C** to copy all the selected polygons into memory.
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.


Step 1:
Open the THERM file containing the spacer, in this case a “half” spacer
Step 2:
Use the **Draw/Locator** (Shift F2) to position the locator in the upper left corner of the spacer
File Edit View Draw Libraries Options Calculation Window Help
Undo Ctrl+Z
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Delete Del
Delete All
Select All
Unselect All
Select Special...
Update All Glazing Systems Ctrl+U
??<multiple sel
Step 3:
Use **Edit/Select All** to select all the polygons in the spacer
Step 4:
Use **Edit/Copy** (Ctrl-C) to copy all the spacer polygons into memory


File Edit View Draw Libraries Options Calculation Window Help
Step 5:
Open the cross section into which the spacer will be copied.
Step 6:
Use **Draw/Locator** (Shift F2) to position the locator in the upper left cover of the cavity the spacer will be pasted into.
Step 7:
Use **Edit/Paste** (Ctrl-V) to paste the spacer polygons into the cavity

8. Use the **Window** menu to select the “half” 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.

Step 8:
Use the **Window** menu to select the "half" spacer file



Step 10:
Use **Draw/Locator** (Shift-F2) to position the locator in the upper right corener of the spacer

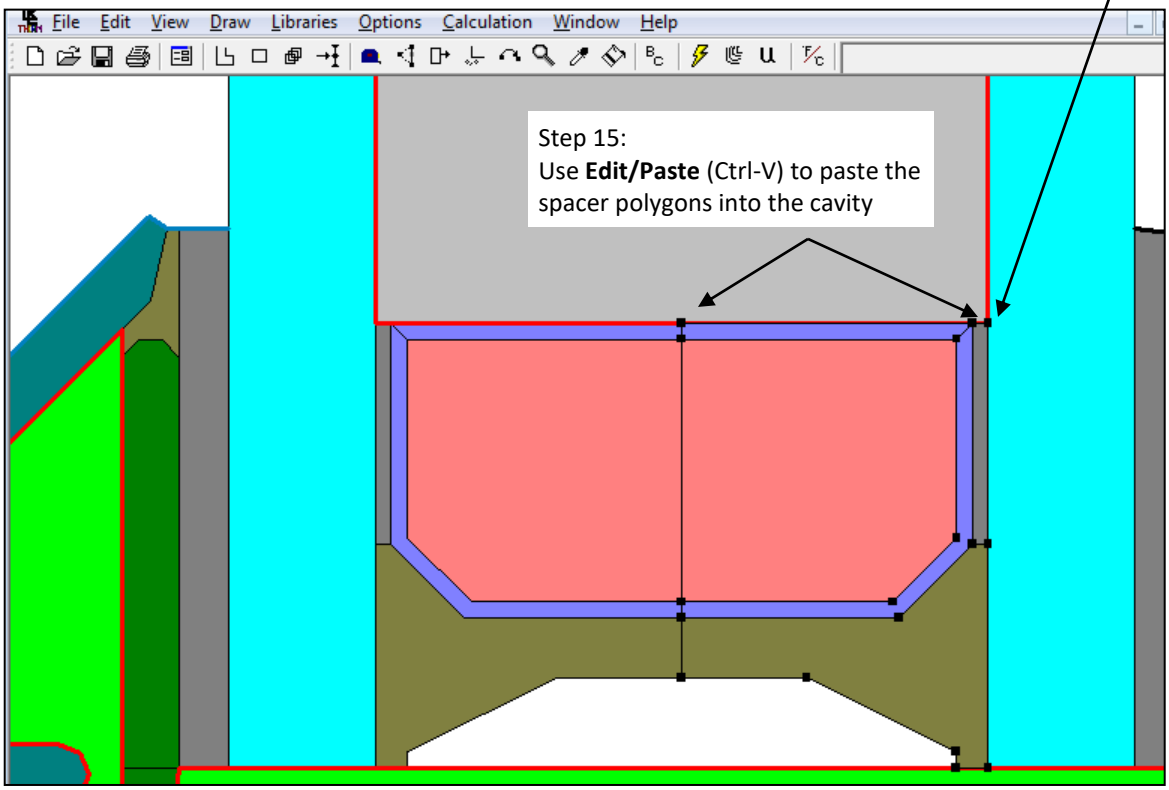
Step 11:
Use **Edit/Select All** to select all the spacer polygons

Step 12:
Use **Edit/Copy** to copy the selected polygons into memory

Step 9:
Use **Draw/Flip/Horizontal** to flip the spacer

Step 13:
Usethe **Window** menu to select the file to copy the spacer into

Step 14:
Use **Draw/Locator** to position the locator in the upper right corner of the cavity where the spacer will be copied



3.6. Specify Boundary Conditions

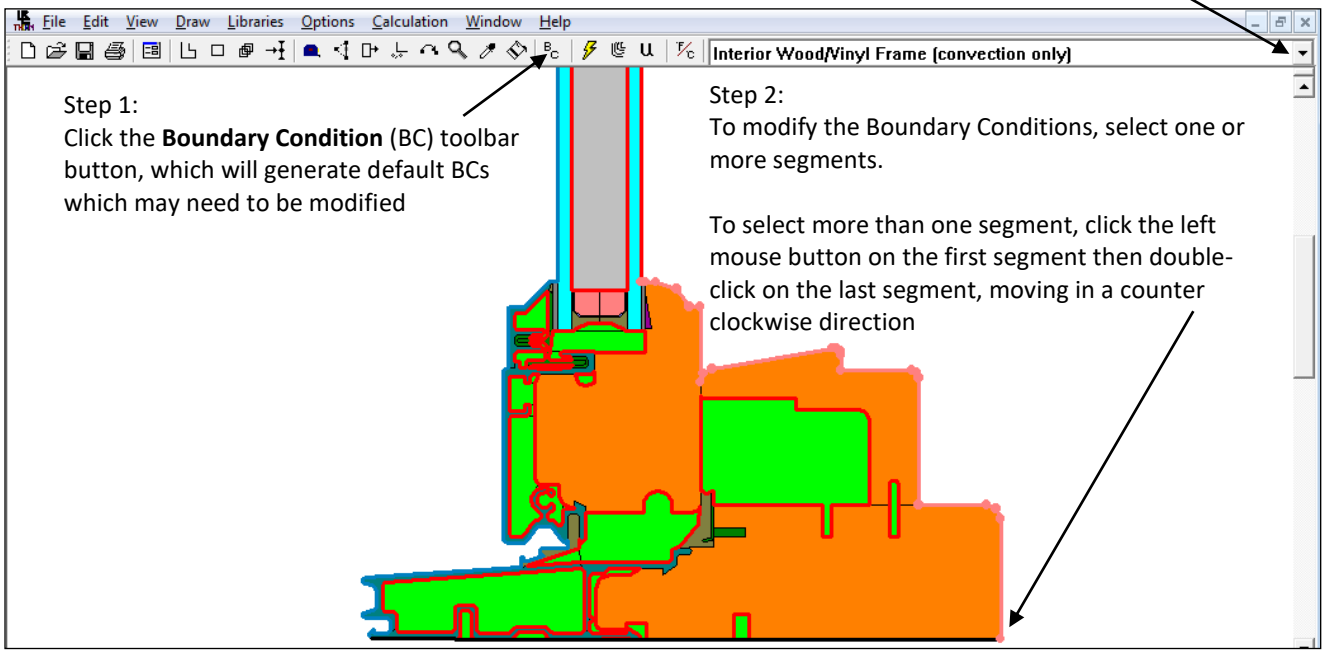
(See *Section 6, Defining Boundary Conditions*)

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).

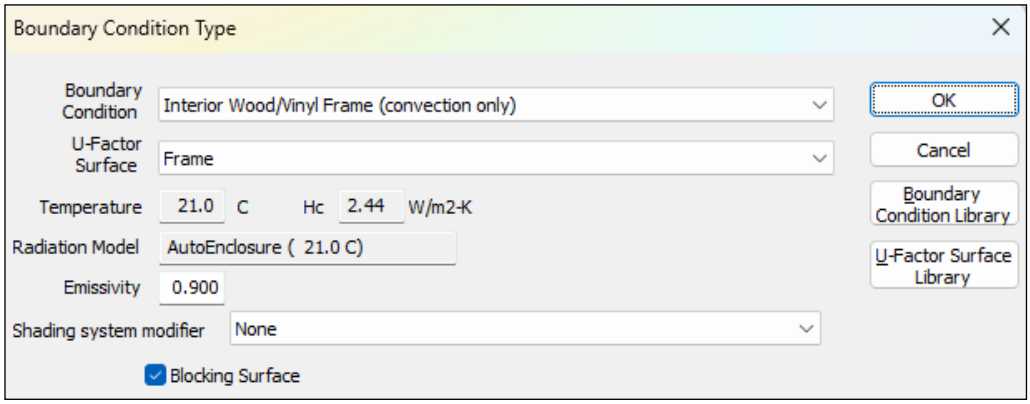
2. To change the default boundary condition, select a boundary segment (double-click the left mouse button on a boundary segment) or multiple contiguous boundary segments (hold the **Shift** key down, click the left mouse button on the first segment and the last segment, moving in a counter-clockwise 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.

Step 3a:
The Boundary Condition for the selected segments can be chosen from the pulldown list in the toolbar



Step 3b:
Double clicking on the last segment in Step 2 will open **the Boundary Condition Type** dialog box, where the Boundary Condition can be selected. Pressing the **Enter** keyboard button with Boundary Condition segments selected will also open this dialog box.

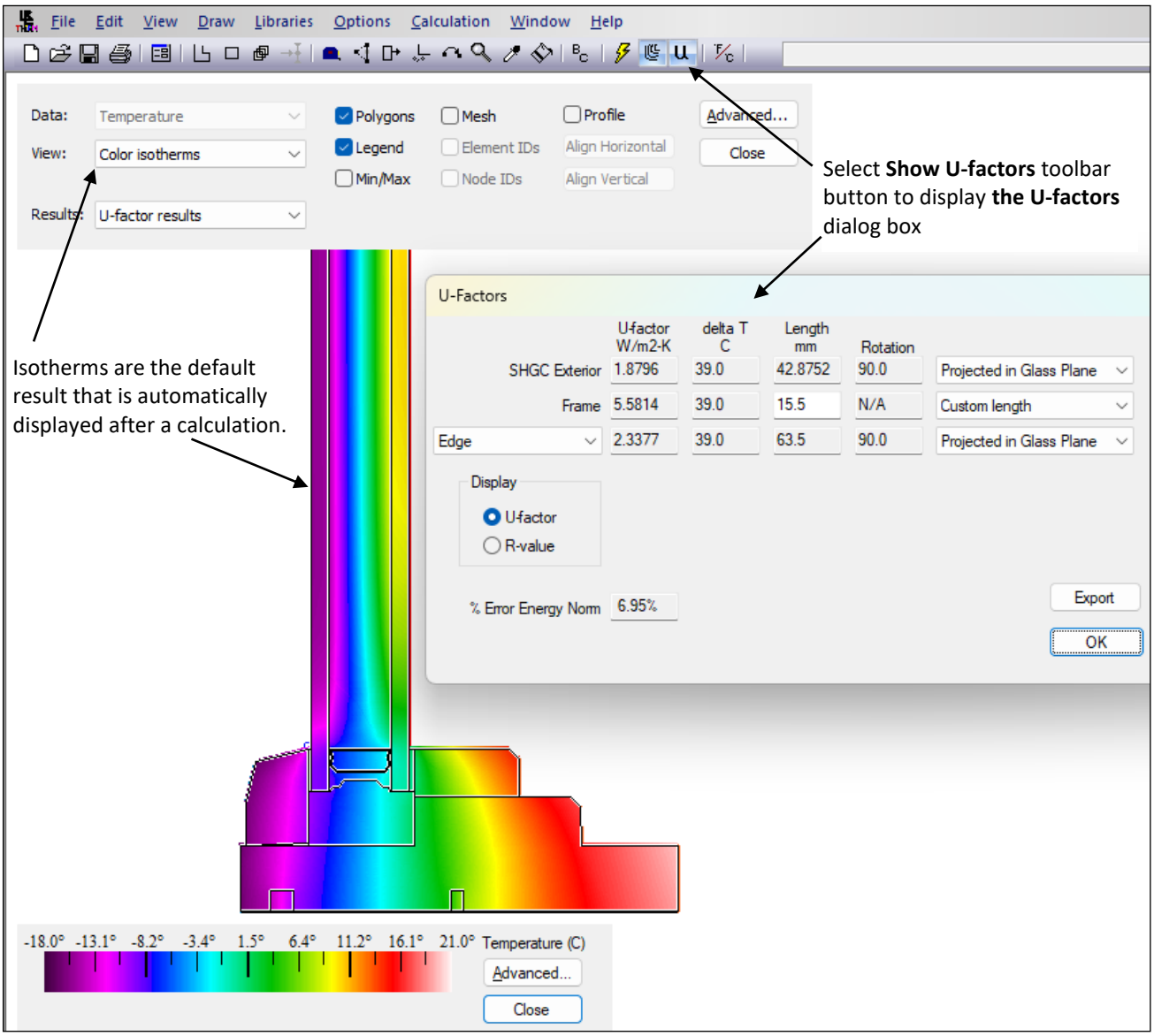


3.7. Run Simulation and View Results

(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 from the control bar when the results are displayed.



3.8. 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
	THERM File Properties	File / Properties	Ctrl+T
	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	Shift+F3
	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		F12 *(opens Options/Preferences)*
	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
	Select Materials or Boundary Conditions		
	Fill void	Draw / Fill Void	
	Draw Boundary Conditions	Draw / Boundary Conditions	F10

	Calculate	Calculation / Calculation	F9
	Show Results	Calculation / Show Results	
	Show U-factors	Calculation/ Show U-factors	Ctrl+F9
	Switch Unit Systems between SI and IP	Options / Switch Units	

3.9. Other Shortcuts

Description	**Menu/Choice**	**Shortcut**
Set Material	Libraries/ Set Material	F4
Material Library	Libraries/ Material Library	Shift-F4
Set Boundary Condition	Libraries/ Set Boundary Condition	F5
Boundary Condition Library	Libraries/ Boundary Conditions	Shift-F5
Glazing Systems	Libraries/ Glazing Systems	F6
Display Options	Calculation/ Display Options	Shift-F9
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
Open the Step Size dialog box		*Press the period button on your keyboard*

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.

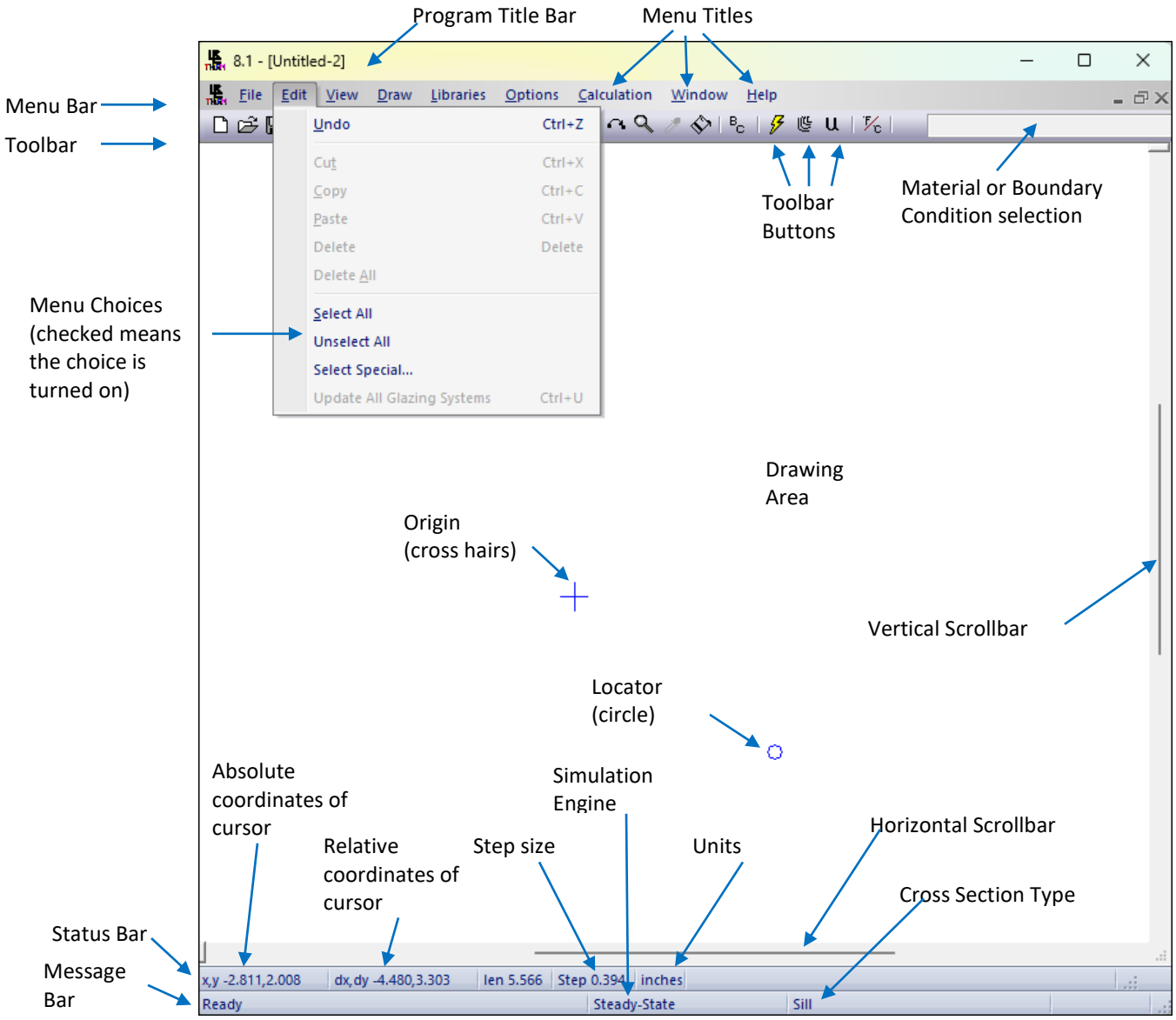


*Figure 4-1. THERM **Main Screen** components*

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.
- **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.
- **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. To change the location of the Origin, use the Draw menu, Set Origin choice.
- **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.
- **Status Bar:** The Status bar displays information about the coordinates of your drawing cursor, as well as the step size and the current units (SI or IP). 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. Status Bar

The **Status Bar** is the top row of information at the bottom of the **Main Screen**. This bar displays information about the cursor position, as well as the Step Size and the Units (SI or IP).

The **x,y** information gives the absolute **x** and **y** coordinates of the drawing cursor (the "cross" that appears when you click the Draw / Polygon or Draw / Rectangle options) when you click relative to the origin (the small cross hairs). The **dx, dy** information gives the **x** and **y** coordinates of the drawing cursor relative to where you most recently began drawing; if you place the drawing 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.

The Units information indicates whether you are in SI (International) or IP (Inch Pound) units.

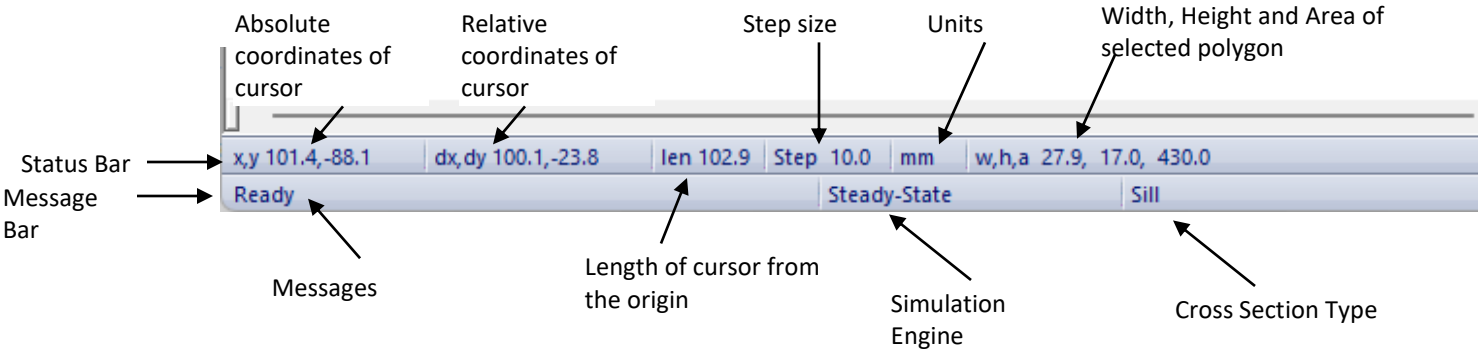


*Figure 4-2. **Status Bar** notations*

4.2. Main Menu

Figure 4.3 shows THERM's main menu choices.

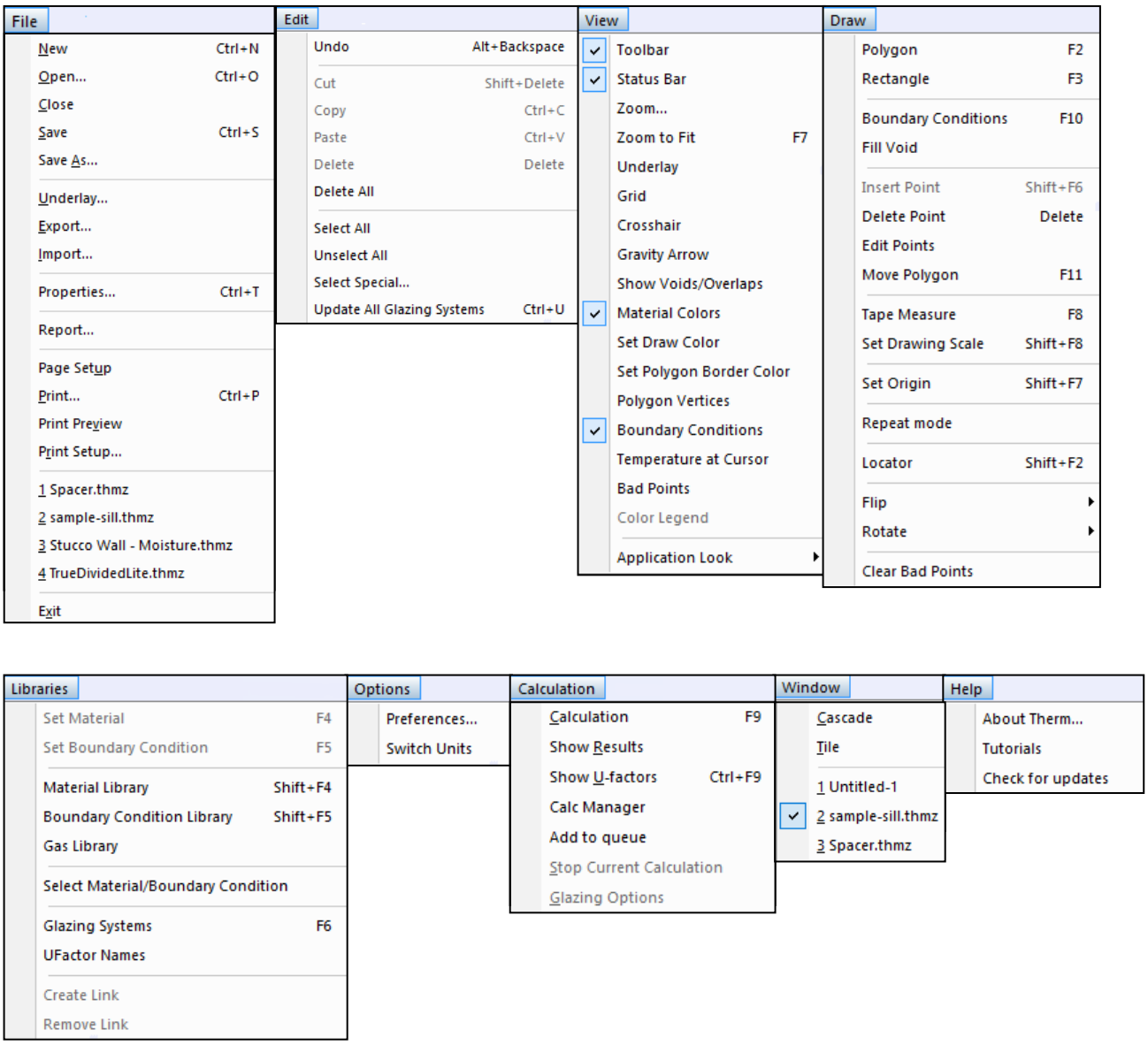


*Figure 4-3. THERM **Main Menu** choices*

See the Toolbar Reference table in *Chapter 3, Getting Started*, to see the keyboard shortcuts for various actions.

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.

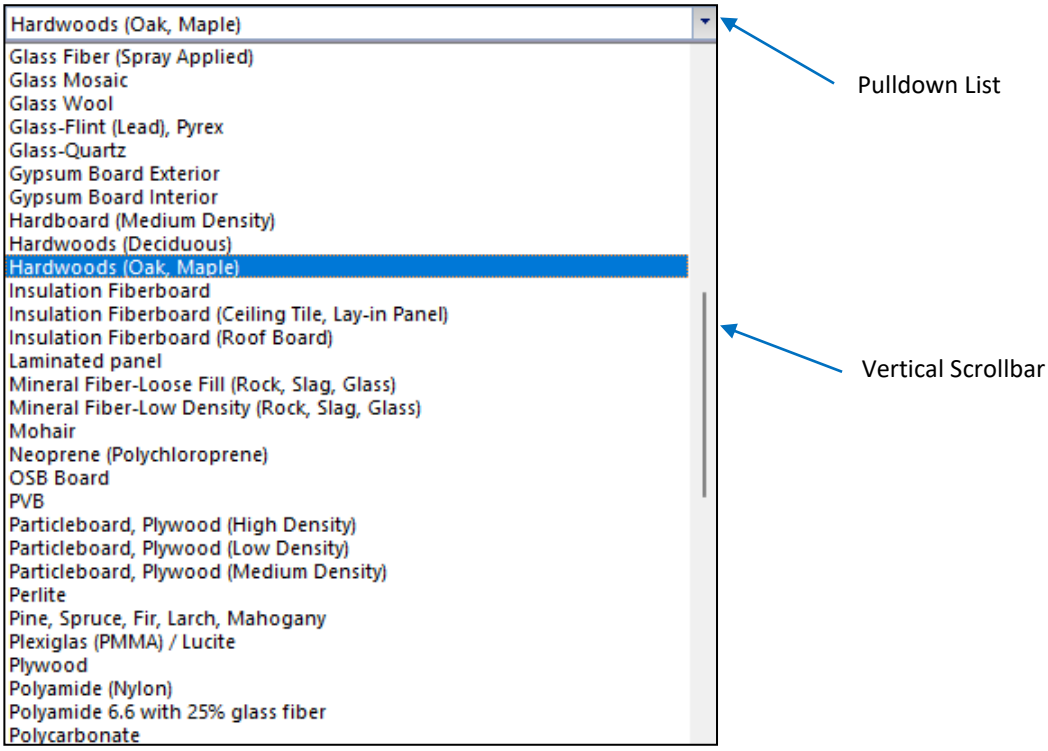


Figure 4-4. Example of a pull-down list

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** toolbar button. THERM will automatically add a THMZ extension to the file name you type. You can also open THM files from earlier THERM versions, and they will automatically be converted to the new THMZ format.

Sample files, located in the Samples subdirectory of the **C:\Users\Public\LBNL\THERM<version#>**, are included in the installation. You can open these, or any other THERM file, using the **File/Open** menu choice, or the **Open** toolbar button. A list of files with a THMZ 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** 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 THMZ extension to the name.

THERM allows you to have multiple THMZ files open at one time. You can see a list of all opened files using the **Window** menu. You can close each open THMZ file independently, using the **File/Close** menu option. You can close all the files, and still have the THERM program active.

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.

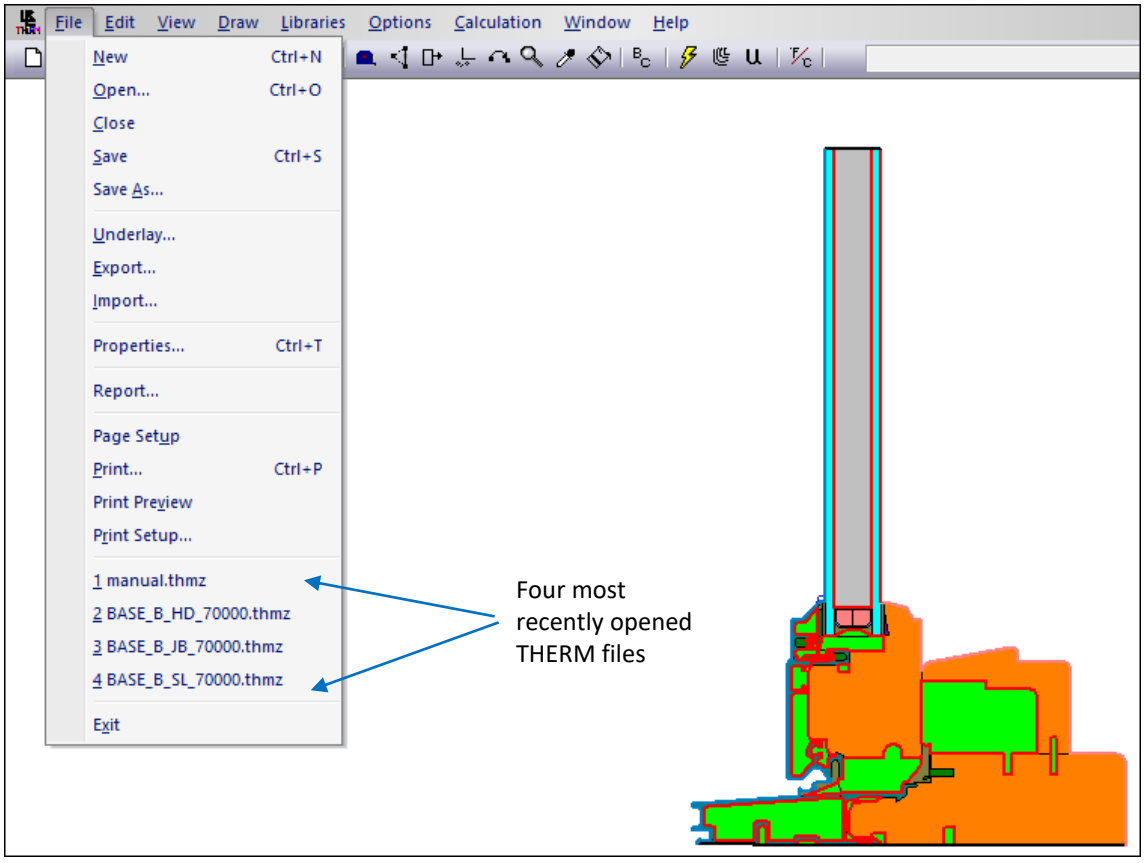


*Figure 4-5. Last four open THERM files are shown at the end of the **File** menu.*

4.2.1. Options / Preferences

The **Options/Preferences** menu opens a dialog box with several tabs which control various areas of the program. Below is a description of each tab and the settings on that tab.

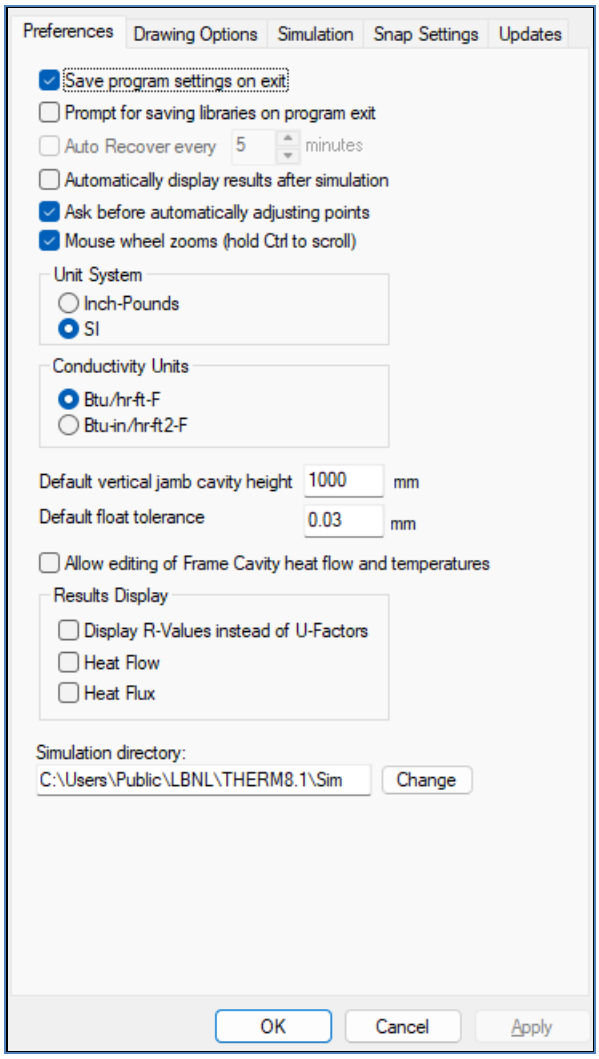


Figure 4-6. The option of having THERM ask before automatically adjusting points can be turned on and off.

Preferences Tab

Below is a description of the fields in the Preferences tab of the Options menu.

Save program settings on exit

Any changes made to the settings in Preferences will be automatically saved when exiting the program.

Default: Checked

Prompt for saving libraries on program exit

If this option is checked, the program will prompt the user whether changes made to libraries should be saved. If this is not checked, the changes will not be saved in the libraries, but will be saved in the THERM file they were applied to.

Default: unchecked

Auto Recovery every "X" minutes

Not currently functional

Automatically display results after simulation

If this option is checked, the program will display results (the default being isotherms) after a simulation.

Default: Checked

Ask before automatically adjusting points

If this option is checked, the program will put up a dialog box asking if the user wants the program to automatically adjust points when the points are within the "checking tolerance" of 0.1 mm. There may be circumstances where it is desired to keep those points, and this option allows the user to not have the program automatically merge them into one point.

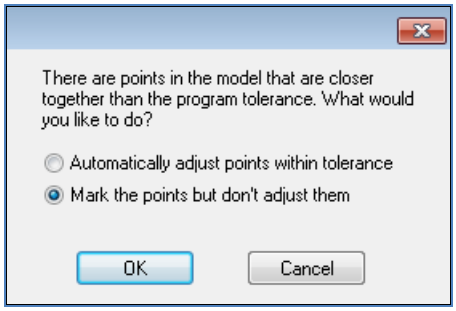


Figure 4-7. The program will ask whether to automatically adjust points, or leave them, if this option is checked.

Mouse Wheel Zooms (hold Ctrl to scroll)

This turns on the ability to control and zoom and scrolling using the computer mouse.

Unit System

There are two choices for the unit system:

- Inch-Pounds
- SI

Conductivity Units

There are two choices for the unit system:

- Btu/hr-ft-F
- Btu-in/hr-ft2-F

Default vertical jamb cavity height

This is the vertical height of cavities when the THERM file properties (set in File/Properties/Model Exposure tab) are set to any vertical model, such as Jamb, Vertical Divider, or Vertical Meeting Rail.

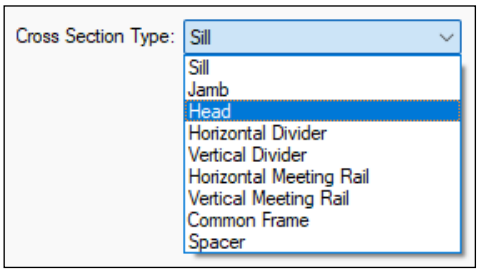


Figure 4-8. The vertical cross section types will use this default cavity height during the simulations

Default float tolerance

Float Tolerance is the geometric snap distance THERM uses when comparing or merging points and edges in a cross-section. Two points or two edge endpoints separated by a distance less than or equal to Float Tolerance are treated as coincident; an edge whose length is below Float Tolerance is treated as zero-length and discarded.

This values is used internally to recognize coincident points and well-formed boundaries; if the tolerance is too coarse, THERM displays the message:

There is an error in the model. Error estimator could not perform correct evaluation *and results are incorrect. Try reducing Float Tolerance in Options → Preferences → Therm File Options."*

Lowering Float Tolerance (e.g., 0.001 mm) preserves fine geometric detail. Vertices that you intentionally placed close together stay distinct. The cost is that geometry imported from CAD or pasted from another file is more likely to leave near-but-not-exactly-coincident vertices, which can produce slivers, unshared edges between adjoining polygons, and meshing problems.

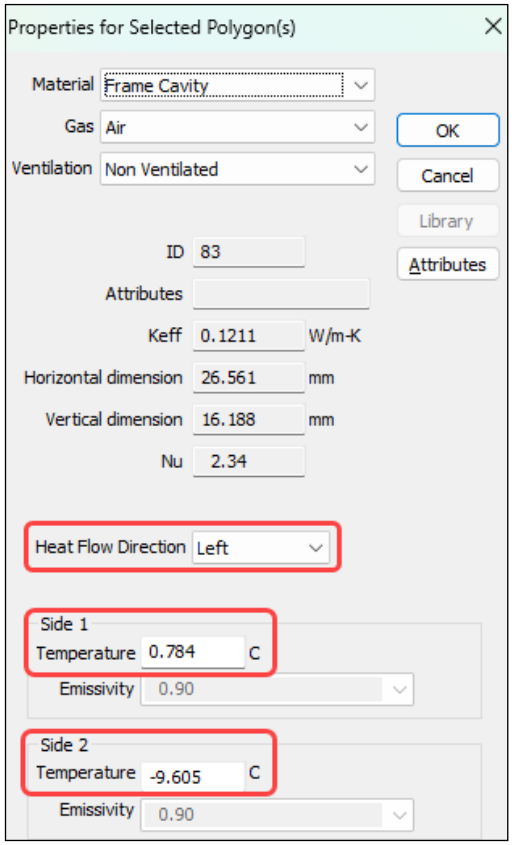
Raising Float Tolerance (e.g., 0.1 mm or higher) is more aggressive about merging. Sloppy geometry is automatically cleaned up so that adjoining polygons share edges, but real detail finer than the tolerance is collapsed away. If the tolerance becomes a noticeable fraction of the smallest feature in the cross-section, the error estimator may report a failure and the simulation results should not be trusted; the prompt above asks the user to reduce the value.

Default: 0.03 mm: a balance chosen so that typical drafting precision in millimeters cleans up correctly while sub-millimeter features (slat thicknesses, films, gaskets) are preserved. Most users do not need to change it.

Units: mm (SI), inches (IP).

Allow editing of Frame Cavity heat flow and temperatures

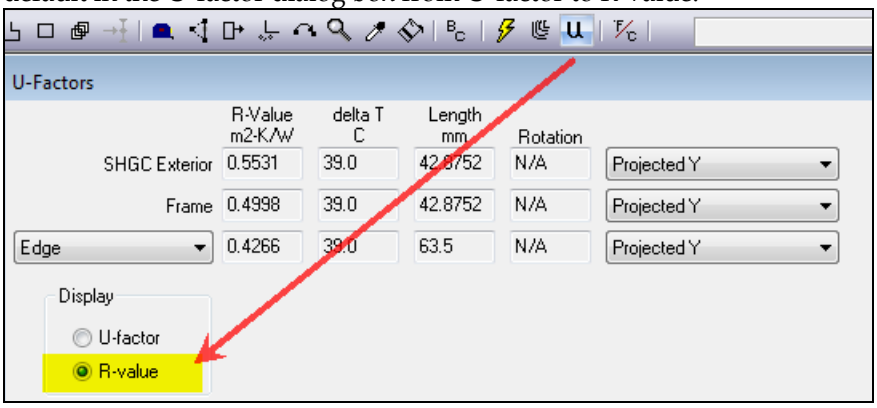
If checked, this option allowed editing the Heat Flow Direction and Side 1 and Side 2 Temperatures for frame cavities.



Results Display

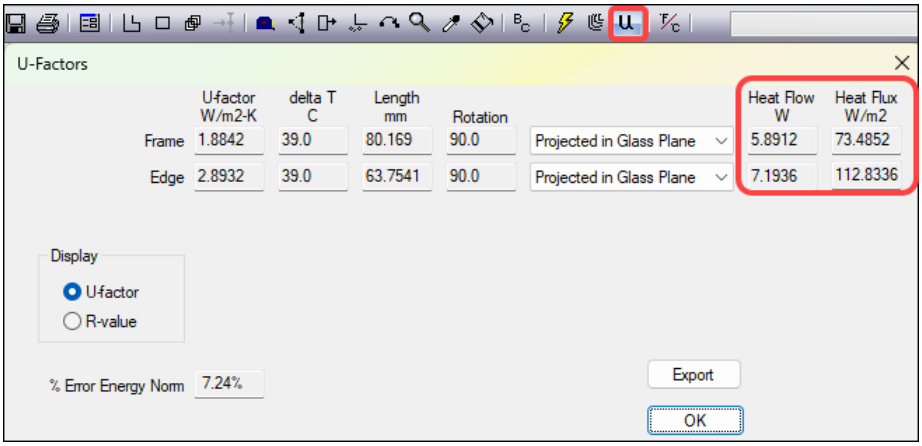
There are multiple options for displaying the results. All of these choices can be checked.

Display R-values instead of U-factors: Selecting this option changes the default in the U-factor dialog box from U-factor to R-value.



Heat Flow: Checking Heat Flow will show Heat Flow results in the U-factors dialog box.

Heat Flux: Checking Heat Flux will show Heat Flux results in the U-factors dialog box

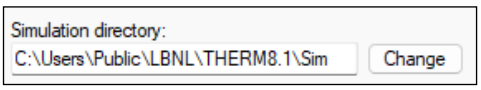


Simulation Directory

This is the directory where the program will write the intermediate simulation files. Click the Change button to change the directory.

Default: The "Sim" subdirectory where the program is installed by default:

C:\Users\Public\LBNL\THERM8.1\Sim



Simulation Tab

See Section 7.2.2 for a detailed description of the values in this tab

Drawing Options Tab

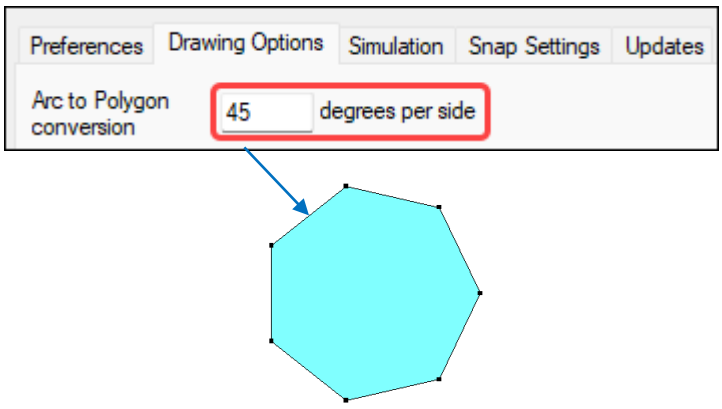
Below is a description of the fields in the Preferences tab of the Options menu.

Arc to Polygon conversion

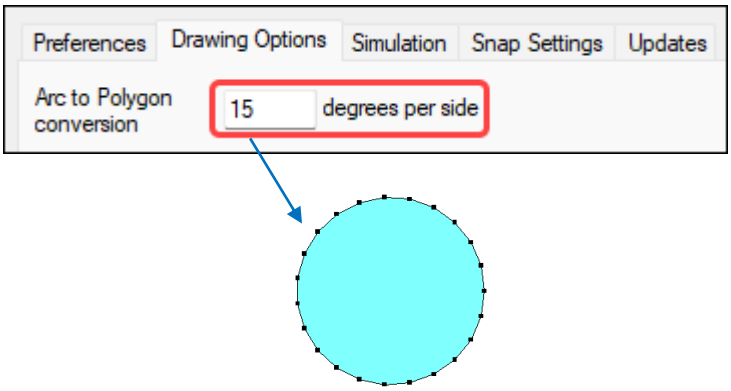
The number of polygon sides that will be created when autoconverting arcs and circles from a DXF file.

Default: 15 degrees

Case 1: Arc to polygon conversion = 45



Case 2: Arc to polygon conversion = 15



Stay in draw mode after drawing

When using the Draw Rectangle or Draw Polygon feature, after having drawn a completed polygon, the program will stay in "draw" mode if this is checked.

Default: unchecked

Always check for overlapping polygons

During the drawing process, the program will put up a message if a polygon segment is going to overlap with an existing polygon.

Default: checked

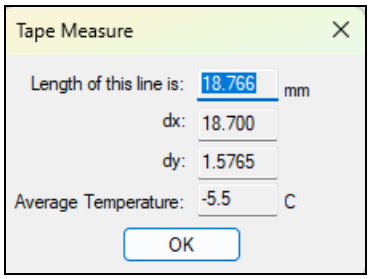
Snap preview

Not currently functional.

Tape Measure Average Temperature

When the tape measure is used, if this box is checked, and the model has simulation results, the tape measure display will include an average temperature calculated over the measured length.

Default: unchecked



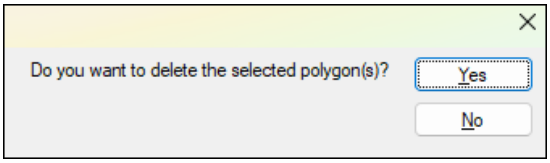
Allow editing of IG polygons

Checking this box allows editing of a glazing system imported from WINDOW. In general, it is not recommended to edit the glazing system, but there are circumstances where it is necessary in order to properly define the model.

Default: unchecked

Prompt before deleting polygons

Checking this box will enable a dialog box to appear, allowing the user to verify that the polygon is to be deleted.



Default: checked

Drawing Size

Height: Height of the overall drawing area.

Default: 500 mm (SI); 19.685 inches (IP)

Width: Width of the overall drawing area.

Default: 500 mm (SI); 19.685 inches (IP)

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:

- Draw polygons and rectangles to represent the cross-section geometry (for more details, see *5.4. Drawing Using a Mouse* and *5.5. Drawing Using the Keyboard)*. When modeling windows, we recommend that you import a glazing system from the WINDOW program (see 5.10. Inserting a Glazing System), but you can also represent the glazing systems with individual rectangles (recommended only for advanced users).
- Assign materials to polygons and rectangles as you draw (recommended) or after the drawing is completed (see *5.11. Assigning Materials After Drawing Polygons*). The THERM default material library (material.lib) is used to model solids or frame cavities and to define new materials as necessary for developing the model. This default material library was developed for modeling fenestration systems. A library called "Material-Arch.lib" is also distributed with the program and contains materials more appropriate for modeling building components such as walls, floors and ceiling/roof assemblies. See *5.12. Material Library* for a more detailed explanation of creating new records in the Material Library.
- Add, delete, or move polygons and points as necessary (see *5.6. Editing Polygons*).
- Insert a glazing system created in WINDOW for modeling fenestration products. Glazing systems from WINDOW are imported into THERM with all material properties, including the effective conductivity of the glazing cavity and the boundary conditions defined in the WINDOW Environmental Conditions (see 5.10. Inserting a Glazing System). Use of the Condensation Index Model in THERM requires that the glazing system be imported from WINDOW.

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. *5.2.3. DXF File Conversion* provides suggestions on how to make a DXF file for use with THERM. A DXF filter function (see *5.2.2. Select DXF or Bitmap File*) in THERM can be used to restrict

the entities (points, polylines, and arcs) 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. *5.3.3. Snap Settings* 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.

The steps below, used to import a DXF file to use as an underlay for your THERM drawing, are explained in more detail in the following sections.

5.2.1. Select Underlay from the File menu

Using the **File** menu, select the **Underlay** choice. This will open the **Underlay** dialog box, discussed in the next section.

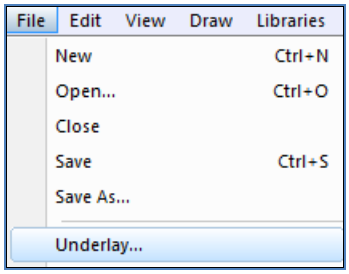


*Figure 5-1. To import a DXF file, pick the **Underlay** option from the **File** menu.*

5.2.2. Select DXF or Bitmap File

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

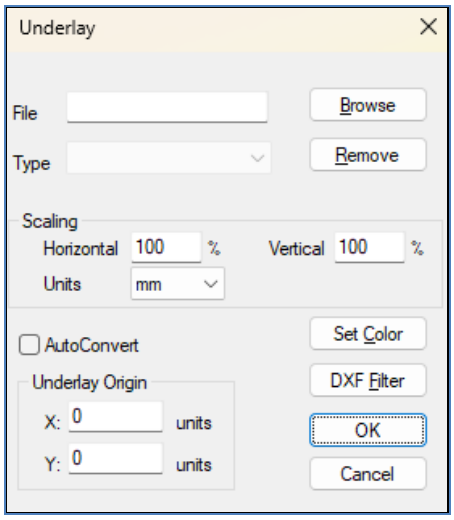


*Figure 5-2. The **Underlay** dialog box*

Click on the **Browse** button to see all files in your current working directory with a file extension of *.DXF or *.BMP. You can browse to any directory where you have the underlay files.

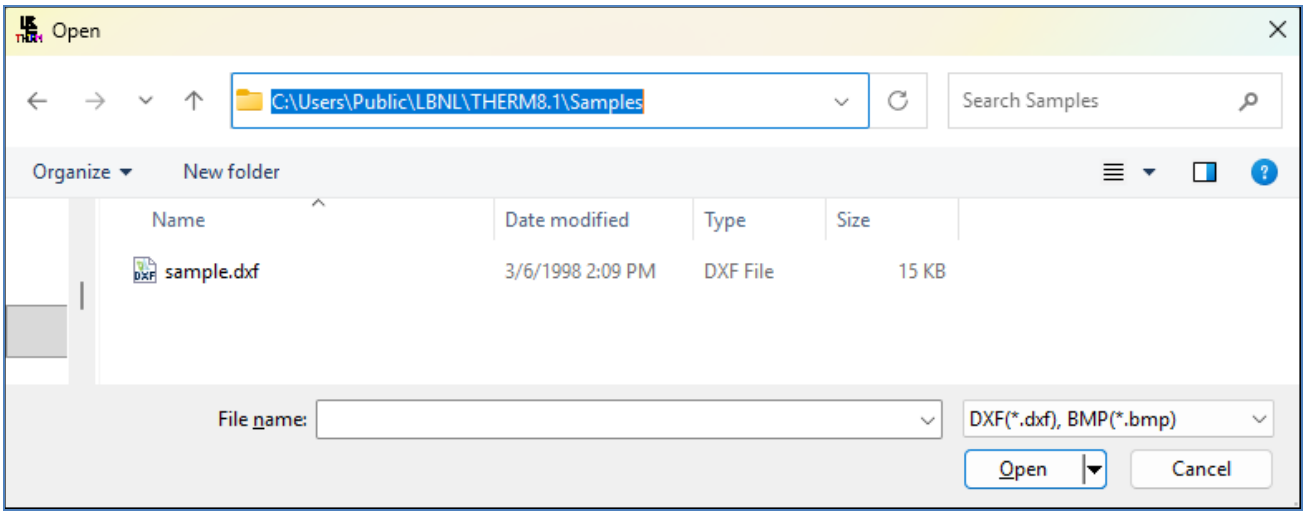


*Figure 5-3. Click on the **Browse** button to get to the **Open** dialog box.*

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

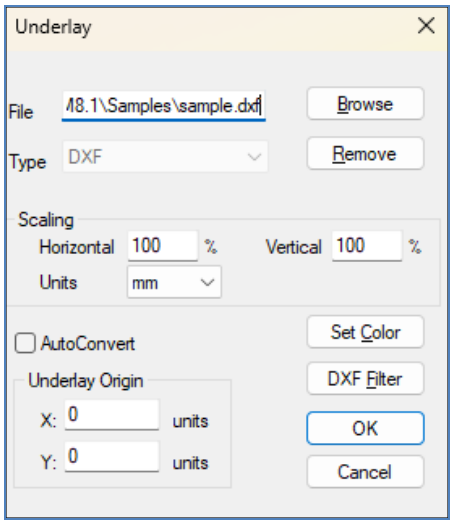


*Figure 5-4. Select the DXF file using the **Underlay** dialog box.*

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.

Horizontal & Vertical — 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.

Units — Make sure the Units option is set to match the units system that the DXF file was drawn in. The default unit is inches (IP), so if you are using an underlay created in mm (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 *5.3.5. Voids and Overlaps*) after using the Autoconvert feature to check for overlapping polygons. See *5.2.3. DXF File Conversion* 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 degrees 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, as this introduces more complexity to the model than is needed and can result in meshing errors during the beginning of the simulation.

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.

Buttons

Browse This button allows you to browse to and select the DXF file to use as an underlay.

Remove This button allows you to remove an underlay after it has been imported into the THERM file.

Set Color This button allows you to set the color of the underlay as it will appear in your drawing area. 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.

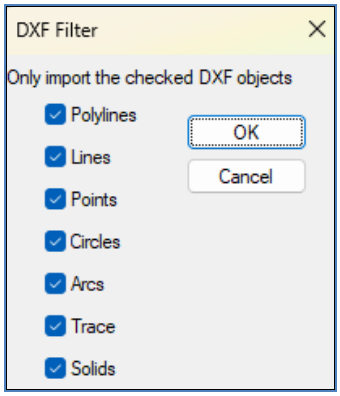


*Figure 5-5. The **DXF Filter** dialog box*

OK Clicking this button will cause the selected options to be implemented, such as importing the DXF file, autoconverting, etc.

Cancel This button will close the Underlay dialog box without implementing any of the specified options.

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 (the cursor will "snap" to points on the underlay using the spacebar; in Figure 5-7, some of the polygons already exist.)

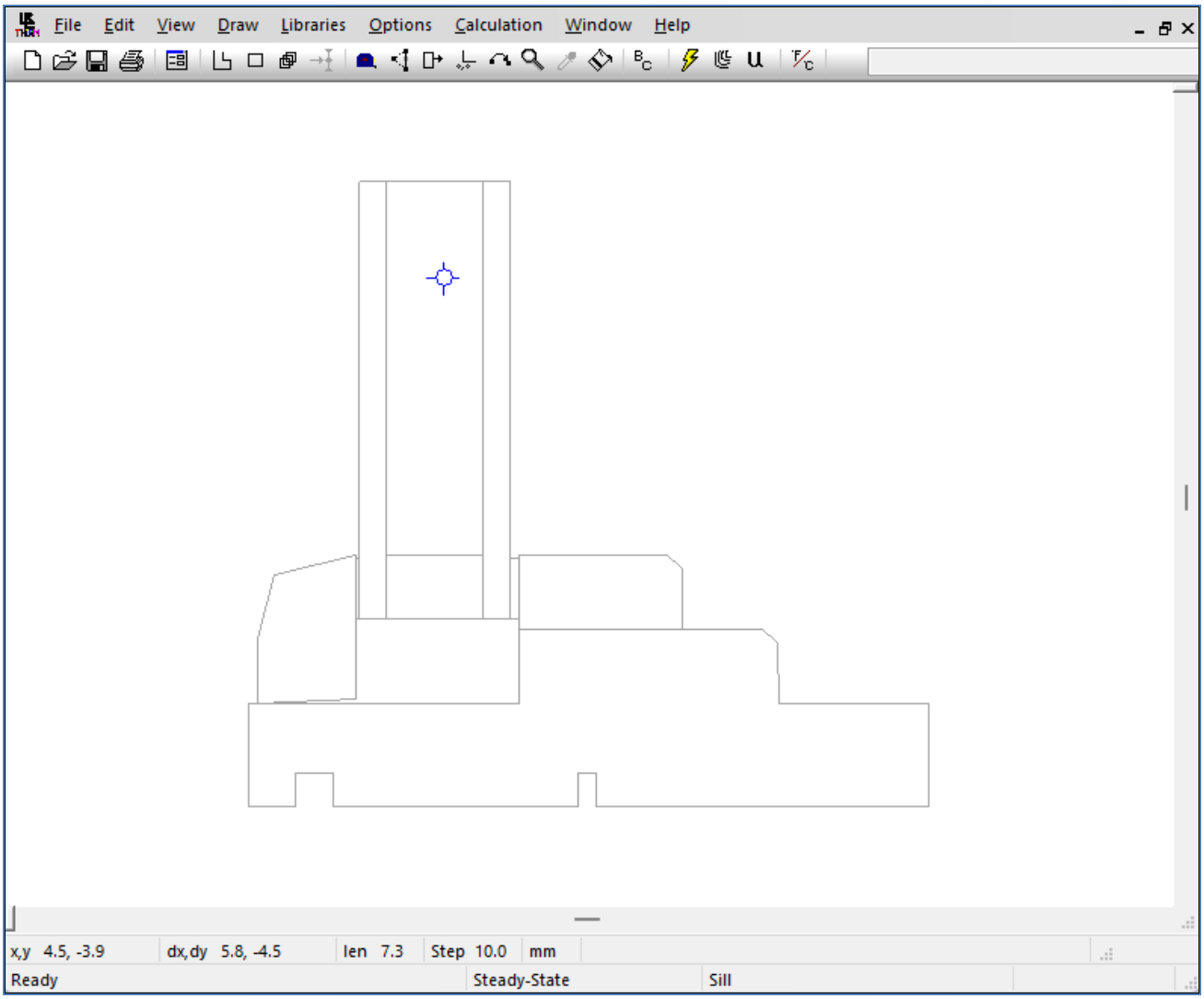


*Figure 5-6. A DXF file imported as an underlay with **Autoconvert** turned off, so no polygons are drawn*

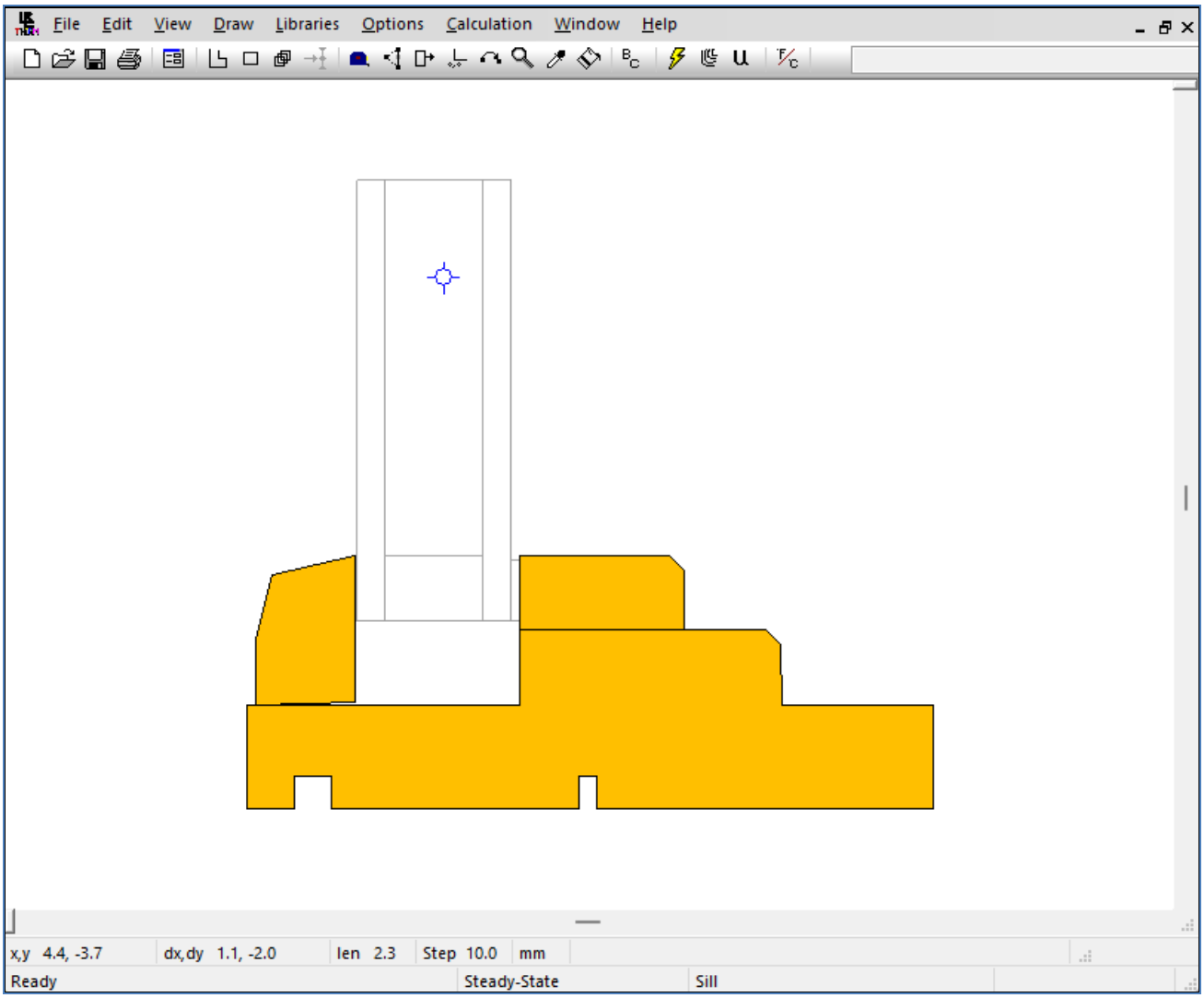


*Figure 5-7. A DXF file imported as an underlay with **Autoconvert** turned on, so closed polylines are converted into polygons*

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

In addition to the information below, consult the THERM Knowledge Base (https://windows.lbl.gov/knowledge-base-therm} for updated information about importing DXF files into THERM.

Note: *The current version of THERM only accepts DXF files from AutoCAD in the AutoCAD12 or AutoCAD13 DXF format files.*

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 3 mm (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 *5.5. Drawing Using the Keyboard* for a discussion of numeric cursor positioning).

5.3.2. Drawing Options

The **Options/Preferences** menu choice, **Drawing Options** tab contains THERM drawing settings.

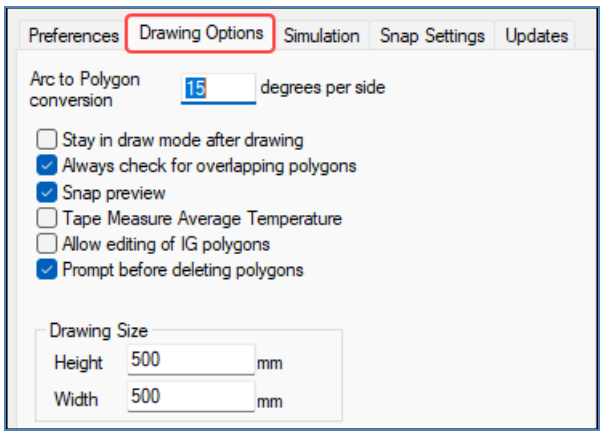


Figure 5-8. Setting the drawing preference options

Below is a description of the settings in this tab.

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. If the arc contains too many segments, the program may have difficulty creating the finite element mesh when starting the simulation, and you may get a "mesh error". Simplifying curves generally will not significantly affect the results of the simulation.

Stay in Draw mode after drawing

This option turns the Repeat Mode on by default. This means that the toolbar option you select after activating Repeat Mode, such as draw polygon or draw rectangle, will remain active so that you can stay in that drawing mode without having to reselect it each time you want to draw.

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 an 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 also check for overlaps at simulation time, at which time you must fix the overlaps before the simulation will complete; 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.

Currently not functional.

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 results.

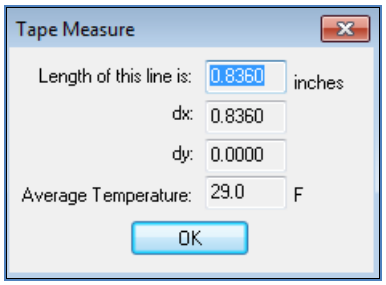


Figure 5-9. When the tape measure feature is used, the dialog box will also show the average temperature across the measured line

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 as well as change the geometry of the glazing system cavity. This is useful for cases where the glass layers in a glazing system may start at different heights in the model, such as in a storm window. 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.

Note: There is only one level of Undo; if all the polygons in a model are deleted at once, the program cannot "undo" that deletion.

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, which decreases the sticky distance. (See *5.3.1. Sticky Distance* for more details).

Use the **Snap Settings** tab under the **Options/Preferences** menu to control the "Snap" features in THERM.

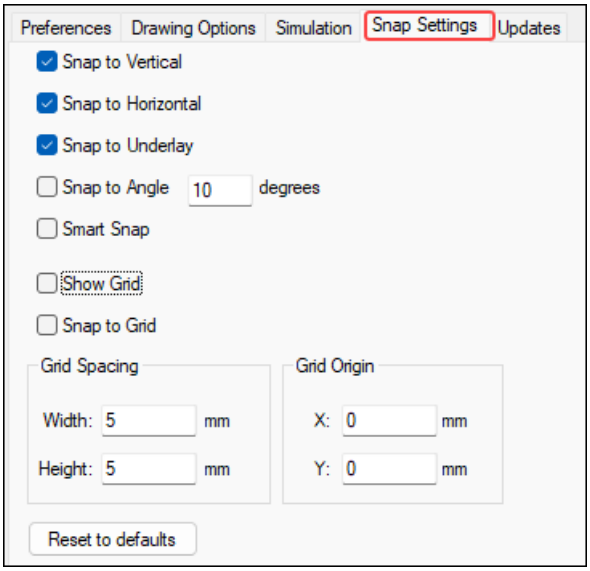


*Figure 5-10. Setting the **Snap Settings** options, accessed from the **Options/Therm File Options** menu*

Here is a description of the Snap Setting options.

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.

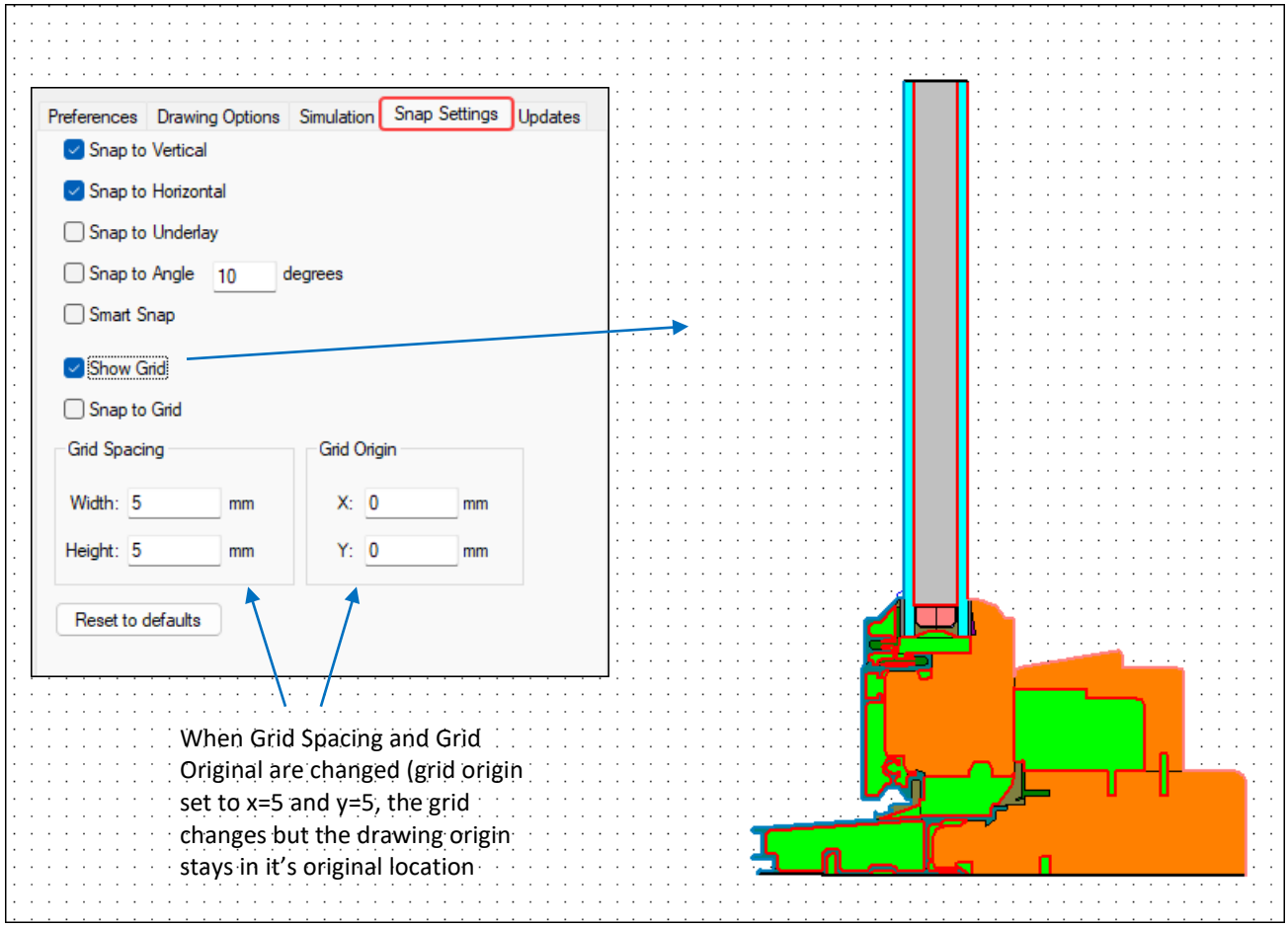


Figure 5-11. Changing the grid origin relative to the drawing origin

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 (F7 or 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%.

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 choice in the View menu called **Show Voids/Overlaps**, which 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 *5.2.3. DXF File Conversion* for more details. Figure 5-12 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 when generating boundary conditions. See Chapter 7 for more details. Also, if there are three polygons on top of each other, the top polygon will be shown as blue. However, if there are only two polygons on top of each other, the top polygone will be shown as white, which makes this type of error difficult to detect.

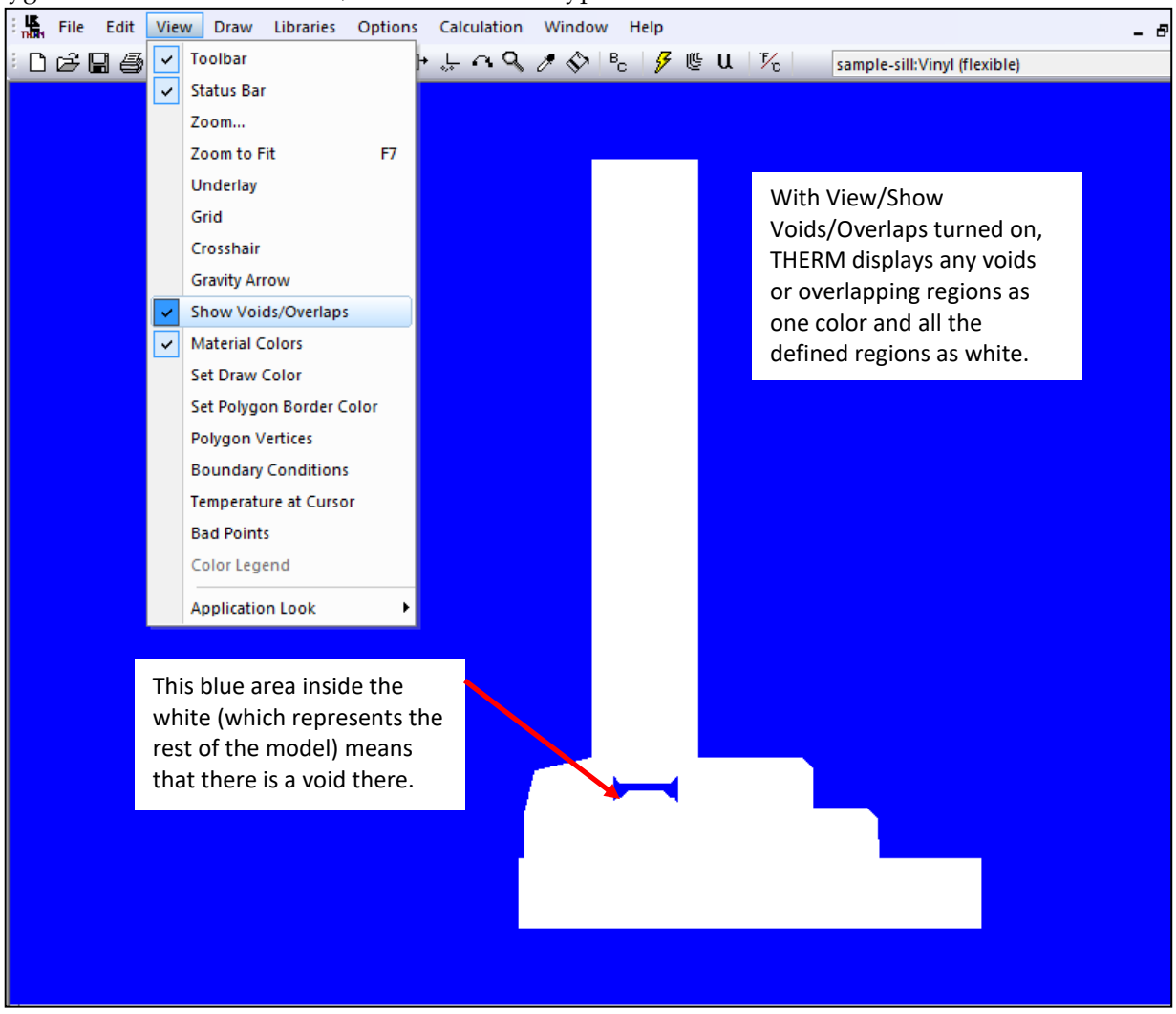


*Figure 5-12. Use the **Draw/ShowVoids/Overlaps** menu to view voids.*

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") or to correct the scale of a file based on an imported DXF file that may be too large or small. 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; these properties are not corrected when the glazing system is rescaled.

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

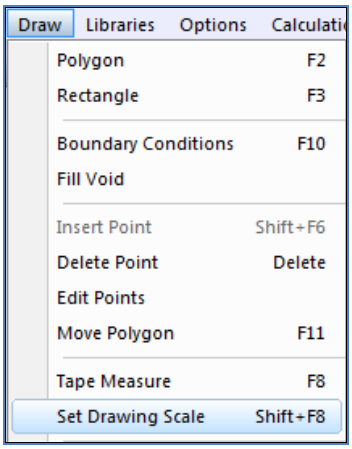


*Figure 5-13. Select **Set Drawing Scale** from the **Draw** menu.*

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. This will then set the scale for the entire model.

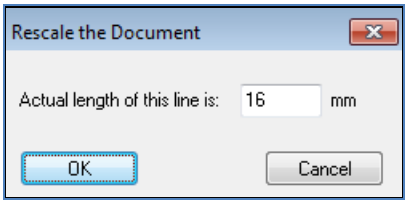


Figure 5-14. After measuring the distance, type in the dimension to which you want that distance to be set.

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 (*5.5. Drawing Using the Keyboard*) for how 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, an underlay is traced and many polygons are drawn to represent the model.

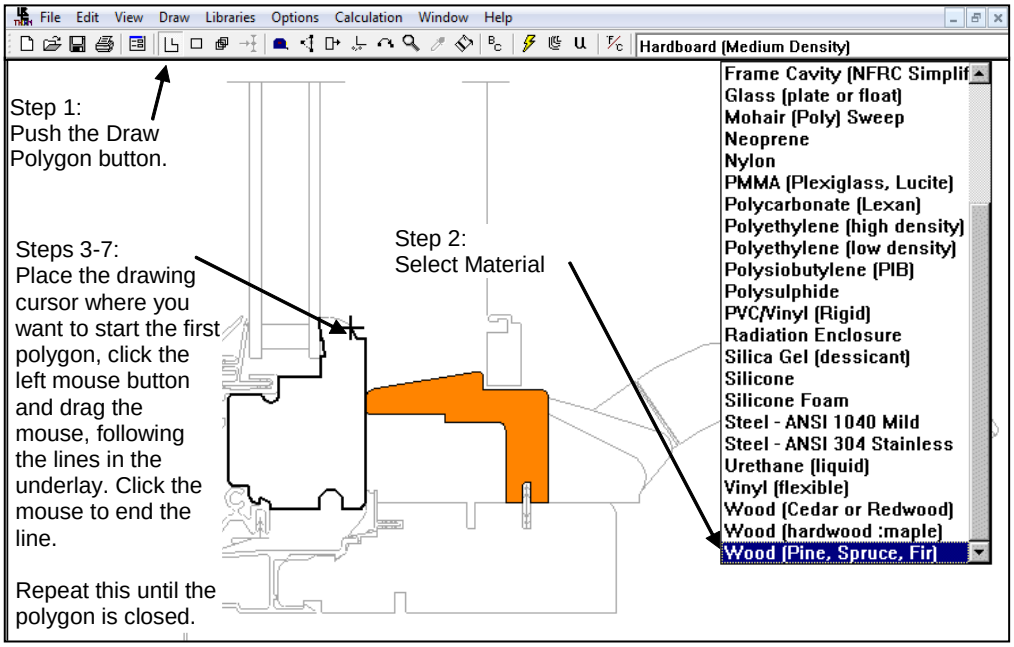


Figure 5-15. Steps for drawing cross-section geometry

- To draw the first polygon, press the **Draw Polygon** toolbar button or use the **Draw/Polygon** menu item.



- Pick the material for the polygon from the material list in the right-hand section of the toolbar.
- Place the drawing cursor where you want to start the first polygon.
- 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). You can also use the spacebar to force the program to snap to the nearest point within the sticky distance of the cursor.

- 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.
- 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.
- 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.

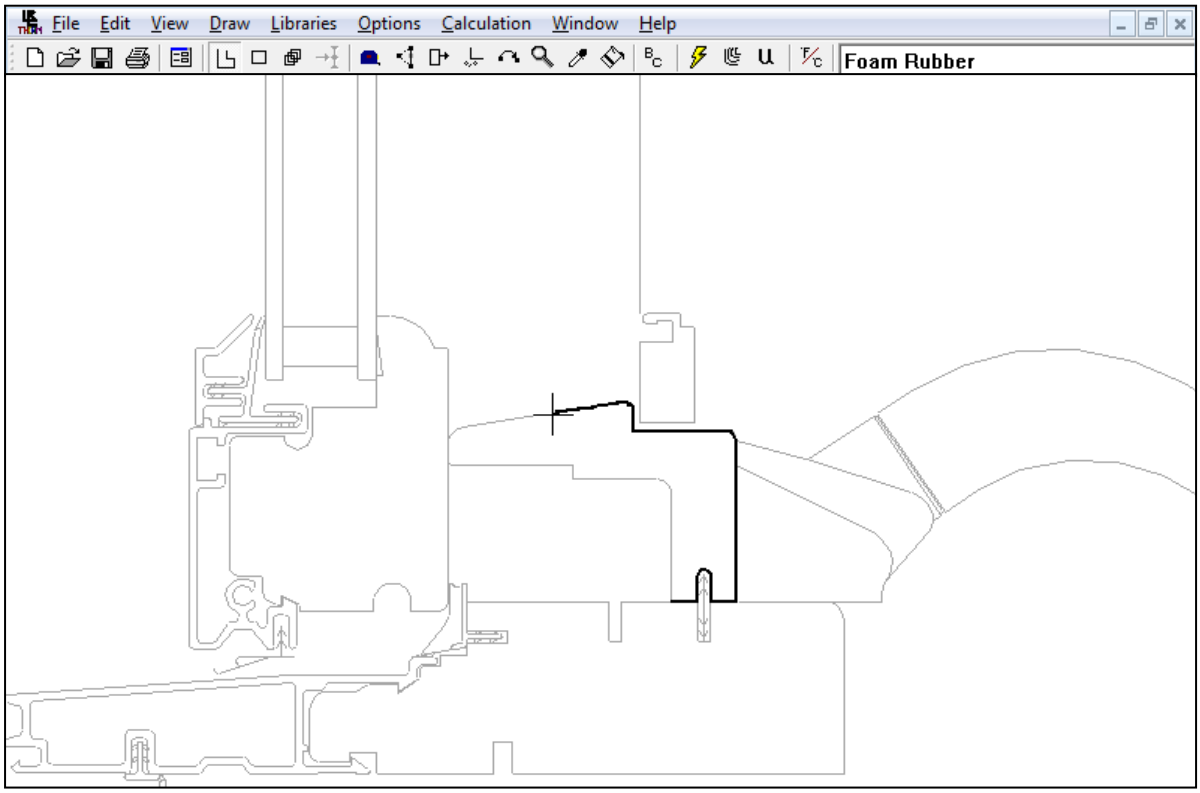


Figure 5-16. Drawing a polygon by tracing an underlay element

If you find that the cursor is "snapping" lines to points that you don't want to snap to, 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 *5.3.5. Voids and Overlaps*, and *6.3. Error Detection in THERM* for more information). The "snap" feature (see *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.

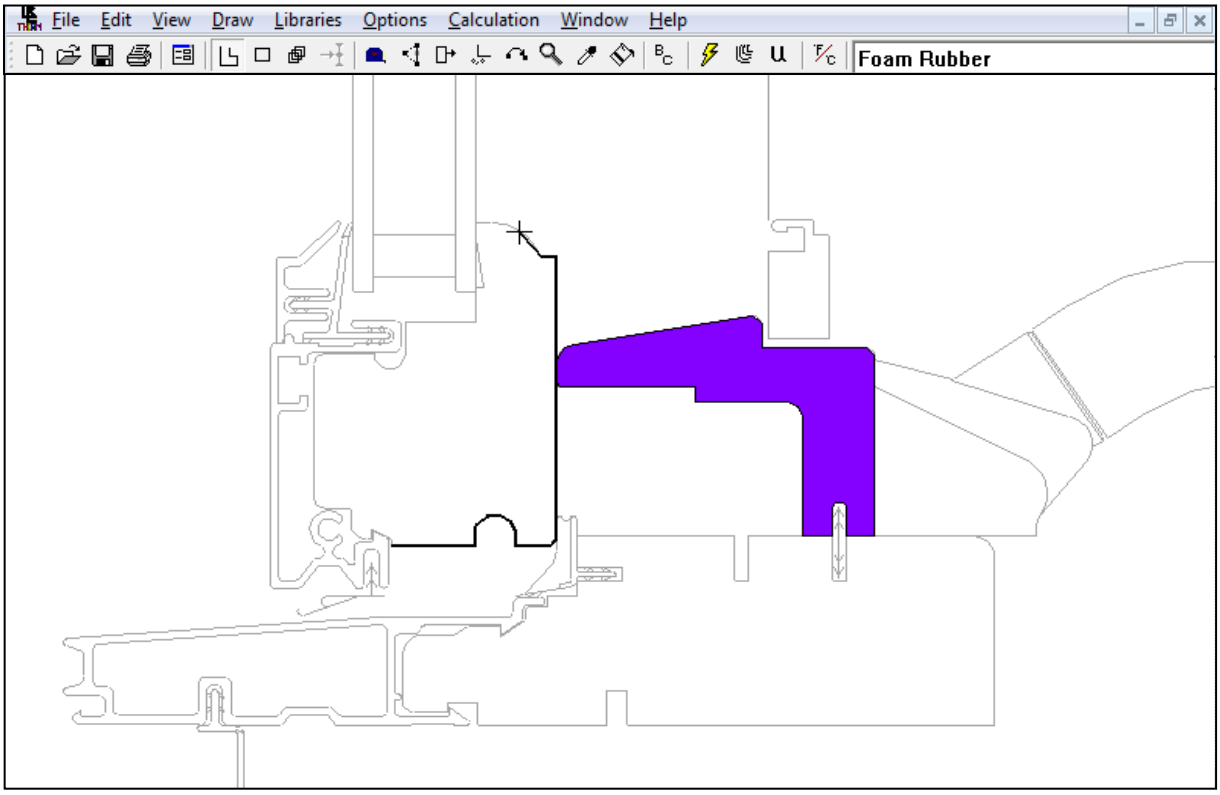


Figure 5-17. The first polygon is complete; start the next polygon.

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:** THERM cannot model 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 affecting the thermal performance of the cross section.
- When breaking up extrusions you should avoid introducing detail into the cross section as follows:
    - Make the breaks at vertices that already exist in the drawing.
    - Avoid creating acute angles.
    - Avoid creating very small or thin polygons.
    - Avoid creating triangles (3 sided polygons).

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

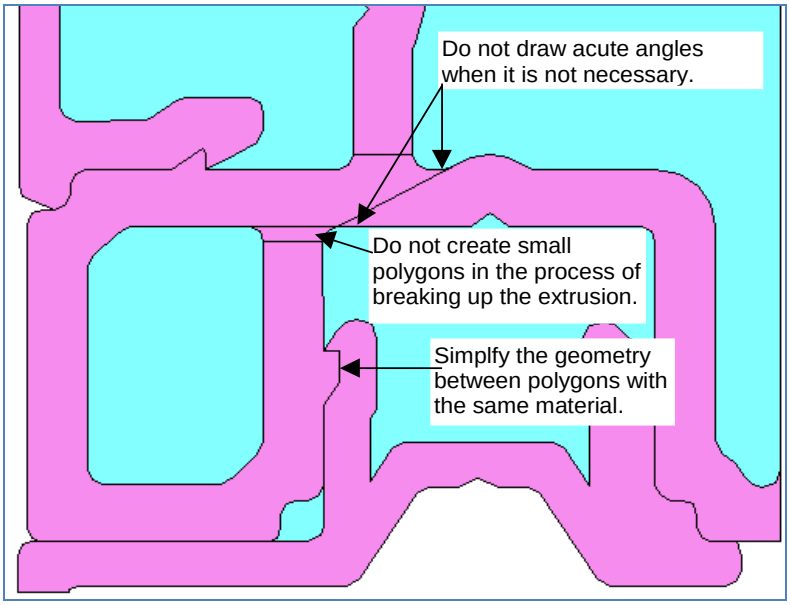


Figure 5-18. This cross section illustrates drawing mistakes when breaking up an extrusion.

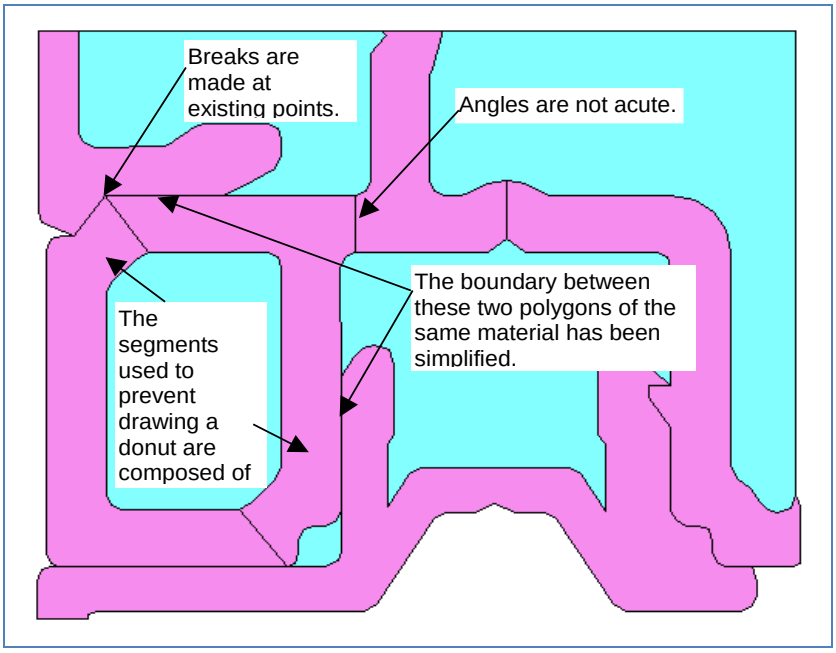


Figure 5-19. This cross section illustrates a better way to draw the extrusion geometry in Figure 5-17.

- **Arcs and curves:** The default "Arc to polygon conversion" value of 15, found in the **Options/Preferences/Drawing Options** tab, converts arcs into line segments every 15 degrees. This level of detail is appropriate for curves with a radius greater than 3 mm. 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:

- Type **Alt-D:** Shows the **Draw** menu.
- 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).
- 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.
- Press **Enter**: Press the **Enter** key to start drawing (the equivalent of clicking the mouse button).
- 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.)

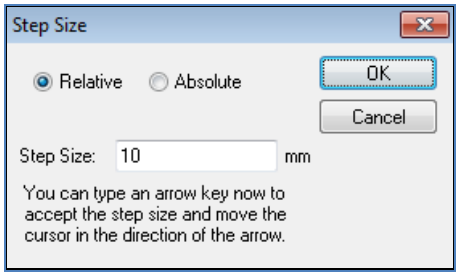


*Figure 5-20. Type a number to bring up the **Step Size** dialog box.*

- 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.
- 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.
- Type the next dimension (a number).
- Press the appropriate arrow key.

- Press **Enter** to commit the line.
- Continue drawing lines until you have only one line remaining to complete the polygon.
- 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 *Chapter 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**.

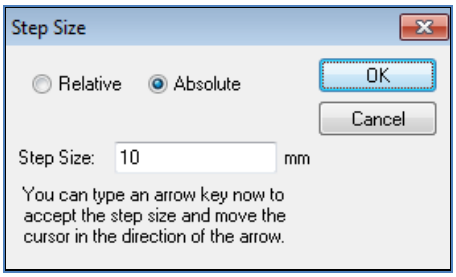


*Figure 5-21. Type the step value to bring up the **Step Size** dialog box, and click on the **Absolute** button.*

- **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.

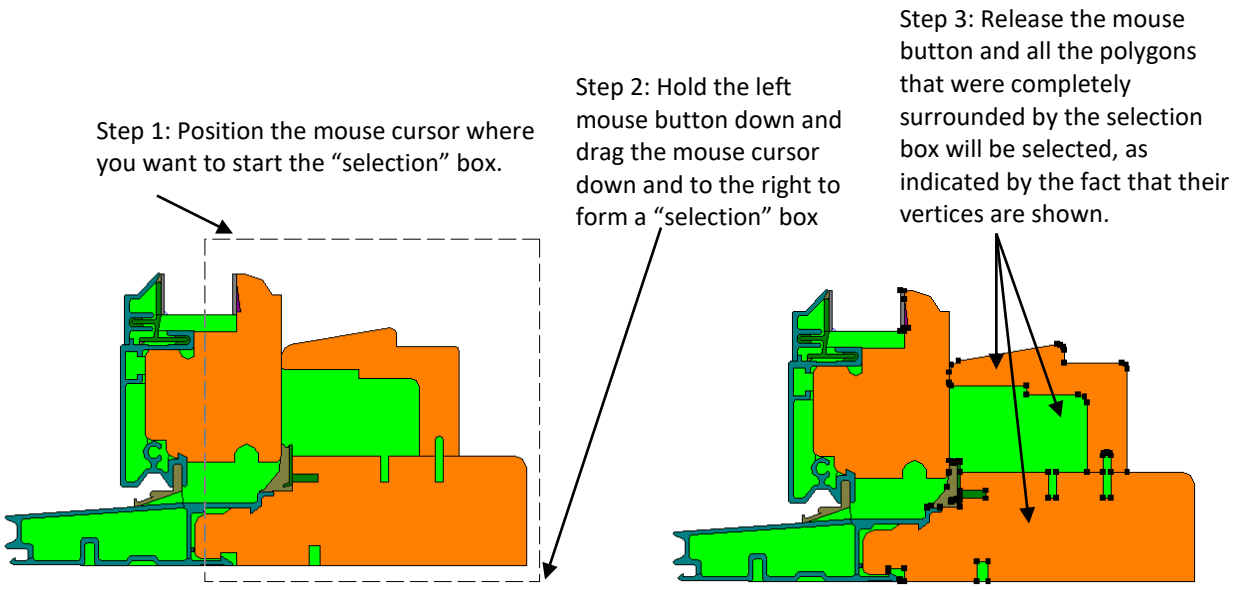


Figure 5-22. Selecting polygons using the selection box

- **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.

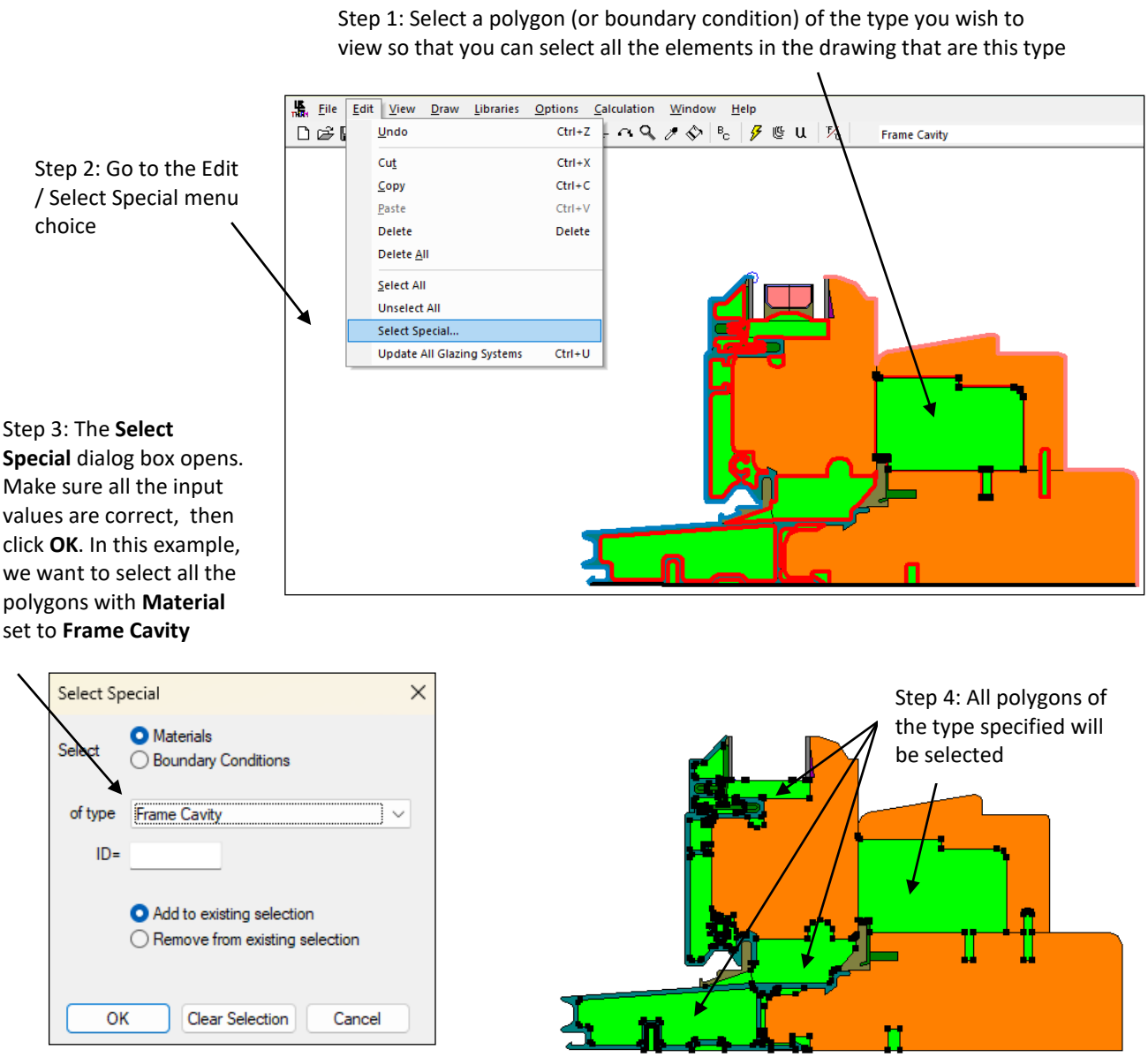


*Figure 5-23. Using **Select Special** for selecting polygons*

5.6.2. Moving a Polygon

To move a polygon:

- Click the mouse on the **Move** toolbar button, press the **F11** keyboard button, or use the **Draw/Move** menu choice.



- 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.

- 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.
- Select all of the polygons to be moved, using any of the methods discussed in *5.6.1. Selecting a Polygon*.
- Cut (**Ctrl-X**) the selected polygons.
- 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.)
- Paste (**Ctrl-V**) the selected polygons. If no overlaps are created, the polygons will be pasted in the new location.

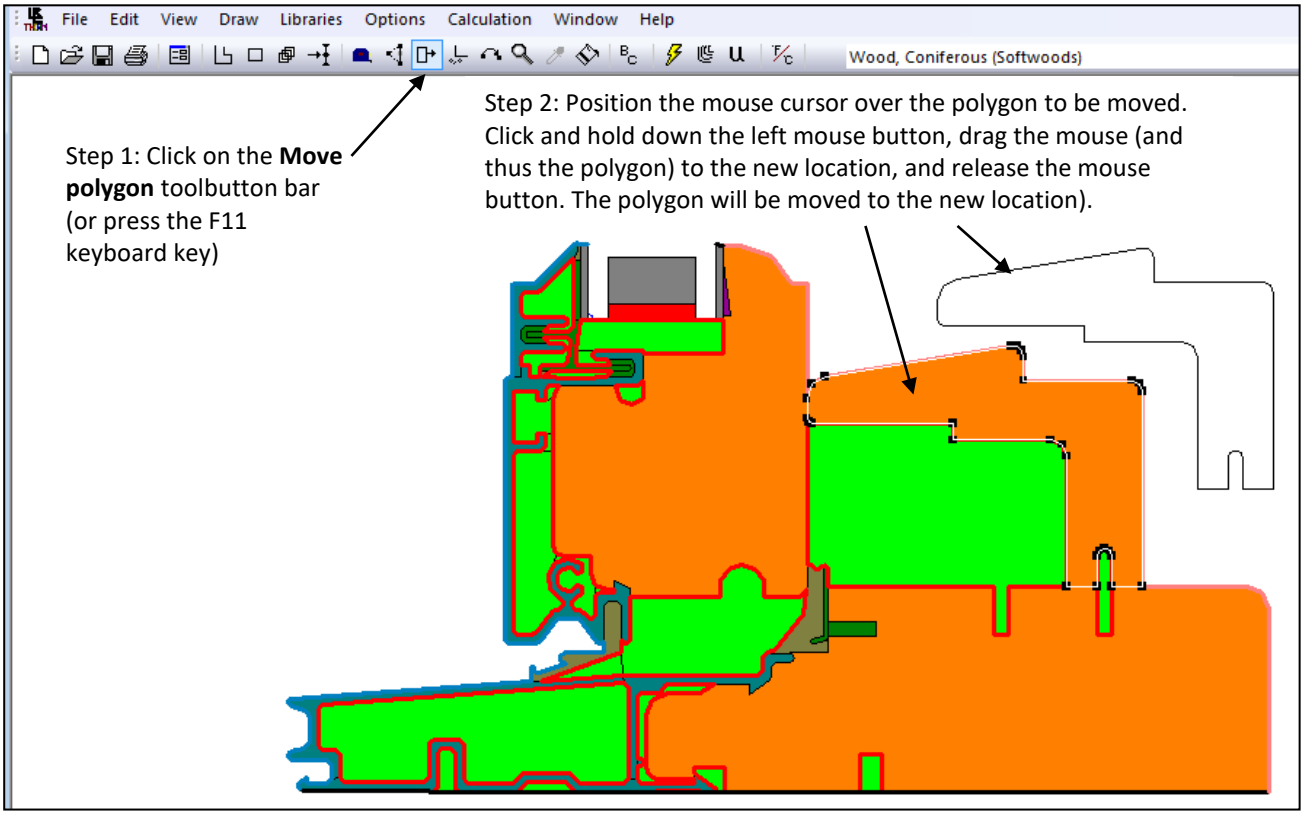


Figure 5-24. Moving a polygon

5.6.3. Deleting a Polygon

To delete a polygon:

- Select the polygon (click the left mouse button while the cursor is inside the polygon) that you want to delete.
- Press the **Delete** key on your keyboard.

- 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.)
- 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:

- Click the mouse on the **Edit Points** toolbar button or use the **Draw/Edit Points** menu choice.
- Select the polygon (click the left mouse button while the cursor is positioned inside the polygon) so that the vertices are displayed.
- Move the cursor to the point to be moved.
- A double-arrow cursor is displayed when you are within sticky distance of the point.
- Use the **Step Size** and **Arrow** keys to move the point, or click and drag the point with the left mouse button.
- 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.

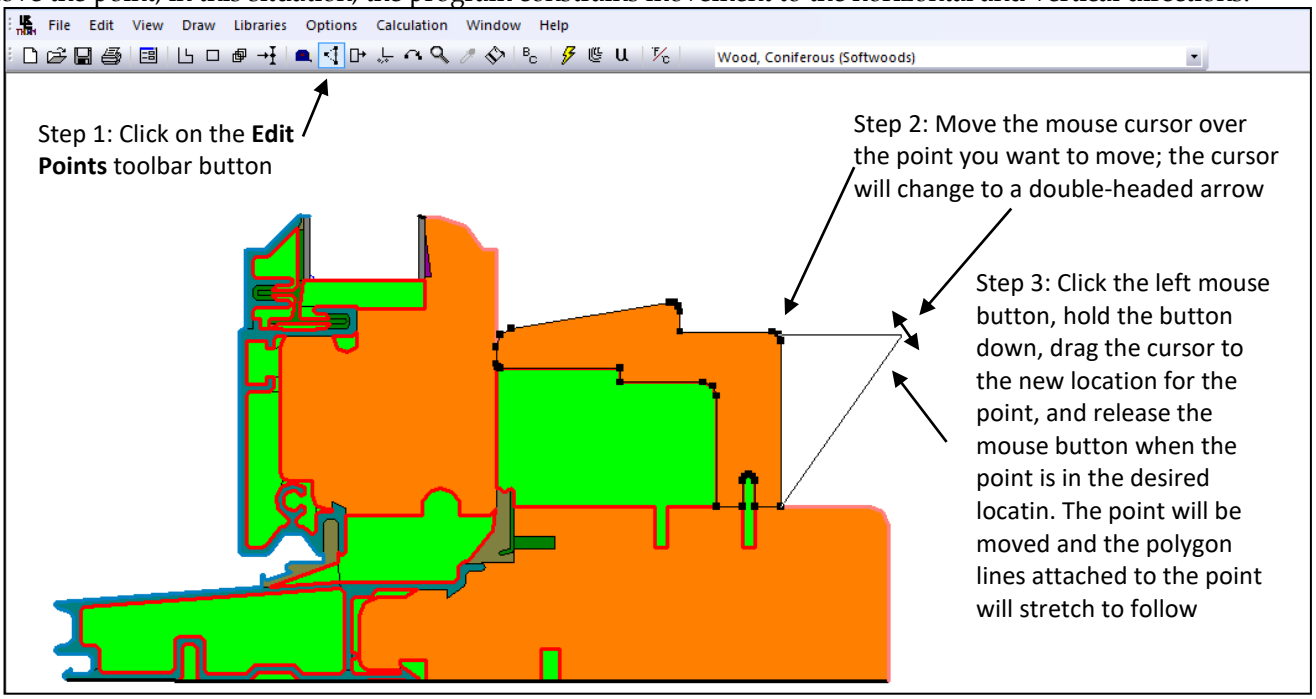


Figure 5-25. Moving a point on a polygon

5.6.5. Moving a Line

To move a line on a polygon:

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

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

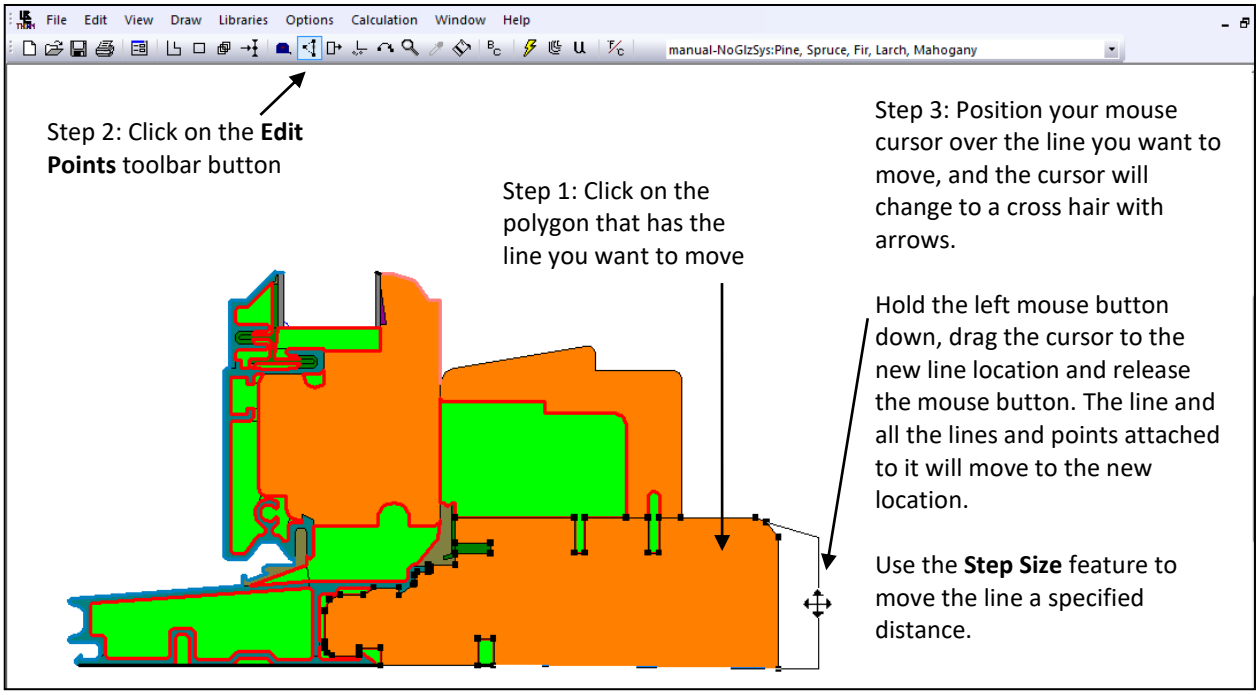


*Figure 5-26. Move a line by pressing the **Edit Point** button and placing the "move line" cursor over the line to be moved; hold the left mouse button, and move the mouse in the desired direction.*

5.6.6. Adding a Point

To add a point on a polygon:

- 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.
- Press the Insert Point toolbar button, or use the **Draw/Insert Point** menu choice (Shift-F6).
- 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. Drawing Using the Keyboard) 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.

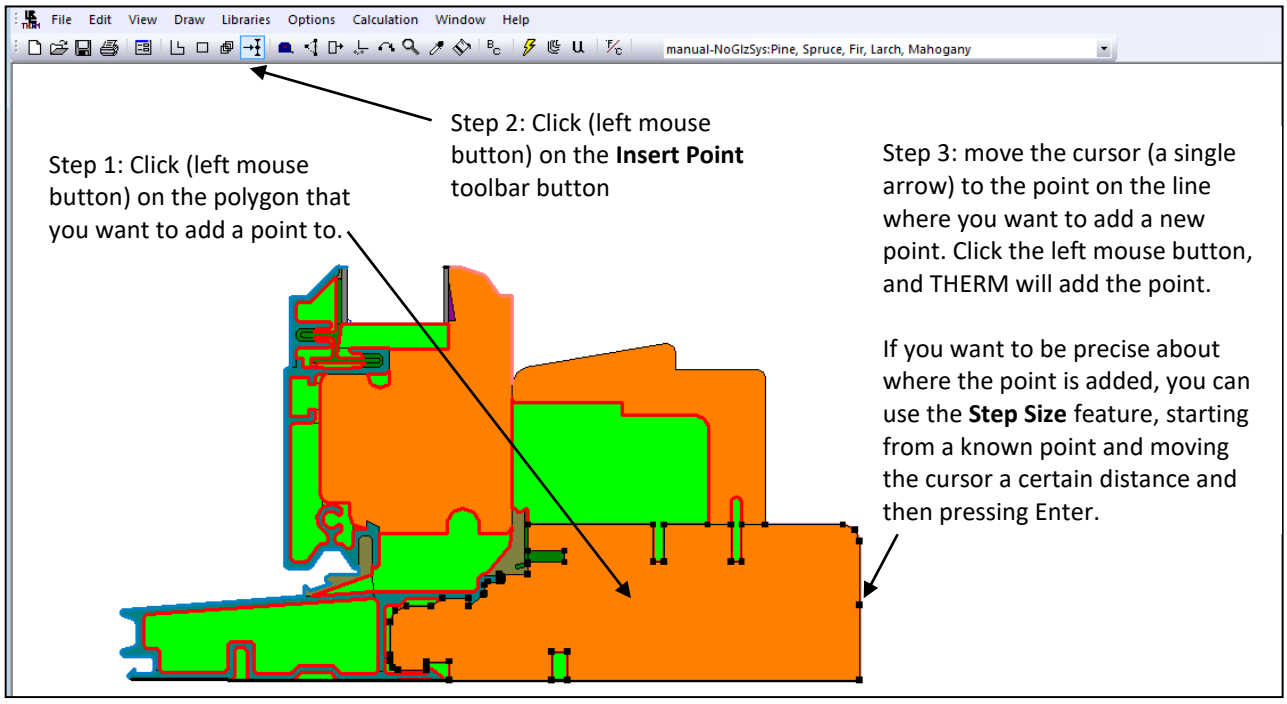


*Figure 5-27. Add a point by pressing the **Insert Point** button and clicking where the point is to be inserted.*

5.6.7. Deleting a Point

To delete a point on a polygon:

- Click the mouse on the **Edit Points** toolbar button, or use the **Draw/Edit Points** menu choice.
- Select the polygon (click the left mouse button) so that the points are displayed.
- Move the cursor to the point to be deleted (get within sticky distance of the point).
- A double-arrow cursor is displayed.
- 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.

It is not generally recommended that you flip or rotate models with the glazing system already inserted. It is better to rotate the frame without the glazing system, and then insert the glazing system into the finally rotated or flipped frame. If you already have a glazing system in a frame and want to rotate it, the safest procedure is to delete the glazing system, select all the polygons (using the "rubber band" method), copy the polygons, paste them into a new file, rotate or flip them and then insert the glazing system.

5.7.1. Flip

From the **Draw** menu, select the **Flip** choice, and then either the **Horizontal** or **Vertical** options. Figure 5-29 shows the result of horizontally flipping the cross section in Figure 5-28. You will get a warning that glazing sysetms should not be flipped, if the model contains one.

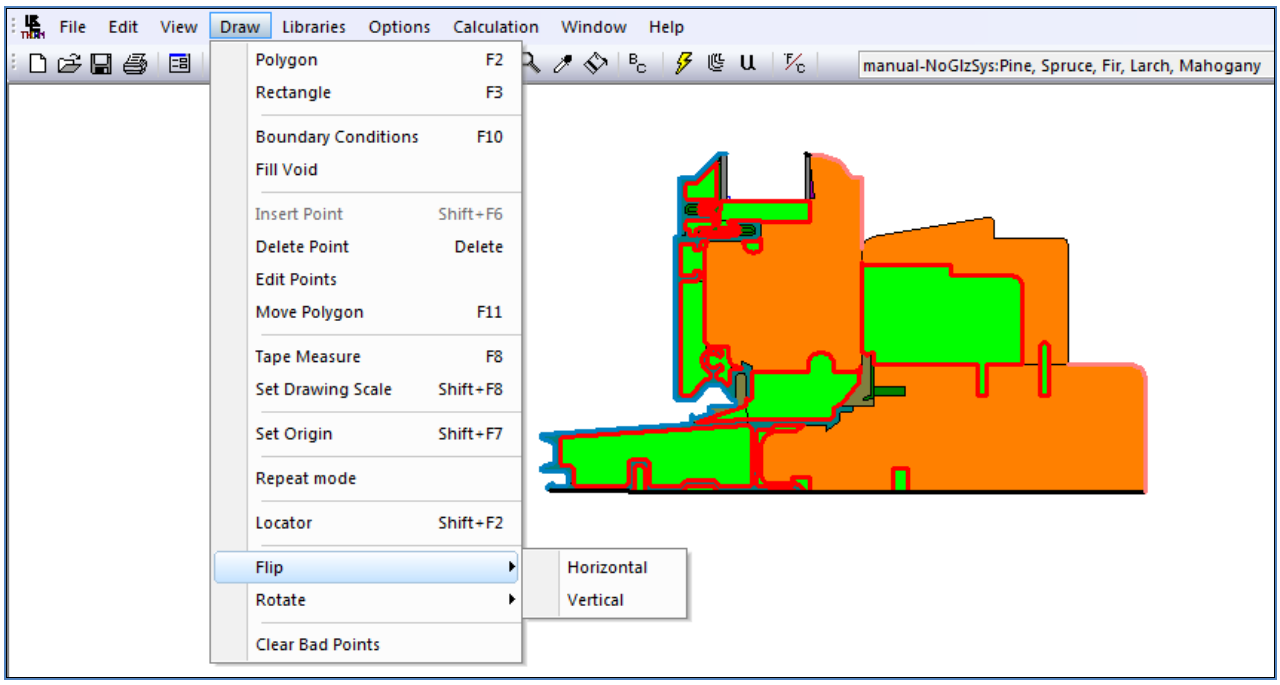


*Figure 5-28. Select **Flip** from the **Draw** menu, then select the **Horizontal** option.*

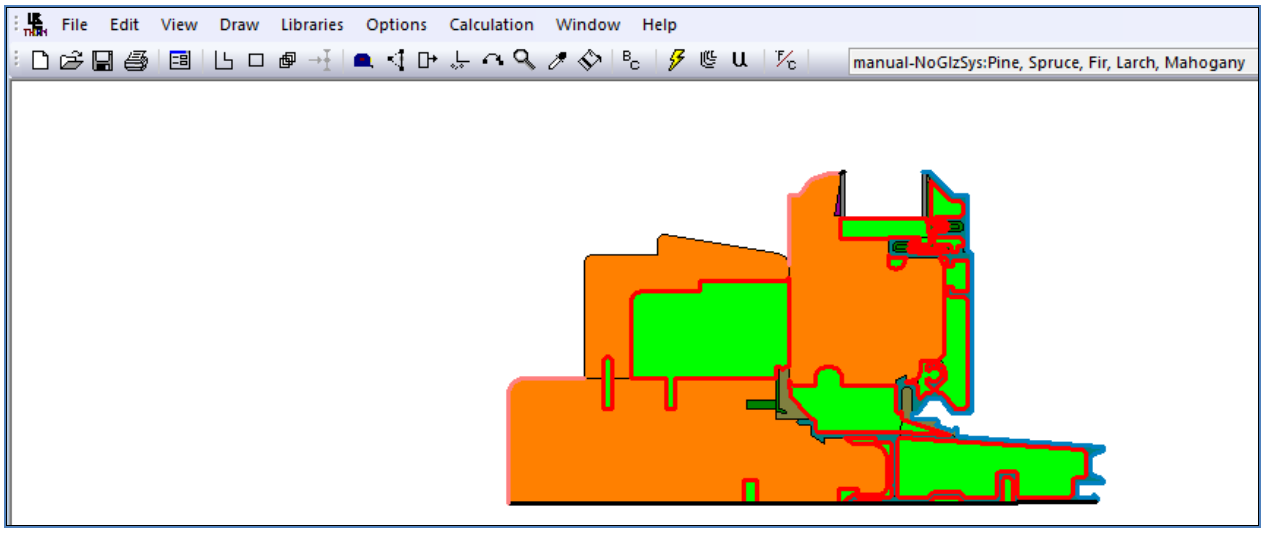


Figure 5-29. The cross section has been flipped in the horizontal direction.

5.7.2. Rotate

From the **Draw** menu, select the **Rotate** choice, and then either the **Left 90º, Right 90º, 180º** or **Degree** options. Choosing **Degree** displays a dialog box which allows you to enter the exact degrees of rotation. Figure 5-31 shows the result of horizontally flipping the cross section in Figure 5-30.

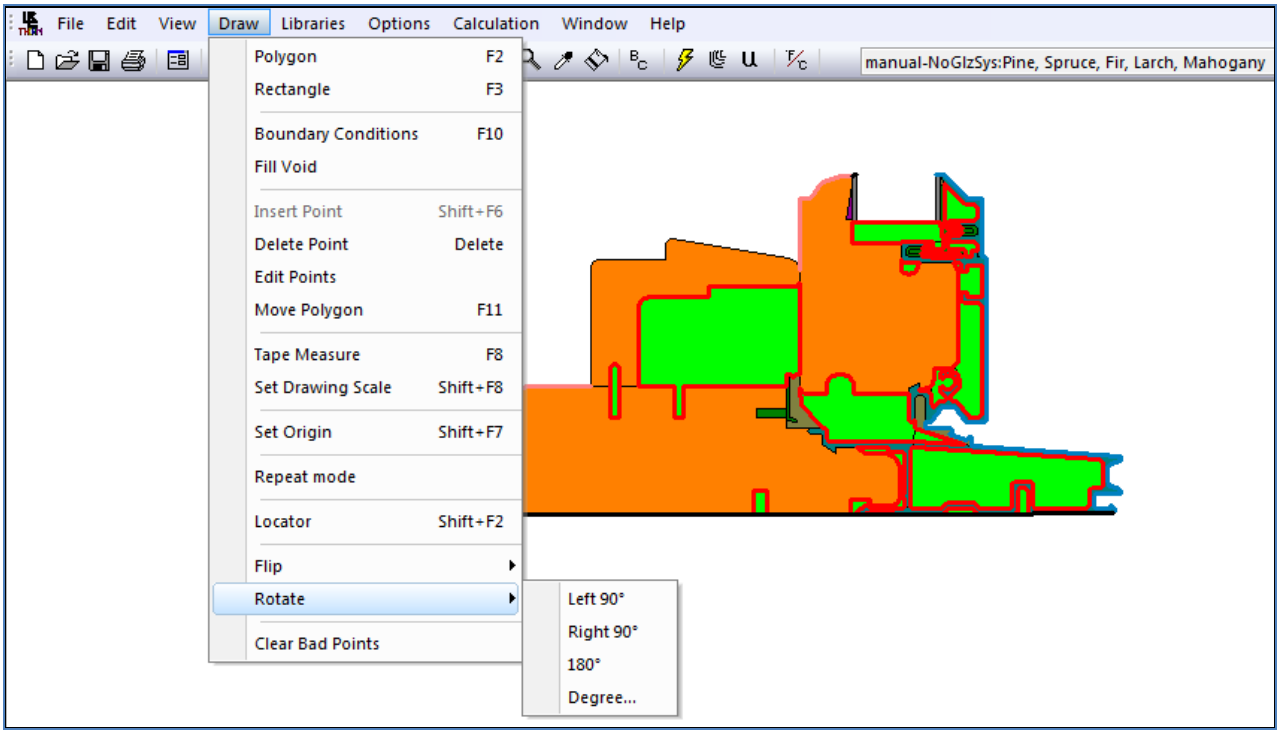


*Figure 5-30. Select **Rotate** from the **Draw** menu.*

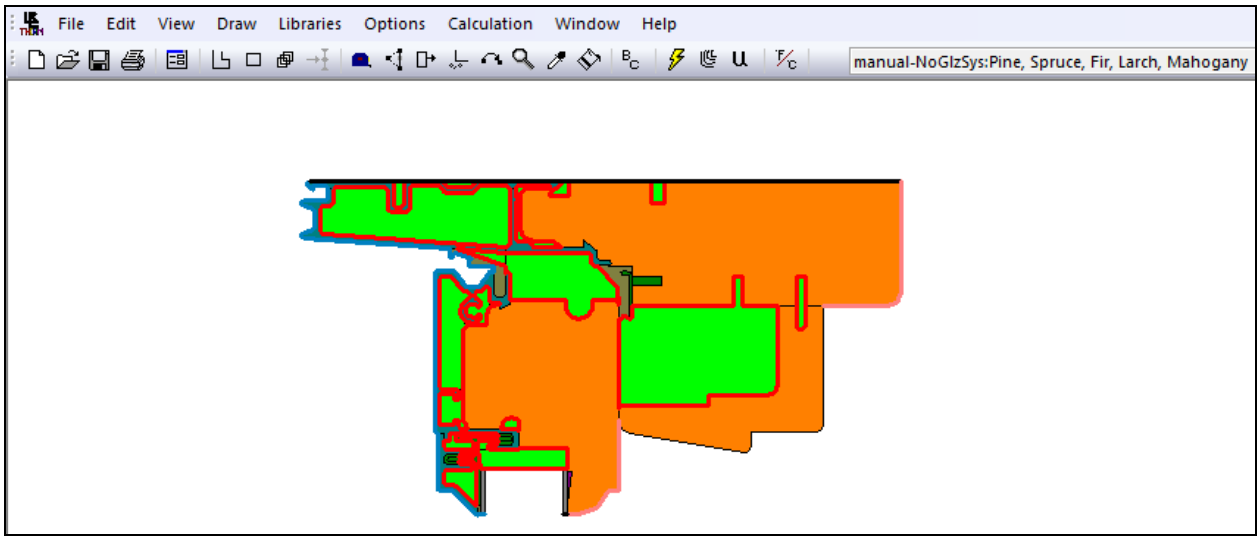


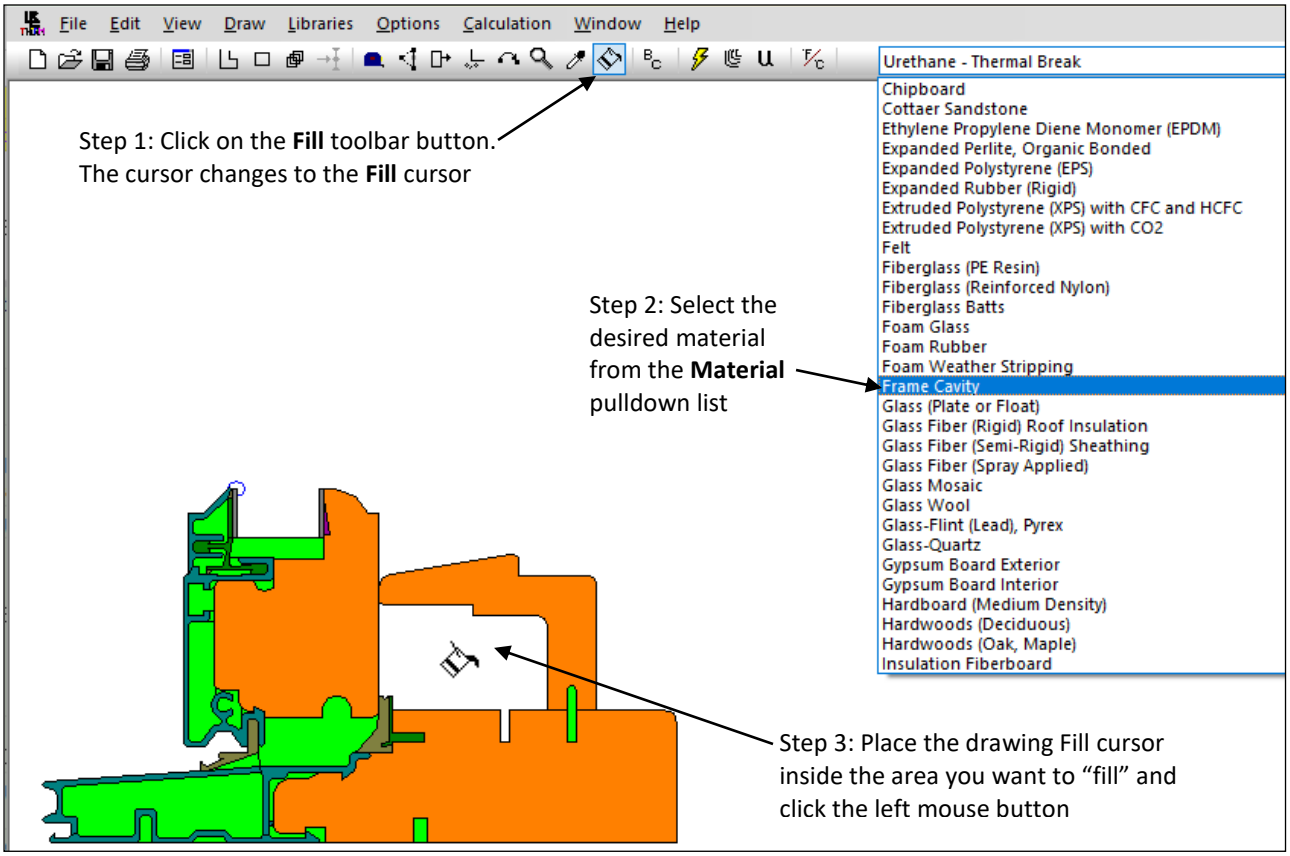
Figure 5-31. The cross section has been rotated 180 degrees.

5.8. Fill Tool

THERM has a drawing tool that allows creation of a polygon by "filling" a void or undefined area in the cross section that is completely surrounded by other polygons. It can also be used to change the material properties of an existing polygon. When a new polygon is created using the fill tool, THERM automatically creates all the necessary points to match the points on the surrounding polygons. This feature eliminates the need for drawing these potentially complicated polygons "by hand".

5.8.1. Using the Fill Tool to Create a New Polygon

- A cross section which has an undefined area completely surrounded by polygons is the perfect candidate for using the fill void feature, as shown in
- Figure 5-32. To create a polygon for this cavity, follow these steps:
- Click the left mouse button on the **Fill** toolbar button. 
- The drawing cursor will change to the **Fill** cursor. 
- Pick a **Material** from the material pulldown list, or from the **Library/Material** menu choice.
- Place the **Fill** cursor inside the undefined area to be filled with a polygon.
- Click the left mouse button, and THERM will draw a polygon, defined with the selected **Material**, that fills the area and contains points matching all the vertices on the adjacent polygons, as shown in



- Figure 5-33.

*Figure 5-32. Click on the **Fill Void** toolbar button, place the drawing cursor inside the void to be filled, and click on the left mouse button.*

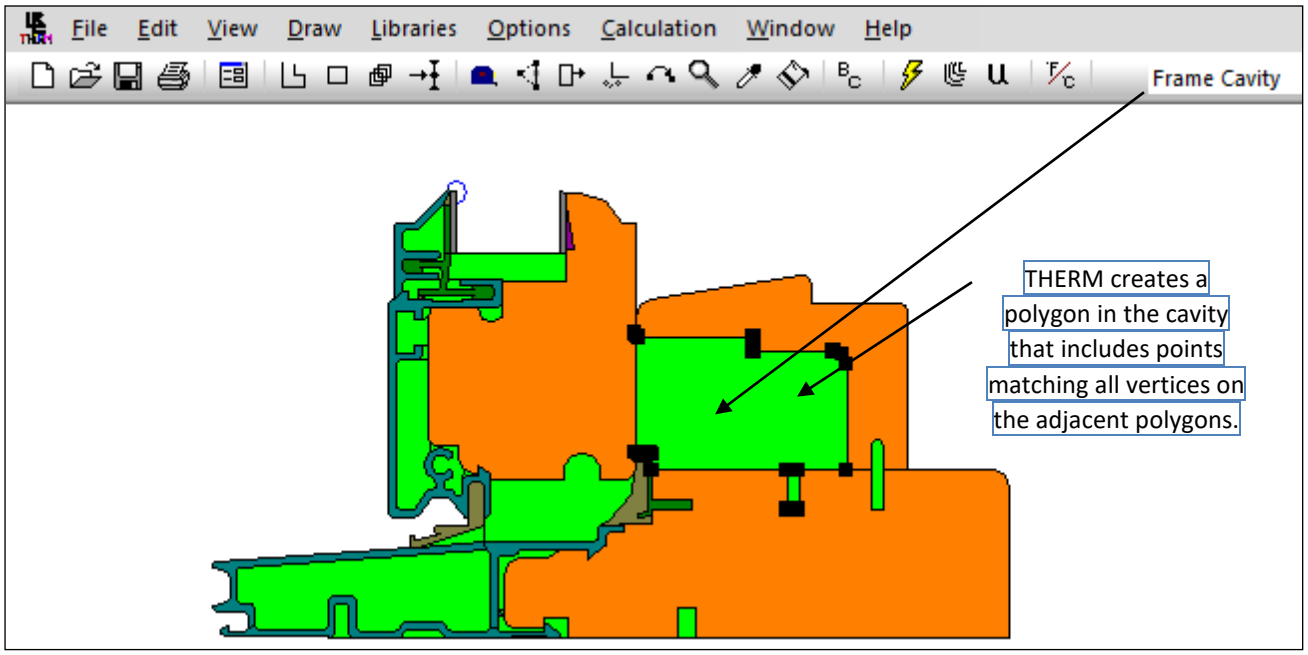


*Figure 5-33. The **Fill Void** feature creates a polygon in the cavity containing points matching all vertices on the adjacent polygons.*

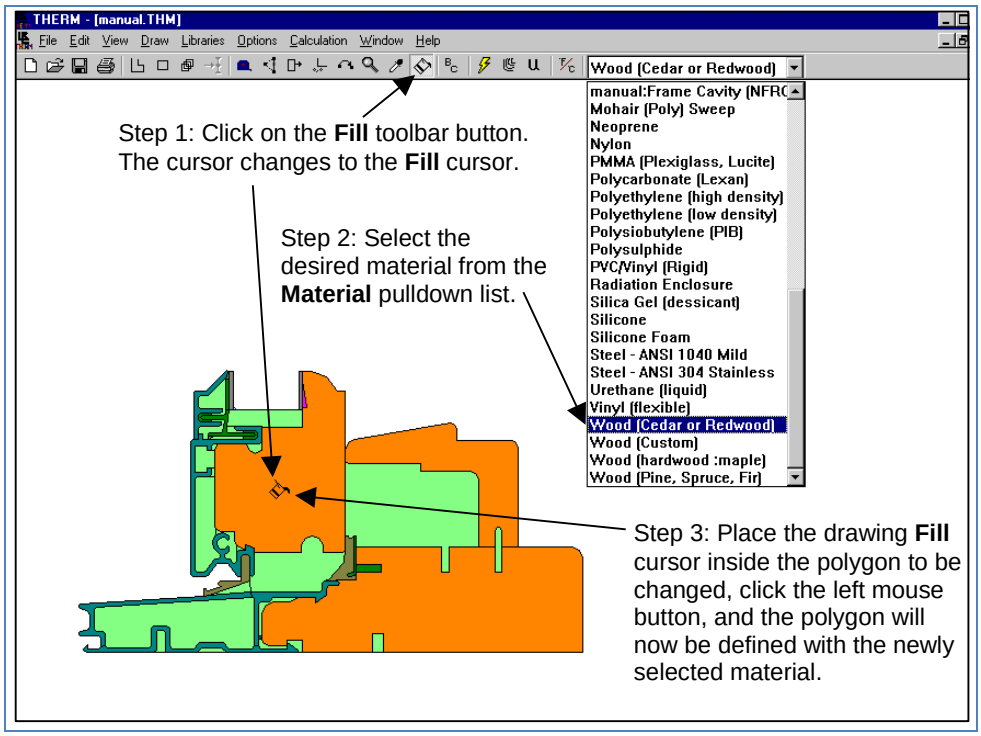


Figure 5-34. The Fill tool can be used to change the material of an existing polygon.

5.9. Eye Dropper Tool

The eye dropper tool is used to select a material in an existing polygon and apply that material choice to another existing polygon.

- Click the left mouse button on the **Eye Dropper** toolbar button. 
- The drawing cursor will change to the **Fill** cursor. 
- Click on the polygon which is defined with the material to be copied.
- The cursor will change to the **Fill** icon  and the **Material Pulldown** list will be set to material to be copied.
- Place the **Fill** cursor on any polygon, or a void to be filled, and click the left mouse button. The material or the void will be defined with the new material.

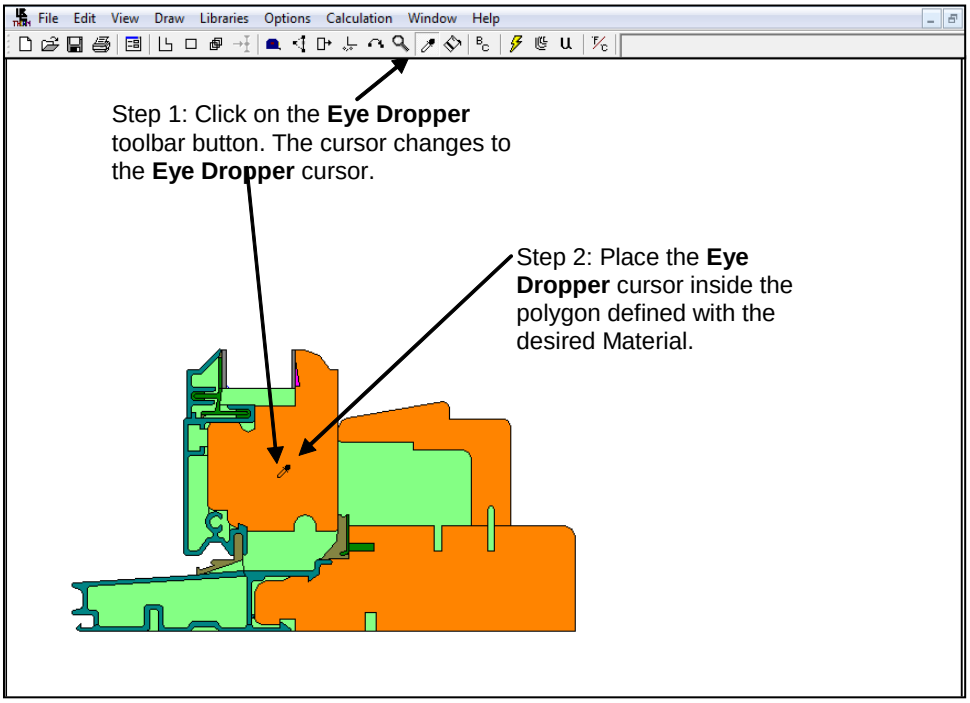


Figure 5-35. The Eye Dropper tool can be used to change the material of an existing polygon.

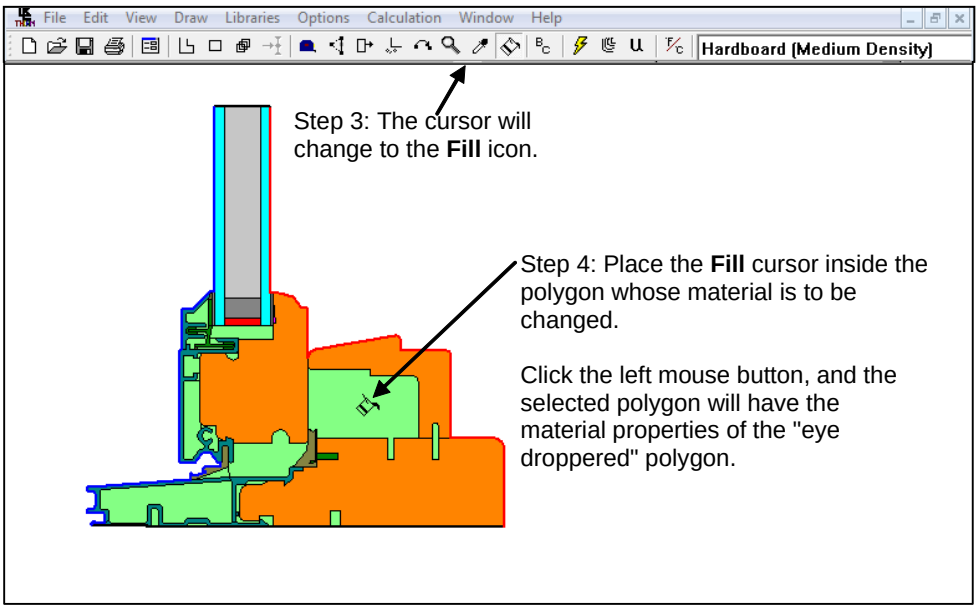


Figure 5-36. The cursor changes to the Fill tool.

5.10. Inserting a Glazing System

When you have drawn the non-glazed 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 (because they come from WINDOW with the glazing system) if you use this feature.

5.10.1. Inserting a WINDOW Glazing System

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

- Specify the location for the glazing system to be inserted by selecting the **Draw/Locator** menu choice or **Shift-F2**.

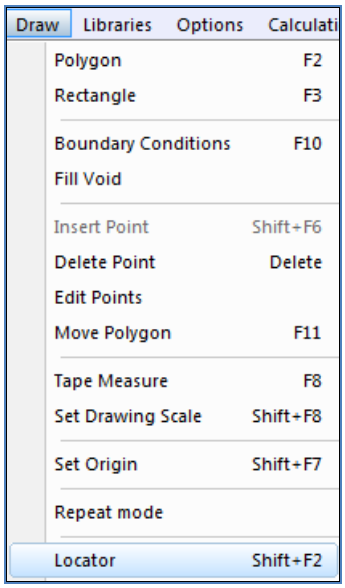


*Figure 5-37. Select **Locator** from the **Draw** menu.*

- 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.

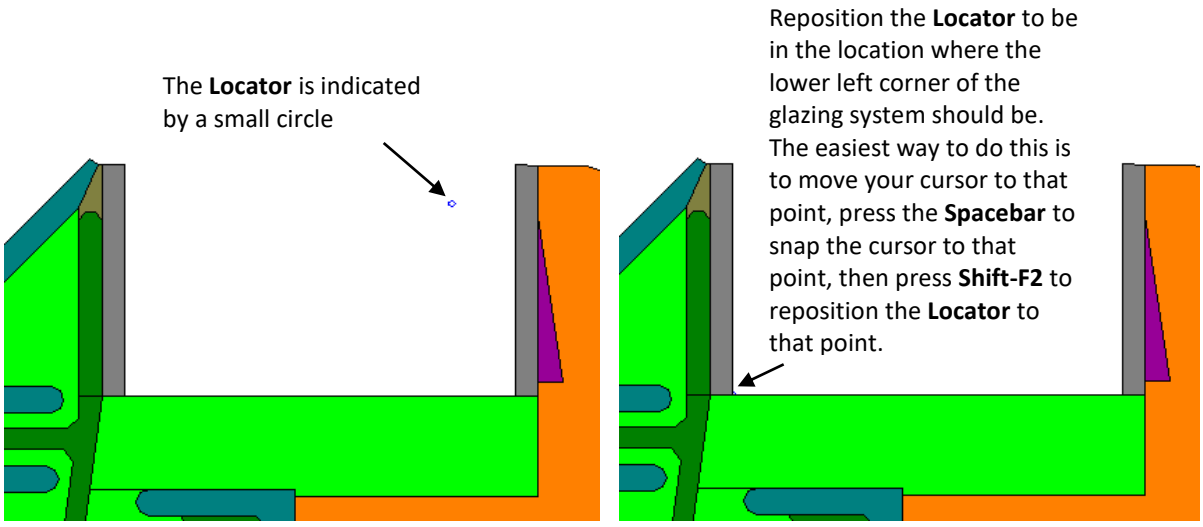


Figure 5-38. A small circle indicates the location of the lower left corner of the glazing system to be inserted.

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

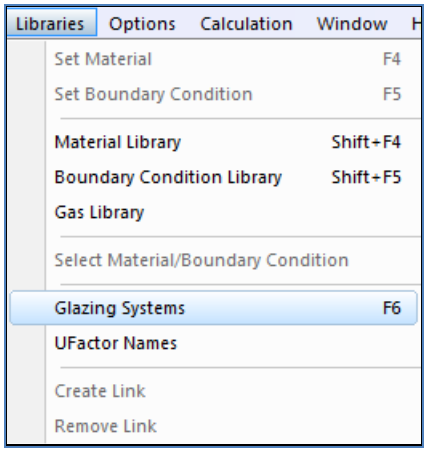


*Figure 5-39. Select the **Glazing Systems** choice from the **Libraries** menu.*

- From the **Glazing Systems** dialog box, shown below, specify the path by which the program can find the **Glazing System Library** in a WINDOW database, such as W7.mdb. Use the **Browse** button to find the WINDOW database; all **Glazing Systems** defined in the specified WINDOW database (in the Glazing System Library) will appear in the **Glazing System** pull-down list.

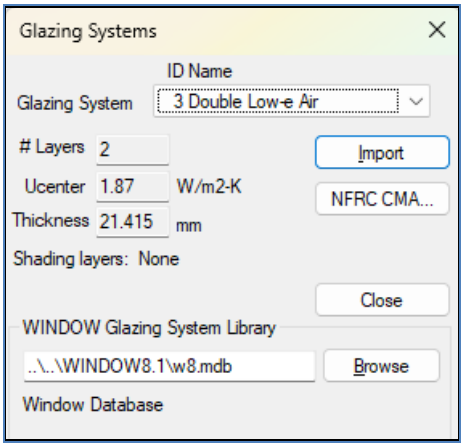


*Figure 5-40. Select a WINDOW glazing system from the **Glazing Systems** dialog box.*

The Glazing Systems dialog box shows the following information, derived from the selected WINDOW Glazing System Library:

Glazing System — Name of the glazing system

 # Layers — Number of layers in the glazing system.

Ucenter — Center-of-glass U-factor of the glazing system. **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)

If the text "Data not current" appears in the dialog box, the import button will be grayed out and the Ucenter value will be N/A. This means that the glazing system must be calculated in the WINDOW database before trying to insert it into THERM.

Thickness — Overall thickness (including all layers and gaps) of the glazing system. **Units:** inches (IP); mm (SI).

Shading layers — Indication of the location of any shading layers in the glazing system. Options are:

- **Interior:** The shading layer is on the interior (room facing) side of the glazing system
- **Exterior:** The shading layer is on the exterior (outside facing) side of the glazing system
- **Integral:** The shading layer is between two of the glass layers in the glazing system

Buttons

Import — Click to import the glazing system currently highlighted in the pulldown list into the THERM model

NFRC CMA — This opens a different version of the Insert Glazing System which has extra options specifically for the NFRC CMA program.

Close — Click to close the Glazing Systems dialog box without importing a glazing system into the THERM model.

Browse — Click to browse to a WINDOW database which contains glazing systems to import into the THERM model.

.

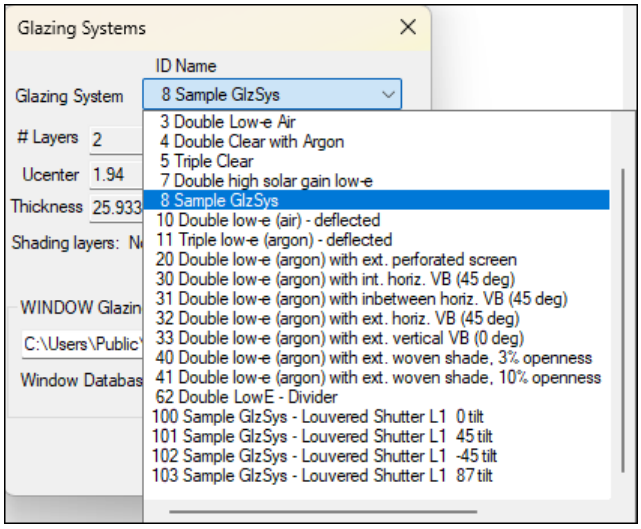


*Figure 5-41. Select the desired **Glazing System** from the pull-down list.*

Pick the desired **Glazing System** from the pull-down list.

Press the **Import** button.

The **Insert Glazing System** dialog box will appear (see Figure 5-42), with the following input values to be specified:

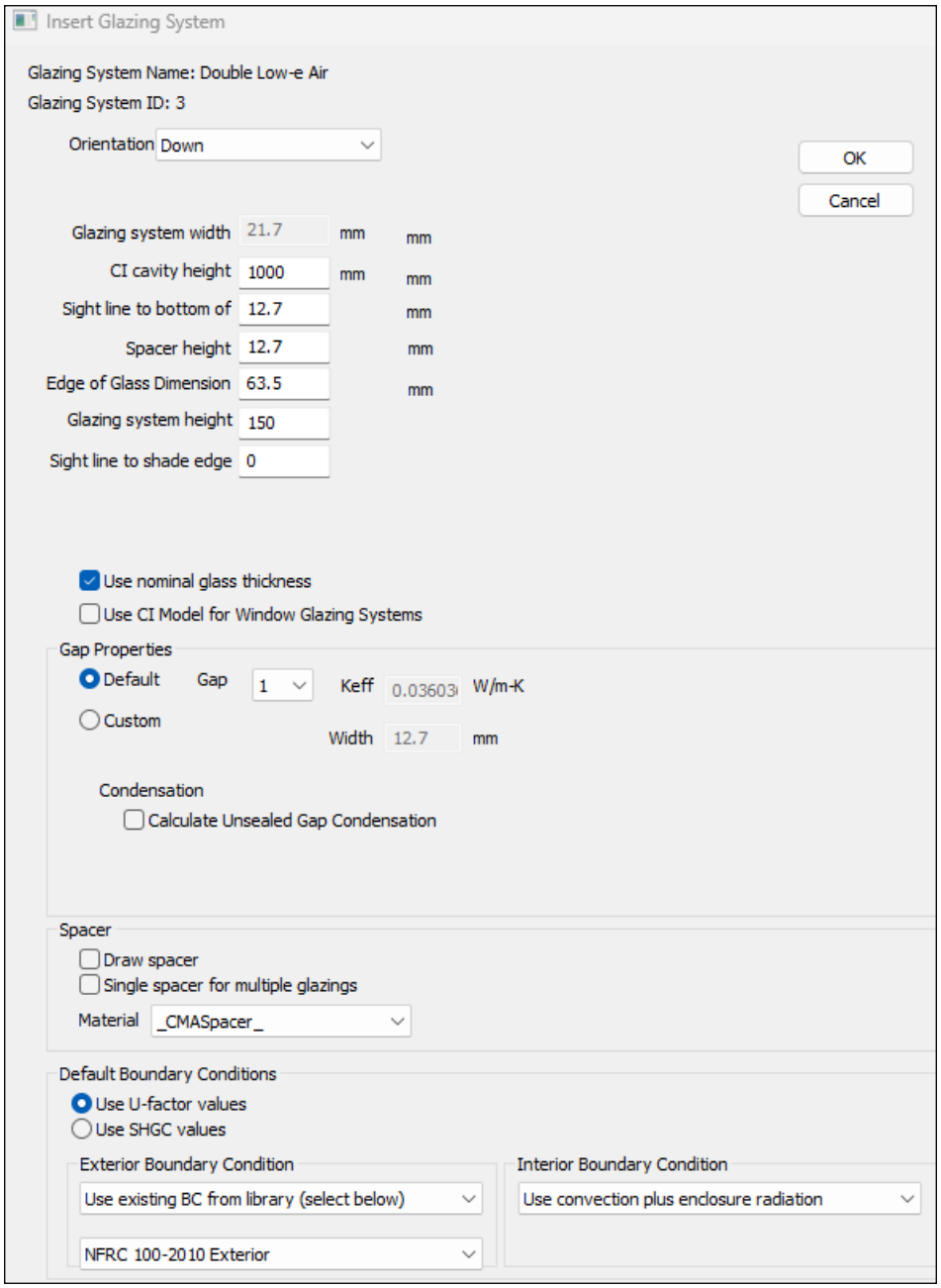


Figure 5-42. The Insert Glazing System dialog box.

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** restuls box, and then press **Ctrl-V** to paste the value into the **Insert Glazing System** input boxes.

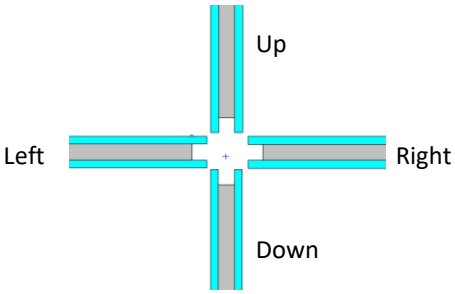
Below are the input options:

Orientation

Orientation of the glazing system. Up means the spacer will be at the bottom of the drawing. Down places the spacer at the top of the drawing. It is best to set the orientation here rather than rotating the glazing system once it has been inserted, which can potentially cause wrong results. Default: Up.

Options are:

- Up
- Down
- Right
- Left



Glazing System Width

This is feedback for the selected glazing system (from WINDOW), showing the total width of the system. **Units:** inches (IP); mm (SI).

CI cavity height

The cavity height the program will use when calculating the Condensation Index value (CI). **Units:** inches (IP); mm (SI). Default: 1000 mm.

Sight 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, which can be used as the demarcation point between the frame and the edge U-factor calculations. **Units:** inches (IP); mm (SI). **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. **Units:** inches (IP); mm (SI). **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." **Units:** inches (IP); mm (SI). **Default:** 2.5 inches (IP); 63.5 mm (SI).

Glazing system Height

The height of the glazing system used for the standard (non-CI) U-factor calculation. **Units:** inches (IP); mm (SI). **Default:** 6 inches (IP); 150 mm (SI)

Sight line to shade edge

Used to calculate where the edge of the shading system (modeled in WINDOW) is, relative to the frame sight line. This value can be negative if the shading system ends below the frame sight line. **Units:** inches (IP); mm (SI).

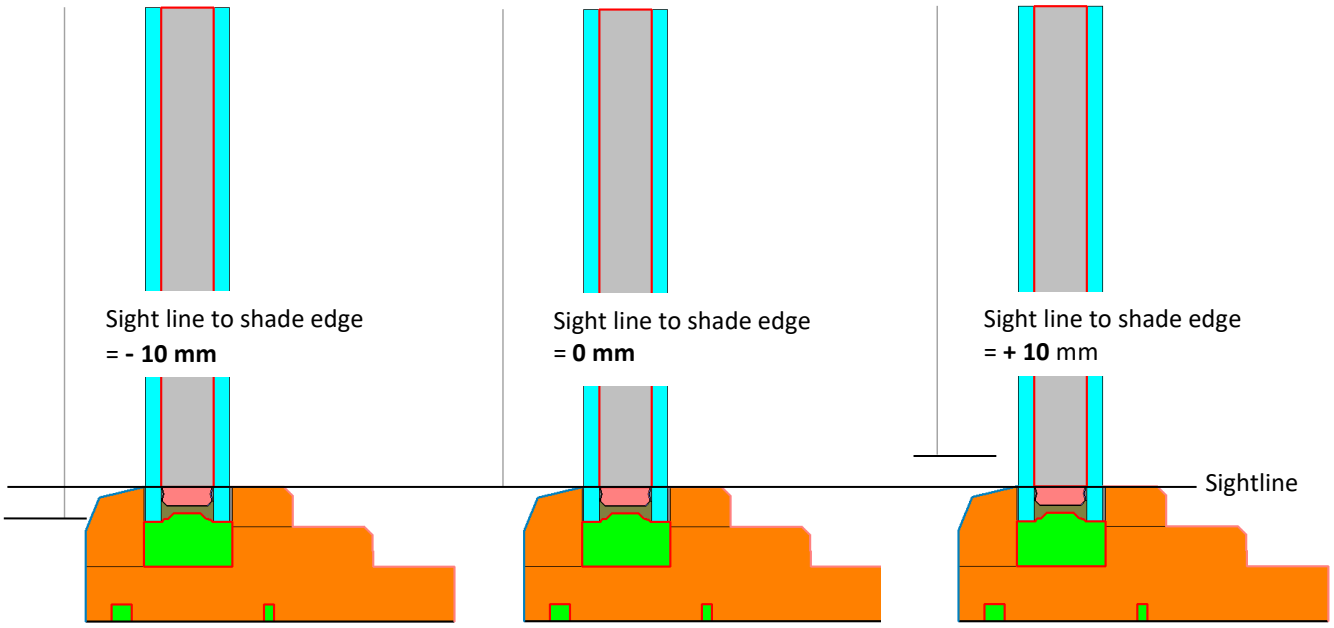


Figure 5-43. Possible settings for the Sight line to shade edge.

Use nominal glass thickness

If this box is checked, the program will use the nominal thickness of the glass layers, based on the values in *Table 5-1. Nominal thickness values included in THERM* when importing the glazing system. Using the nominal thickness feature will affect the overall thickness of the glazing system. If this option is used, it may be useful to also use that setting in WINDOW for the glazing system being inserted for consistency.

Use CI Model For Window Glazing Systems

Check the **Use CI Model for Window Glazing System** check box to allow THERM to use the **Condensation Index (CI) Model** when simulating the cross section. 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 **File/Properites/Calculation Options tab**. If you select this option, you will get U-factors based on the CI algorithms which are slightly different from the algorithms used in WINDOW; the CI U-factor algorithm is consistent with current NFRC U-factor procedures for calculating condensation .

The results default (see Calculation menu, Display Options choice) is to display the result based on the standard U-factor calculation. However, it is possible to select "CI Results" if those results are what is desired to be displayed in the results areas of the program (the U-factor results dialog box, etc).

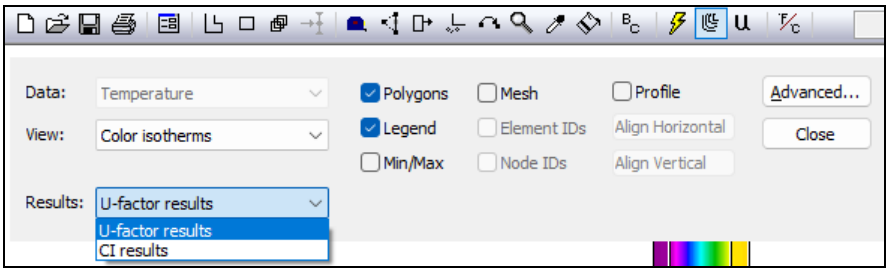


Figure 5-44. The standard U-factor results are the default display, but CI results can be selected instead.

Gap Properties

This section can be used to change the Keff (effective conductivity) and width of each gap in the glazing system when it is imported into the THERM file. The standard state is that the "Default" radio button is on, which means THERM will use the default values for Keff and Width from WINDOW. If the "Custom" radio button is clicked, the fields for Keff and Width can be edited for each gap in the glazing system.

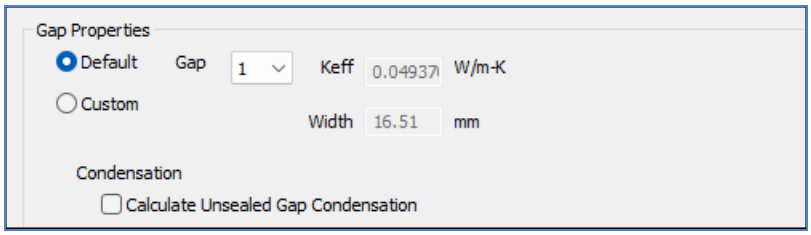


Figure 5-45. Gap Properties section of the Insert Glazing System dialog box

It has the following input choices:

Default — The Default radio button (this is the default of the Default and Custom radio button choices) shows the Keff and Width values, for each gap in the glazing system, that come from WINDOW.

Custom — Activating the Custom radio button makes the Keff and Width values editable, so that non-default values can be entered in those fields.

Gap — The cavity gap number, in a pulldown list. Selecting the Gap number causes the Keff and Width fields to display the values for the chosen gap

Keff — The keff (effective conductivity), from WINDOW, of the gap selected in the Gap pulldown list. This value can be edited if the Custom radio button is activated. **Units:** Btu/h-ft-F (IP); W/m-K (SI).

Width — The width, from WINDOW, of the gap selected in the Gap pulldown list. This value can be edited if the Cutoms radio button is activated. **Units**: inches (IP); mm (SI).

Condensation Calculate Unsealed Gap Condensation — This is a metric for condensation in unsealed spaces, such as the space between the existing window and window attachment. Window attachment can be a shading system or secondary window system. This index is equivalent to condensation index for windows (CI), which predicts the ability of the product to resist condensation. In the case of windows, CI predicts resistance to condensation on the room facing window surface. In the case of window attachments, CIU predicts resistance to condensation on the existing window when window attachment is placed over the existing window on the room (indoor) side. If the attachment is paced on the outdoor side, CI is governing index of performance. The higher the CI or CIU, the more resistive the window system is to condensation occurance.

Unitless.

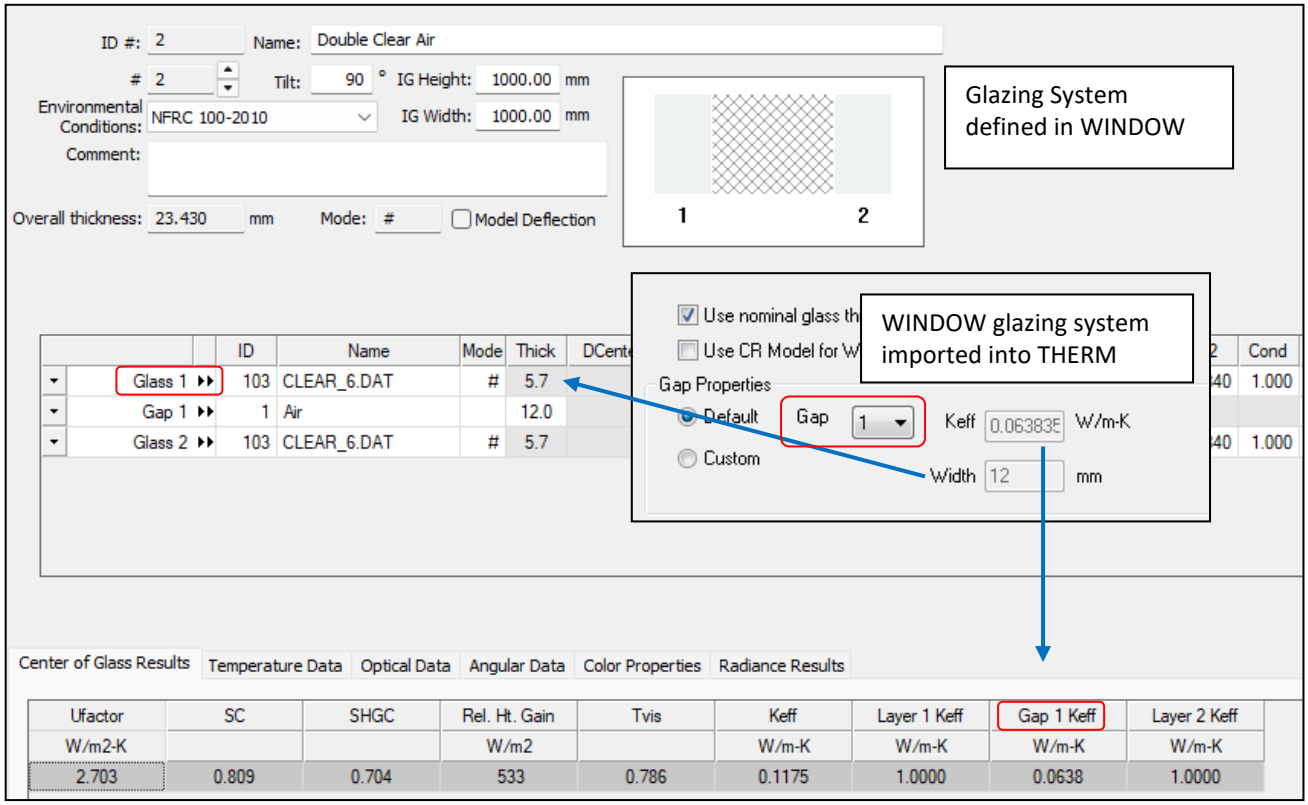


Figure 5-46. The gap Keff values calculated for a glazing system in WINDOW are imported into THERM when the WINDOW glazing system is imported.

Spacer

This section can be used to specify whether the program generates a default spacer, or just leaves a blank space for the spacer based on the "Spacer Height" input value.

It has the following input choices:

Draw Spacer — If this box is checked, the program will draw a rectangle, of the material chosen from the pulldown list next to Material, below each of the glazing system cavities. If it is unchecked, the program will leave the spacer area blank, and a custom spacer can be drawn or copied and pasted from another THERM file (see 5.13. Creating a Component Library later in this section). The program will assume that there are separate spacers for each glazing system gap, unless the "Single spacer for multiple glazings", discussed below, is checked. **Default:** unchecked.

Single spacer for multiple glazings — If this box is checked, the program will assume that there is one spacer that spans all the glazing system cavities.

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 program will draw a solid block of this material for the spacer.

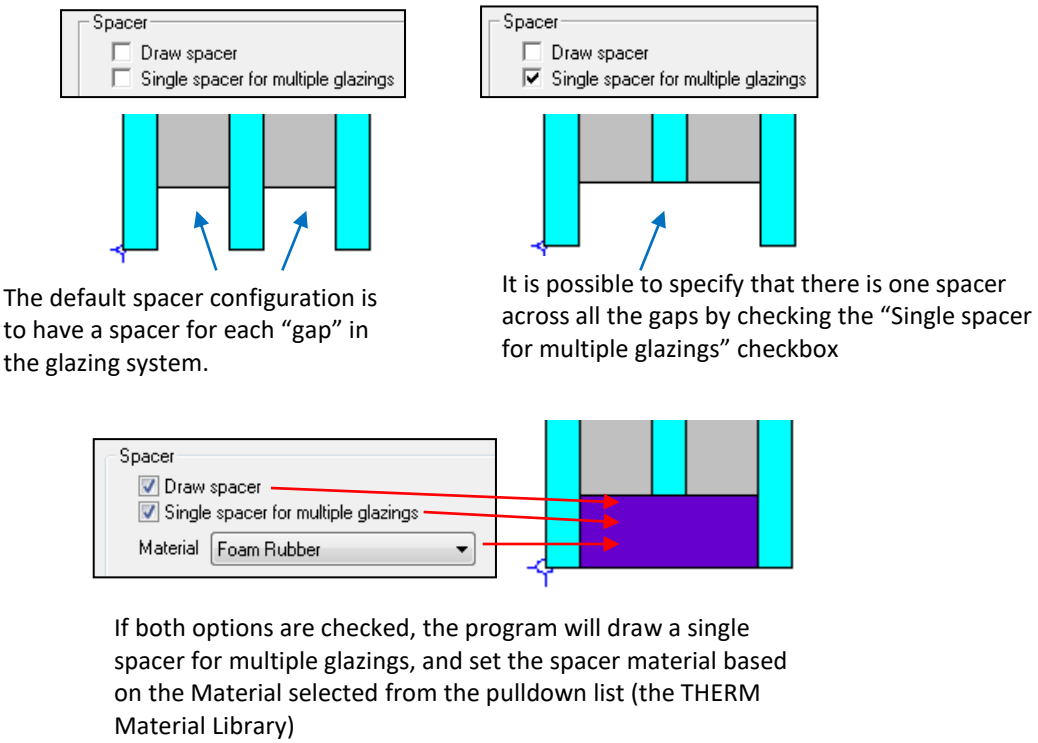


Figure 5-47. Space section of the Insert Glazing System dialog box

Default Boundary Conditions

This section determines the default boundary conditions that are applied to the WINDOW glazing system when it is imported into the THERM model.

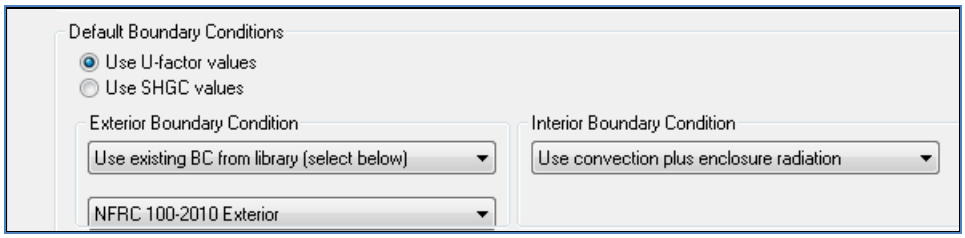


Figure 5-48. Default Boundary Conditions section of the Insert Glazing System dialog box.

It has the following input choices:

Use U-factor Values — If this option is selected, the program will apply the U-factor boundary condition values applied to the glazing system in WINDOW to the glazing system imported into THERM. This is generally the default choice when modeling WINDOW glazing systems in THERM.

Use SHGC Values — If this option is selected, the program will apply the SHGC (Solar Heat Gain Coefficient) boundary condition values applied to the glazing system in WINDOW to the glazing system imported into THERM.

Exterior Boundary Conditions — This section allows the following options for assigning the default exterior boundary conditions as follows:

- Use combined film coefficient
- Use convection plus linear radiation coefficient
- Use convection plus blackbody radiation
- Use convection plus enclosure radiation

Use existing BC from library (select below): This is the default Exterior Boundary Condition choice, and can be used to select a boundary condition from the pulldown list below this option. **Default:** "NFRC 100-2010 Exterior" is the default boundary condition selected in the pulldown below this input.

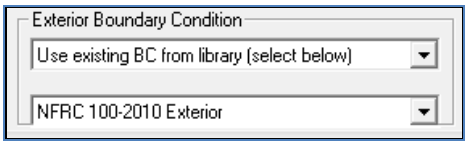


Figure 5-49. Default settings for the Exterior Boundary Condition section.

Interior Boundary Conditions — This section allows options for assigning the default interior boundary condition as follows:

- Use combined film coefficient
- Use convection plus linear radiation coefficient

- Use convection plus blackbody radiation
- Use convection plus enclosure radiation: *This is the default choice for the Interior Boundary Condition*, and will cause THERM to set the glazing system interior boundary condition **Radiation Mode**l to "AutoEnclosure". THERM will then model the cross section with the radiation enclosure feature without having to actually draw the geometry of a Radiation Enclosure.
- Use existing BC from library (select below): This option displays a pulldown list of the standard THERM choices from the THERM Boundary Condition Library which can be selected as the Interior Boundary Conditions

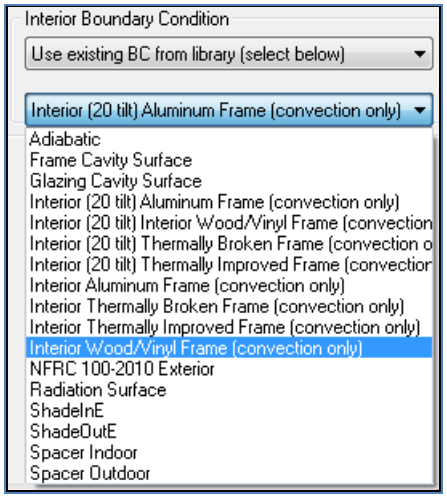


Figure 5-50. For the choice "Use existing BC from Library" the program displays the records from the THERM Boundary Condition Library.

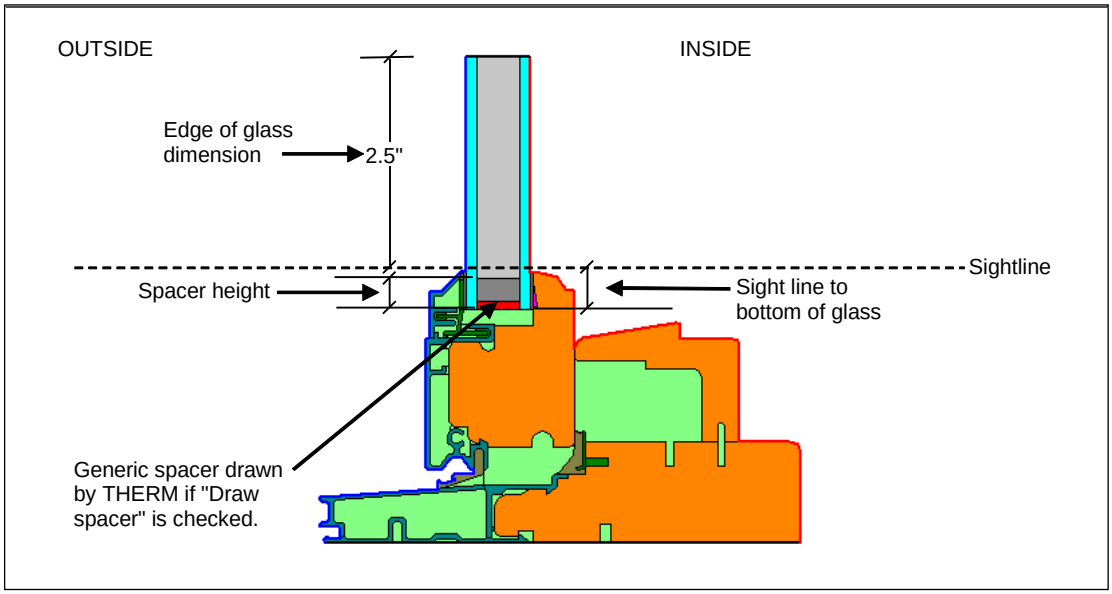


*Figure 5-51. Illustration of dimension definitions used in the **Insert Glazing System** dialog box*

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 *5.13. Creating a Component Library* for more details).

Representation of the Shading System

If the WINDOW glazing system includes a shading system, it will be displayed as a graphic representation, rather than an actual THERM polygon that can be edited or otherwise manipulated in THERM.

In THERM, the shading system is not explicitly modeled. When a WINDOW glazing system with a shading system is inserted into a THERM file, THERM draws a graphic representation of the shade, but does not create a polygon for it. The space between the shade and the glazing system or frame is not modeled as a frame cavity, and the effect on the glazing system or frame is accounted for by assigning a Shading Modifier when defining the boundary conditions. The Shading Modifier is automatically created when the glazing system is inserted, and will be available from the Shading modifier pulldown menu in the Boundary Condition dialog box.

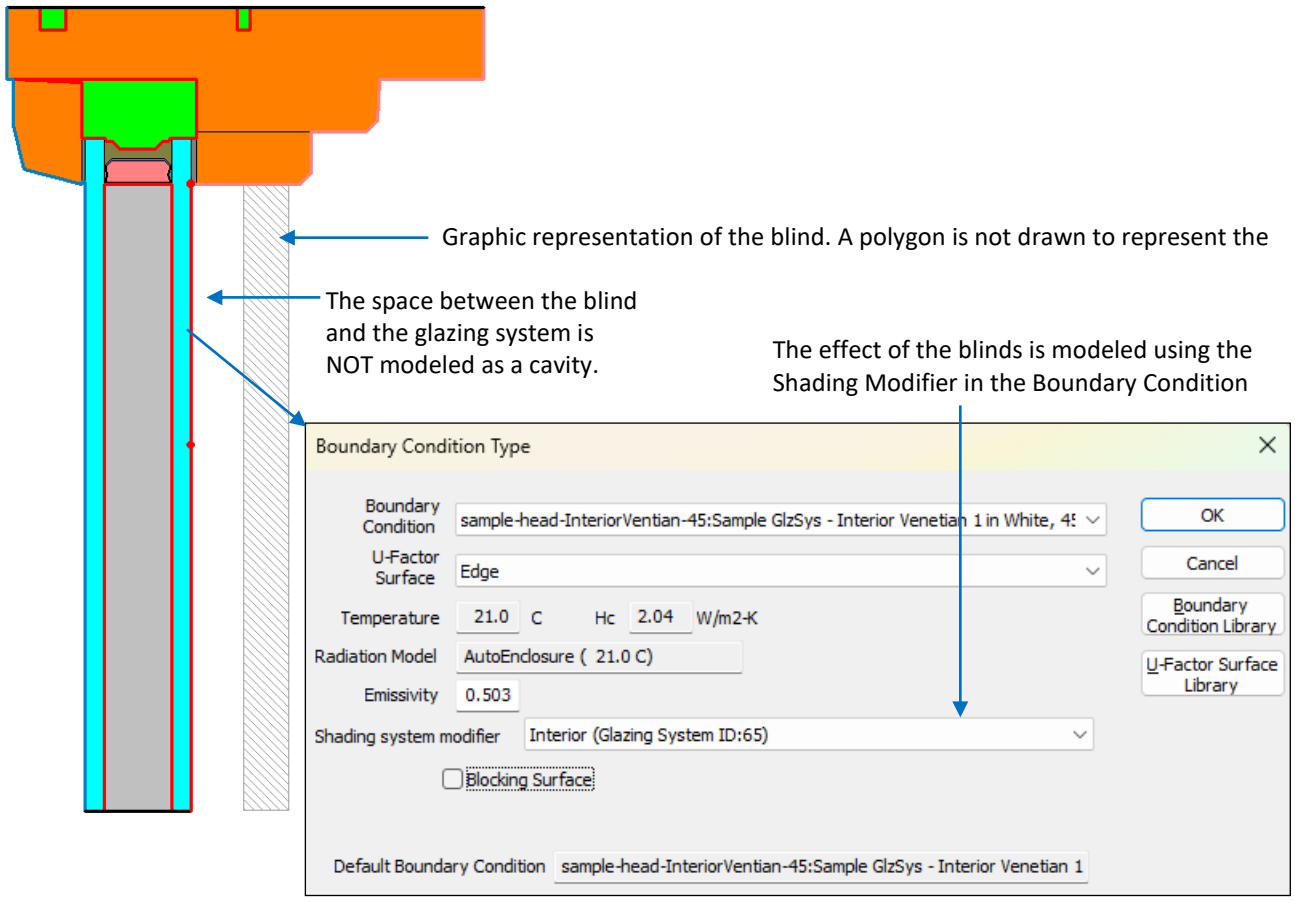


Figure 5-52. The shading system is not explicitly modeled in THERM.

When the glazing system(s) has been added to the THERM model, it is possible to double click on the glazing system to view the Glazing System Info dialog box, which contains information about the glazing system, as shown in Figure 5-53. Double click the glazing system in the THERM model to see detailed information about the glazing system.

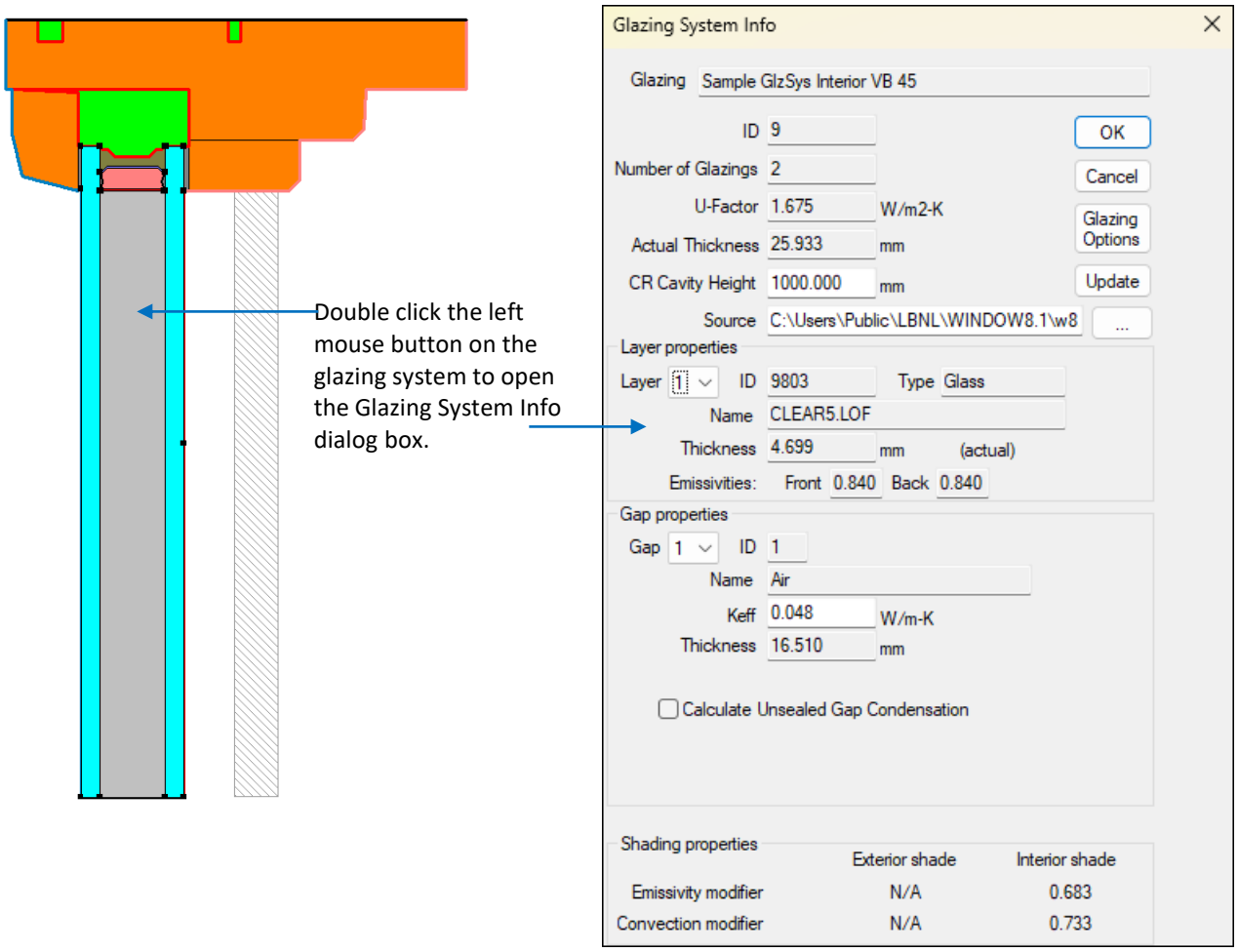


Figure 5-53. Double click the glazing system in the THERM model to see detailed information about the glazying system.

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 unless it is required to model special cases such as storm windows or other complicated glazing system whose geometry can't be completely specified in WINDOW.

Generally, 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 WINDOW. 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, particularly in defining the Keff values for the glazing cavities.

If you checked the **Use CI Model for Window 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. See Section *5.10.2. Condensation* below for more detailed information.

5.10.2. 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 WINDOW 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. These assumptions have been shown to yield acceptable results for the average heat loss (U-factor), but they are inadequate for predicting local temperatures. The condensation index model included in THERM 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; these U-factors are not consistent with current NFRC U-factor procedures.

- 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 Glazing Systems" checkbox in the **Therm File Options** tab of the **Options/Preferences** menu, as shown in Figure 5-55, 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 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.

Click on this checkbox when importing a glazing system from WINDOW to use the Condensation Index Model for glazing cavities.

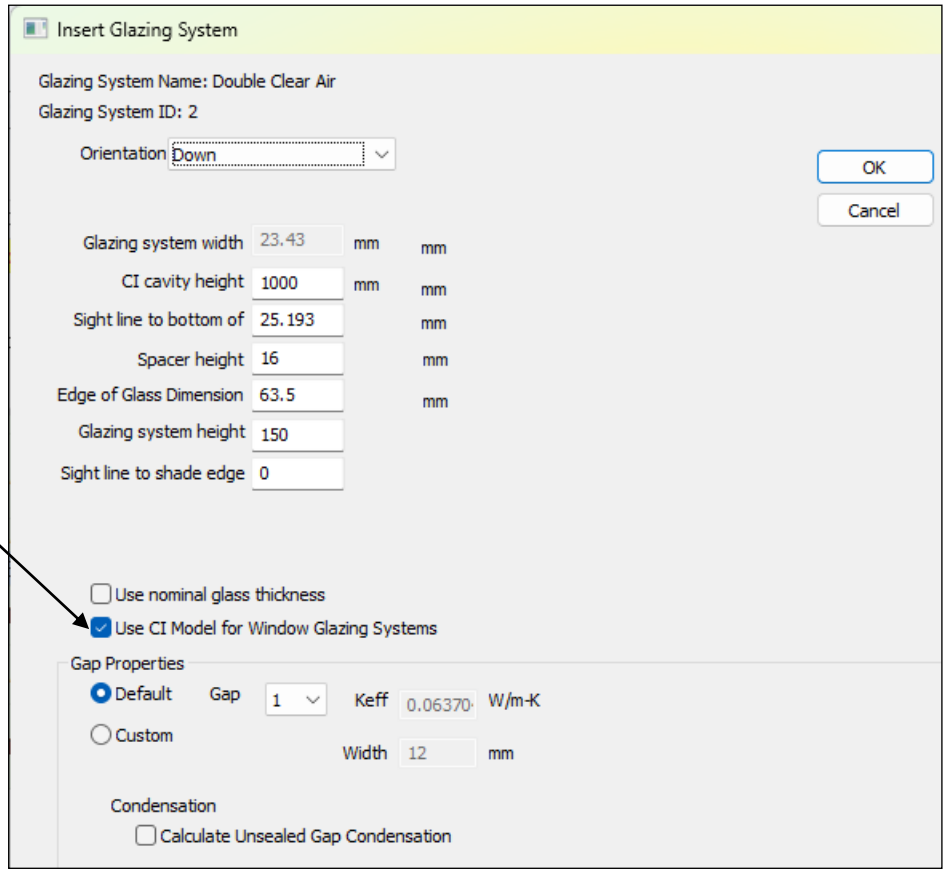


Figure 5-54. CI can be turned on for individual glazing systems.

If the "Use CI Model for Window Glazing Systems" is not checked when the glazing is first imported, it can be changed before the final calculation in the **Calculation Options** dialog box (from the **File / Properties** menu); checking the "Use CI Model for Glazing Systems" checkbox at that time will change the model of a glazing system already imported into THERM.

Click on this check box (either before or after importing a glazing system from WINDOW) to use the Condensation Index Model for glazing cavities

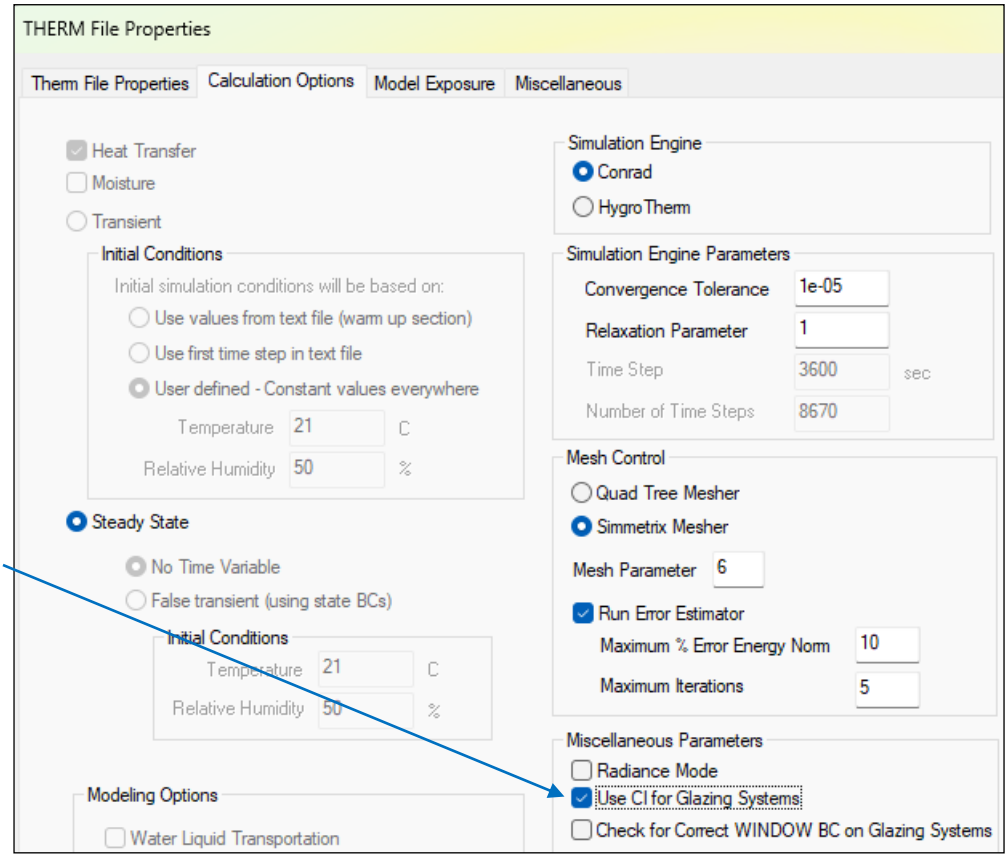


Figure 5-55. Options menu / Preferences option / THERM File Options tab to specify whether to run the Condensation model for Glazing Systems when simulating the file.

Import a glazing system from WINDOW. The glazing system appears with the glazing cavity surrounded by a red line when it is imported with the CI Model turned on, which is an easy way to tell whether the condensation model is being used. The condensation 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.

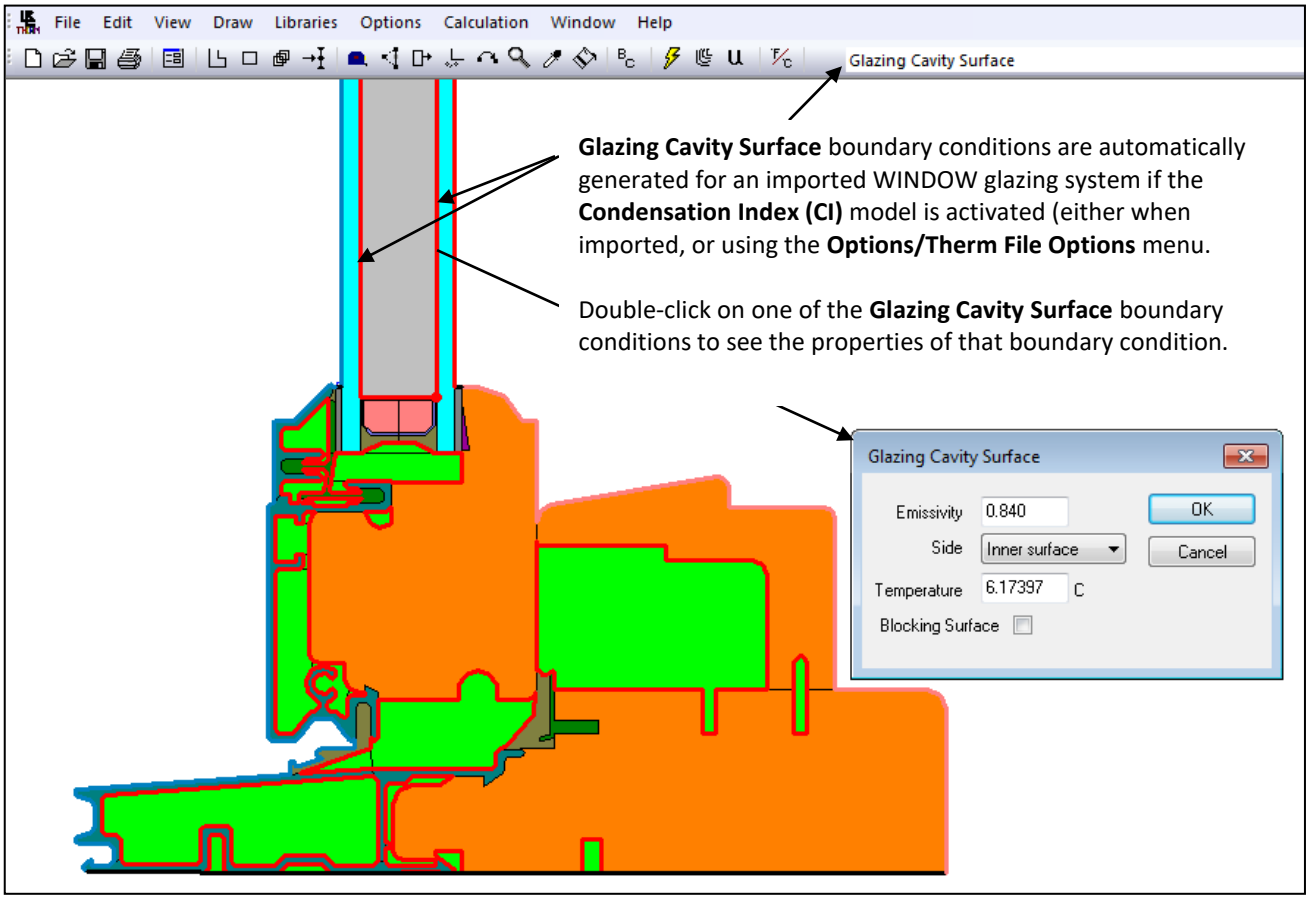


Figure 5-56. Condensation Index procedure for glazing cavities

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, as shown in Figure 5-56 and Figure 5-57. The fields are the following:

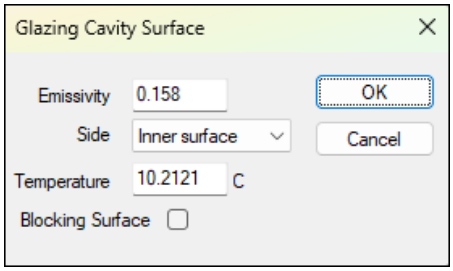


Figure 5-57. Glazing Cavity Surface dialog box for the Boundary Conditions in a glazing system cavity modeled with Condensation Index calculation on.

Emissivity — The emissivity of the surface. This value comes from the WINDOW glass library for glazing systems imported from WINDOW. 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.
- **Outer Surface:** assigned to the outer surface of the glazing cavity.
- **Inner Surface:** assigned to the inner 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 for glazing system interior boundary conditions is that this option is unchecked, meaning there are no blocking surfaces.

5.10.3. If the WINDOW Glazing System Doesn't Fit

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

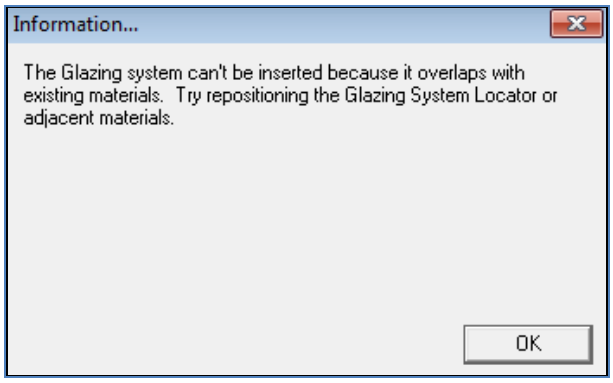


Figure 5-58. Message displayed if the Glazing System cannot be inserted

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 WINDOW 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 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.

5.10.4. Nominal Thickness for Glazing Systems

When importing a glazing system, it is possible to specify that THERM should use a nominal glass thickness rather than the actual glass thickness when importing a glazing system. Keep in mind the following:

- THERM adjusts the glass layer thickness to the nominal value upon import (see the Nominal Thickness values table below).
- THERM does not change the thickness of the glazing system cavity
- This means that the overall thickness of the IG will change by the amount of the difference between the actual and the nominal thicknesses for each glass layer. So the frame cross section may need to be adjusted to account for this
- The Nominal Thickness values are contained in the THERM "INI" file (which generally contains the version and subversion of the program, such as "therm8.1.ini"), located in

  C:\Users\Public\LBNL\Settings

Table 5-1. Nominal thickness values included in THERM.

Minimum Value (mm)	Maximum Value (mm)	Nominal Thickness (mm)
2.16	2.57	2.50
2.92	3.4	3.00
3.78	4.19	4.00
4.57	5.05	5.00
5.56	6.2	6.00
7.42	8.43	8.00
9.02	10.31	10.00
11.91	13.49	12.00

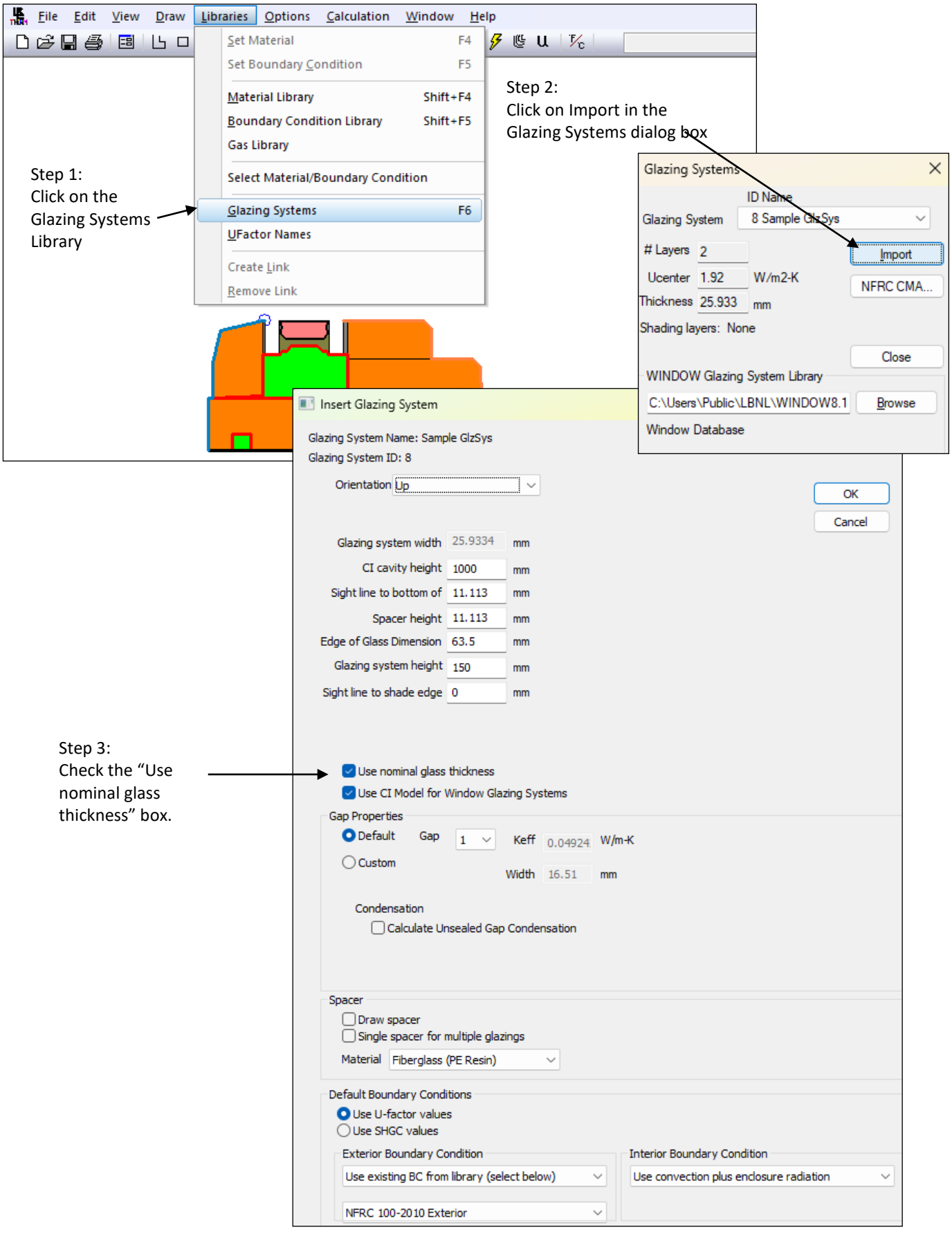


Figure 5-59. Nominal Thickness

5.10.5. Inserting More Than One WINDOW 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.

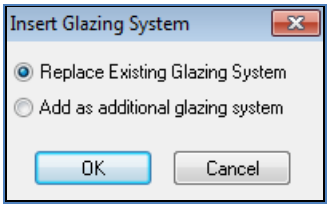


Figure 5-60. Select "Add an additional glazing system" to insert more than one glazing system into a model.

5.10.6. 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:

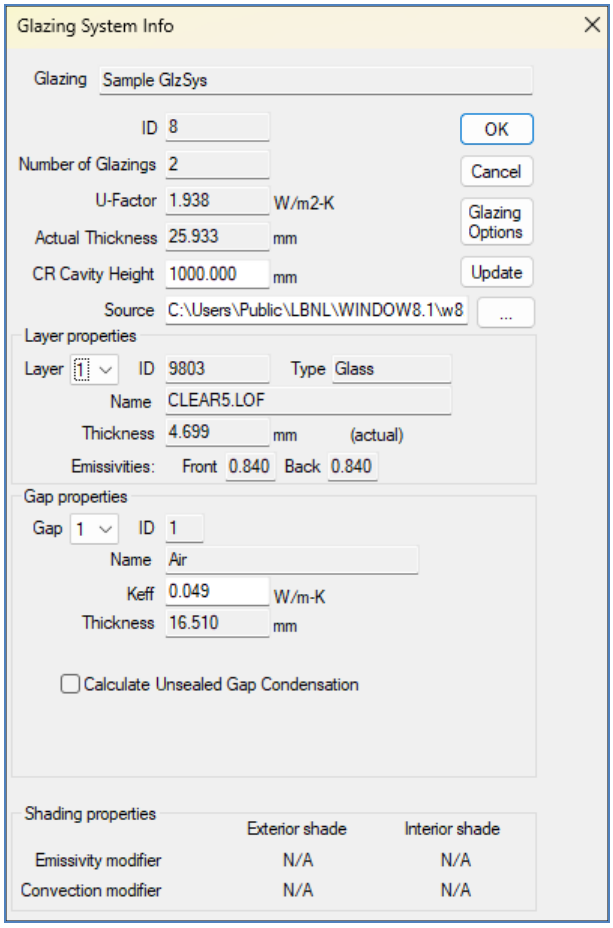


Figure 5-61. Double-click on the inserted glazing system to see information about the system.

Many characteristics of the glazing system are displayed in the Glazing System Info dialog box, some of which can be edited and some that cannot. The fields on this dialog box are the following:

Glazing The name of the glazing system, from the Name of the glazing system in the WINDOW database it was imported from. This cannot be edited.

ID The ID of the glazing system, from the ID of the glazing system in the WINDOW database it was imported from. This value cannot be edited

Number of Glazings The number of glass layers in the glazing system, from the glazing system in the WINDOW database it was imported from. This value cannot be edited.

U-factor The U-factor of the glazing system (center of glass) from the glazing system in the WINDOW database it was imported from. **Units:** Btu/h-ft² (IP); W/m²-C (SI).

Actual or Nominal Thickness Thickness of the glazing system, from the glazing system in the WINDOW database it was imported from. The program will show either "Actual thickness" or "Nominal thickness" depending on whether the glazing system was imported with the "Use nominal glass thickness" checkbox checked or not. **Units:** inches (IP); mm (SI).

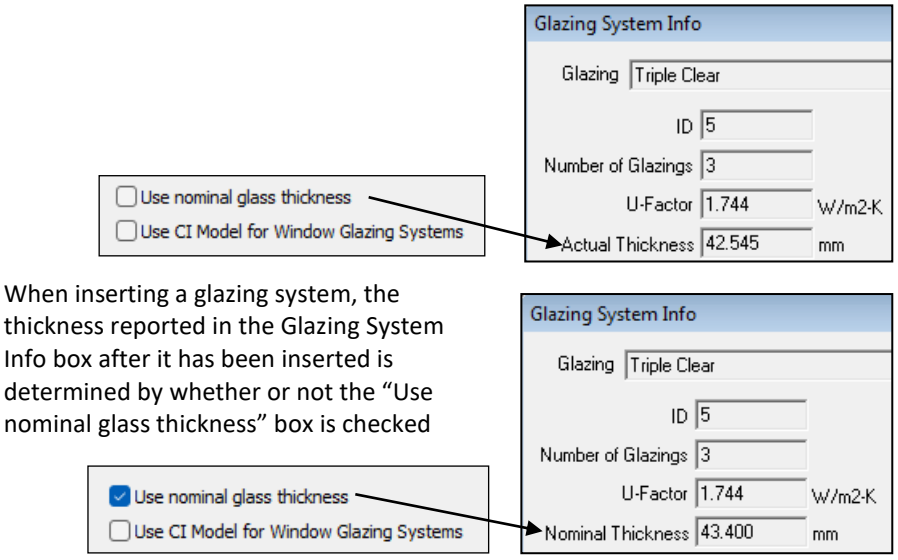


Figure 5-62. How the glazing system was imported determines the thickness reported in the Glazing System Info dialog box.

CI Cavity Height The height of the glazing system cavity used to calculate the CI value. Default value: 39.37 (IP); 1000 (SI). **Units:** inches (IP); mm (SI).

Source The WINDOW database name, including the full directory path the glazing system was imported from.

Layer Properties

This section displays the properties of each of the layers (usually glass) in the glazing system, based on the definition of the glazing system in the WINDOW database it was imported from.

Layer — A pulldown list showing the ID of each layer. Selecting an ID value will show the properties for that layer. The Layer ID numbering system starts with the outside layer (the layer exposed to outside conditions) to the inside. Therefore, ID = 1 will be the outermost layer. See the figure below for an illustration of the ID numbering system.

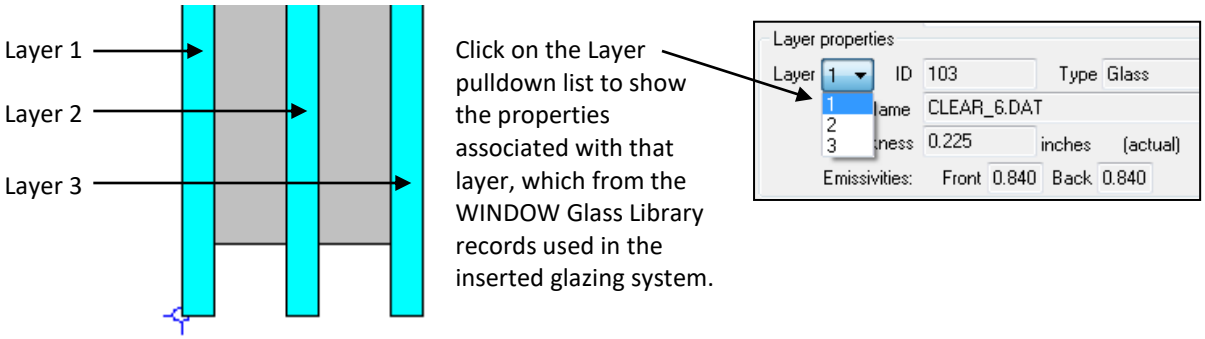


Figure 5-63. How the glazing system was imported determines the thickness reported in the Glazing System Info dialog box for each layer.

ID — The ID of the layer of the glazing system, from the **WINDOW Glass Library**.

Type — The type of layer, from the WINDOW database the glazing system was imported from. It is generally set to "Glass".

Name — Name of the layer from the **WINDOW Glass Library**

Thickness — Thickness of the layer from the **WINDOW Glass Library**. This value will depend on whether the glazing system was imported with "Nominal Thickness" checked.

If Nominal Thickness was checked, the word "(nominal)" will be displayed to the right. If it was not checked, the word "(actual)" will be displayed to the right. Units: inches (IP); mm (SI).

Emissivities — The emissivity values for the front (facing the outside) and back (facing the inside) of each layer

- Front
- Back

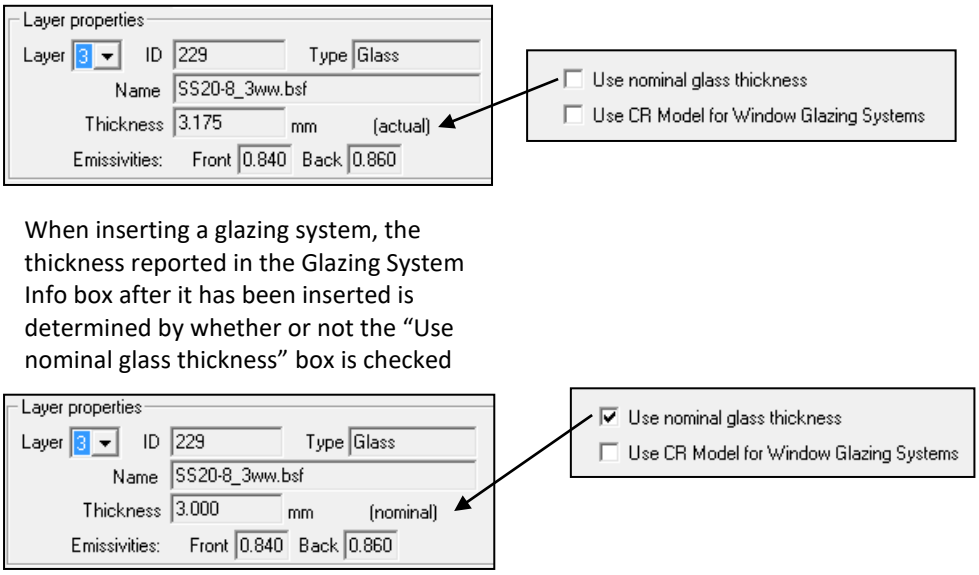


Figure 5-64. How the glazing system was imported determines the thickness reported in the Glazing System Info dialog box for each layer

Gas Properties

Gap The glazing cavity "Gap" number. This field is a pulldown and will have one number for every glazing system cavity, and will cause the Keff value to the right to be displayed for the gap number selected. For double pane glazing systems, there is only one gap, which will be number "1", with the corresponding Keff value for that gap. For triple pane glazing systems, there are two gaps, and there will be two Gap numbers, "1" and "2", one for each gap in the glazing system.

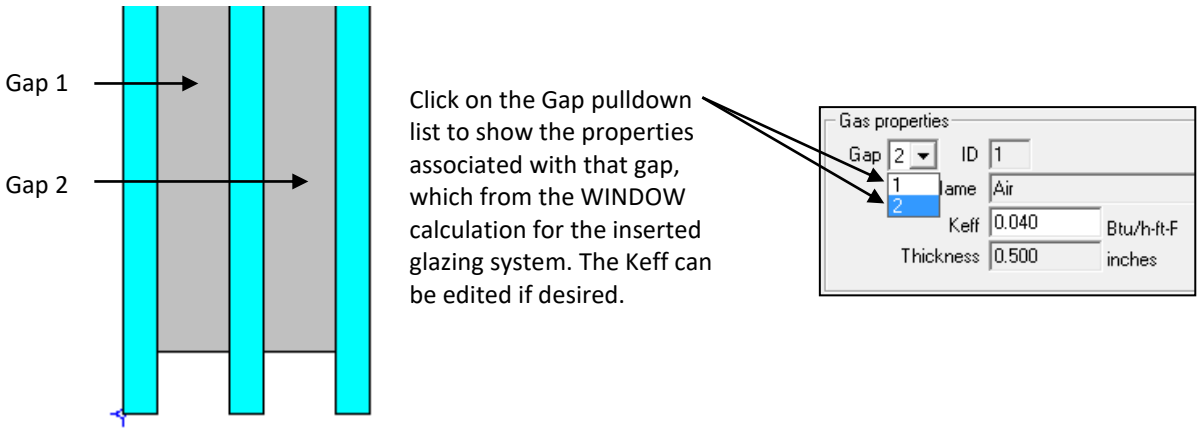


Figure 5-65. The Glazing System Info dialog box allows some glazing system values to be changed.

ID The ID of the gas fill (from the **WINDOW Gap Library**) for the selected gap.

Name The Name of the gass fill (from the **WINDOW Gap Library**) for the selected gap.

Keff The effective conductivity (Keff) of the glazing system gap selected in the Gap field. This field can be edited so that the the Keff value for each gap can be changed from the value imported from the WINDOW program. **Units:** Btu/h-ft-F (IP); W/m-K (SI).

Thickness

Overall thickness of the glazing system. The label on this field will depend on whether or not the glazing system was imported with nominal thickness checked.

If the glazing system was imported with the Nominal Thickness choice checked, this field will display a label "Nominal Thickness" and the value displayed will be the overall thickness for the glazing system calculated using the nominal glass layer thicknesses. If the glazing system was imported without the Nominal Thickness choice checked, this field will display a label "Actual Thickness" and the value displayed will be the overall thickness for the glazing system calculated using the actual glass layer thicknesses. **Units:** inches (IP); mm (SI).

Shading Properties

If a shading system is modeled, modifiers will appear in this section, depending on whether the shading system is modeled on the exterior or interior of the glazing system.

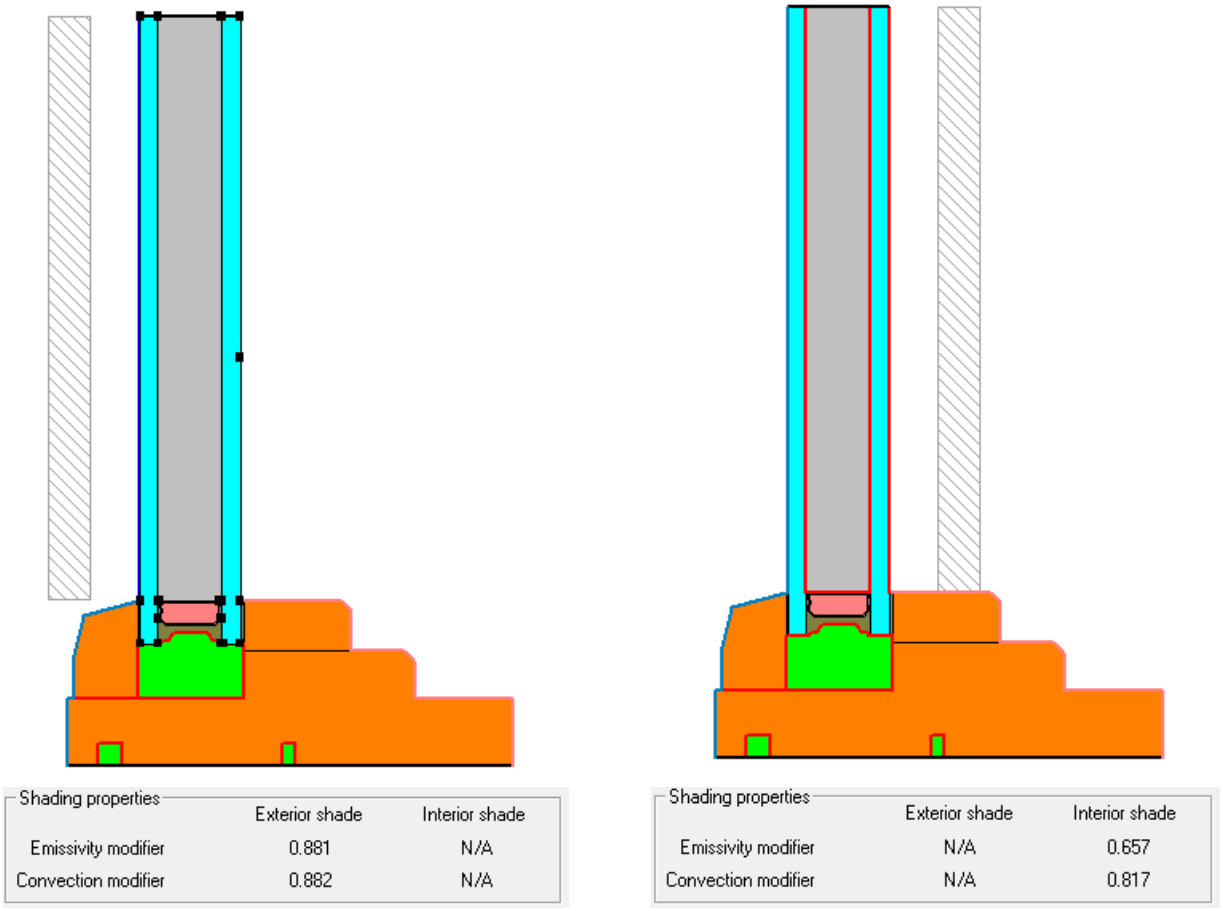


Figure 5-66. The Glazing System Info dialog box (accessed by double clicking on the glazing system) reports the modifiers for Exterior and Interior shading systems.

Emissivity Modifier	For either the Exterior or Interior Shade, based on what is being modeled, the value used to modify the glazing system emissivity.
Convection Modifier	For either the Exterior or Interior Shade, based on what is being modeled, the value used to modify the glazing system emissivity.

5.10.7. Multiple Glazing Options

In THERM it is possible to define many different glazing systems for one cross-section. When the cross-section is calculated, THERM will create a separate file for each glazing system - cross-section combination, and simulate each file. This allows many different glazing options to be calculated in one operation.

To create a cross section with multiple glazing options, follow the steps below (also explained in the accompanying figures):

- Make a cross section that is complete, including the initial, or base case, glazing system.
- Define the multiple glazing system options using either of these two methods:
  - Highlight the glazing system (click on it once with the mouse) and then click on the **Calculation/Glazing Options** menu choice

  OR

  - Double click on the glazing system

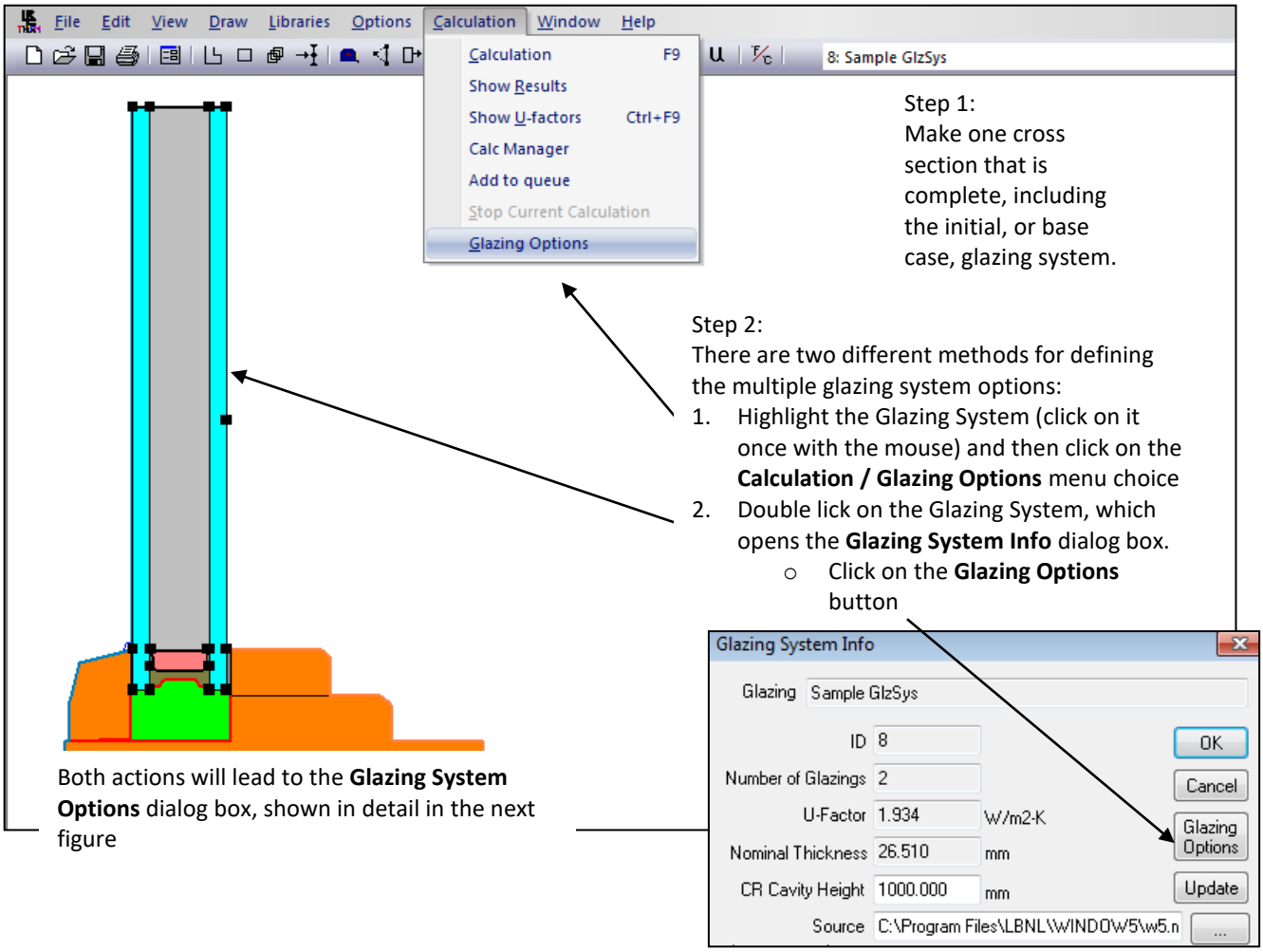


Figure 5-67. The Glazing System Info dialog box, click the Glazing Options button to set up multiple glazing systems for a model.

- In the **Glazing System Options** dialog box, click on the **Add** button.
- The **Add Glazing Option** dialog box will show all the glazing systems that match the base glazing system in terms of glass layer thickness and overall thickness.
- Select the individual glazing systems (use Shift with the left mouse button to select multiple contiguous system or Control with the left mouse button to select non-contiguous systems), or click on the **Select All** button to select all the glazing systems listed.
- To select another WINDOW glazing system file, click on the **Browse** button and select the desired file.
- The selected **Glazing Systems** will appear in the **Glazing System Options** dialog box, and are now associated with the file.
- Select the desired naming convention radio button:
- **Glazing System Name:** the program will name each of the glazing options files by append the glazing system name onto the base case THERM file name.
- **Glazing System ID:** the program will name each of the glazing options files by append the glazing system ID number onto the base case THERM file name.

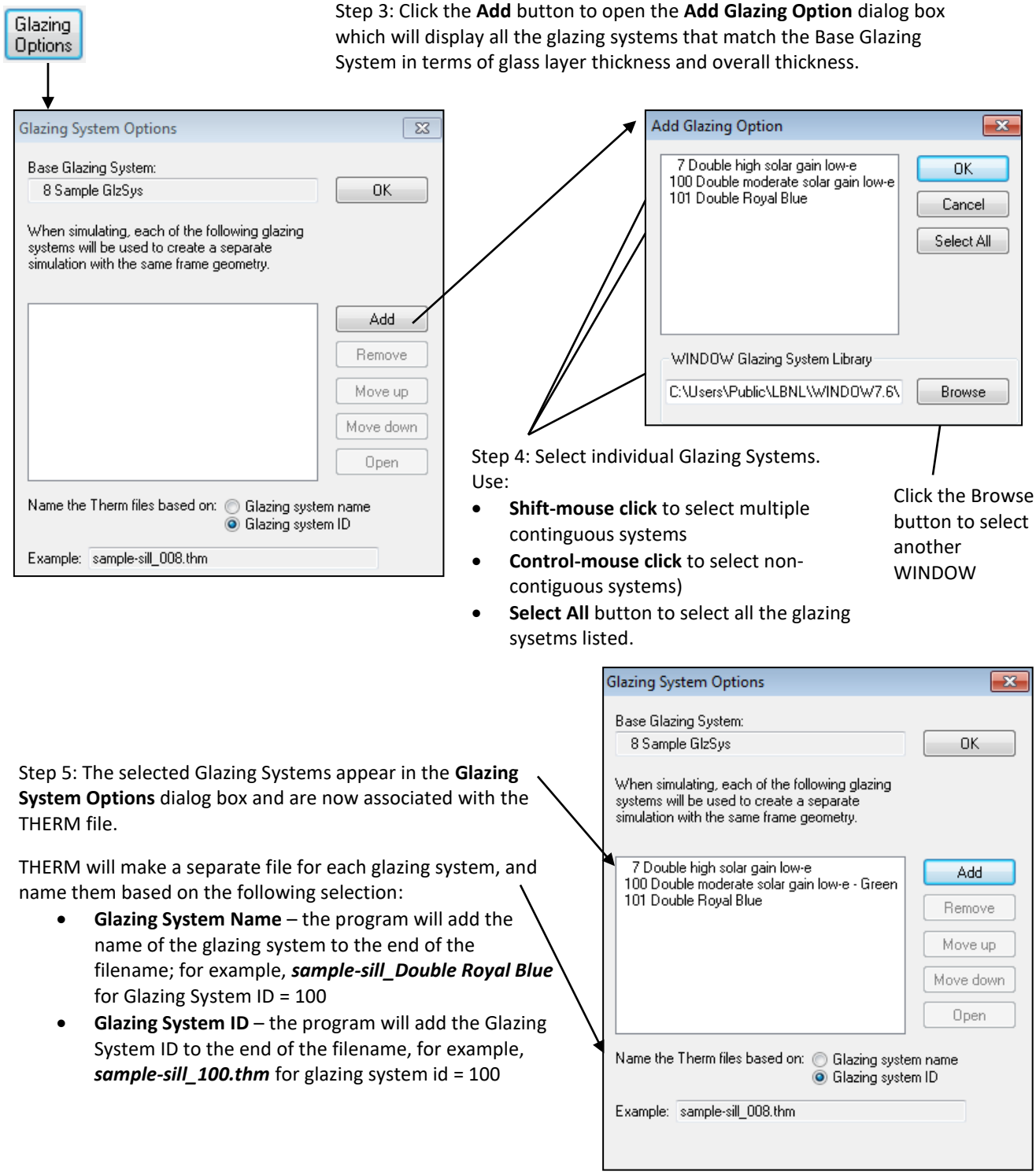


Figure 5-68. Select the glazing systems to be associated with the base frame cross section.

- To start the simulation for the multiple glazing options, click on the **Calc** toolbar button (or select the **Calculation/Calculation** menu choice, or press the **F9** function key).
- The **Glazing Option Simulation** dialog box will appear. Select one of the following options:
  - **Create the glazing options files and perform all simulations:** this is the default setting, and will create an individual THERM file for each glazing option that has been defined, naming it as specified

in the **Glazing Systems Options** dialog box (either the **Glazing System Name** or **ID** appended onto the base file name), and then add all the created files into the **Calc Manager** to be simulated.

- **Simulate the current file only:** this choice will not create the individual glazing option files, but instead will only simulate the base case file. This option can be used to make sure that the base file is properly defined before sending all the glazing option files to the **Calc Manager**, where they might fail to simulate because of a problem with the base case file.
- **Create the glazing option files but do not perform the simulations:** this choice will create the individual glazing option THERM files, but will not simulate them. This may be useful in the unusual cases where the individual files need to changed after they are generated and before they are simulated. These files can be individually added to the **Calc Manager** after they are created.

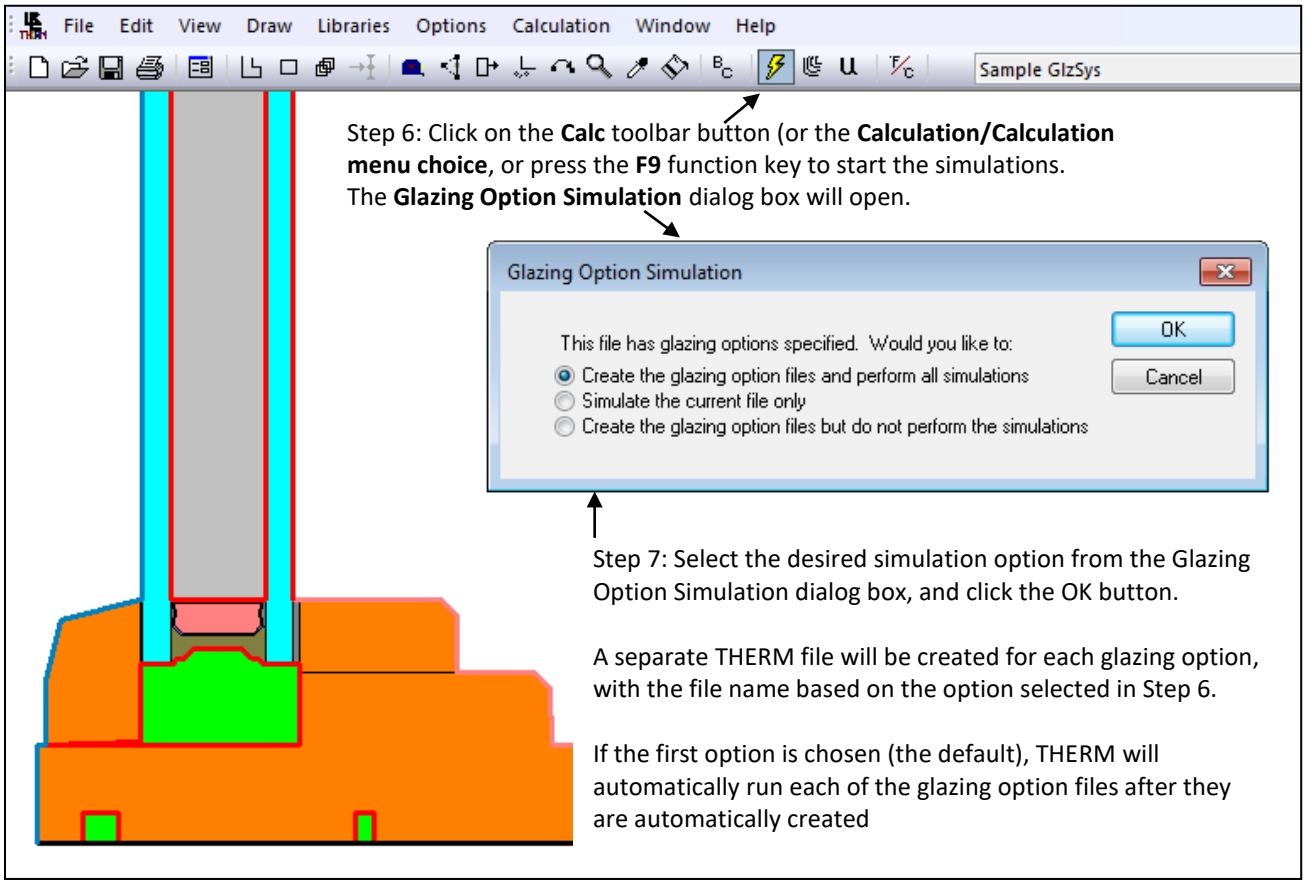
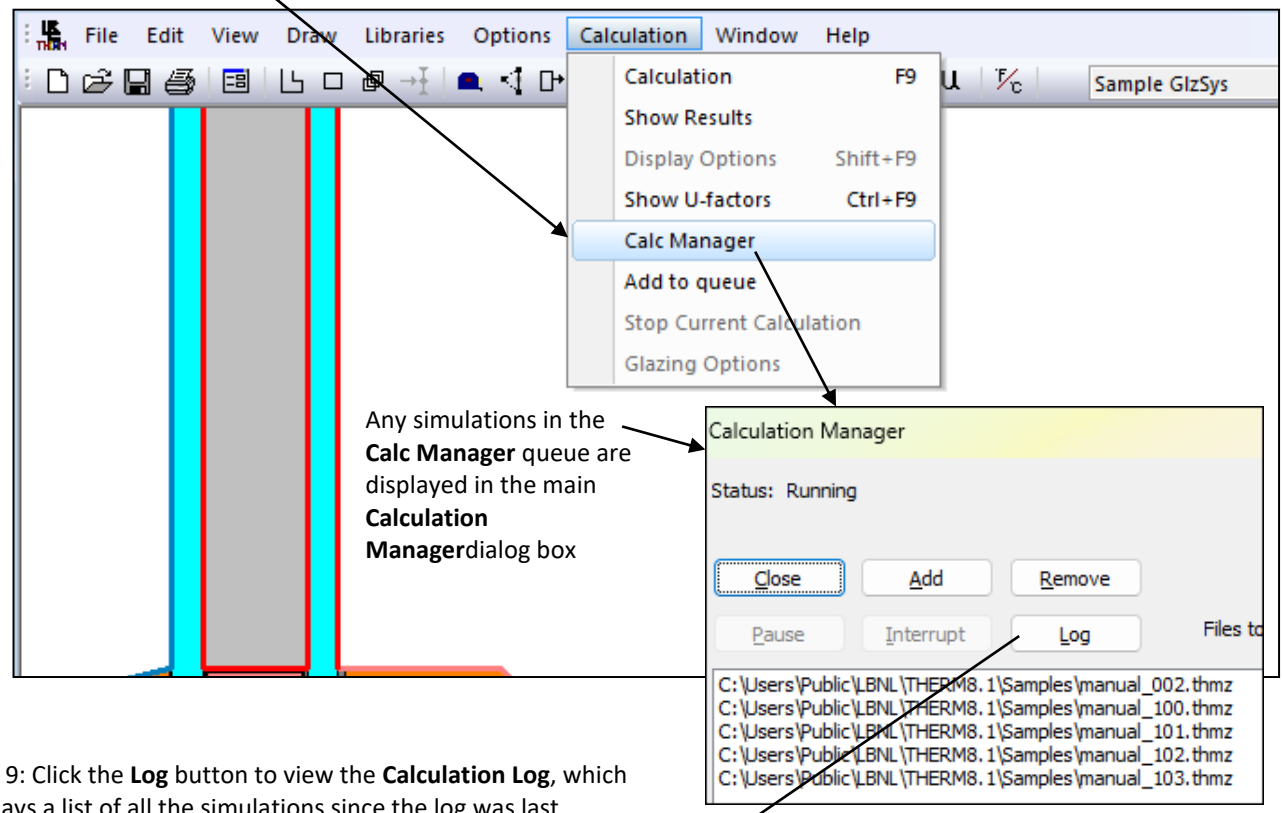


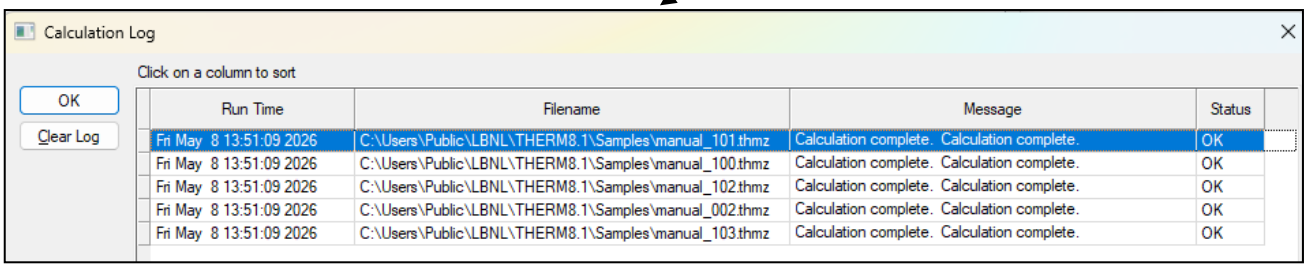
Figure 5-69. Calculate the multiple glazing options.

- If the glazing options files are automatically created and simulated, the status of the runs can be viewed by going to the **Calculation/Calc Manager** menu choice.
- From the **Calculation Manager** dialog box, click on the **Log** botton to see the **Calculation Log**. In the **Calculation Log**, all files that have been submitted to the program for calculation will be displayed. This log can be sorted by each of the column headings (by clicking on the column). The **Status** column will display whether the files simulated, showing either "OK" if the simulation completed, or "Error" if it did not. The **Message** column will display text explaining briefly why the file did not simulate if the **Status** is "Error".

Step 8: To see the status of the simulations,click the **Calculation/Calc Manager** menu option.



Step 9: Click the **Log** button to view the **Calculation Log**, which displays a list of all the simulations since the log was last cleared. The "Status" column will show whether the simulation completed successfully.



The individual THERM files generated from the Glazing Options feature will be in the same directory as the base case file.

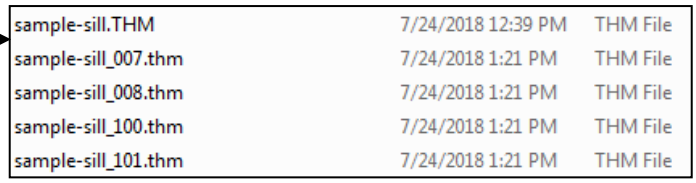


Figure 5-70. Calculate the multiple glazing options.

5.10.8. Editing Keff Value of Glazing System Cavity

The effective conductivity (Keff) values for the glazing system cavities can be edited if the glazing systems have been imported from WINDOW. This can sometimes be necessary for special cases. The example shows below is for a storm window model where a specific keff value has been calculated for one of the cavities.

The Keff values can either be edited at the time the glazing system is imported, or after the glazing system is already in the cross section. To edit the Keff values of the glazing system cavities, follow the steps listed below.

To Edit the Keff Value When Importing the Glazing System:

- Import the desired glazing system using the **Library/Glazing Systems** menu choice, or pressing the **F6** key.
- Select the glazing system from the **Glazing Systems** dialog box and click on the **Import** button.
- The **Insert Glazing System** dialog box will appear.
    - In the Gap Properties section, click on the **Custom** radio button, which will make the **Keff** and **Width** fields editable.
    - Pick the **Gap** number to edit from the **Gap** pulldown list.
    - Enter the new **Keff** and **Width** values.

To Edit the Keff Value For a Glazing System Already in a Cross Section:

- Double click on the glazing system to display the Glazing System Info dialog box.
    - Select the **Gap** to be edited using the **Gap** pulldown list
    - The **Keff** values are displayed in the **Keff** input box, and can be edited.

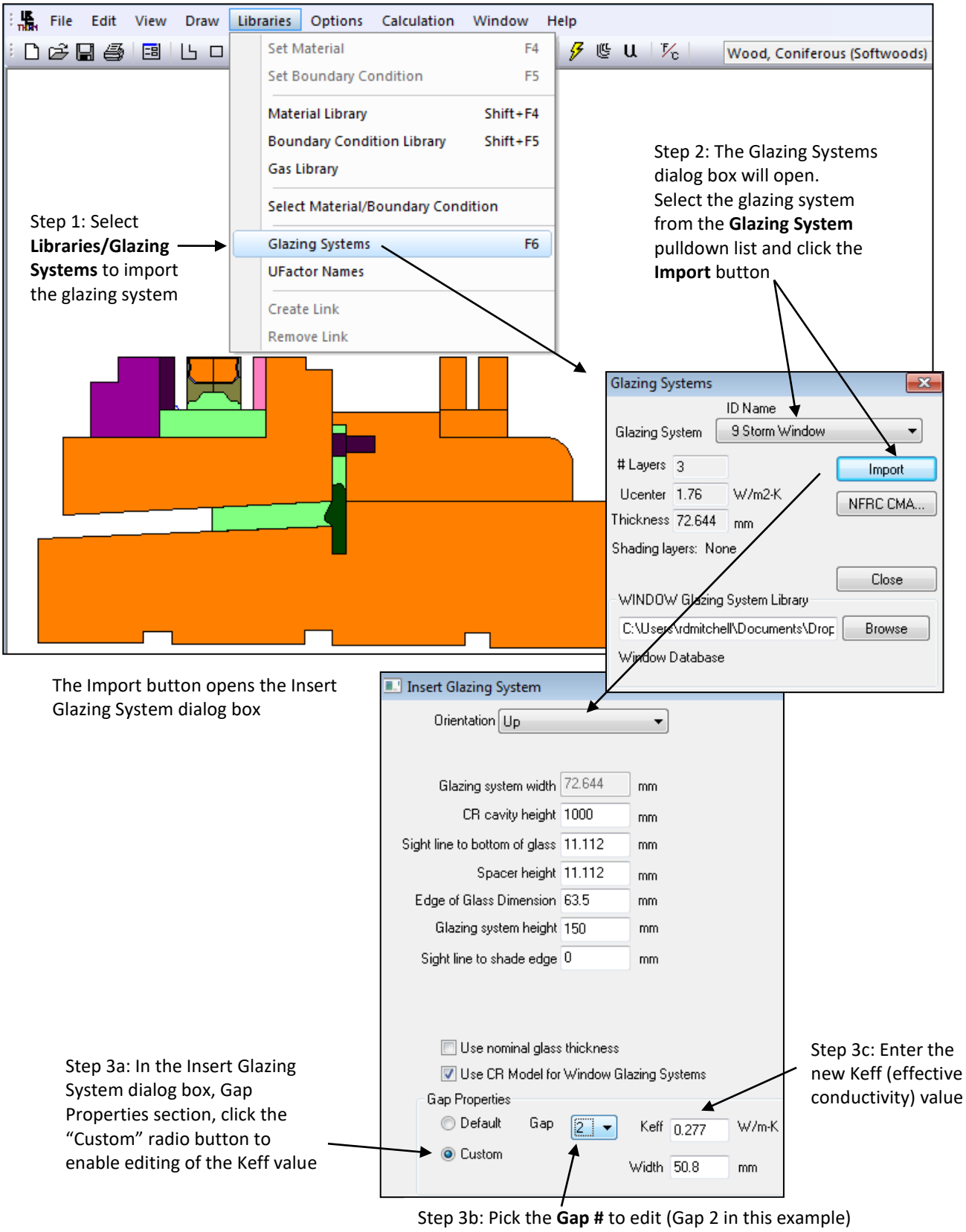


Figure 5-71. Change the keff value of glazing cavities when importing the glazing system.

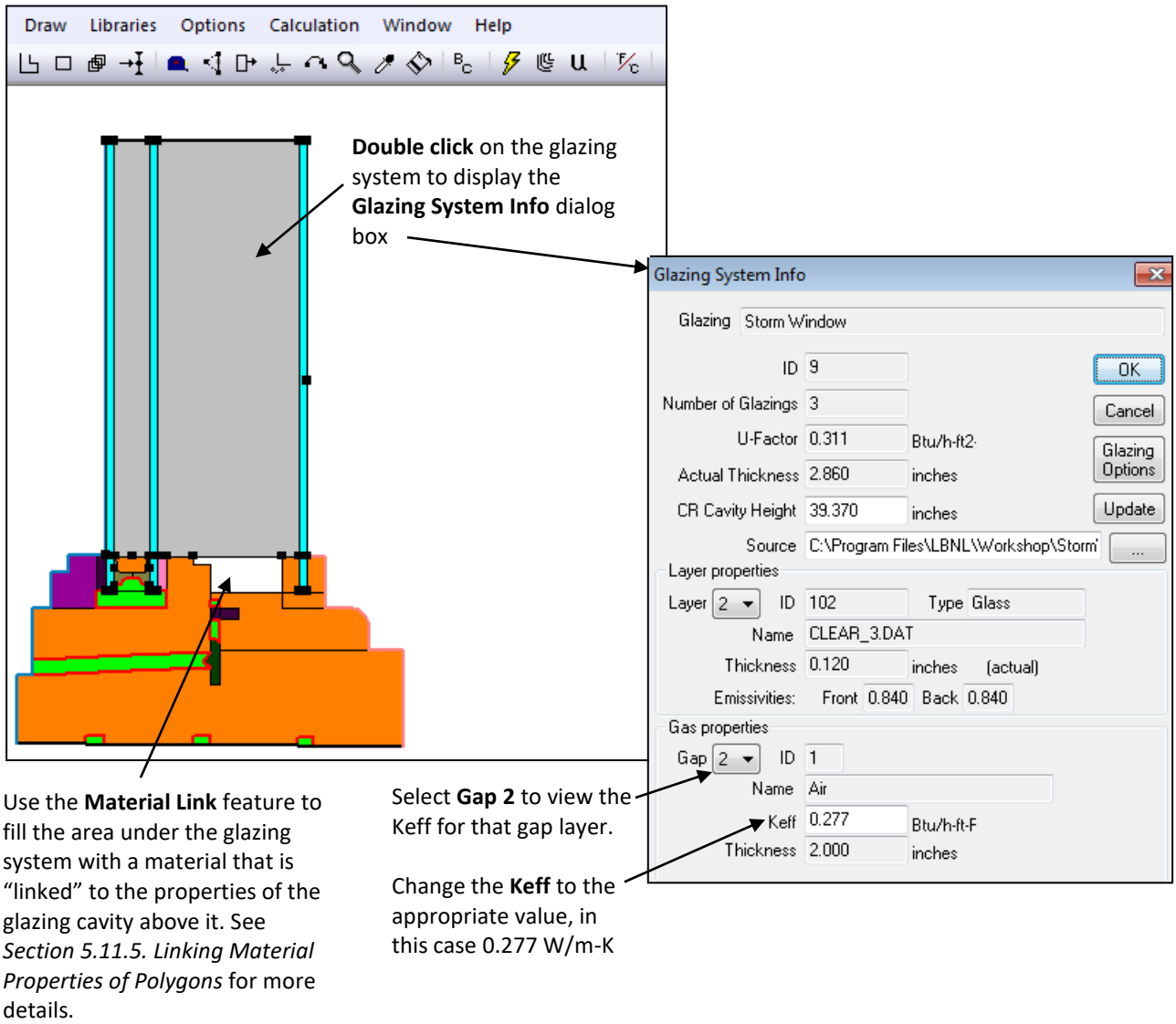


Figure 5-72. Change the keff value of glazing cavities after a glazing system is already imported.

5.11. 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 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 gap cavity). It is recommended that the glazing cavity be modeled by importing a glazing system from WINDOW in which case THERM automatically creates the glazing cavity based on the properties from WINDOW.

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."

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

Select the polygon by placing your mouse inside the object and clicking the left mouse button.

The vertices of the selected polygon are indicated as squares.

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. 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.

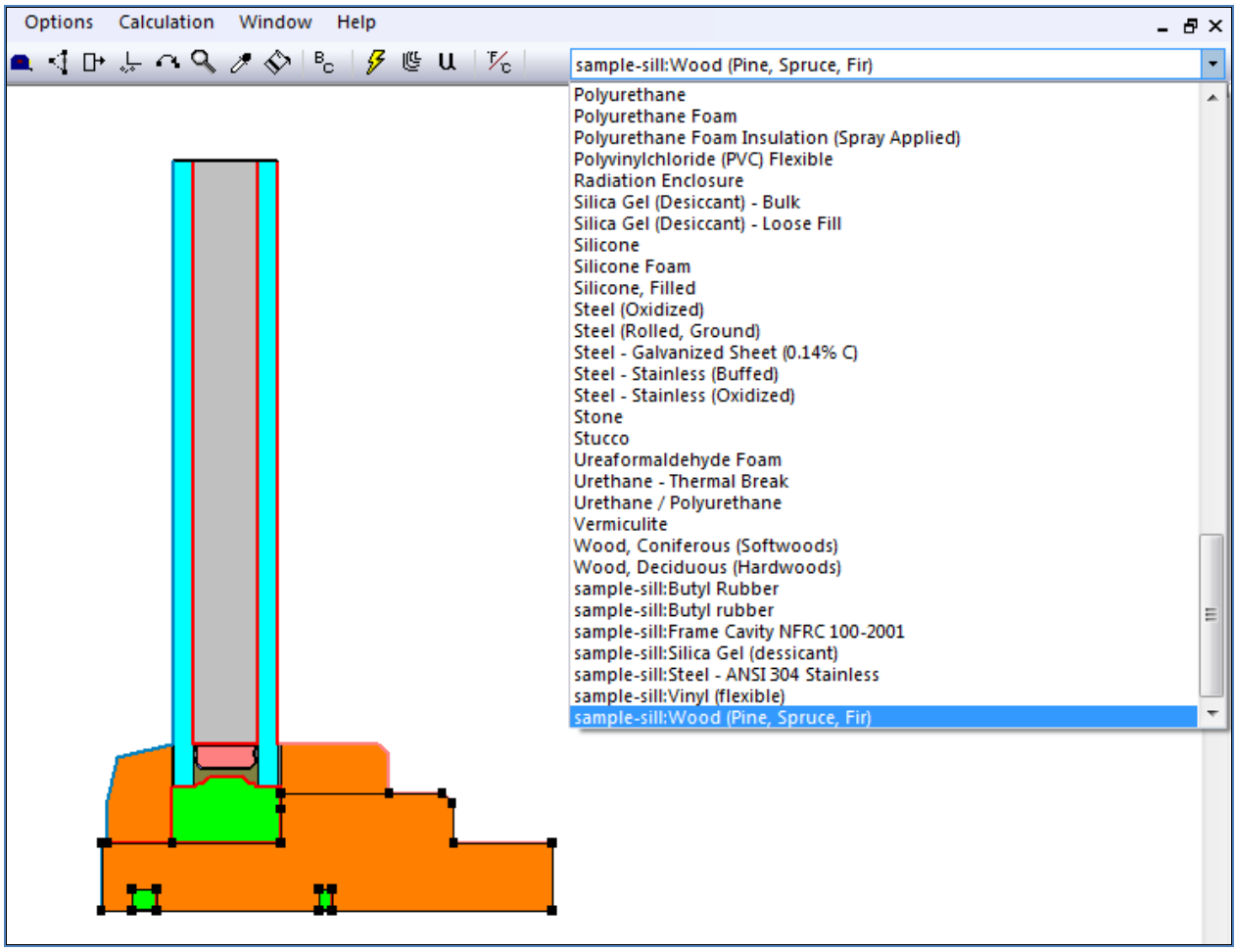


Figure 5-73. You can select polygons after they are drawn and assign materials to them.

5.11.2. 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.

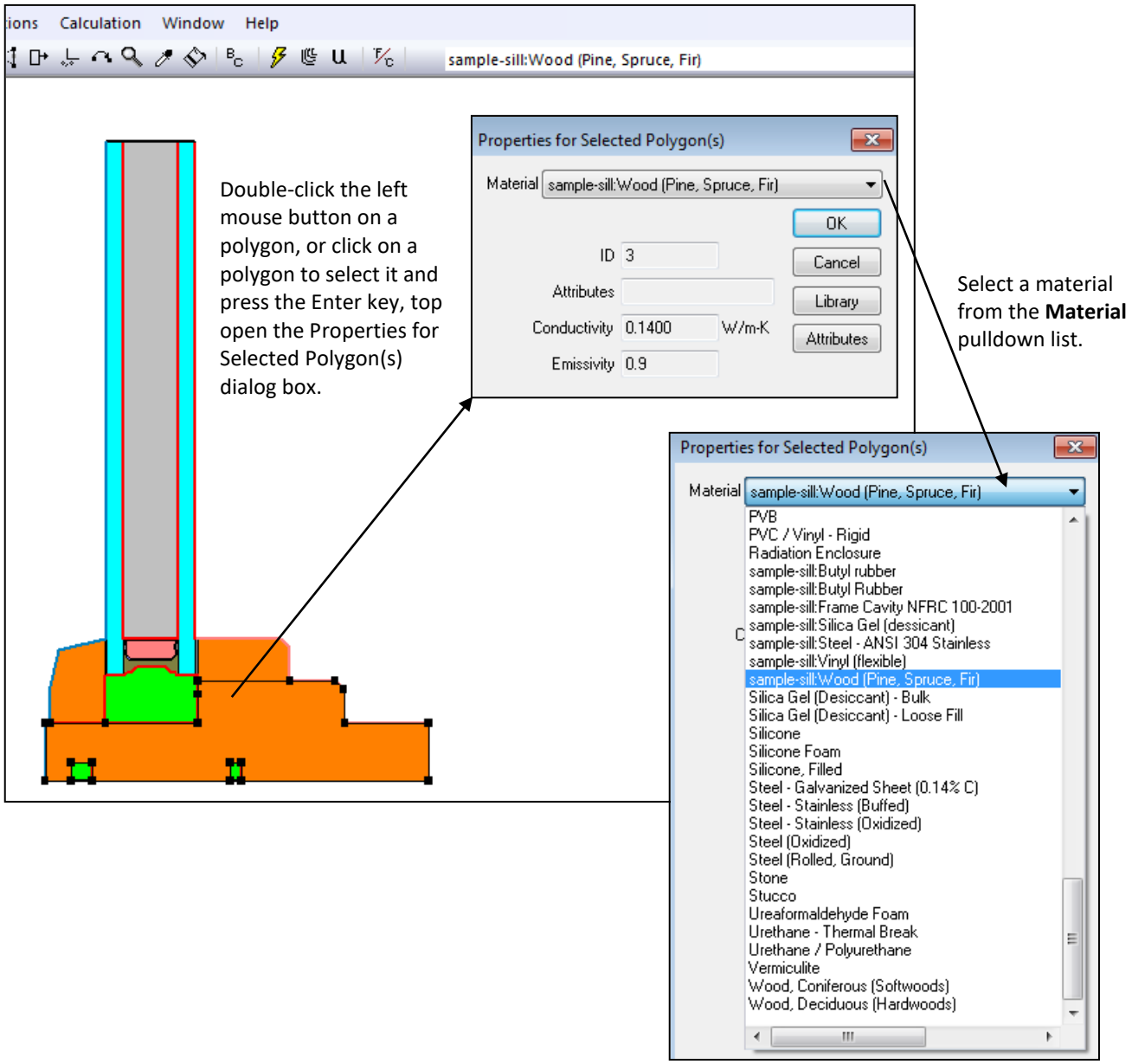


Figure 5-74. Material properties for solid material are accessed by double-clicking mouse on polygon.

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 *5.12. Material Library*. However, all material types will have the following fields:

Material A pull-down list which contains 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 The unique ID number assigned to every polygon in the cross section. This number is useful for identifying specific objects in the THERM reports and when using the **Edit/Select Special** option.

Attributes Selecting an attribute for a polygon is used for the NFRC CMAST program.

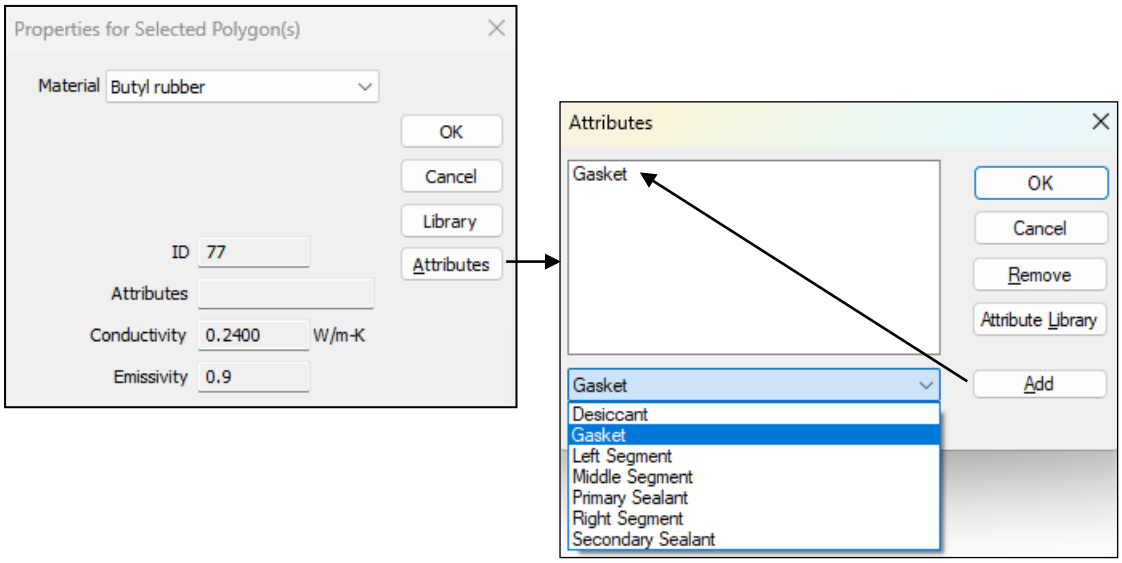


Figure 5-75. The Attributes dialog box

Conductivity The conductivity of the material for the selected polygon. Units: Btu/h-ft-oF (IP); W/m-K (SI).

Emissivity The emissivity of the material for the selected polygon. The emissivity of a material can be changed after the material is assigned if it is next to a frame cavity.

5.11.3. Solid Materials

For solid materials, you can press the **Library** button to access the **Material Definitions** dialog box for that material to see how the material is defined. For solid materials, the primary characteristics defined are Conductivity and Emissivity.

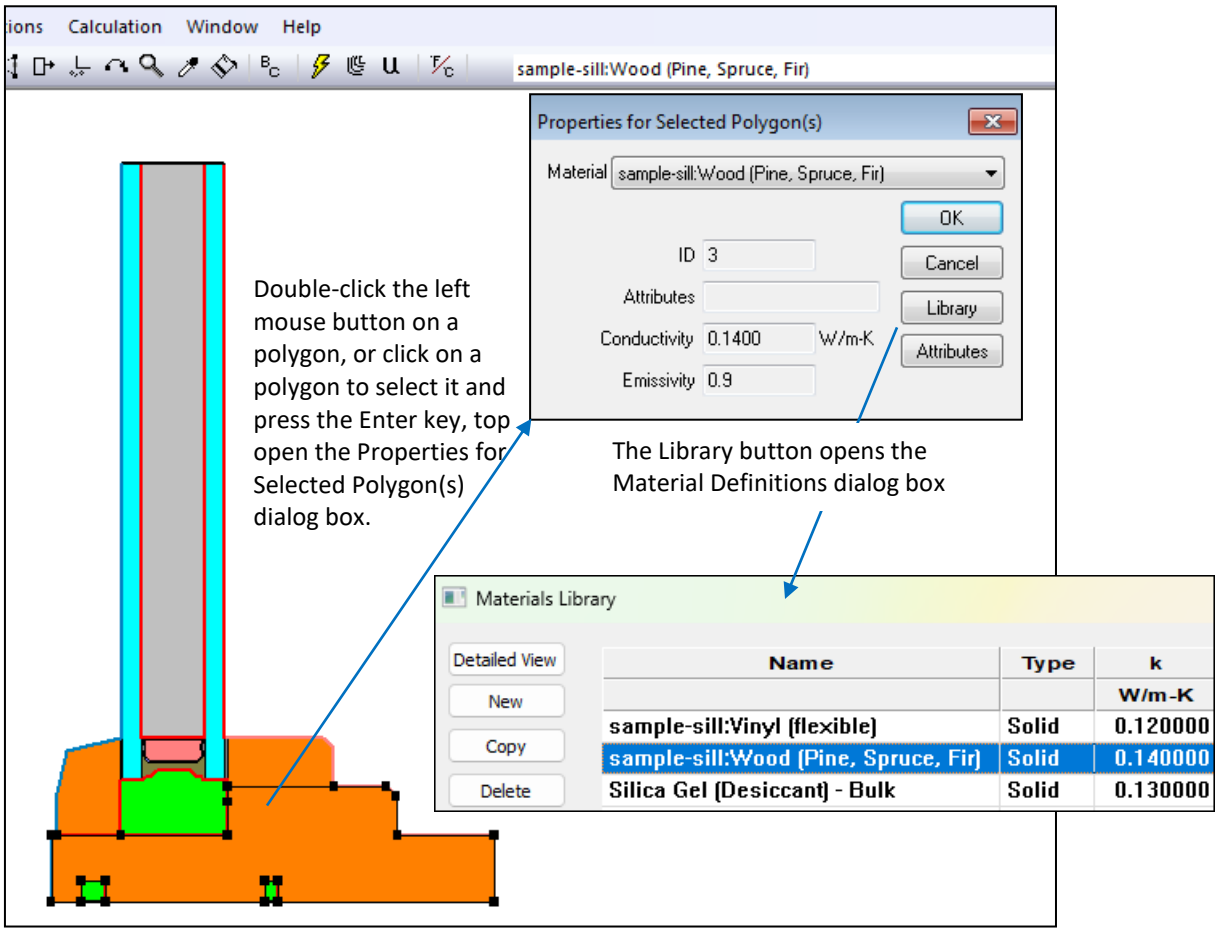
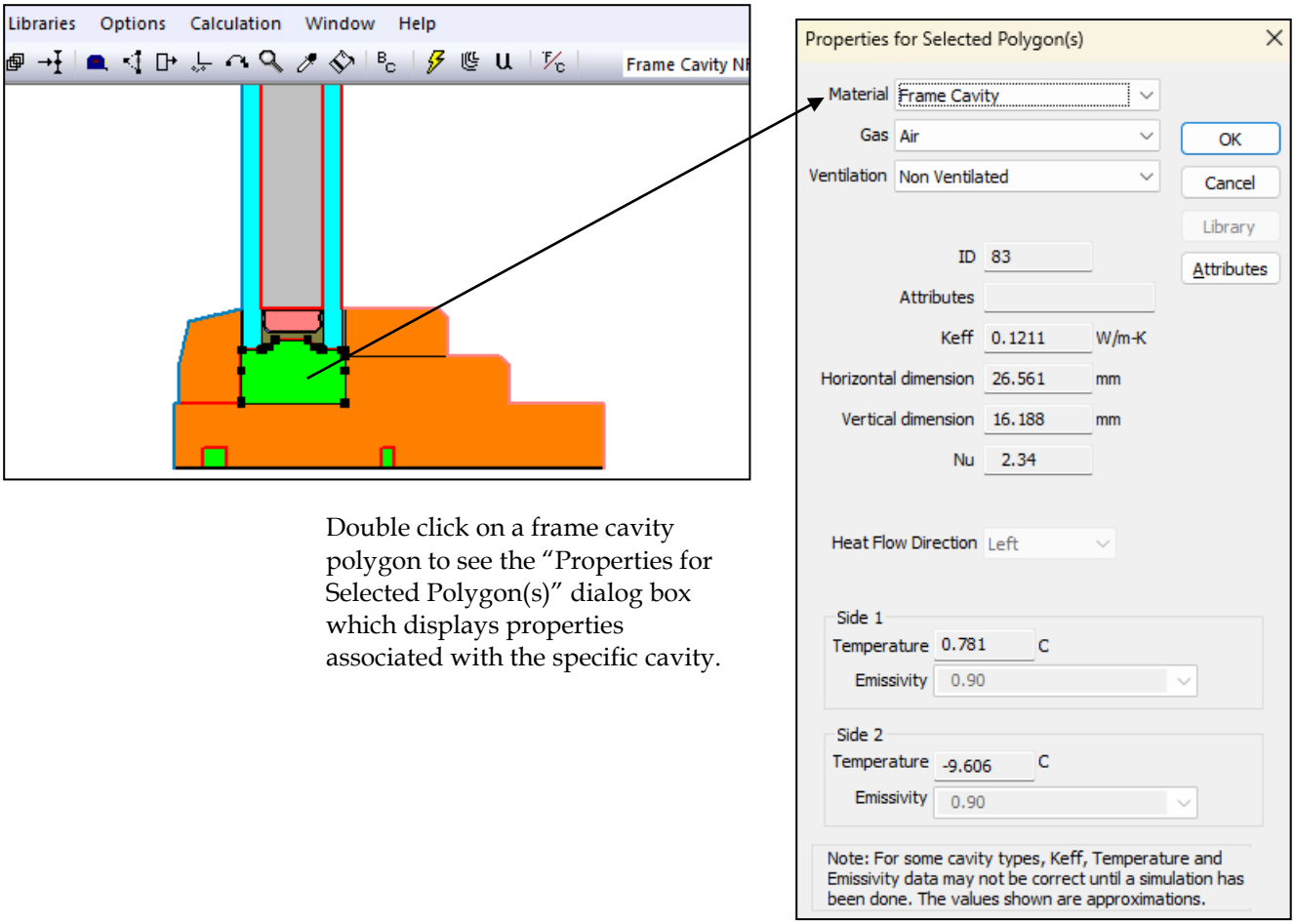


*Figure 5-76. **Material Definitions** dialog box accessed from the **Library** button shows the **Conductivity** and **Emissivity** for **Solids**.*

5.11.4. Frame Cavities

Frame cavity have a more complicated definition than solid materials. After they have been defined, if a Frame Cavity material is selected for a polygon, the following information can be viewed for each specific cavity in the model.



Double click on a frame cavity polygon to see the "Properties for Selected Polygon(s)" dialog box which displays properties associated with the specific cavity.

Figure 5-77. Frame cavity details can be edited by double-clicking the mouse on polygon, or by highlighting the polygon and pressing the Enter key.

Keff	Effective conductivity, calculated by THERM based on the Cavity Model choice specified in the Material Library.
Nu	Nussult number, calculated by THERM based on the Cavity Model choice in the Material Library.
Vertical and Horizontal dimension	These are the 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, in the Material Library definition for the cavity, the Cavity Model (under Cavity Properties) is set to "NFRC with user dimensions".
Heat Flow Direction	The heat flow direction in the frame cavity. The possible directions are Left, Right, Up and Down. This choice determines what correlation will be used to

calculate the convective film coefficient. Left and Right heat flow 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.

Side 1 and 2 Temperatures

Before the simulation is run, these are the starting temperatures for the iterative calculation. After the simulation is run, these are the average temperatures of the vertical sides of the cavity.

Side 1 and 2 Emissivities

The Emissivities 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 Left or Right heat flow. The default values associated with the Material can be edited by double clicking on specific Frame Cavity boundary conditions and "hand editing" the emissivities These will then be the emissivities used in the calculations and should be reflected in the resulting emissivities reported in this dialog box after the simulation.

5.11.5. Linking Material Properties of Polygons

The material properties of one polygon can be linked to other polygons, including the polygons which define glazing cavities. This is useful in an example such as in the figure below, where the area below the glazing system cavity has the same properties as the cavity.

If the properties of the polygon that is linked from change, then the properties of the polygons linked to it will also change.

To Link the properties of two materials, follow these steps:

- Select the polygon that is to be linked to another polygon
- From the Libraries menu, select the Create Link option.
- The cursor will become an Eye Dropper. Click the Eye Dropper cursor in the polygon to be linked to. The material properties of the first polygon are now linked to the material properties of the second polygon.

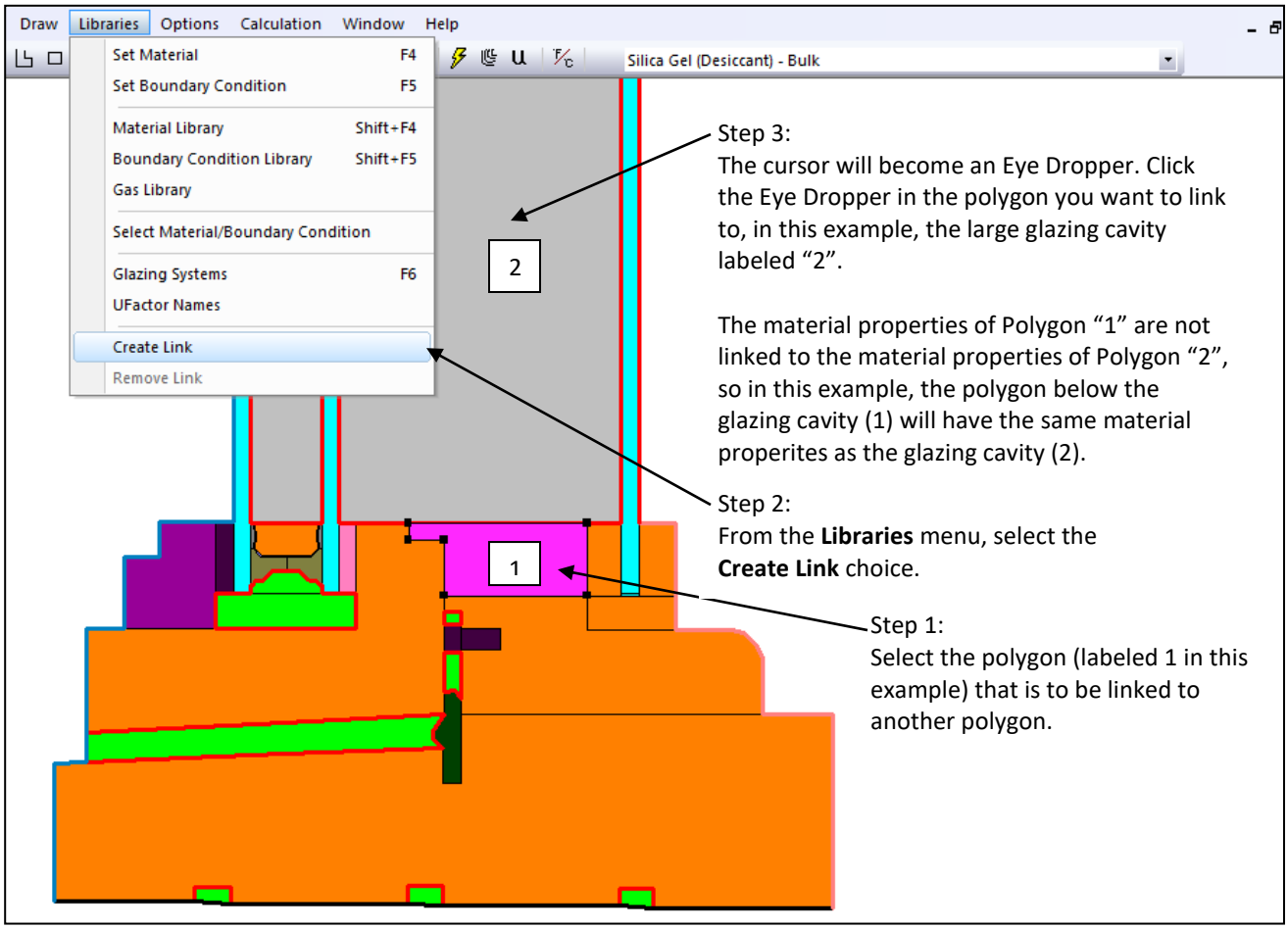
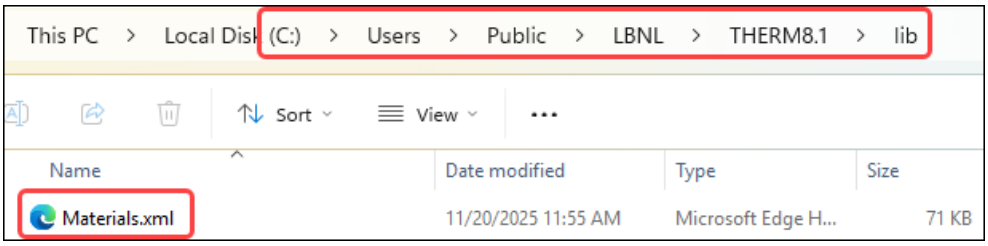


Figure 5-78. Material properties for cavities accessed by double-clicking mouse on polygon.

5.12. Material Library

The Material Library is an XML file that is stored in the "lib" folder of the THERM working directory, which by default is:

C:\Users\Public\LBNL\THERM8.1\lib



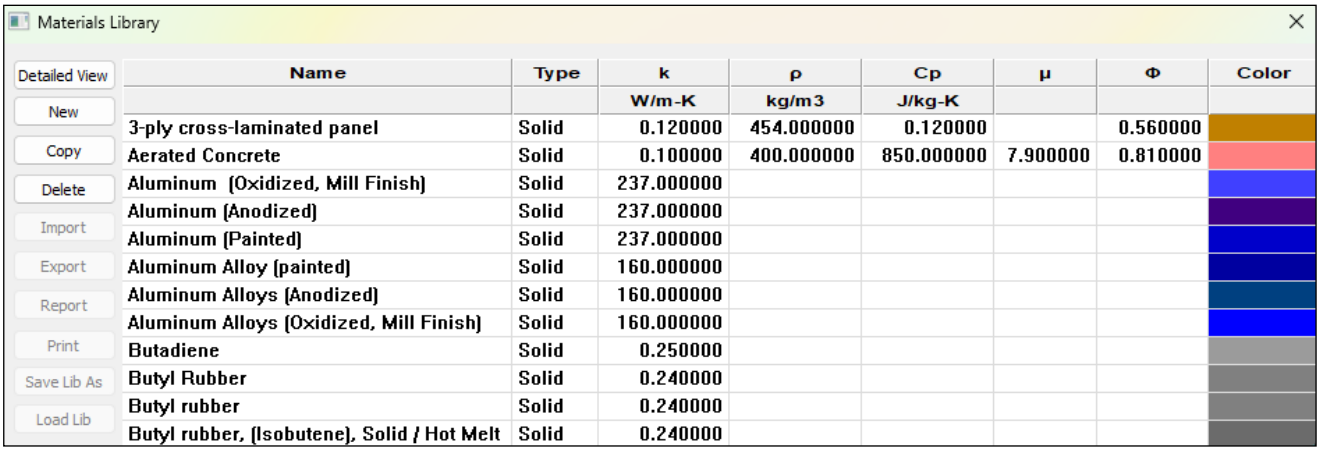
There are four material types in the THERM Material Library:

- Solid
- Shade Material: This is a special option in THERM used to define materials that can be used when modeling shading systems with unusual geometries, such as cellular and pleated shades, louvered shutters, and roller shutters. This option will only appear in the Material Library if the Material Type is set to "Shading Material".

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

5.12.1. Material Library List View

Open the **Material Library** by selecting the **Library/Material Library** menu choice, and the **Materials Library** dialog box is displayed. It contains a tabular list (List View) of all the materials currently defined in the library.



Name	Type	k	ρ	Cp	μ	Φ	Color
		W/m-K	kg/m3	J/kg-K			
3-ply cross-laminated panel	Solid	0.120000	454.000000	0.120000		0.560000	
Aerated Concrete	Solid	0.100000	400.000000	850.000000	7.900000	0.810000	
Aluminum (Oxidized, Mill Finish)	Solid	237.000000					
Aluminum (Anodized)	Solid	237.000000					
Aluminum (Painted)	Solid	237.000000					
Aluminum Alloy (painted)	Solid	160.000000					
Aluminum Alloys (Anodized)	Solid	160.000000					
Aluminum Alloys (Oxidized, Mill Finish)	Solid	160.000000					
Butadiene	Solid	0.250000					
Butyl Rubber	Solid	0.240000					
Butyl rubber	Solid	0.240000					
Butyl rubber, (Isobutene), Solid / Hot Melt	Solid	0.240000					

Figure 5-79. The default Material Library that comes with THERM has only "Protected" materials that cannot be edited.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.
Import	Used to import records from THERM 7.8 Material.lib files. This import will bring in all records in the THERM 7.8 file into the THERM 8.1 library.
Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Type	The material type, from the options of ▪ Solid ▪ Shading Material
k	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI)
ρ	Density of the material. **Units:** lb/ft3 (IP); kg/m3 (SI)
Cp	Specific heat of the material. **Units:** Btu/lb-F (IP);J/kg-K (SI)
μ	Water Vapor Diffusion Resistance Factor. **Units**: unitless
Φ	Porosity of the material. **Units:** unitless
Color	The color assigned to the material.

5.12.2. Material Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each material.

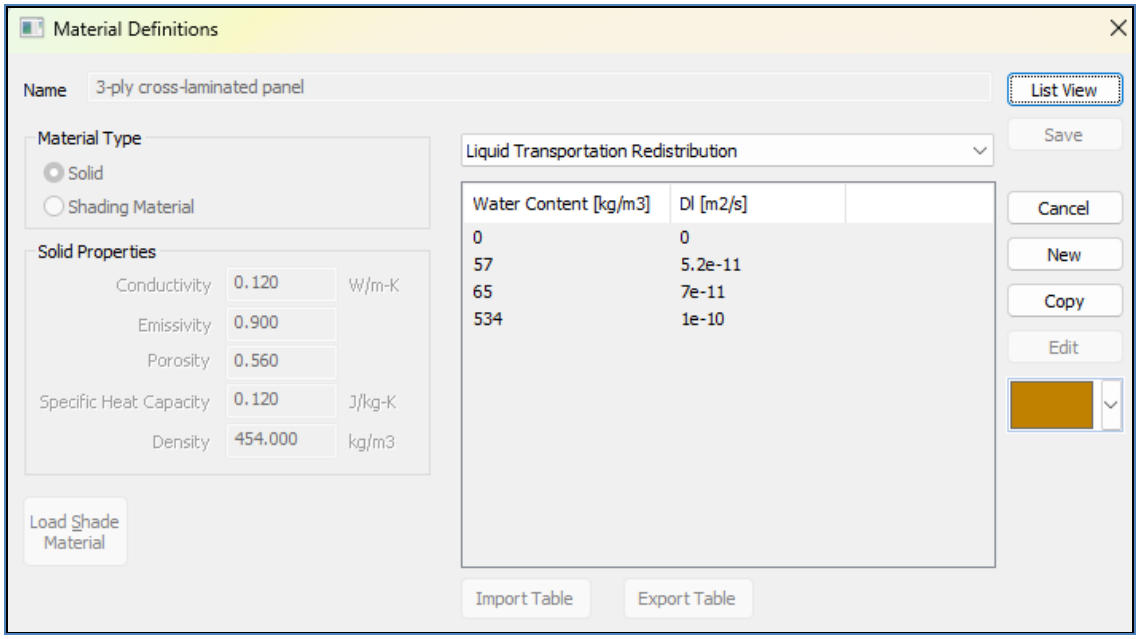
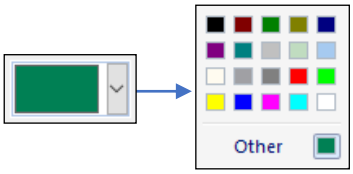


Figure 5-80. Material Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Material Library.
Save	Saves any changes made to the Materials XML file. The default materials in the library are "protected" from editing, but if you make a new record, or copy an existing record (with a different name), the values can then be changed.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. The current record will be copied and all the parameters will be editable. Change the name, set the parameters accordingly and Save the copied record.
Edit	*Currently not implemented.*

Color	The color assigned to the material. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.  *Figure 5-81. Color Settings in the Material Library Detail View.*

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Material Type	
	The material type, from the options of ▪ Solid ▪ Shading Material
Solid Properties *These input values are for Material Type = Solid*	
Conductivity	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Emissivity	Emissivity of the material. **Units:** unitless
Porosity	Porosity of the material. **Units:** unitless
Specific Heat Capacity	Specific Heat Capacity of the material. **Units:** Btu/lb-F (IP); J/kg-K (SI).
Density	Density of the material. **Units:** lb/ft3 (IP); kg/m3 (SI).

Liquid Transportation Redistribution

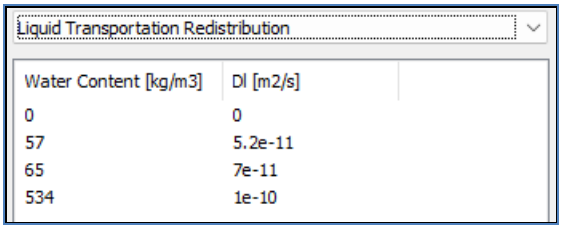
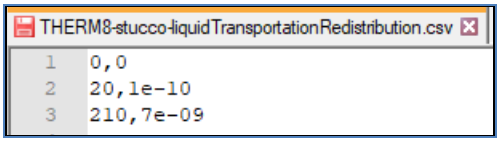


Figure 5-82. Liquid Transportation Redistribution variables for Stucco material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Dl	Liquid transportation coefficient. Units: ft2/s (IP); m2/s (SI).
Import Table	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content ▪ Dl (liquid transportation coefficient)  *Figure 5-83. Import CSV file format for Liquid Transportation Redistribution variables, always in SI units.*
Export Table	The Liquid Transportation Redistribution properties for the material will be exported as a CSV in the same format as shown for importing the file.

Liquid Transportation Suction

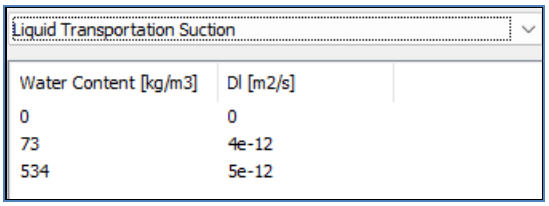
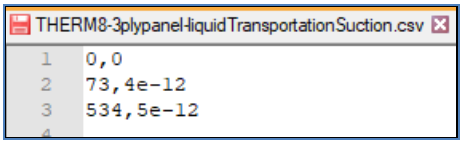


Figure 5-84. Liquid Transportation Suction variables for 3-ply cross-laminated panel material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Dl	Liquid transportation coefficient. Units: ft2/s (IP); m2/s (SI).
Import File Format	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content (kg/m3) ▪ Dl (liquid transportation coefficient, m2/s)  *Figure 5-85. Import CSV file format for Liquid Transportation Suction variables for 3-ply cross-laminated panel material, always in SI units.*
Export Table	The Liquid Transportation Suction properties for the material will be exported as a CSV in the same format as shown for importing the file.

Moisture Dependent Thermal Conductivity

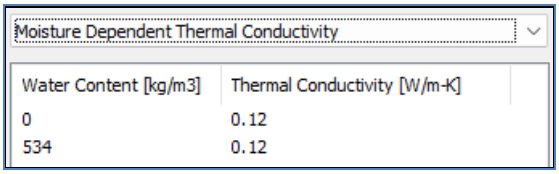
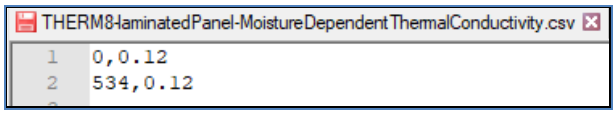


Figure 5-86. Moisture Dependent Thermal Conductivity variables for Laminated Panel material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Thermal Conductivity	Thermal conductivity. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Import File	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content (kg/m3) ▪ Thermal Conductivity (W/m-K)  *Figure 5-87. Import CSV file format for Moisture Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.*
Export Table	The Mosture Dependent Thermal Conductivity properties for the material will be exported as a CSV in the same format as shown for importing the file.

Moisture Storage Function

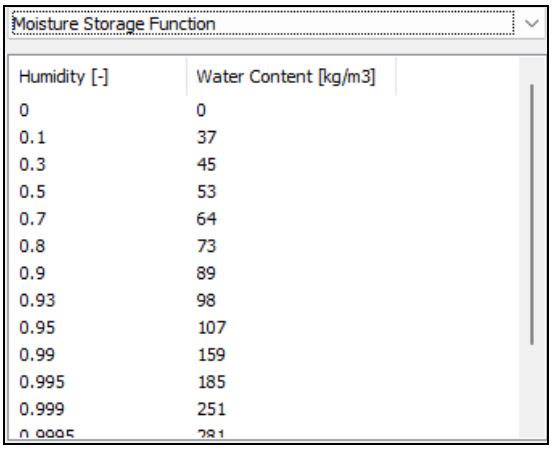


Figure 5-88. Moisture Storage Function variables for Laminated Panel material.

Humidity	Humidity fraction. Legal values: 0 – 1. Units: unitless.
Water content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Import File *Export File*	To import data into this function, create a CSV file with the two values separated by a comma. The data can also be exported into a CSV file of the same format as the import file. The values (always in SI units) are: ▪ Humidity (fraction, 0-1) ▪ Water Content (kg/m3) THERM8-LaminatedPanel-MoistureStorageFunction.csv 1 0,0 2 0.1,37 3 0.3,45 4 0.5,53 5 0.7,64 6 0.8,73 7 0.9,89 8 0.93,98 9 0.95,107 10 0.99,159 11 0.995,185

Figure 5-89. Import CSV file format for Moisture Storage Function variables for Lamianted Panel material, always in SI units.

Temperature Dependent Thermal Conductivity

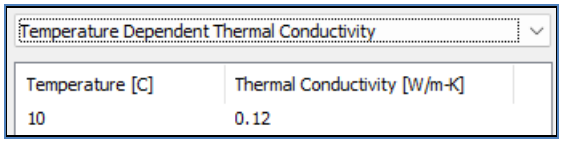


Figure 5-90. Temperature Dependent Thermal Conductivity variables for Laminated Panel material.

Temperature	Temperature. Units: F (IP); C (IP).
Thermal Conductivity	Thermal conductivity. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Import File *Export File*	To import data into this function, create a CSV file with the two values separated by a comma. The data can also be exported into a CSV file of the same format as the import file. T The values (always in SI units) are: ▪ Temperature (C) ▪ Thermal Conductivity (W/m-K)

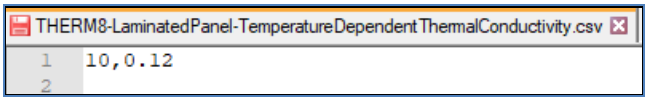


Figure 5-91. Import CSV file format for Temperature Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.

File Format and Location

The Material Library is stored in an XML file called "Materials.xml" which is located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

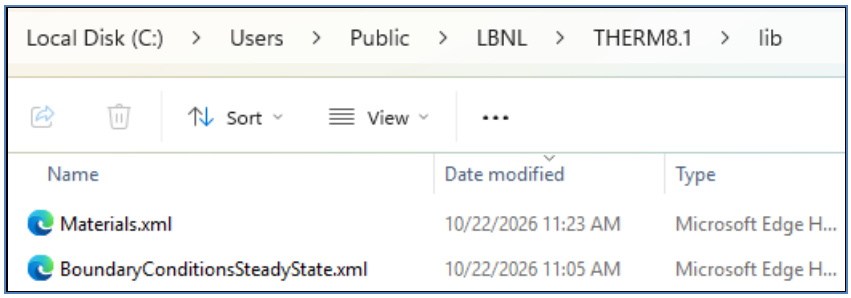


Figure 5-92. The Materials.xml file is located in the "lib" subfolder of the THERM working directory.

The definitions of all the materials displayed in the THERM Material Library are stored in this XML file.

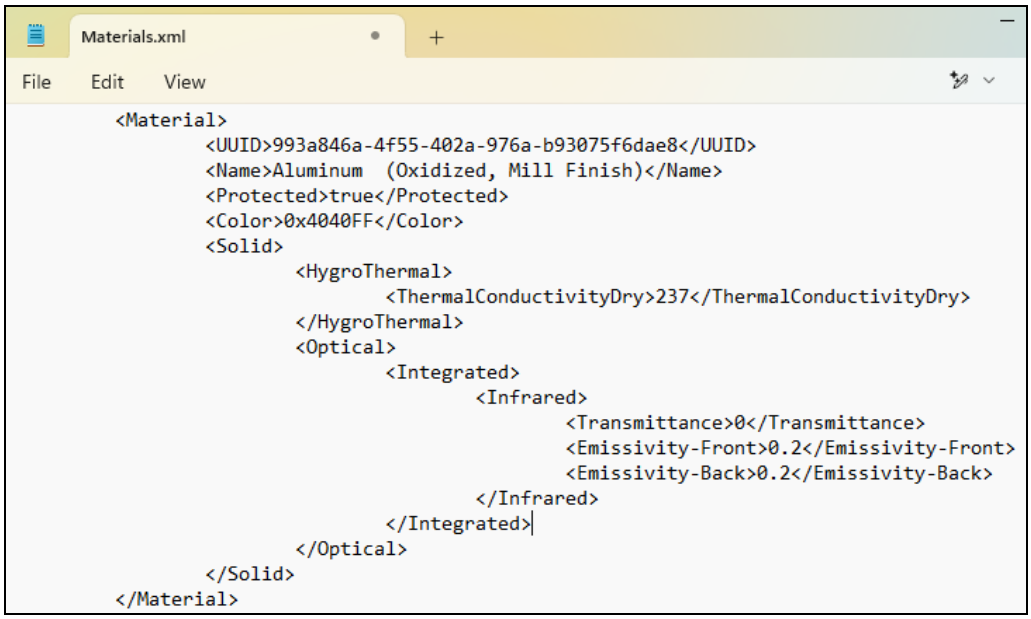


Figure 5-93. The Materials.xml file contains all the values for all the material records in the Material Library.

5.12.3. Defining New Materials

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the Material.xml file.

For example, if a new "solid" material record is created, and values added for the Solid Properties section as well as the different functions on the right side, all those changes are then saved in the Materials.xml file.

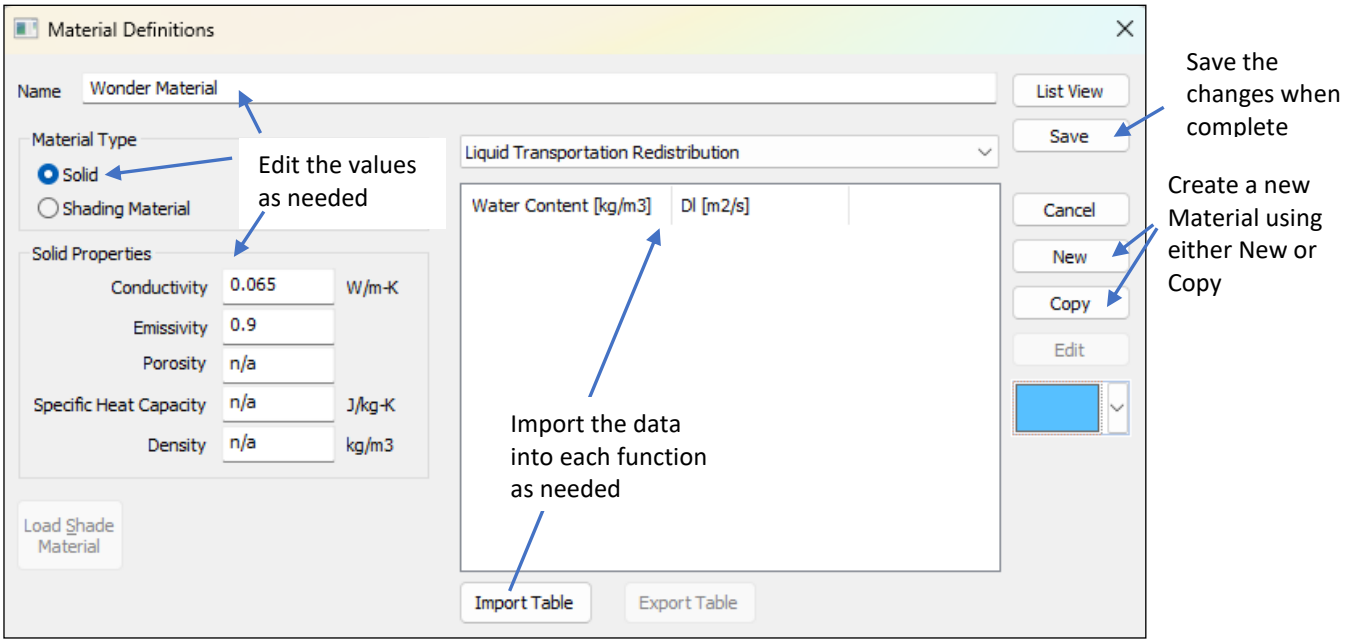




Figure 5-94. Make a new material using either New or Copy and change the input values as needed.

5.12.4. 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.

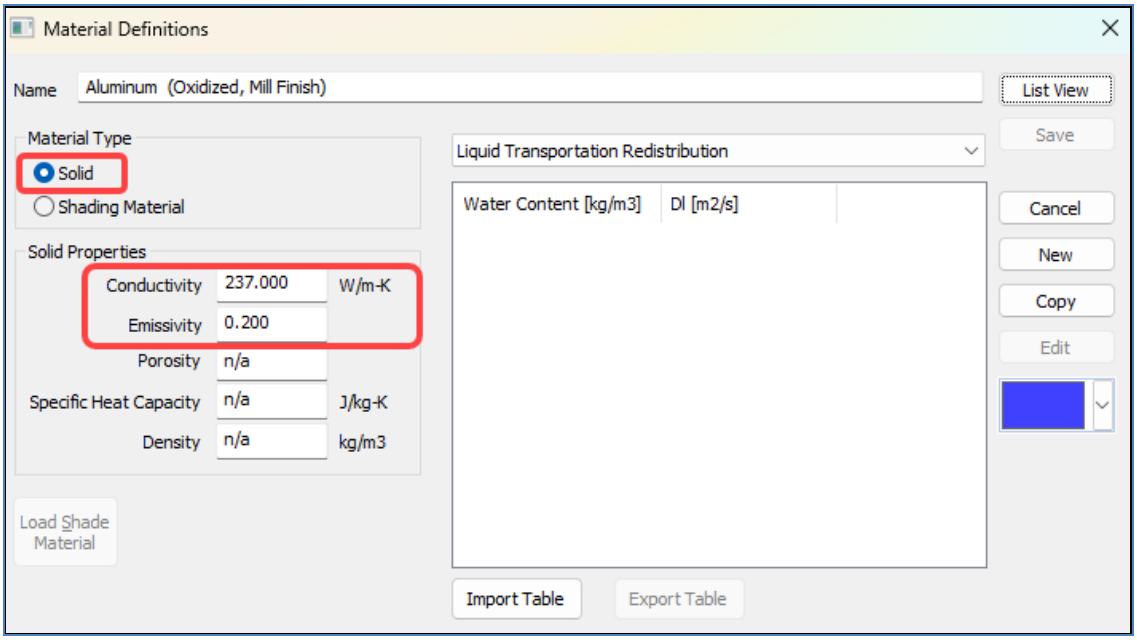


Figure 5-95. Material Library definition for solids

The properties which define a Solid Material are:

Conductivity

Thermal conductivity of the material, an intrinsic material property that does not depend on the dimension (thickness) of the material. Therefore when making new materials it is important to understand how to derive this value.
Units: Btu/h-ft-°F (IP); W/m-K (SI)

Here is an example for making a new material for insulation that has an R-value of 8.6 per inch.

First, convert the R-value to a U-factor by calculating the inverse of the R-value

1 / 8.6 h-ft2-F/Btu = 0.1162791 Btu/h-f2-F

Then divide that U-factor by 12 (12 inches per foot) to get the value in feet

0.1162791 Btu/h-f2-F / 12 ft = 0.009690 Btu/h-ft-F

This is the value to use as the conductivity for this material in the THERM Material Library

It is then possible to draw a polygon of any dimension with this material assigned, and THERM will use the conductivity to do the appropriate calculation for the dimensions drawn.

Emissivity Emissivity of the material. **Default value:** 0.9

This is the default emissivity of the material. For materials next to polygones defined as a "Frame Cavity", that default emissivity can be over-ridden one boundary segment at a time

For the predefined materials, these values are not editable. If you define a new material by clicking on the **New** or **Copy** button, you will be able to edit these values.

5.12.5. Frame Cavity

Frame cavitis are used to define elements in the THERM model that are gas-filled (air being a gas).

The program no long has frame cavities defined in the Material Library.

The characteristics of each frame cavity are defined in the polygon definition. Therefore, to edit the characteristics of a specific frame cavity, double click on a cavity and the following dialog box will be displayed.

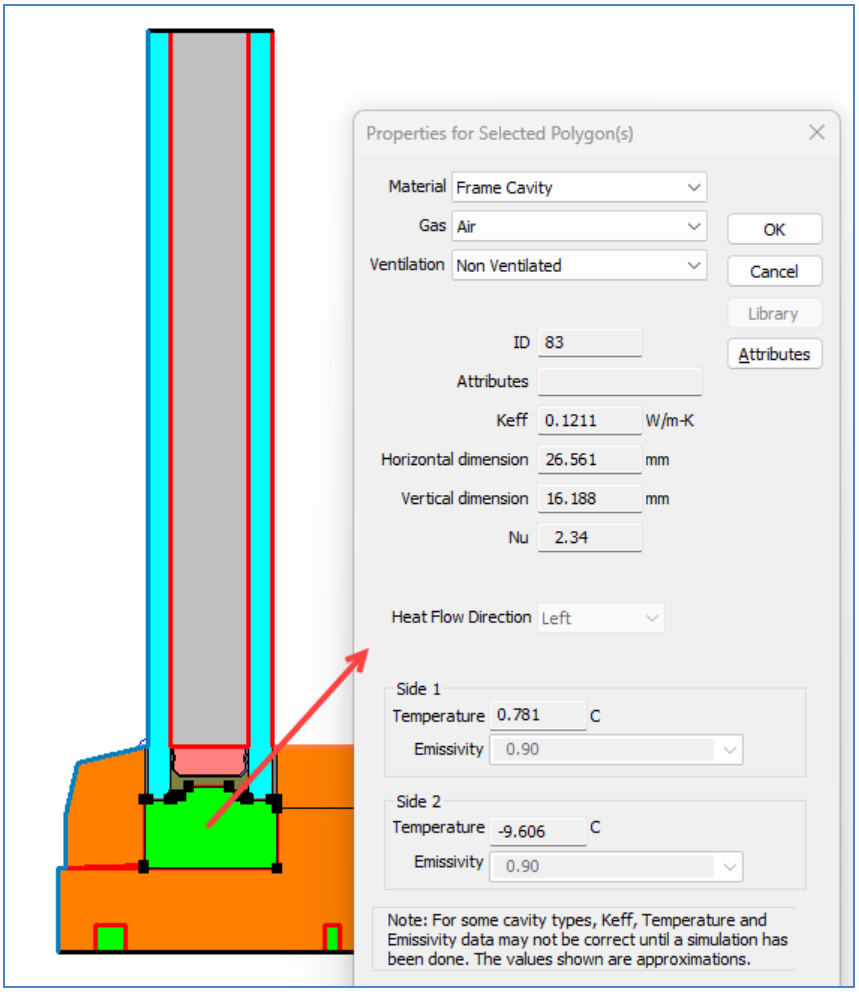


Figure 5-96. Click on a frame cavity to define the properties of that cavity.

To make new frame cavities, draw the frame cavity polygon and use the pulldown list in the tool bar to select the Frame Cavity option. Then you can double click on the polygon to edit the individual frame cavity characteristics.

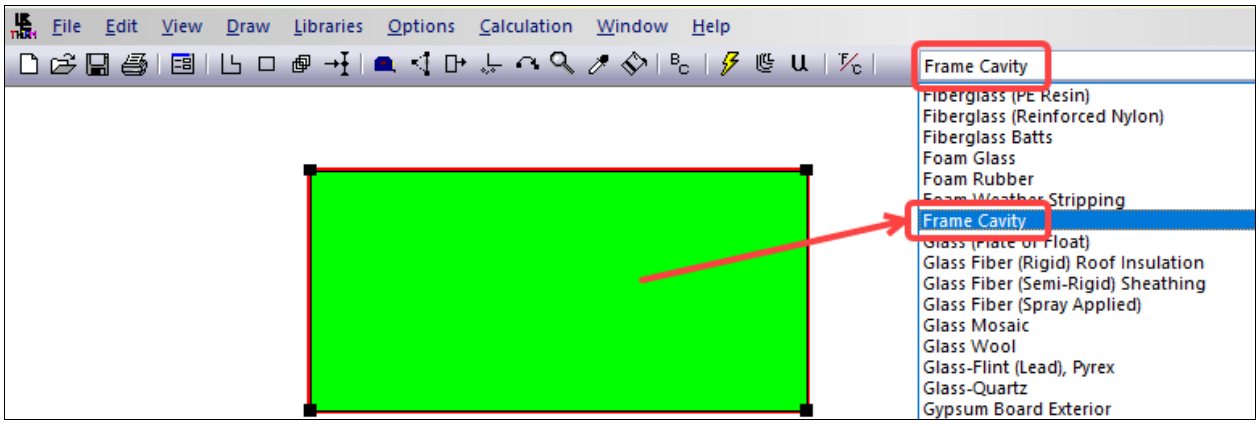


Figure 5-97. Creating a new frame cavity material in the ***Material Library for a gas-filled frame cavity.***

Frame Cavity Properties

For each frame cavity, the following properites can be defined.

Gas

The gas contained in the frame cavity. The values in this pull-down list come from the THERM **Gas Library**.

The default value is "Air".

Ventilation

The type of ventilation in the frame cavity:

- Non Ventilated
- Slightly Ventilated

The effective conductance of frame cavities is obtained from standard correlations from ISO 15099 (ISO 2007). Frame Cavity heat flow direction is calculated by THERM. For a rectangular cavity, the governing temperature difference is assumed to be between the opposing walls of the cavity in the direction of the heat flow. An average emissivity for these surfaces is determined by THERM. 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. In some cases, the vertical and horizontal dimensions will have to be switched (or the cross section rotated) for these definitions to be correct. For cavities that are not rectangular, a procedure is applied to rectangularlize the cavity, according to ISO 15099 (ISO 2007).

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 *5.11. Assigning Materials After Drawing Polygons*).

5.12.6. Glazing Cavity

This material type is not an option in the Material Library. Glazing cavity properties should be obtained from WINDOW, which occurs automatically when glazing systems are imported from the WINDOW software.

5.12.7. External Radiation Enclosure

In general, this type of material is not needed in THERM, because there is an option in the Boundary Conditions to automatically use a Radiation Enclosure model.

However, the functionality exists to define a specific external radiation enclosure material type in case that is every needed for a THERM model.

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:

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

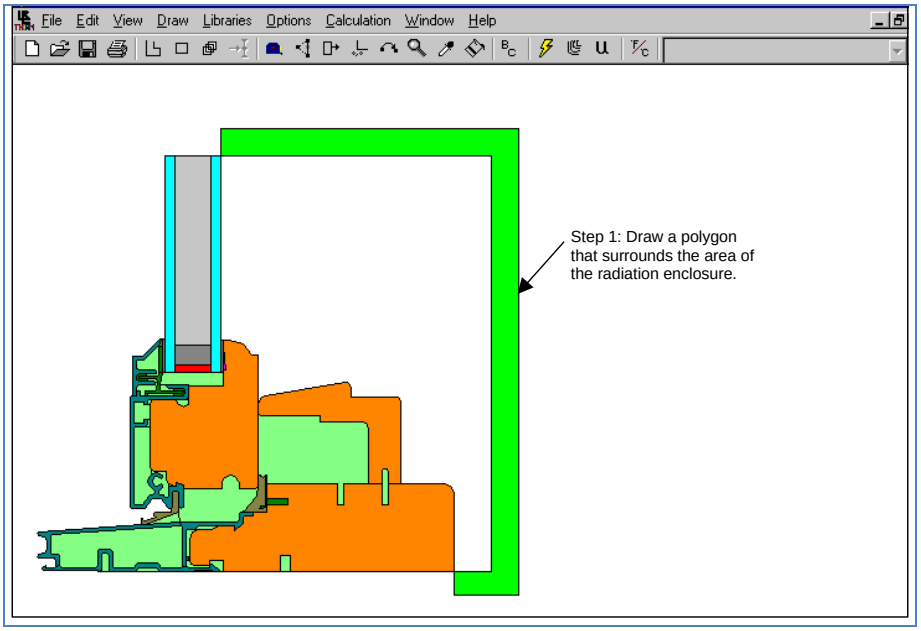


*Figure 5-98. Draw a temporary polygon around the area that will become the **External Radiation Enclosure**.*

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

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.

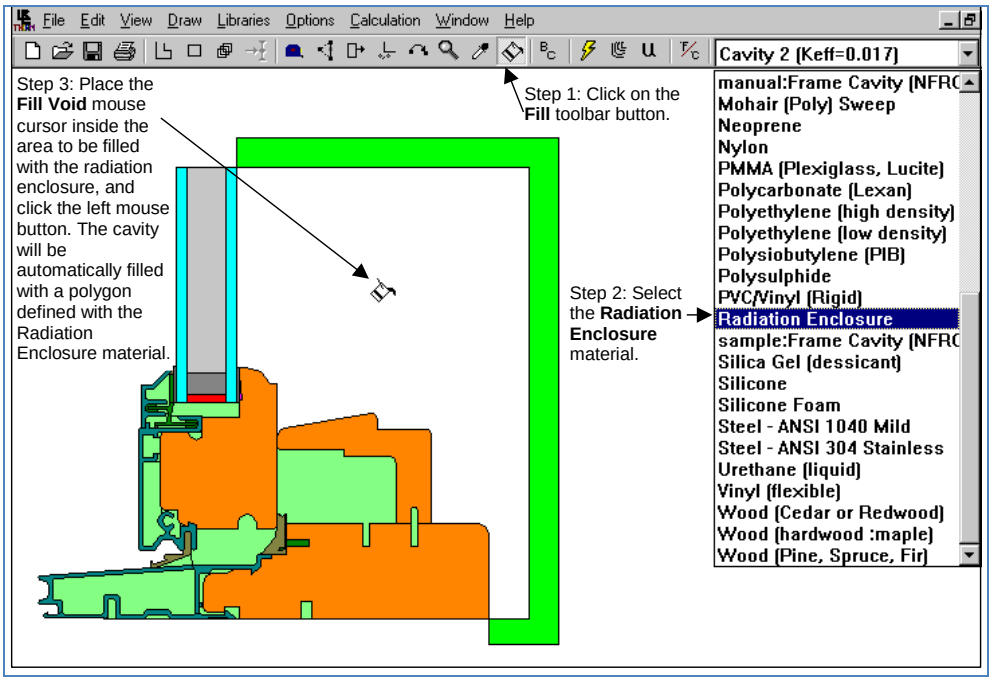


*Figure 5-99. Use the **Fill Void** toolbar button to create the **External Radiation Enclosure** polygon.*

Select the polygon that is created by using a single click of the left mouse button, as shown above. Assign the radiation enclosure polygon from the pull-down list on the right side of the toolbar by selecting the choice called **Radiation Enclosure**, shown in the figure below. Any material of this type will have special boundary conditions automatically assigned to it; these are called Radiation Surface.

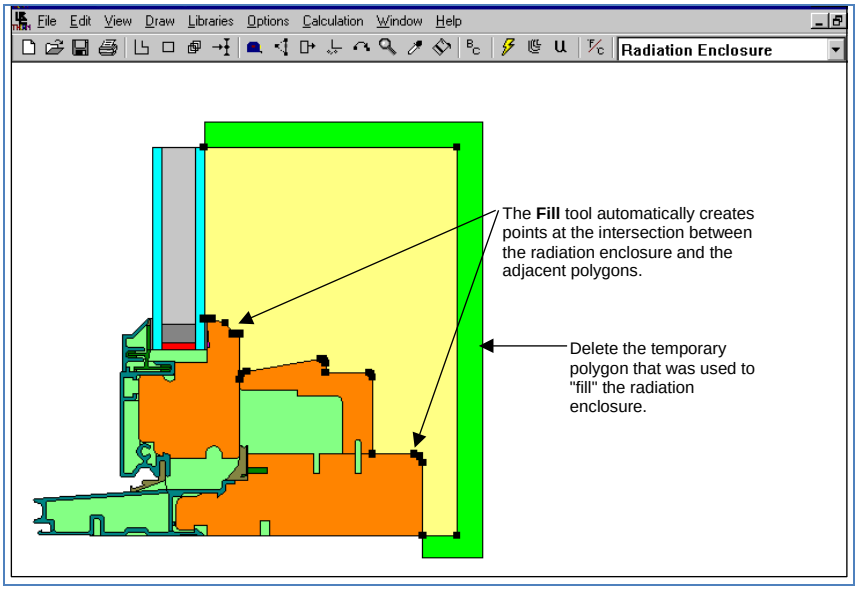


*Figure 5-100. Assign the predefined material **Radiation Enclosure** to the Radiation Enclosure polygon.*

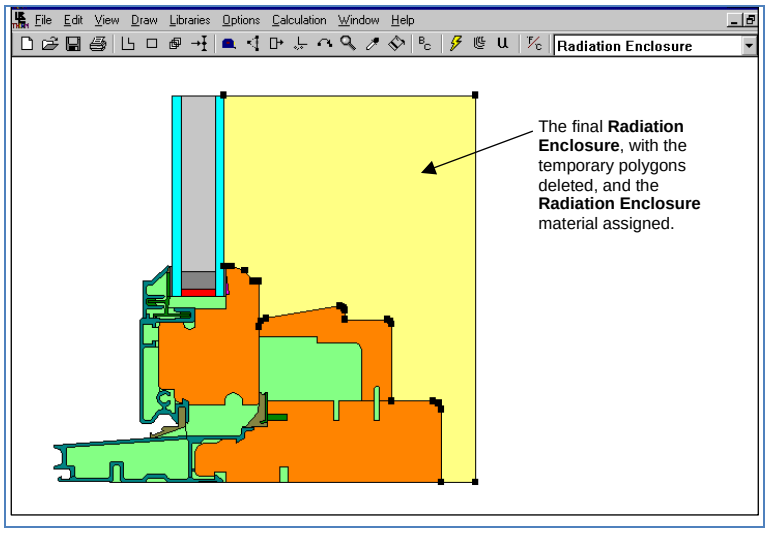


*Figure 5-101. The completed **Radiation Enclosure (the temporary surrounding rectangle deleted)**.*

Complete the analysis by defining boundary conditions and calculating the results.

5.12.8. Shade Material

THERM can be used to model cellular shades, pleated shades and other shading system with complicated geometry that cannot be easily characterized in WINDOW. The THERM models for those shading systems can then be used in the WINDOW Shading Layer Library. The modeling procedure for these shading systems is explained in detail in the *WINDOW User Manual, Section 15, Shading Layer Library*. The discussion here explains how to add materials to the THERM Material Library in order to create these models.

The materials for these shading models are first defined in the WINDOW Shade Material Library, and then imported into the THERM Material Library. The geometry of the shading system is then created in THERM, using the imported materials for the appropriate polygons in the model.

The description here does not include how to create the WINDOW Shade Material Library entries. We assume they have already been created and can be imported into THERM.

Importing WINDOW Shade Materials

To import a WINDOW Shade Mateiral into THERM, in the Material Definitions dialog box (opened by clicking on the Libraries menu / Material Library option and clicking the New or Copy button), do the following:

- Material Type:
- Set to Shading Material – this will cause the Load Shade Material button in the lower left to be highlighted.

Load Shade Materials

Click the **Load Shade Materials** button in the Materials Definition dialog box to open the Shade Materials dialog box. Shade Materials defined in a WINDOW database can be imported into THERM from this dialog box. These added materials can then be used to define shading systems in THERM.

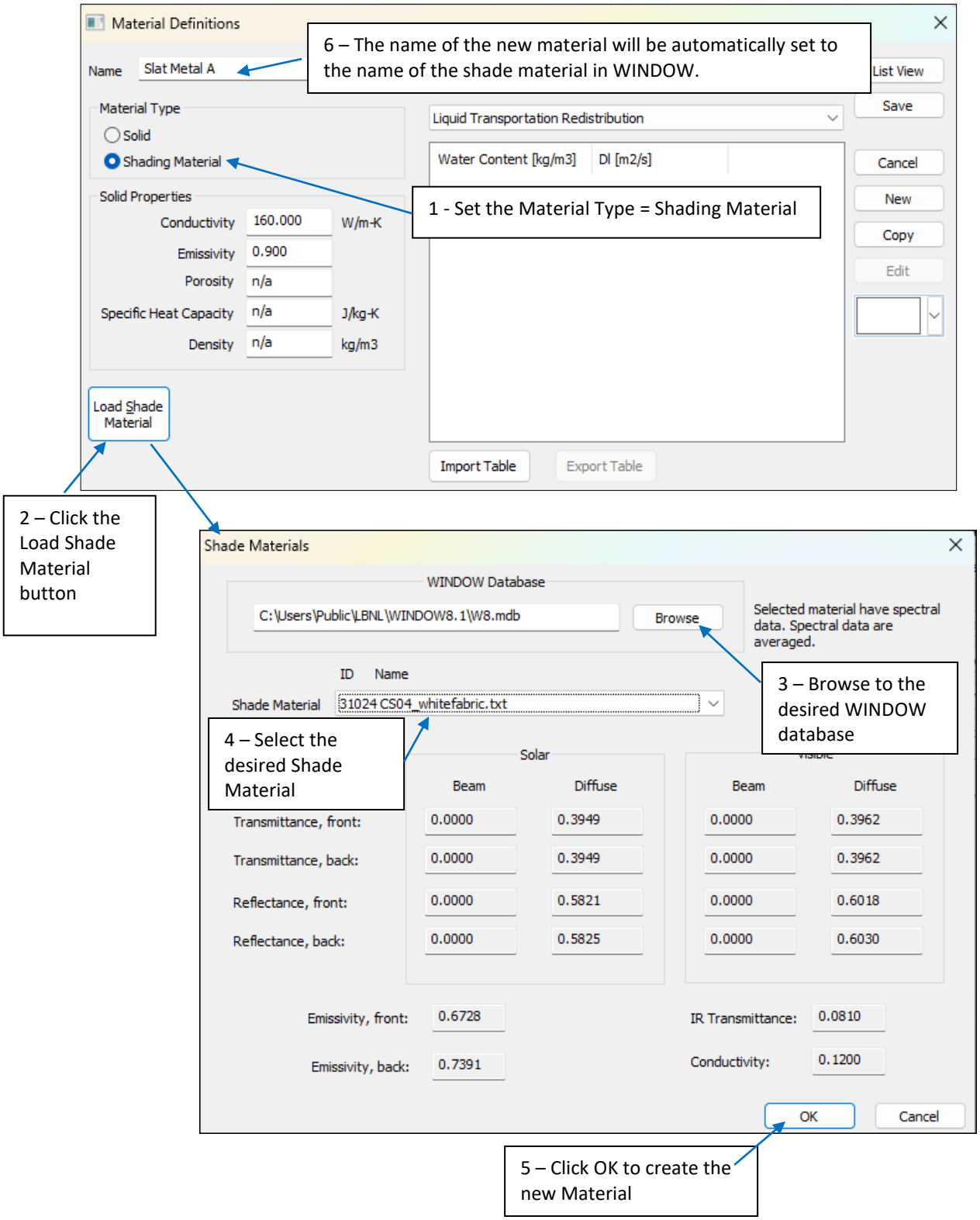


Figure 5-102. Create a Shading Material record in the THERM Material Library.

5.13. 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.

The THERM installation includes a sample spacer, which can be found in the user directory for THERM, in the Samples directory.

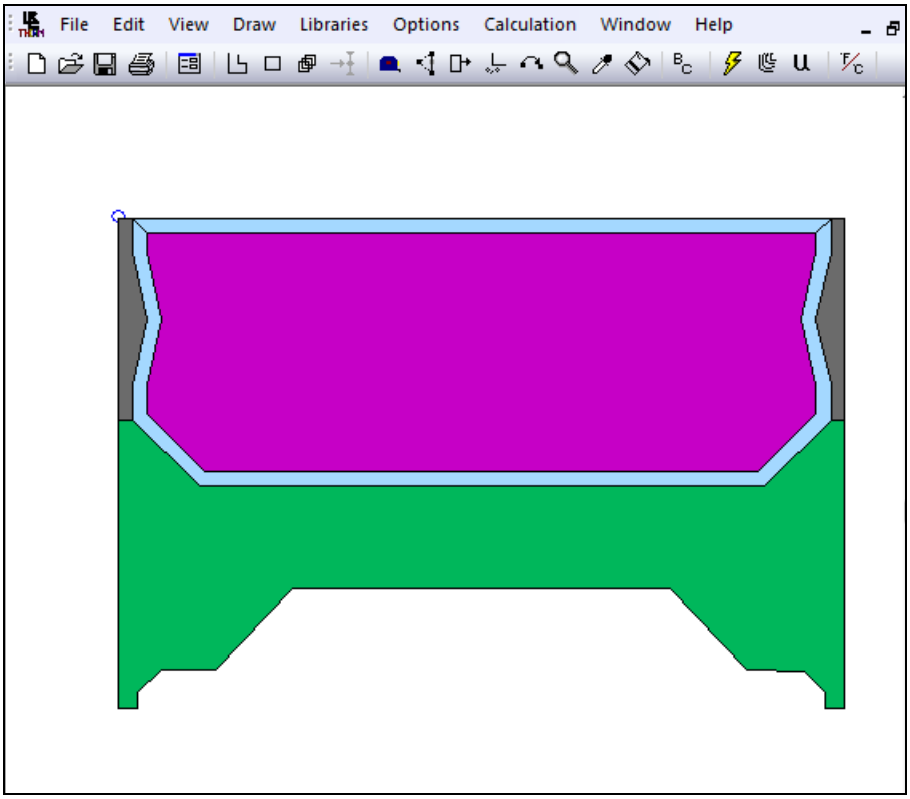


Figure 5-103. THERM file containing just a spacer

To cut and paste components between THERM files:

- Open both a current working THERM file and the THERM file containing a component you wish to copy and paste.
- In the working file, place the locator (**Shift F2**) on the point where the matching point on the component will be placed.
- 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).
- Place the locator (use **Shift F2** and click the left mouse button) at the point which will match the location in the working file.
- Press **Ctrl-C** to copy the selected polygons into the Windows™ clipboard.
- Using the **Window** menu, switch to the working file.

- 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.

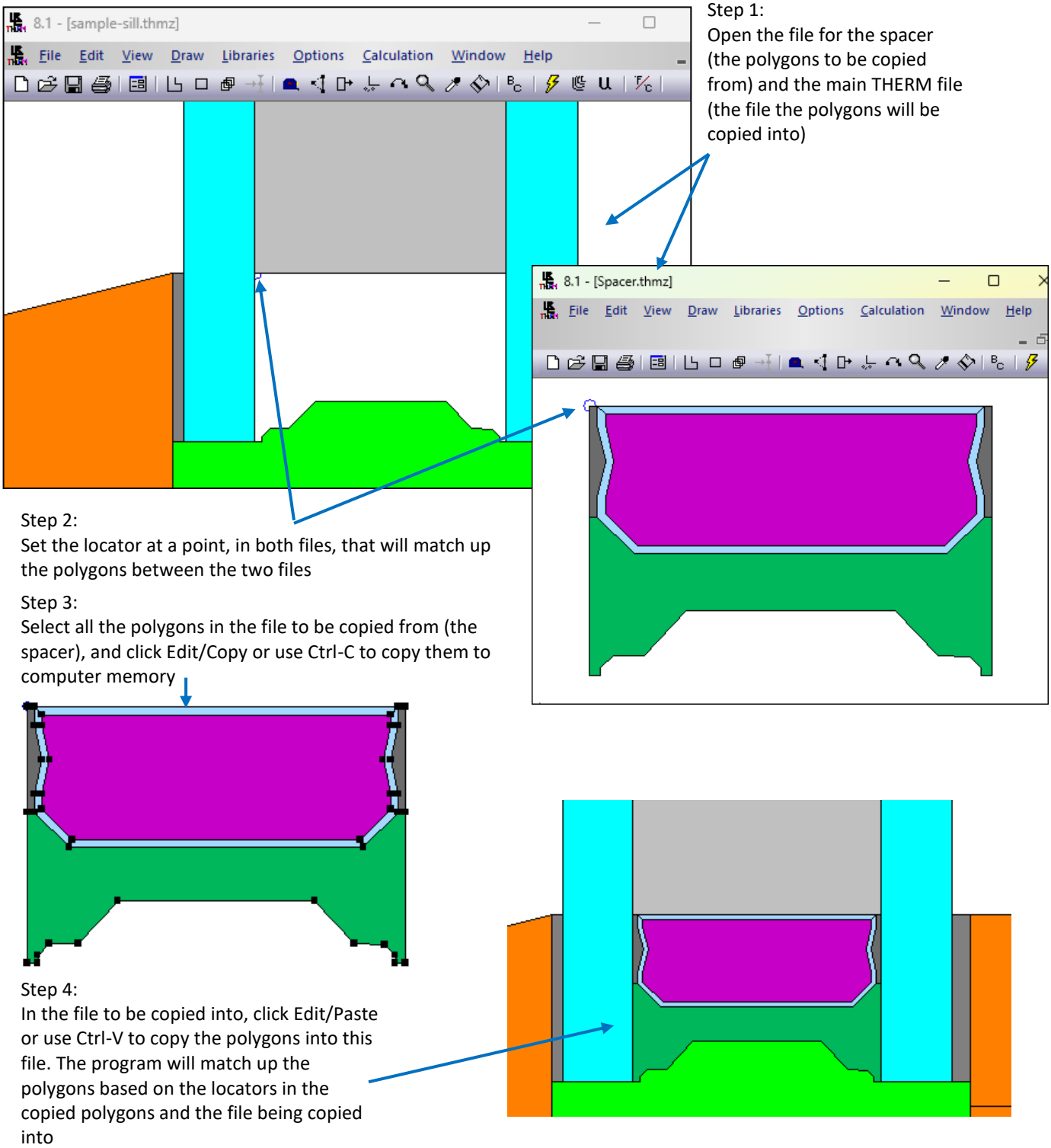


Figure 5-104. Copying polygons from one file to another.

5.14. Drawing Tips

Much of the information below is already covered in previous sections, but this is a summary of the most important aspects of drawing a model, with some examples about how to deal with some more complicated DXF files.

5.14.1. Drawing Methods

There are a few different ways that cross sections can be created in THERM. They are:

- Draw the cross sections from dimensioned drawings
- Draw the cross sections by tracing a DXF file used as an underlay
- Use the autoconvert feature to convert a DXF file into a THERM drawing

Each method has its advantages, and the method you choose will depend primarily on the type of information you have about the product to be analyzed. The following sections contain specific drawing tips that are particularly useful when drawing fenestration cross sections.

5.14.2. DXF Files

THERM can interpret the polygons in DXF files, if the files are created correctly, and thus save much drawing time. There is extensive discussion of DXF files earlier in this section; the following add to, reiterate, or clarify some of the information in that section:

- DXF files are imported using the **File/Underlay** command (See Section *5.2. Importing a DXF or Bitmap File as an Underlay*).
- The **Arc to Polygon Conversion** command under **Options/Preferences/Drawing** options determines how many straight line segments will be used to approximate curves in the DXF files. The value can be set from 1 to 45 degrees. The program is conservative in its determination of how many segments are needed to represent the curve. For example if Arc to Polygon Conversion is set to 45 degrees, and the angle is ≥ 90 degrees, the curve will be divided into a minimum of 3 segments. The *circles.dxf* file, which can be found on the THERM Documentation web page under Sample Files, can be used to discover how changing the arc to polygon value changes the way the DXF file is imported. (See Section *5.2. Importing a DXF or Bitmap File as an Underlay*).
- The horizontal and vertical scaling factors are only used with DXF files that are drawn at a scale other than 1:1. (See Section *5.2.2. Select DXF or Bitmap File*).
- The choice of units is a scaling factor for DXF files that were originally drawn in SI units; by default THERM assumes that DXF files are drawn in IP units. This unit setting is different than the overall units setting for THERM. (See Section *5.2.2. Select DXF or Bitmap File*).
- Note you will need to re-import the DXF file for changes in the arc conversion or scaling settings to take effect.
- By default, the THERM origin and DXF origin are the same. You can use the origin setting to change this if needed. (See Section *5.2.2. Select DXF or Bitmap File*).
- Only one DXF file at a time can be used as an underlay. In a file that already has an underlay, you can pick a new underlay, but it will supercede the previous one.
- In THERM, the base unit is millimeters when drawing in SI units, and inches when drawing in IP units.

5.14.3. Representing Curves with Line Segments

THERM can draw straight lines at any angle but it currently must approximate curves with a series of straight line segments. The guidelines for geometric representation of curved surfaces are still being discussed by the ASHRAE committee charged with coming up with a standard for calculating heat transfer in windows (SPC 142P). Some parameters to consider in evaluating the accuracy of the geometric representation are path length, displacement, and area.

For NFRC simulations, the model must be within 5 mm (0.2 in) of the actual product. In most cases, using either 30 or 45 degrees per side for the **Arc to Polygon Conversion** parameter will create a drawing that is within 0.2 inches (5 mm) of the product and eliminate unnecessary complexity in the drawing. A drawing with many points very close together may result in problems when creating the boundary conditions or during the simulation when the mesh is being generated by THERM.

Interpreting Arcs in DXF Files

Sample DXF files referenced in this discussion can be found on the THERM Documentaiton web page under Sample Files. The **Arc to Polygon** conversion setting in the **Options/Preferences** menu option, **Drawing Options** tab controls how many line segments THERM will use to translate an arc in a DXF file. THERM treats every curve the same regardless of radius. The *circles.dxf* file, which contains many arcs, can be used to illustrate how using the Arc to Polygon conversion setting when autoconverting a DXF file can affect the resulting THERM polygons. If you decide to model curves in detail setting the Snap to Angle value to the value chosen to divide up the arc will make drawing easier. The *lowhoriz.dxf,* can also be used to learn how THERM interprets arcs when autoconverting a DXF file.

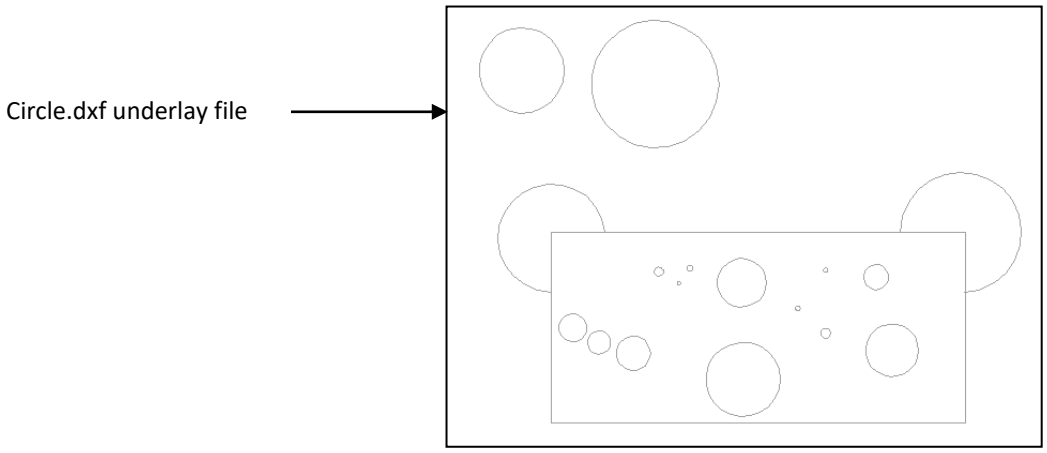


Figure 5-105. Circles.dxf sample file.

DXF Filter Settings

The DXF Filter settings in the DXF Underlay also control how the dxf file is translated by THERM. This can be seen using the *circles.dxf* file.

- When the Arc to Polygon conversion setting is 45 degrees per side, the autoconverted polygon for circles.dxf looks like this. Setting this value to 30 or 45 is recommended because it results in sufficient detail for proper modeling and eliminates unnecessary complexity which may result in problems with boundary conditions or the mesh generation.

- When the Arc to Polygon conversion setting is 15 degrees per side, the autoconverted polygon for *circles.dxf* looks like this. There are many more line segments representing the arc. This level of detail is usually unnecessary and may lead to problems when generating boundary conditions or during simulation meshing.

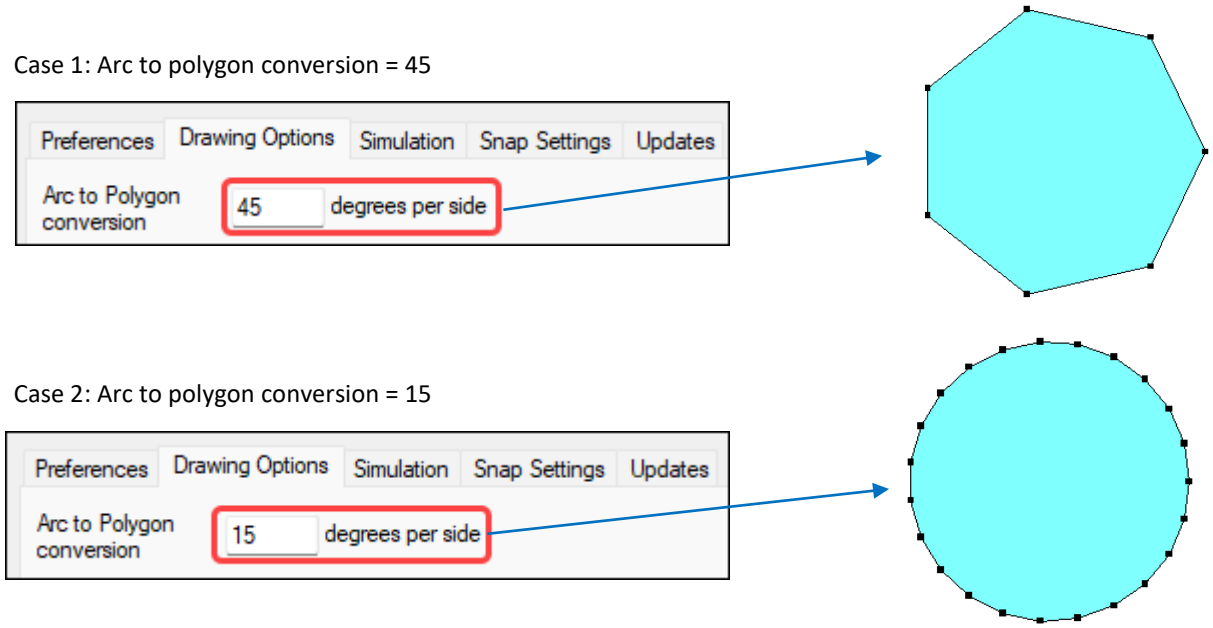


Figure 5-106. Arc to Polygon Conversion of underlay files

5.14.4. DXF Files -- AutoConvert

- For Autoconvert to work the DXF file needs to be created with closed polylines and curves. A polyline is a shape created in a CAD program made up of joined line segments and curves. (See Section *5.2.2. Select DXF or Bitmap File*).
- Autoconvert may create overlapping polygons if polylines are defined multiple times in the CAD program and if the polylines in the CAD files overlap. Use the **View/Show Voids and Overlaps** (See Section *5.3.5. Voids and Overlaps*) to check if this has happened. A single overlapping region will show as an overlap but if yet another region exists it will look like there is no overlap. A further check can be made by deleting (or moving) one of the polygons. If there is nothing underneath it the polygon can be restored using the **Edit/Undo** feature.
- Extrusions are often drawn using polylines. The external profile is one polyline and each of the cavities is a separate polyline. (See Sections *5.14.9. Breaking Up Extrusions into Polygons* and *5.14.10. Drawing Extrusions from DXF Files -- False Die Mold Method*)
- Import the DXF file *auto.dxf*, which can be found on the THERM Documentation web page under Sample Files, using the Autoconvert feature. Check for overlapping polygons. Delete any overlapping polygons. Use the Undo feature to restore any inadvertently deleted polygons.
- Import *auto.dxf*, which can be found on the THERM Documentation web page under Sample Files, with only polylines checked using the DXF filter. The rounded corners of the extrusion are chopped off, which greatly simplifies the model.

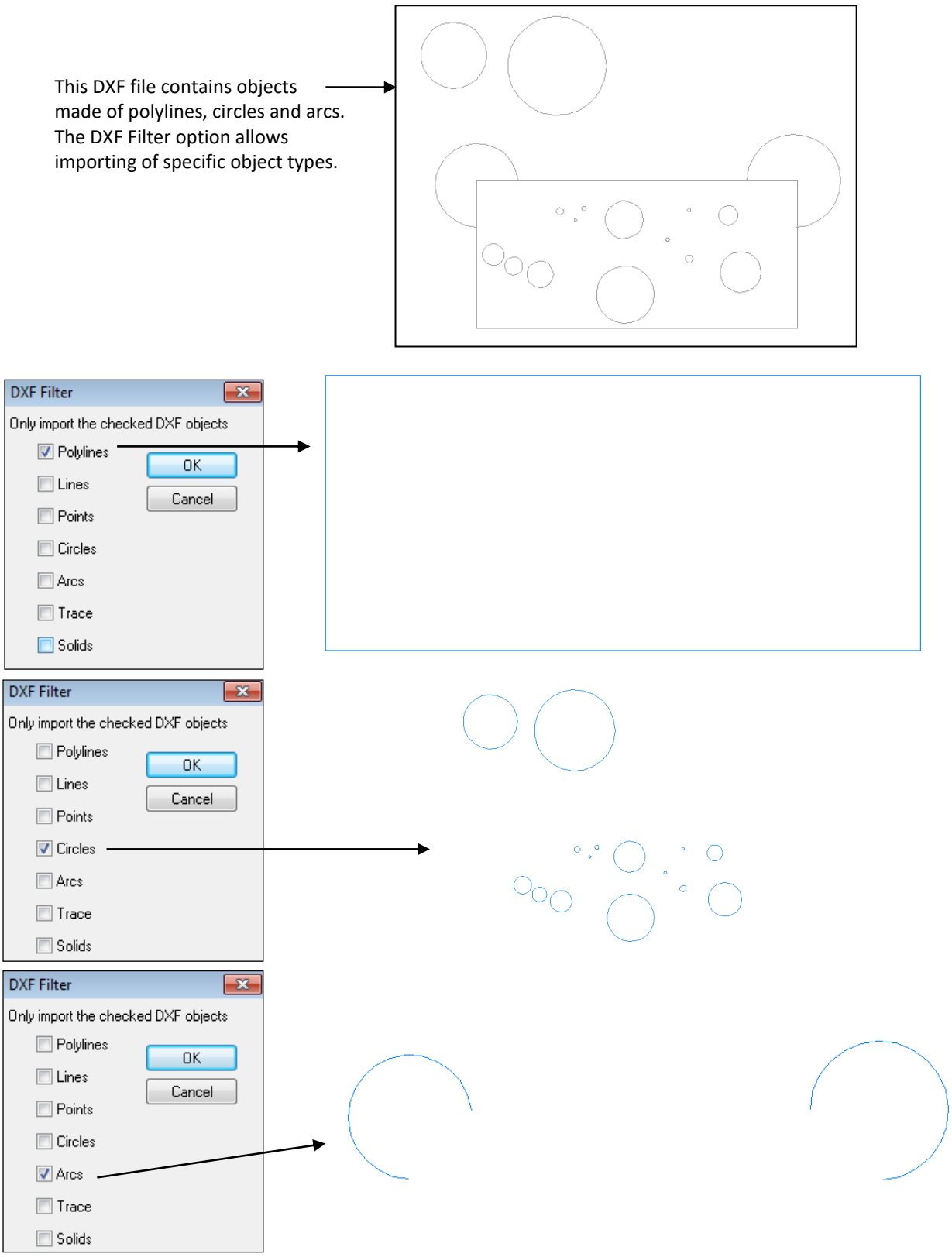


Figure 5-107. This DXF file contains objects made of polylines, circles and arcs. The DXF filter option allows importing of specific object types.

5.14.5. Sticky Distance and Snapping

- Drawing in THERM is facilitated by the fact that the cursor will jump to a DXF file vertex, polygon vertex or polygon line, if the cursor is within the "sticky distance" from those objects.
- The "sticky distance" is equal to one half of one side of the drawing cross-hair. This is equal to 3mm if the drawing is viewed at 100%. (See Section *5.3.1. Sticky Distance*).
- When zooming in on the drawing, the sticky distance is still equal to one half of one side of the drawing cross hair, but the drawing has become more detailed. If there are 2 points next to each other that are both within the sticky distance of the cursor at 100%, so it is not possible to control which point the cursor sticks to, zoom in to that area of the drawing and the the cursor will stick to the desired point.
- While drawing, if an error message appears saying that a point can not be placed on top of another or that overlapping polygons cannot be created, the sticky distance may be causing the cursor to stick to points that are not desired, such as a point that is within the sticky distance of the previously drawn point. In this case, zoom in to the drawing and it is usually possible to get to the desired points.
- There are several snap features that can help with drawing. **Snap to Vertical and Horizontal** and **Snap to Underlay** are the defaults. **Snap to Angle** is helpful for drawing detailed curves. **Smart Snap** extends the edges of existing polygons and helps to line up polygons that aren't adjacent. The **Snap** features can be easily turned on and off through the toolbar. (See *Section 5.3.3. Snap Settings*).

5.14.6. Drawing Tips

- **Moving points and lines:** Press the **Modify Points** button on the toolbar to be able to modify points. A double headed arrow is the cursor for moving points. (See Section *5.6.4. Moving a Point*). A four headed arrow is the cursor for moving lines. (See *Section 5.6.5. Moving a Line*). Holding down the shift key will constrain the movement to the horizontal and vertical directions. Sometimes line move has trouble sticking to adjacent polygons and it is possible to draw overlapping polygon errors when moving.
- **Space Bar to snap:** Use the space bar to snap to a point (and also to DXF file vertices) without changing the model. This is useful for determining the start of a line when measuring, and to move or measure relative to a certain point.
- **Copying and Pasting:** When using the **Edit/Copy** and **Edit/Paste** features within the same drawing, it is possible to create overlapping polygons. Make sure when pasting a polygon that the **Locator** (Shift F2) is moved before the paste is performed, or that the pasted polygon is moved after pasting if it is located directly over the copied polygon.
- **Scooting:** While drawing, **Zoom Out** (Shift right click) to see a point that is off the drawing area, move the cursor to that point, then **Zoom In** (right click) to actually draw the point. This is the best way to move the cursor large distances while drawing.
- **Scroll Bars:** The Scroll Bars can be accessed while drawing a polygon (a much better method than pushing the drawing cursor against the edge of the drawing window, which can result in display problems.)
- **Switching Units:** The **° F / °C** toolbar button switches between IP/SI units.
- **Display clean-up:** Sometimes when deleting points from a polygon there is residue left on the screen. Press Escape to force the screen to redraw and the extraneous images will go away.

  Scrolling up and down makes the labels of the isotherms get bunched together. Escape will redraw them with the labels positioned correctly.

- **Editing Points on Multiple Adjacent Polygons:** Move, add or delete the same point on all adjacent polygons (by selecting all the affected polygons) at the same time -- moving them separately may cause voids and overlaps.
- **Select Material/BC toolbar button:** The **Select Material/BC** toolbar button can be used to set the **Material Library** or **Boundary Condition Library** choice. To use this feature, click on the Select **Material/BC** toolbar button, move the cursor to the material or boundary condition to be selected, and click the left mouse button. This sets the material or boundary condition to the value of the object selected. The cursor will change to the **Fill** cursor. Move it to the object (polygon, void or boundary segment) to be set to the selected material or boundary condition, and click the left mouse button.

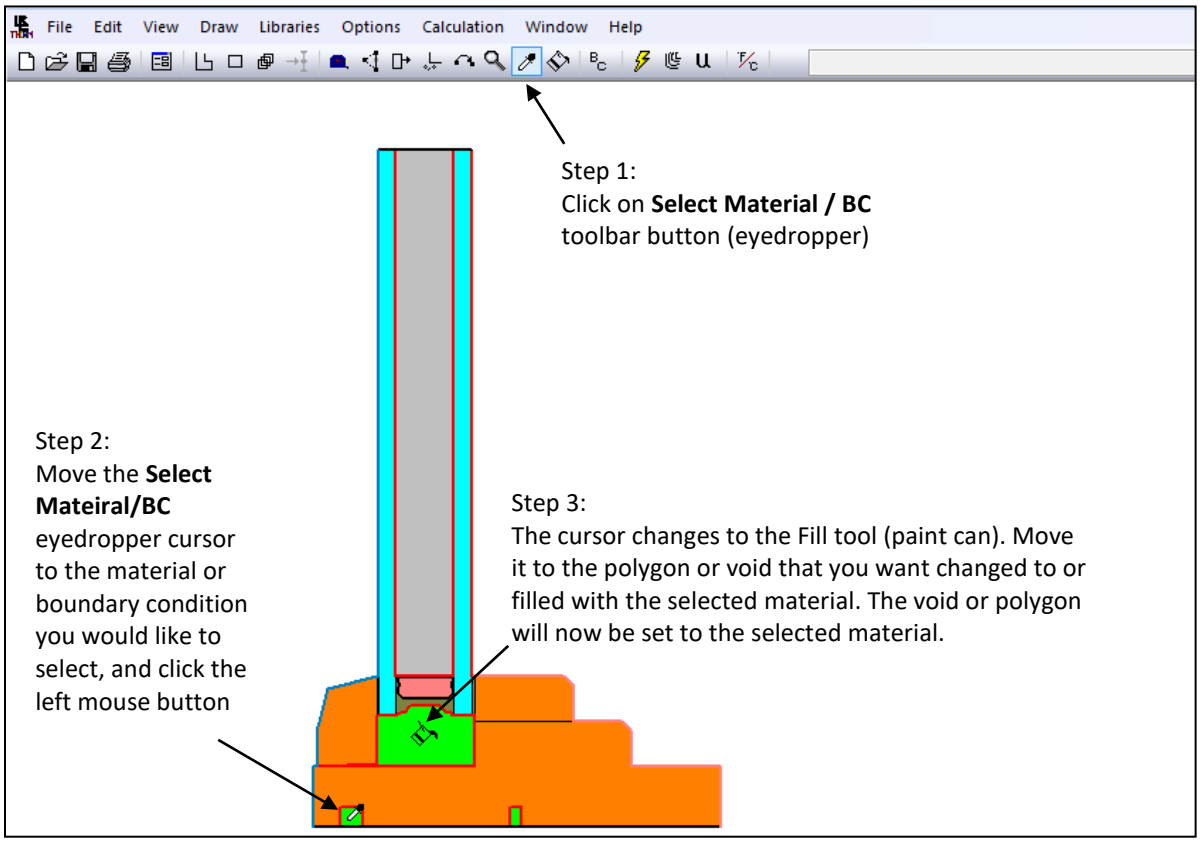


Figure 5-108. Steps for using the Select Material/BC toolbar button.

- **Fill toolbar button:** The **Fill** toolbar button can be used to either fill a void with a selected material, or change a polygon from one material to another. To use this feature, click on the **Fill** toolbar button, select the desired choice from the Material list, and move the **Fill** cursor to the void or polygon to be set to the selected material. Click the left mouse button, and the void or polygon will be set to the selected material.

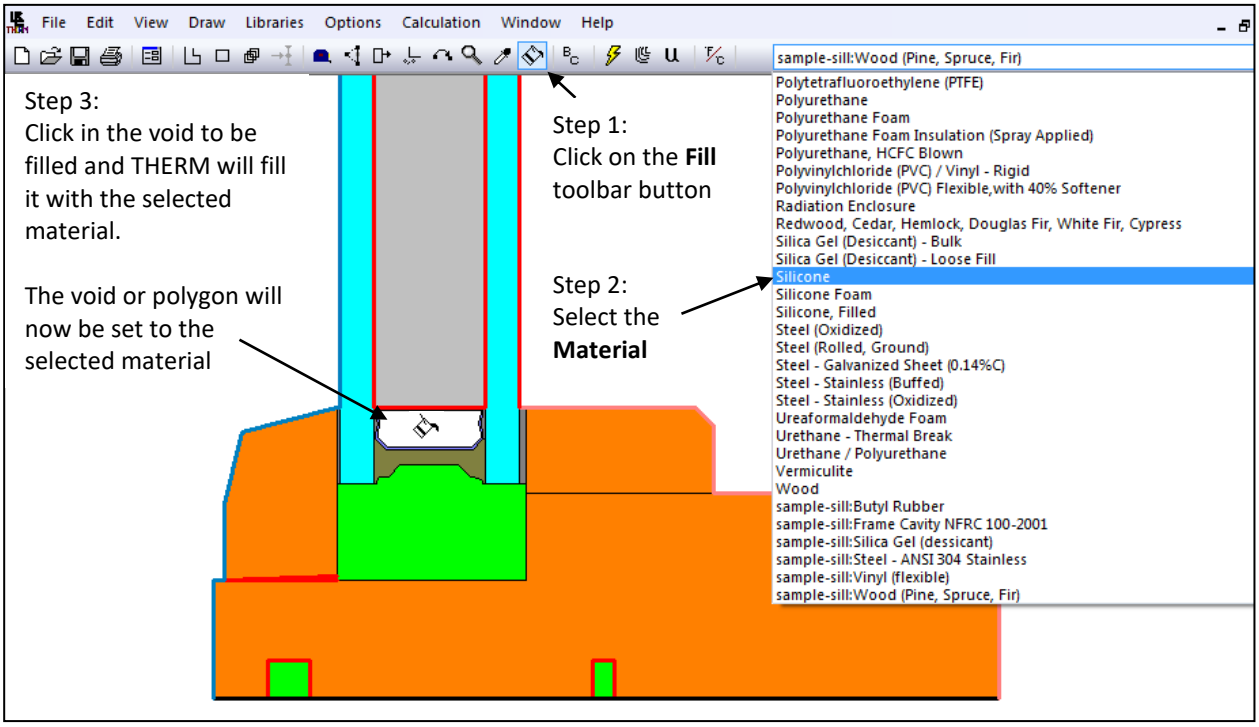


*Figure 5-109. Steps for using the **Fill** toolbar button*

5.14.7. Drawing with the Keyboard

Typing a number will bring up the **Step Size** box, and set the **Step Size**. Then the arrow keys will move the distance set in the **Step Size. Rectangles** and polygons can be drawn entering dimension values into the step size box and moving the arrow keys in the appropriate direction. See Section *5.5. Drawing Using the Keyboard*, for more information about drawing with the keyboard.

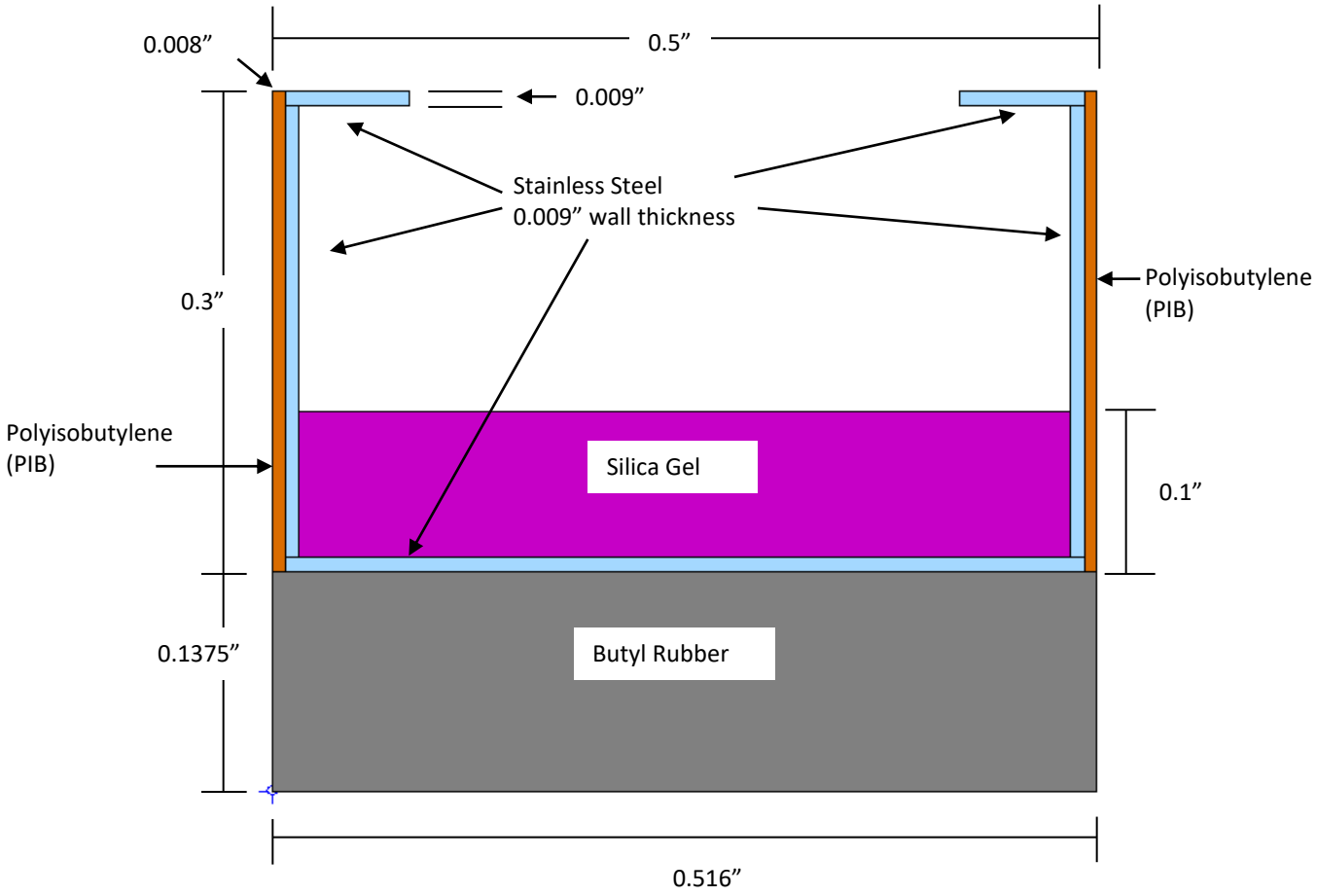


Figure 5-110. A sample spacer to use for drawing with the keyboard from a dimensioned drawing.

5.14.8. Donuts

- A region that connects back onto itself, such as a “donut”, must be broken into multiple polygons in THERM
- When breaking up the "donut" regions, try not to create angles that are extremely acute. (There are many examples of extremely acute angles in Bad4.thm, which can be found on the THERM Documentation web page).

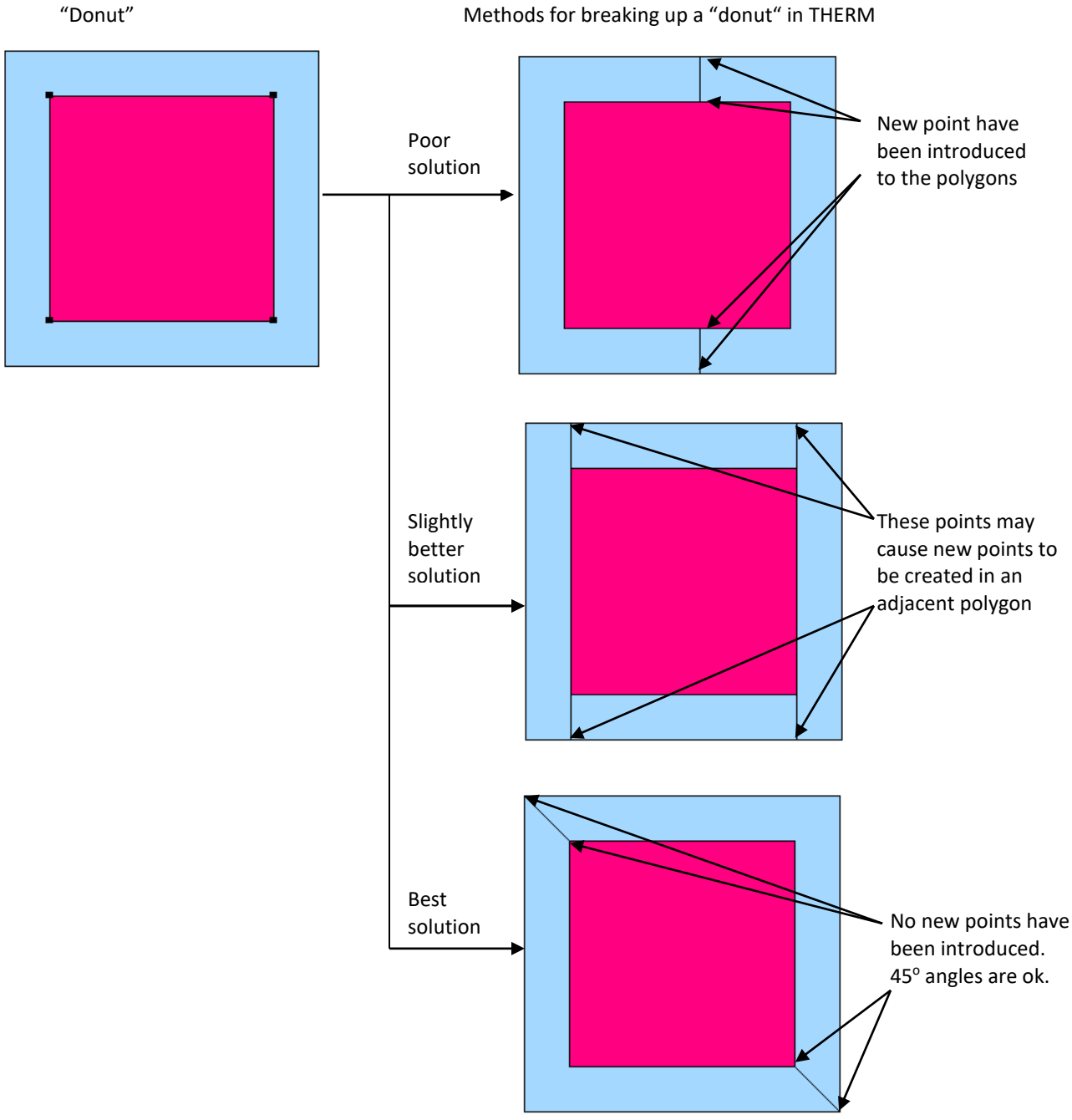


Figure 5-111. Breaking up "donut" regions in a cross section.

5.14.9. Breaking Up Extrusions into Polygons

- THERM currently does not allow polygons that come back on themselves such as "donuts".
- When working with a CAD drawing it is usually necessary to break extrusions into polygons that THERM can handle.
- THERM can handle very complicated polygons. The method for breaking up the extrusion is a matter of personal working style, although for generating boundary conditions and the mesh during simulation, it is usually better to have more simple polygons that just a few very complicated one:
  - Make many simple rectangles and polygons OR
  - Make fewer, more complicated polygons
- Basic rules in breaking up extrusions
  - Try not to create new points-- use points (vertices) that are already there
  - Try not to create extremely acute angles (less than 5o)
  - If an extrusion cannot be broken up without creating an extremely acute angle, then introduce a point.
- It's easy to get lost while making very complicated polygons, which is one reason to make smaller, more simple polygons.
- The cen1.dxf, cen2.dxf, autofill.dxf and autofill2.dxf underlays, which can be found on the THERM Documentation web page under Sample Files, can be used to practice breaking up extrusions.

The *Cen2.dxf* file, which can be found on the THERM Documentation web page under Sample Files, contains donuts which must be broken up when drawing the extrusion.

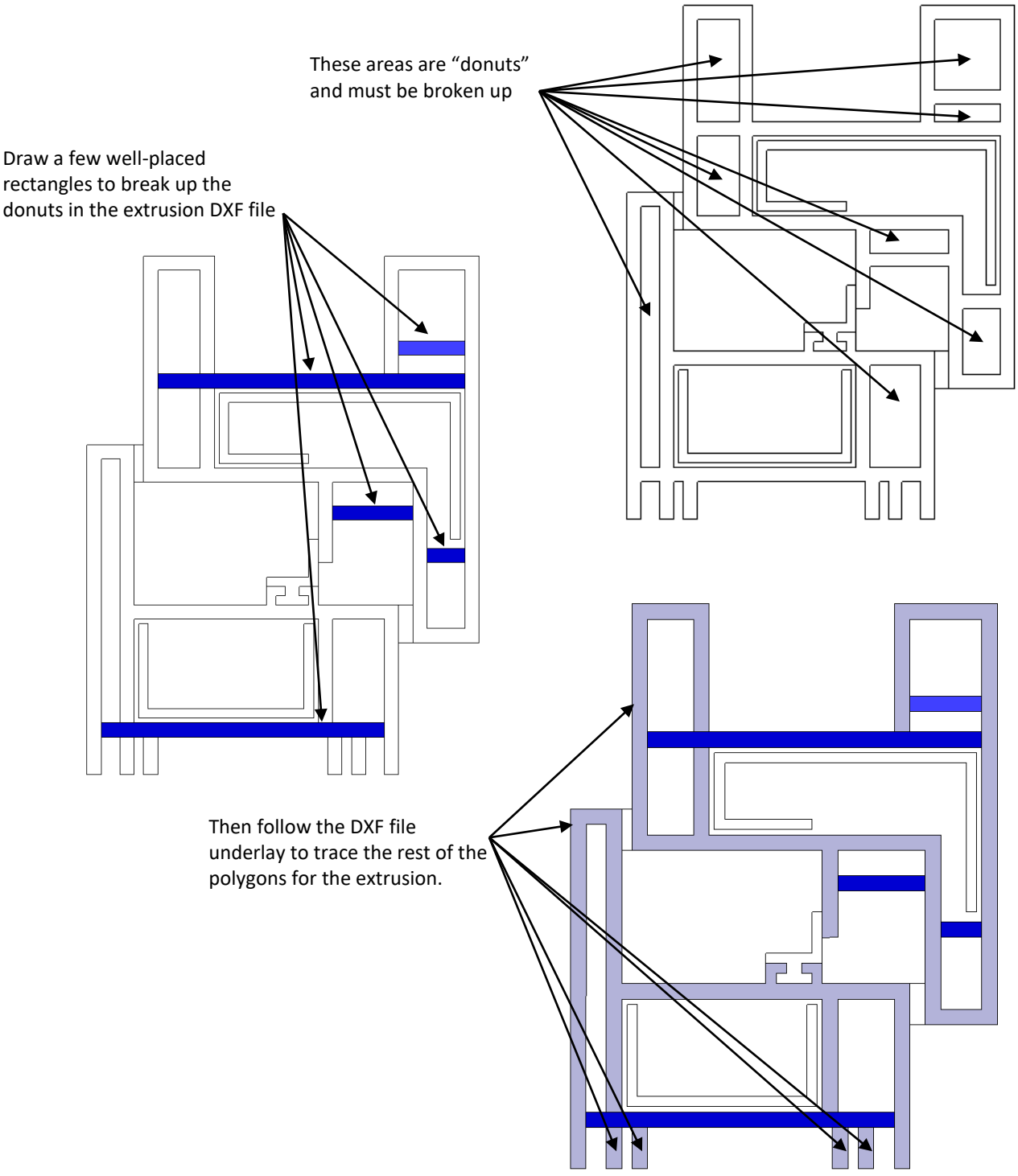


Figure 5-112. Breaking up “donut” regions in an extrusion cross section.

5.14.10. Drawing Extrusions from DXF Files -- False Die Mold Method

- Many DXF files of extrusions are drawn as polylines, with the external profile as one polyline and each of the air cavities is a separate polyline -- auto.dxf, which can be found on the THERM Documentation web page under Sample Files, is an example of this.
- Keeping the external profile polygon in place overlapping the cavity polygons, create a series of polygons that enclose the external profile (remember that you can't draw donuts).
- Use Fill Void to fill in the areas between the enclosing polygons and the external extrusion profile.

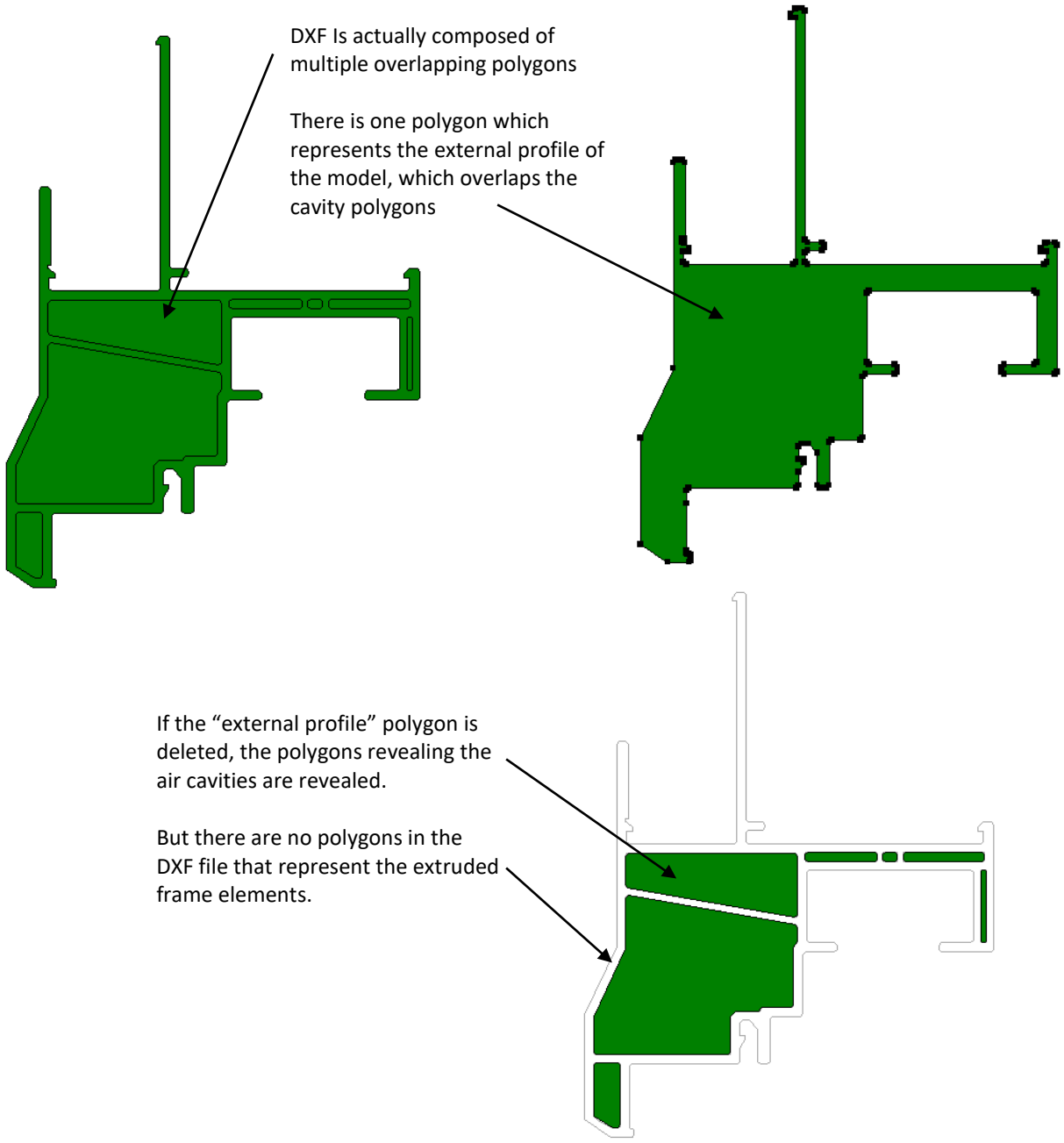


Figure 5-113. There are several overlapping polygons, some representing the outline of the entire extrusion, and some representing the cavities. There are no polygons that represent the actual extruded elements of the frame.

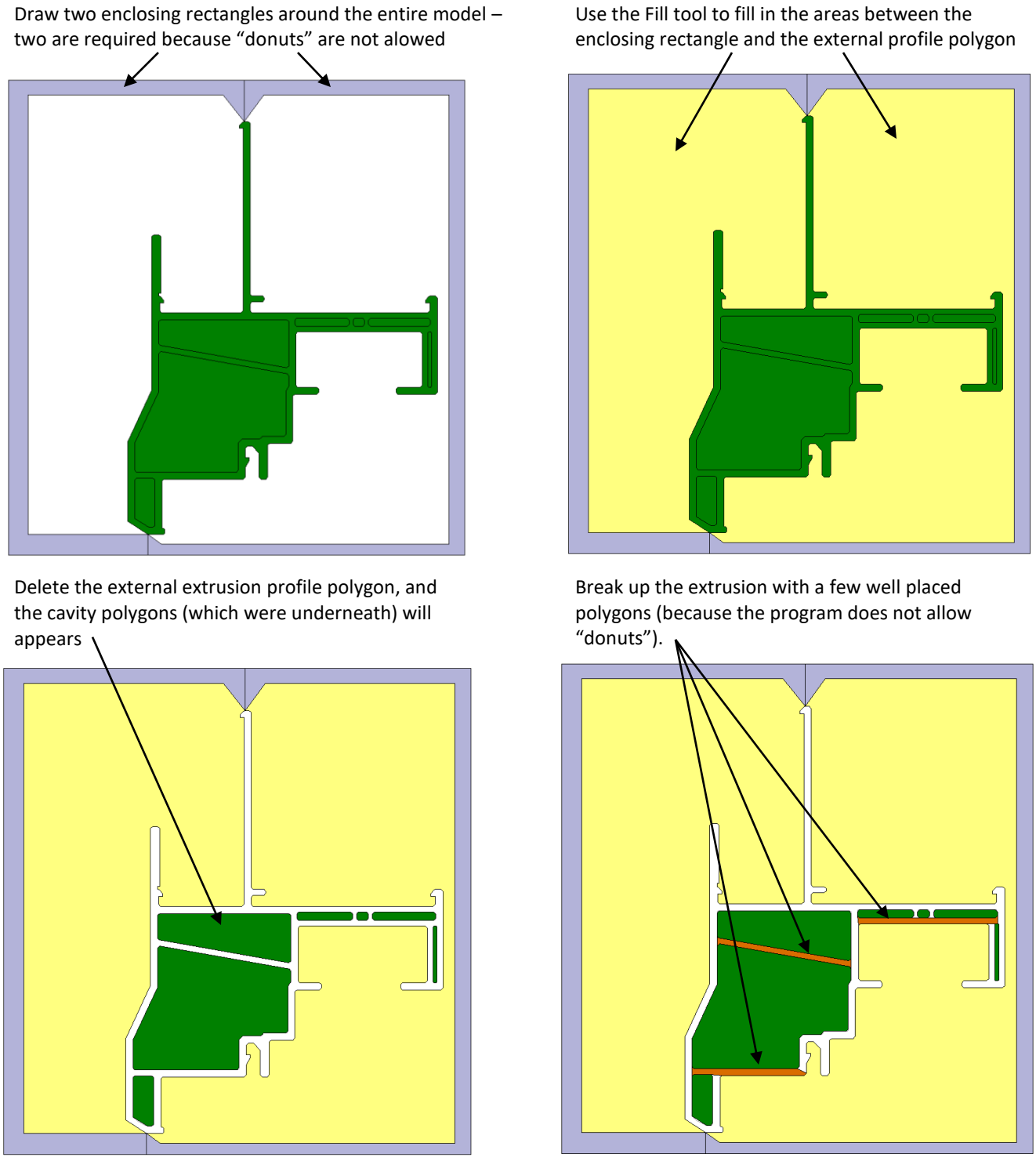


Figure 5-114. By surrounding the external profile polygon with other polygons, it is possible to create the extrusion components of the frame fairly easily.

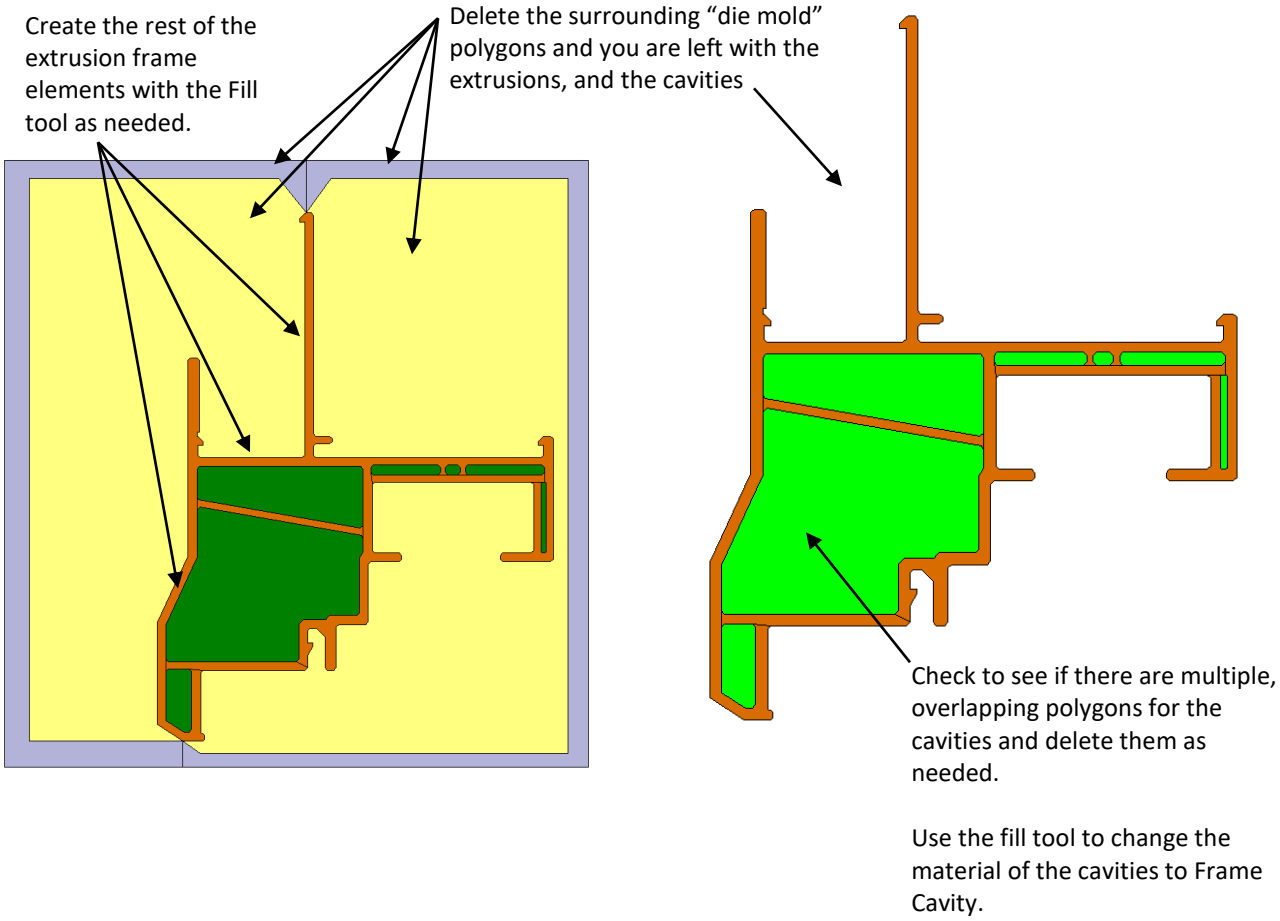


Figure 5-115. Use the fill tool to create the extrusion elements of the frame, delete the surrounding “die mold” polygons, delete any overlapping cavity polygons, and change the cavities to Frame Cavity as appropriate.

5.14.11. Cuttins, Relocating, Flipping and Rotating Discussion

There are several ways to select polygons in THERM.

- Edit/Select Special
- The Drag Box
- Mouse clicks, shift clicks, the keyboard Tab key

There are two ways to move polygon

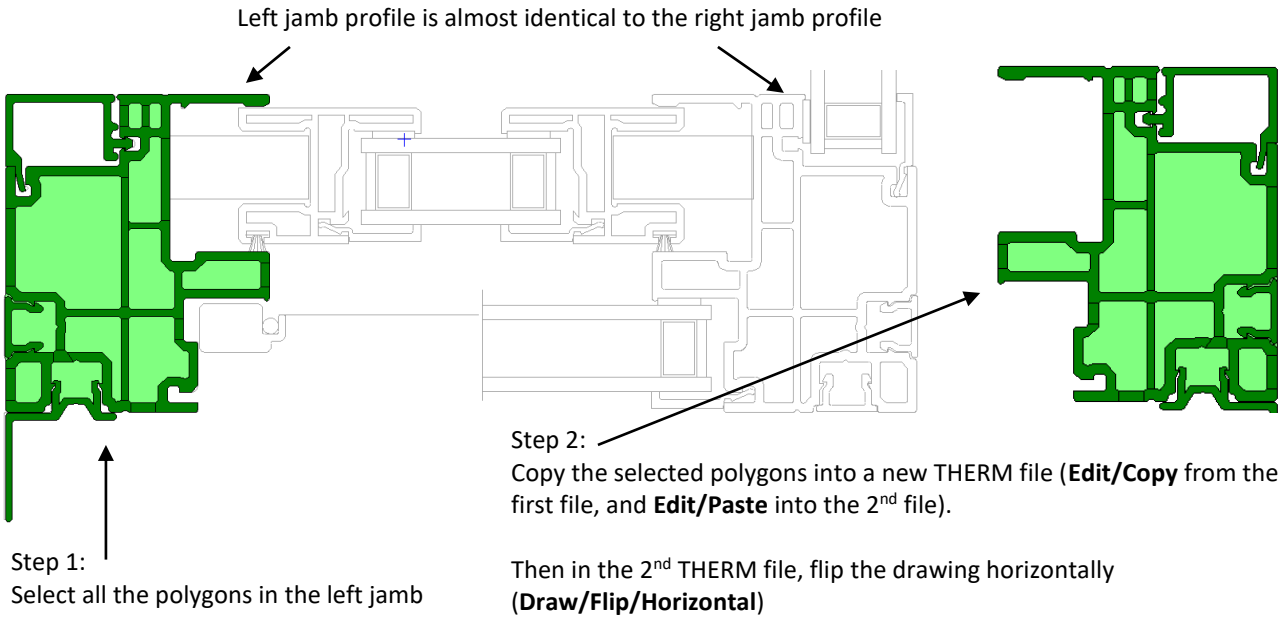
- The Draw/Move Polygon command
- Using the Draw Locator - this is recommended for precise moves

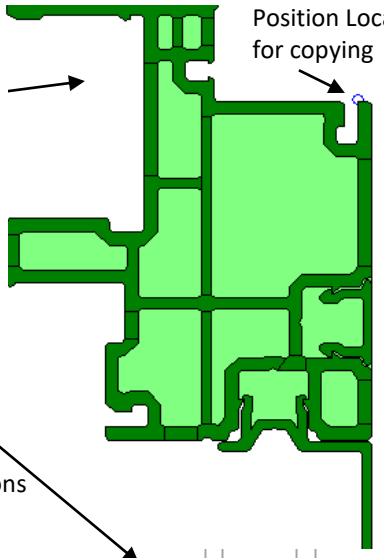
THERM has Flip and Rotate features but they rotate and flip everything in the drawing so to flip or rotate individual polygons you need to:

- **Cut** and **Paste** the polygons that need to be manipulated to a temporary file
- Perform the operation (Flip or Rotate) in the new file on the individual elements
- Paste them back into the original drawing, using the **Locator** functionality

The Cut and Paste feature in THERM eliminates unnecessary redrawing of parts that are exactly duplicated in multiple cross sections. You can make libraries of spacers and other components in THERM files to be used in many cross sections.

The files *spacerhalf.thm* and *cut&paste.thm*, which can be found on the THERM Documentation web page under Sample Files, can be used to practices these features.





Step 3:
Still in the 2nd THERM file:

- Delete unwanted polygons
- Set the Locator (**Shift F2**) on a point that exists in both files.
- Select all polygons and copy them (**Edit/Copy**)

Step 4:
In the 1st THERM file

- Set the Locator (Shift F2) on the same point as in the 2nd file
- Paste the polygons from the 2nd file (**Edit/Paste**) into this file

Because the Locators for both files are set to the same point, the new polygons will be pasted in matching those locators, and they will be in the correct locations.

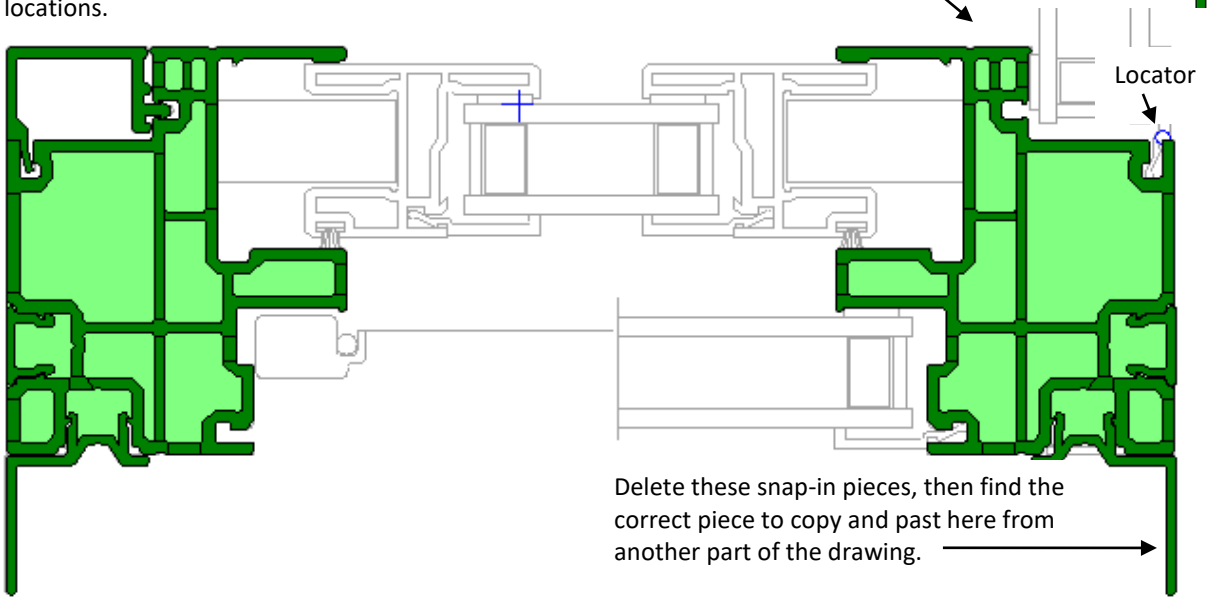


Figure 5-116. Use the Copy, Paste, Flip and Rotate functions to create model elements without redrawing each from scratch

6. DEFINING BOUNDARY CONDITIONS

6.1. Overview

Chapter 5 describes procedures for drawing 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. Boundary conditions will define the environmental conditions (such as air temperature, surface film coefficients, and wind speed next to the surface) for the model. The records in the Boundary Condition library will depend on whether the simulation is "steady state" or "transient", as defined in **File/Properties/Calculation Options**. *Section 9, Transient Moisture Modeling*, discusses the transient modeling in detail. This section will be primarily appropriate to steady state modeling, which at this time cannot model glazing systems.

The procedures for defining boundary conditions are not dependent on how the cross section was drawn, although the general convention is to model the exterior face of the model to the left and the interior face of the model to the right in the THERM drawing.

In addition to defining the environmental conditions, the boundary conditions can define surfaces to be included in U-factor and SHGC calculations. These are defined with "U-factor tags" which are part of the boundary condition definition. The boundary conditions can also be used to determine the "condensation potential" on a model surface.

6.1.1. Boundary Condition Definitions

Boundary conditions must be defined for all the surfaces on the perimeter of a THERM model. These boundary conditions define the temperatures and film coefficients for each element of the perimeter. Different boundary conditions are defined for the surfaces on the interior and the exterior of the cross section. Surfaces which are assumed to have no heat transfer are assigned **Adiabatic** boundary conditions.

There are three main categories of boundary conditions:

- **Interior:** An interior boundary condition is used to define the conditions for interior surfaces, and assumes that the surface is exposed to natural convection, and radiation (depending on the radiation model defined in the boundary condition).
  - **For Glazing Systems:** For glazing systems that are imported from WINDOW, THERM will create a Boundary Condition definition based on the center-of-glazing surface temperatures calculated in WINDOW. This can be used for the the interior glazing system boundary condition in THERM, or a custom boundary condition can be used. At the time the glazing system is imported from WINDOW (in the Insert Glazing System dialog box), set the Interior Boundary Condition pulldown to "Use Convection plus enclosure radiation", which will cause the program to set the glazing system interior boundary condition **Radiation Mode**l to "AutoEnclosure".

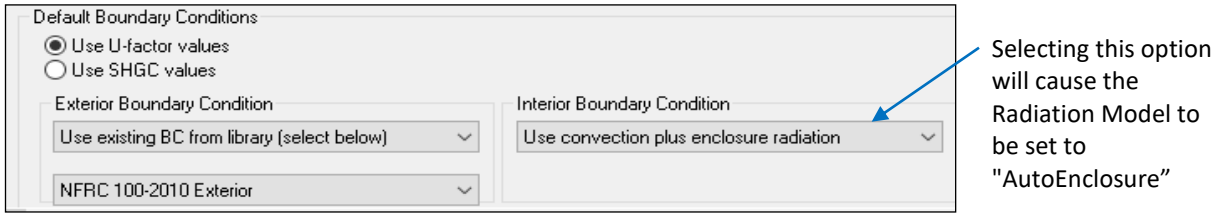


Figure 6-1. Setting the Interior Boundary Condition to "Use convection plus enclosure radiation" will cause the Radiation Model to be automatically st to "AutoEnclosure"

THERM will then model the cross section with the radiation enclosure feature without having to actually draw the geometry of a Radiation Enclosure. The boundary condition is named "<filename>:<glazing system name> U-factor Inside Film". Double click on an interior boundary conditions to see its characteristics, as shown in the figure below.

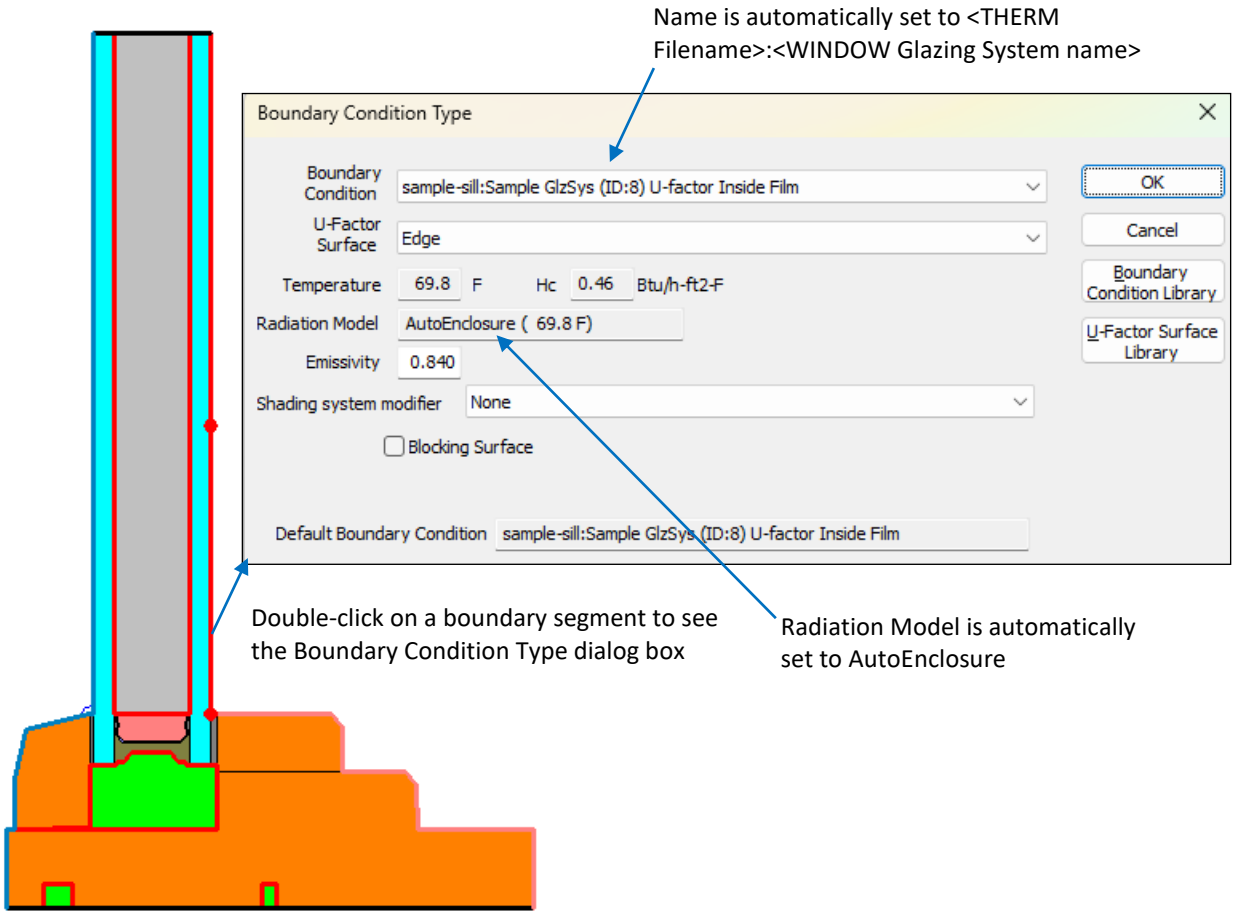


Figure 6-2. Double click on a glazing system boundary condition segment to display the Boundary Condition Type dialog box which displays the setting for that segment.

- **For Frame Elements:** It is possible to create custom boundary conditions, but for most fenestration modeling situations, the pre-defined (non-glazing system) boundary conditions (which all have the Ratiation Model set to "AutoEnclosure"), shown in the list below, can be used. There is one set for cross sections at a 90° tilt and another set for cross sections at a 20° tilt
  - Interior Aluminum Frame (Convection only)
  - Interior Thermally Broken Frame (Convection only)
  - Interior Thermally Improved Frame (Convection only)
  - Interior Wood/Vinyl Frame (Convection only)
  - Interior (20 tilt) Aluminum Frame (Convection only)
  - Interior (20 tilt) Thermally Broken Frame (Convection only)

- Interior (20 tilt) Thermally Improved Frame (Convection only)
- Interior (20 tilt) Wood/Vinyl Frame (Convection only)

- If there are frame materials that fall into more than one category, a single boundary condition must be applied to the entire interior surface of a given frame section, even if it contains different material components (i.e. vinyl sash and aluminum frame). In this case, choose a boundary condition that represents the case with the greatest interior exposed area of that boundary condition segment.

- **Exterior:** The "NFRC 100-2010 Exterior" boundary condition can be used for exterior surfaces, including the glazing system. This boundary condition assumes that the surface is exposed to a 5.5 m/sec (12.3 mph) air velocity, which corresponds to a convective film coefficient of 26 W/m^2-^{o}C (4.58 Btu/h-ft^2-^{o}F). It is always possible to create a custom boundary condition if different conditions need to be modeled.
- **Adiabatic:** The adiabatic boundary condition is used for any surface assumed to have no heat flow. This is used for the top of the glazing system at the boundary between the edge-of-glazing and the center-of-glazing, because the assumption is that the heat transfer between the two sections are independent of each other. Adiabatic is also used for the bottom of the frame that, for example, would sit in the mask wall of a thermal test chamber.
- **Shading Systems:** THERM models with WINDOW glazing systems that include shading systems have a "Shading system modifier" that is applied to the glazing system boundary conditions next to the shading system.

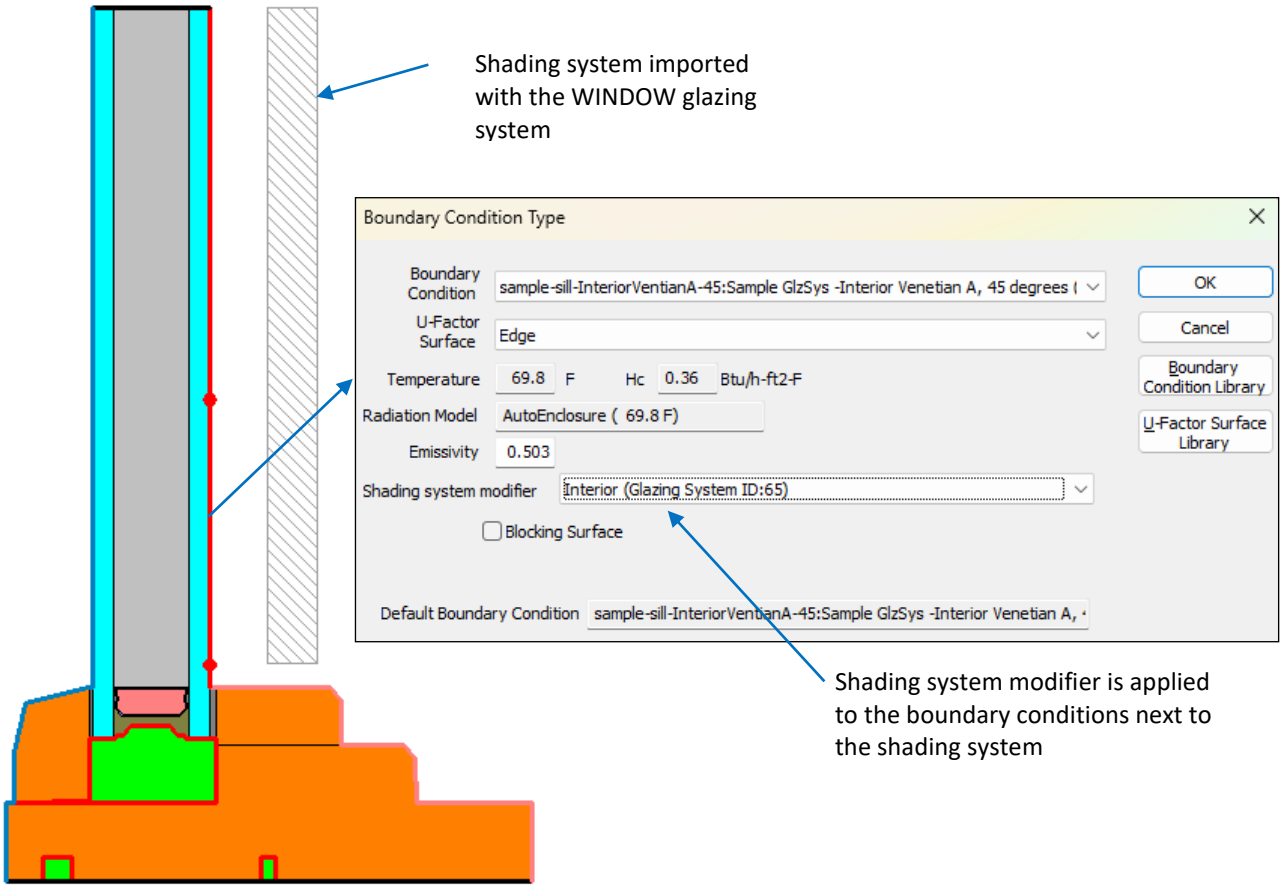


Figure 6-3. Boundary conditions next to shading systems imported with the WINDOW glazing system should have a "Shaing system modifier" applied.

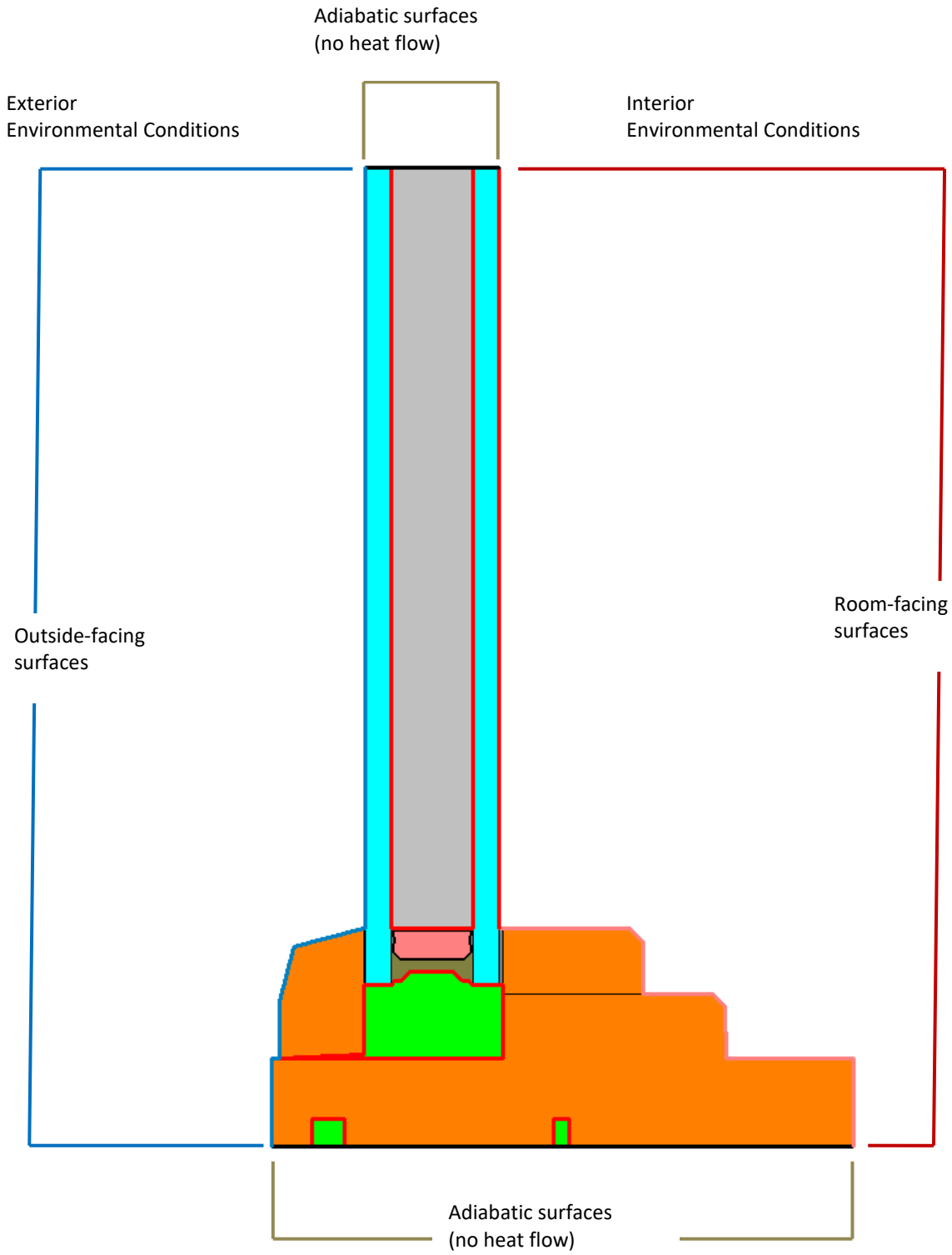


Figure 6-4. A standard convention for constructing a THERM model is to have the exterior surfaces to the left and the interior surfaces to the right, although that is not required.

6.1.2. Boundary Condition Library

THERM has a **Boundary Condition Library** (accessed from the **Library/Boundary Conditions** menu, or by double clicking on a boundary condition) which contains a default set of standard boundary conditions, shown in Table 6-2. It is also possible to define new boundary condition definitions as needed based on what is being modeled. By default in THERM, the exterior boundary conditions are blue, interior boundary conditions are red, and the adiabatic boundary condition is black. For custom boundary conditions, other colors can be assigned.

Table 6-2. Default records in the Boundary Condition Library

Boundary Condition	Radiation Model	Convective Film Coefficient			
		Tilt = 90º		Tilt = 20º	
		W/m²-ºK	Btu/h-ft²-ºF	W/m²-ºK	Btu/h-ft²-ºF
NFRC 100-2010 Exterior	Blackbody	26	4.578	26	4.578
Interior Aluminum Frame (convection only)	Automatic Enclosure Model	3.29	0.579	4.65	0.819
Interior Thermally Broken Frame (convection only)	Automatic Enclosure Model	3.00	0.528	4.09	0.72
Interior Thermally Improved Frame (convection only)	Automatic Enclosure Model	3.12	0.549	4.32	0.761
Interior Wood/Vinyl Frame (convection only)	Automatic Enclosure Model	2.44	0.429	3.09	0.544
WINDOW Glazing System boundary condition <filename>:<glazing system name> U-factor Inside Film	Automatic Enclosure Model	Depends on the WINDOW calculations for the imported glazing system			

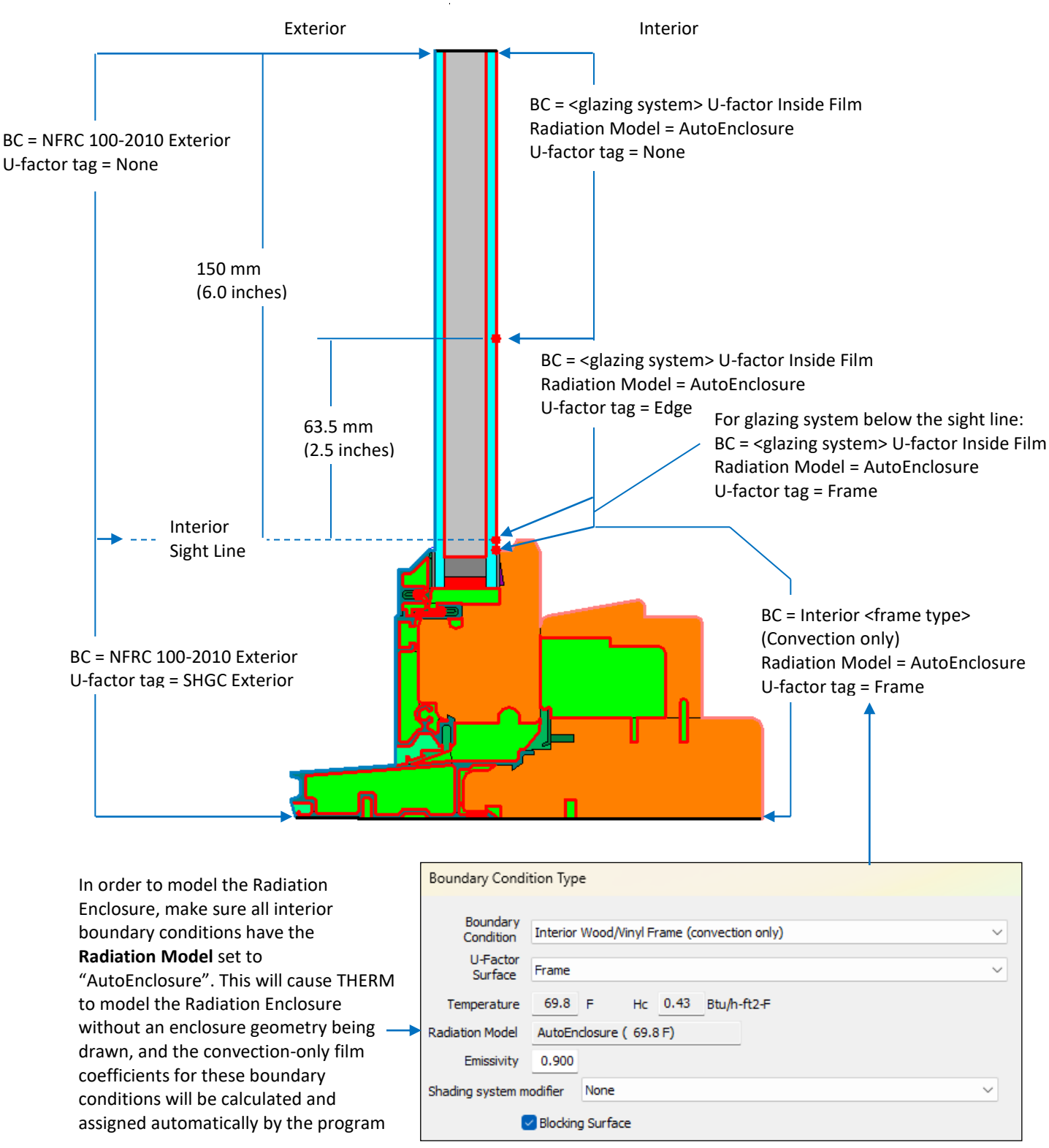


Figure 6-5. A window model with typical boundary condition and U-factor surface definitions.

6.2. Assigning Boundary Conditions and U-factor Surface Tags

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. THERM indicates the boundary by a heavy line with different colors for different boundary condition definitions.

  - THERM will generate a separate boundary condition segment between each point on the outer surfaces of the model. The program will make assumptions about what Boundary Condition and U-factor Surface tags to apply -- *it is important to check all the boundary conditions to make sure the correct definitions are applied to all segments.* If the program did not make the correct default assumption, the boundary condition and U-factor Surface tags can be changed after they are generated.

  - Glazing systems inserted into THERM models from WINDOW will automatically be assigned boundary conditions based on the "environmental conditions" assigned to the glazing system in WINDOW. These boundary conditions will be named with the glazing system ID and name from WINDOW.

  - For interior boundary conditions, under most circumstances, the Radiation Model should be set to "AutoEnclosure". All the default interior boundary conditions are defined with this radiation model.

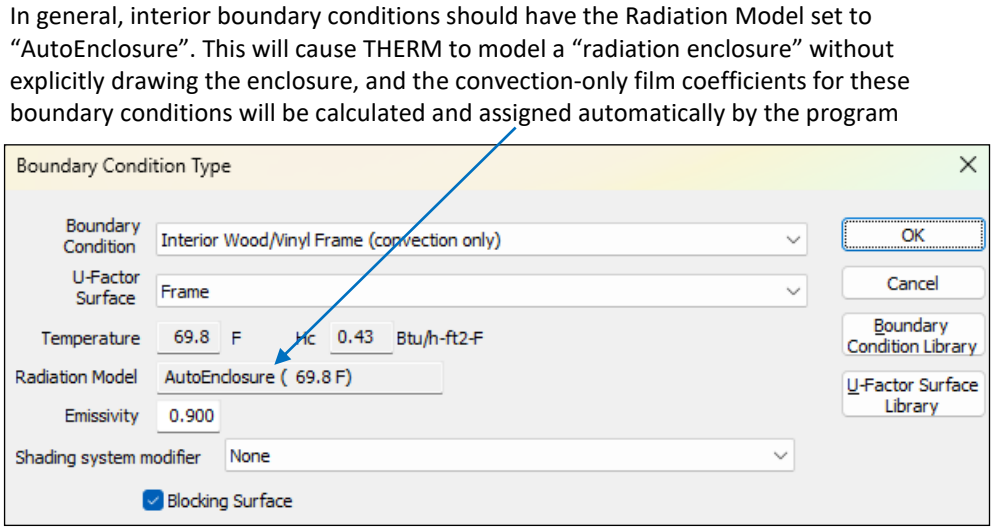


Figure 6-6. In general, interior boundary conditions have the Radiation Model set to "AutoEnclosure".

- THERM will also assign default U-factor Surface tags to each boundary condition segment. THERM will calculate a U-factor for each U-factor surface tag that is defined. The same U-factor surface tag can be applied to multiple boundary condition segments and a total U-factor will be calculated across the entire surface with the same tag.

- When generating the boundary conditions, the program checks for characteristics in the geometry that will cause a problem in the heat-transfer simulation. These include undefined voids, overlapping polygons 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.

- The boundary is actually composed of many small boundary segments (created by points in the model) 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 as described below in Section *6.2.4. Reassign Boundary Conditions*.

- THERM does not delete boundary conditions once they have been assigned, which is convenient in many situations. However, all the boundary conditions can be deleted at once by deleting one boundary condition.
- The **Eye Dropper** tool can be used to assign the properties of one boundary condition to other boundary conditions.
- The **Edit/Undo** feature will undo (one level) a change to a boundary condition.
- Define U-factor tags if desired. U-factor tags allow you to specify a group of boundary segments as one unit for which a single U-factor will be calculated. Generally the U-factor tags are applied to the interior (room-facing) side of the model.

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.

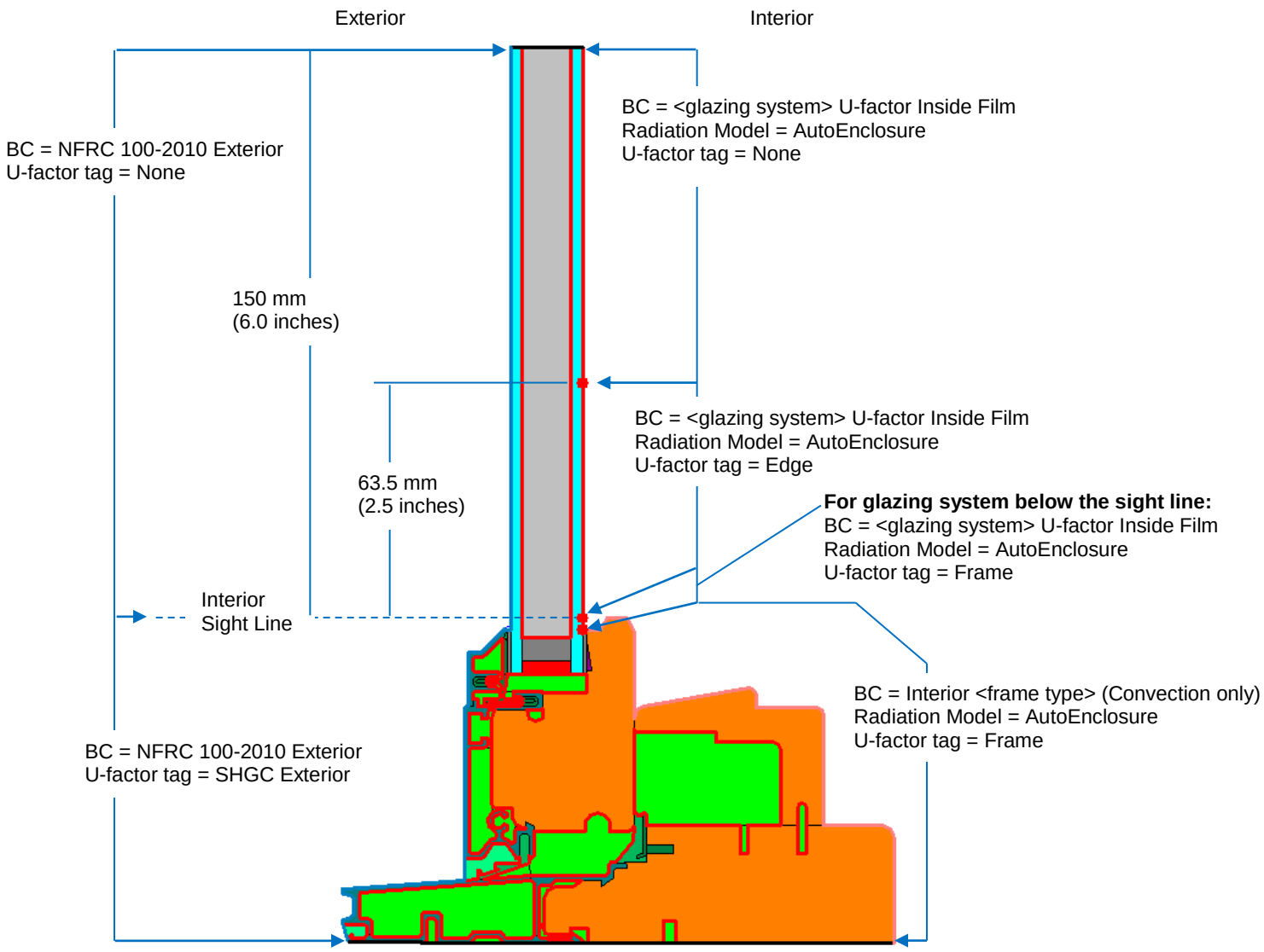


Figure 6-7. Standard boundary condition assignments for most models.

6.2.1. Generate Boundary Conditions

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

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, defined by each point in the exterior polygon surfaces, each of which is individually assigned a boundary condition.

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 or orange.

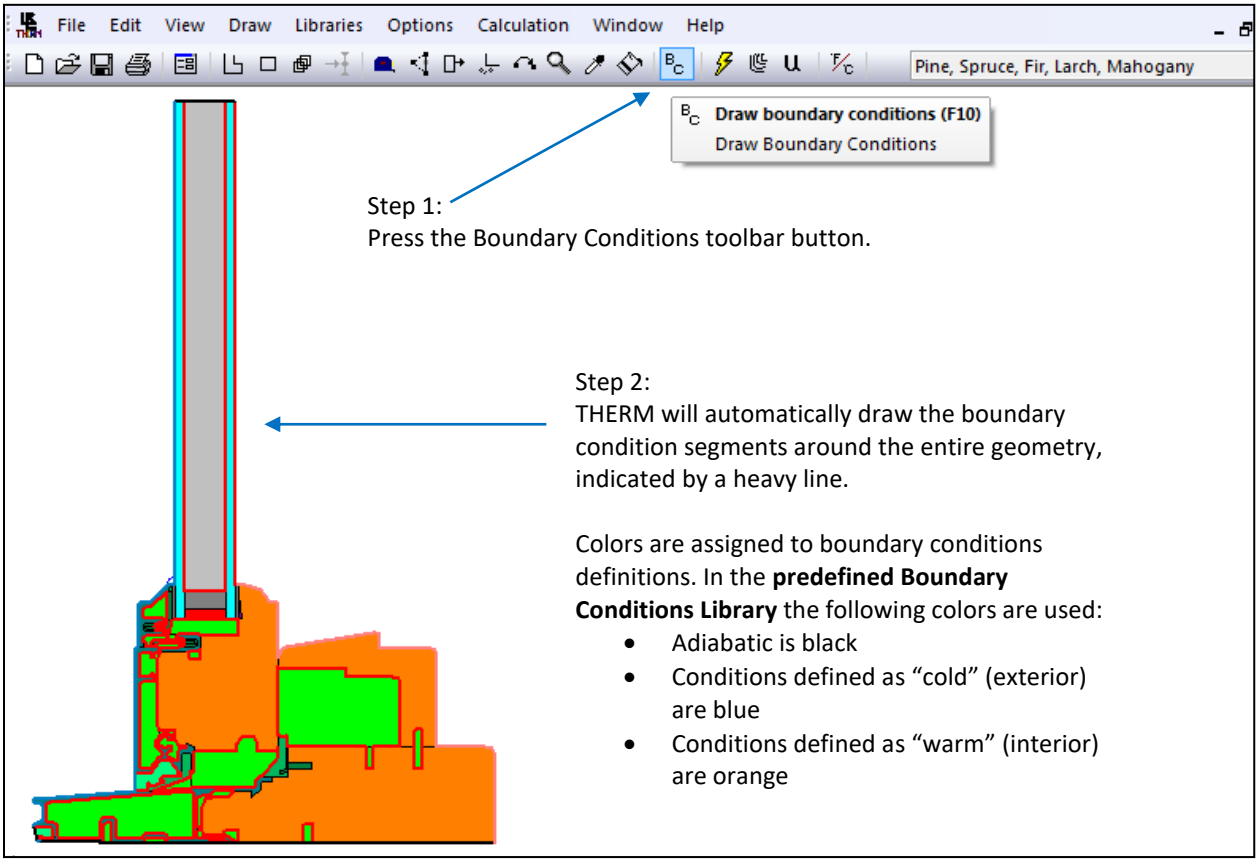


*Figure 6-8. Click on the **Boundary Condition** (BC) toolbar button; THERM will automatically draw the boundary condition segments, indicated by a heavy line.*

When boundary conditions are generated for a model, the program keeps them in the model until they are regenerated. All the boundary conditions can be deleted by deleting one boundary segment.

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. Review the Boundary Conditions

After THERM has generated the boundary conditions it is important to review them to make sure each boundary segment has the correct boundary conditions definition.

You can view the boundary condition for each boundary segment one at a time, but it is also possible to select multiple boundary segments that should have the same definition, and view the boundary condition for that selection at the same time.

If you double click on a boundary segment, or multiple boundary segments, the Boundary Condition Type dialog box appears.

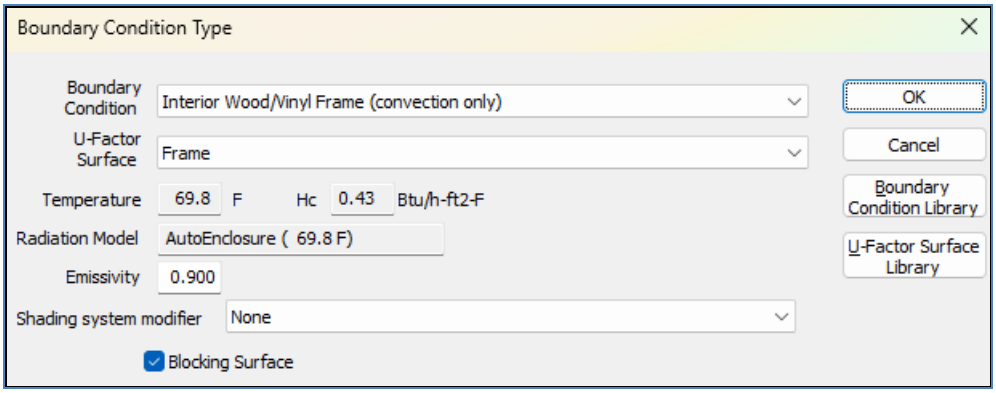


Figure 6-9. Double click on one or more boundary segments (or select the segment(s) and press the Enter key) to view the Boundary Condition Type dialog box.

Below is a description of the fields in the Boundary Condition Type dialog box.

Boundary Condition — A pulldown list showing the records in the Boundary Condition Library.

U-Factor Surface — A pulldown list showing the records in the U-Factor Names Library.

Temperature — The temperature of the boundary condition chosen from the Boundary Condition pulldown list. This field is informational only and cannot be changed here. It would need to be changed in the record referenced from the Boundary Condition Library. **Units:** F (IP); C (SI).

Hc — The film coefficient of the boundary condition chosen from the Boundary Condition pulldown list. This field is informational only and cannot be changed here. It would need to be changed in the record referenced from the Boundary Condition Library. **Units:** Btu/h-ft2-F (IP);W/m2-K (SI)

Radiation Model — The radiation model defined for the boundary condition chosen from the Bounardy Condition pulldown list. This field is informational only and cannot be changed here. It would need to be changed in the record referenced from the Boundary Condition Library.

For interior boundary conditions, this model should generally be set to "AutoEnclosure". For exterior boundary conditions, this model should generally be set to "Blackbody".

Emissivity The emissivity of the boundary segment. This value will default to the emissivity of the material defined for the polygon which the boundary segment is associated with. It can be edited if that default value is not appropriate.

Shading System Modifier A pulldown list showing shading systems being modeled, if relevant. If there are no shading systems being modeled, the value will be set to "None" and there will be no other choices in the pulldown list.

Blocking Surface A checkbox to indicate whether the selected boundary segment(s) is a blocking surface. This box should generally be checked for interior frame boundary conditions when the Radiation Model is set to "AutoEnclosure". These are the default settings that THERM will generally apply.

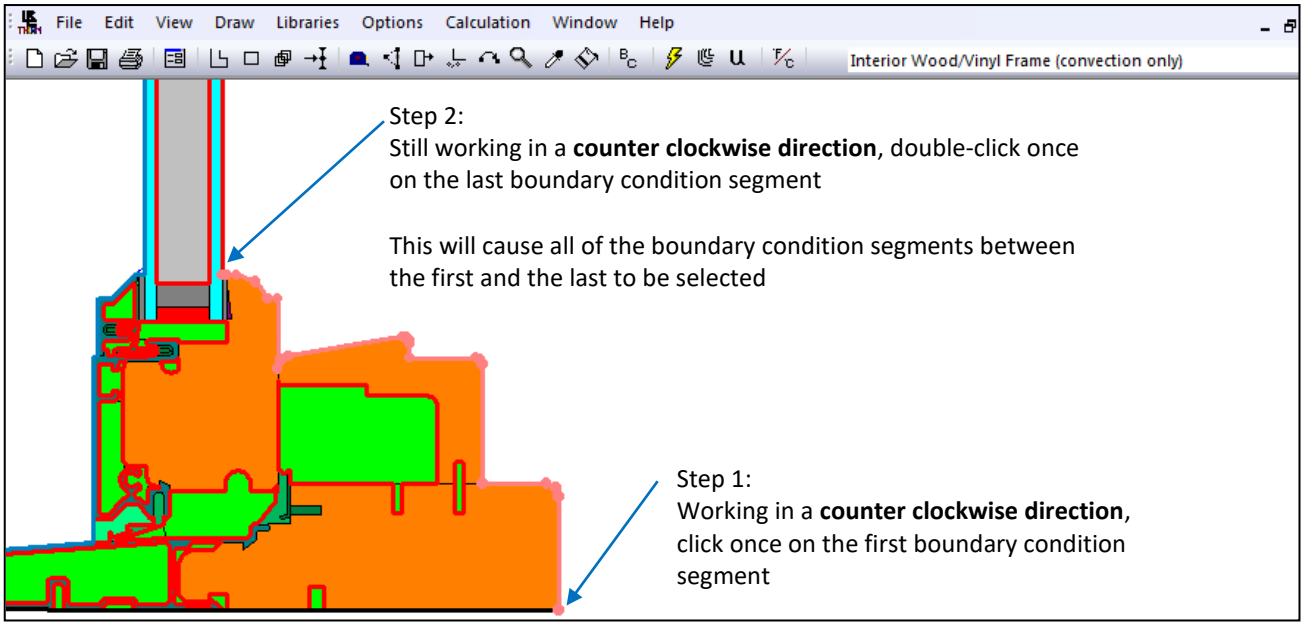


Figure 6-10. Select multiple boundary segments by pressing down the shift key and clicking the mouse on segments as you move in a counter-clockwise direction, OR click on the first segment and then double-click on the last segment, in a counter-clockwise direction; all segments in between will be selected

6.2.4. 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* ***Error! Reference source not found.***. THERM cannot perform the simulation without at least two non-adiabatic boundary conditions, and will issue a message if this condition is not met. To reassign boundary conditions:

- 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.

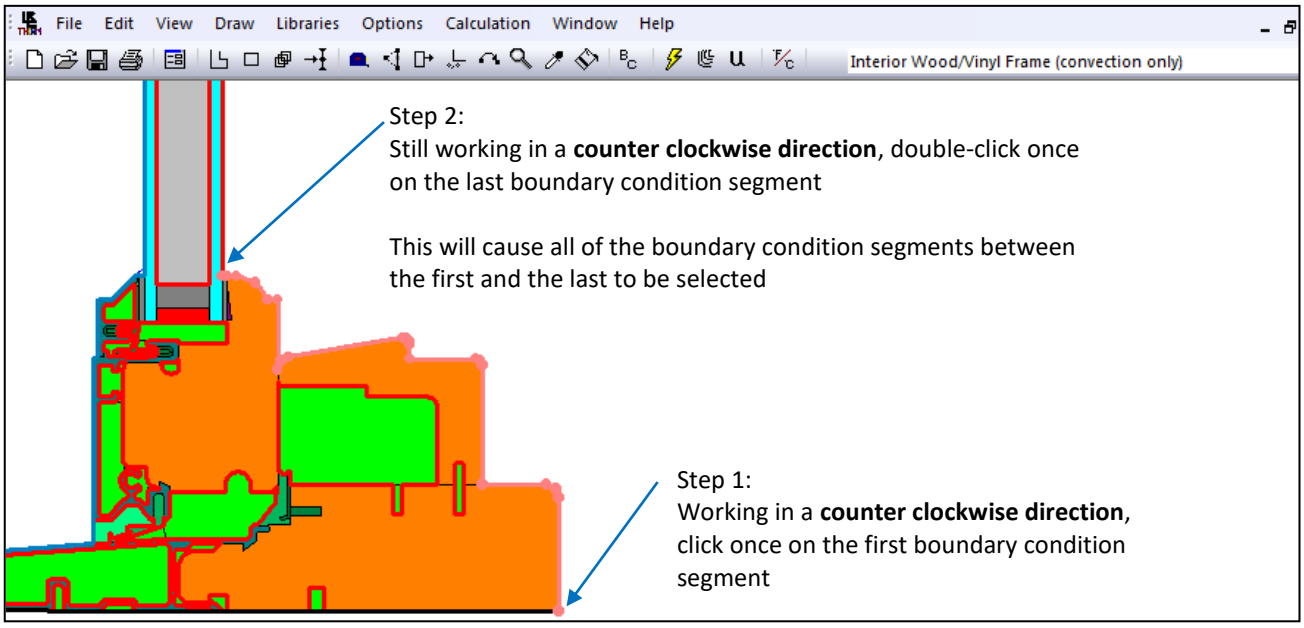


Figure 6-11. Select multiple boundary segments by pressing down the shift key and clicking the mouse on segments as you move in a counter-clockwise direction, OR click on the first segment and then double-click on the last segment, in a ***counter-clockwise*** *direction; all segments in between will be selected.*

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**.

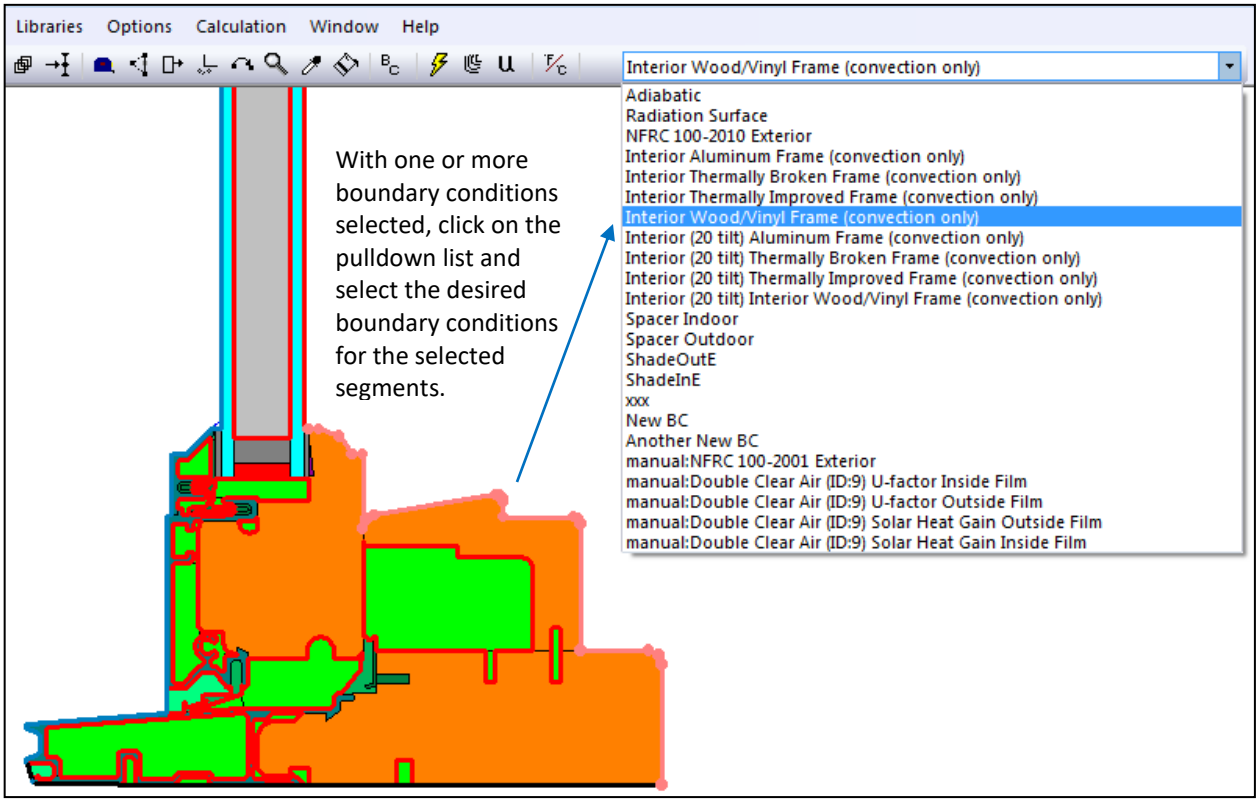


Figure 6-12. One method for defining the boundary condition properties is to click on one or more boundary segments and pick the desired boundary condition definition from the pull-down list.

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 access 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.

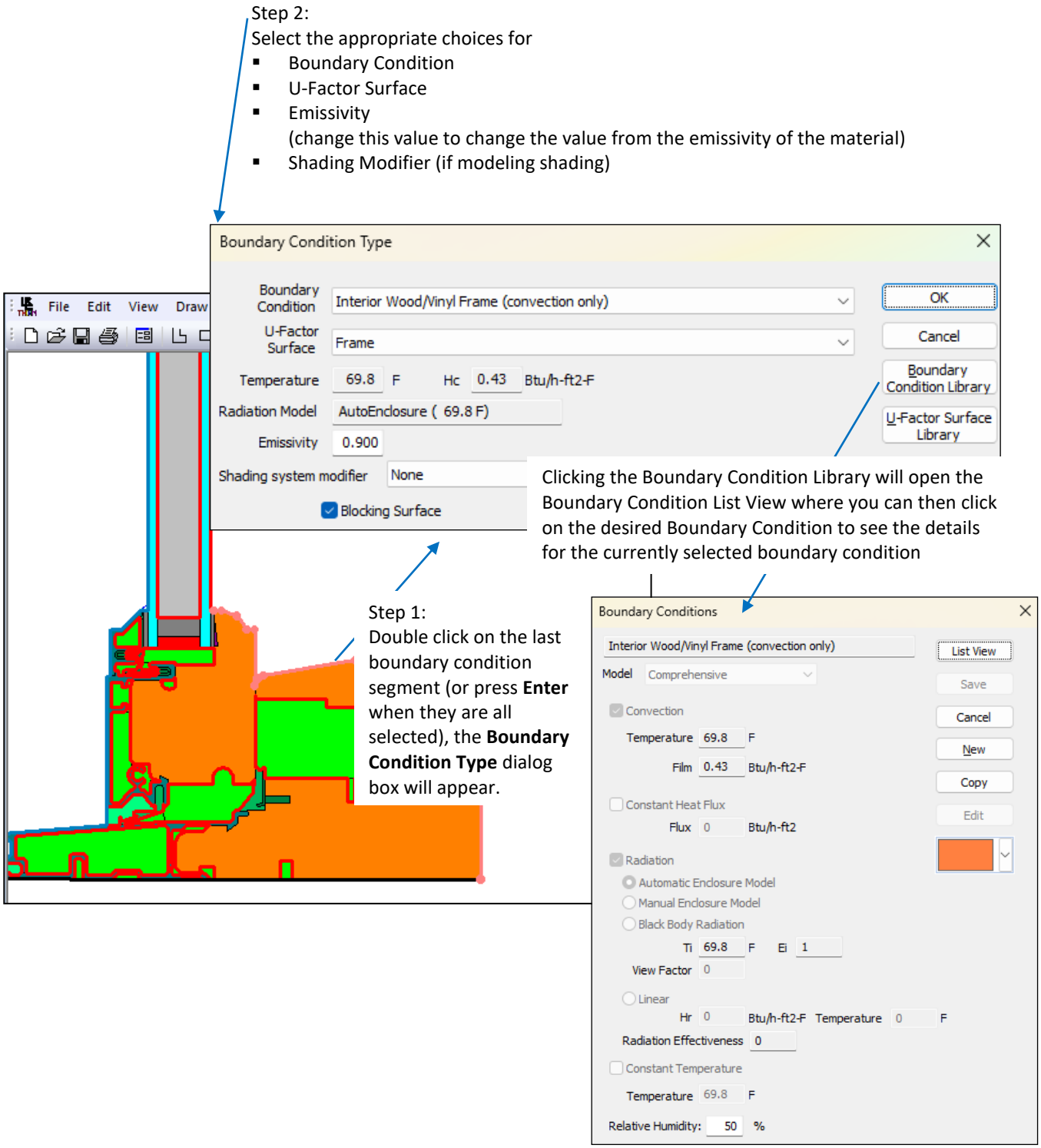


Figure 6-13. Another method for assigning boundary conditions is to double-click on a segment to access the ***Boundary Condition Type*** *dialog box where you can select the* ***Boundary Condition*** *as well as the* ***U-factor Surface*** *tag name.*

Eye Dropper Tool:

The **Eye Dropper** tool can be used to assign the properties of one boundary condition to another boundary condition.

- Click the left mouse button on the **Eye Dropper** toolbar button.
- The drawing cursor will change to the **Eye Dropper** cursor.
- Click on the boundary condition which is defined with the boundary condition to be copied.
- The cursor will change to the **Fill** icon. and the **Boundary Condition** pulldown list will be set to material to be copied.
- Place the **Fill** cursor on any boundary condition, and click the left mouse button. This boundary condition will be defined with the new boundary condition.

Glazing System Boundary Conditions

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.

- <THERM filename>:<Glazing system name> (ID:<Glazing System ID) U-factor Inside Film

For example, the name of an inside glazing system boundary condition would be:

- Sample-sill:Sample GlzSys (ID:8) U-factor Inside Film

Where

- THERM filename = Sample-sill
- WINDOW Glazing System Name = Sample GlzSys
- WINDOW Glazing System ID = 8

6.2.5. 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 required to calculate the appropriate U-factor. You can create as many tags or groups as needed. 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 both the interior and exterior surfaces are given the same **U-factor Tag**, the resulting U-factor will be zero.

The default choices for the **U-factor Tag** pull-down list are:

- **Frame:** This tag can be used with interior boundary conditions of a frame system for defining the boundary segments used to calculate a frame U-factor. If a glazing system is imported from WINDOW into a frame model, THERM will automatically assing the U-factor tag of "Frame" to the portion of the glazing system that is below the sight line (which is defined when inserting the glazing system into the THERM model).
- **Edge:** This tag can be used with interior boundary conditions of a glazing system for defining the boundary segments used to calculate an edge-of-glazing U-factor. If you are importing this model into the WINDOW Frame or Divider Library, the edge of glass must be defined. A standard definition for edge-of-glazing is 63.5 mm (2.5 inches) above the frame sight line, which is specified when importing the WINDOW glazing system into a THERM model.
- **None:** This tag can be applied to any boundary condition segment where a U-factor results is not desired. One common use of this tag is to apply it to the area of the glazing system above the segment tagged as Edge. Another place where this tag should always be used is for all adiabatic boundary segments.
- **SHGC Exterior:** This tag can be applied to the exterior non-glazing surfaces of a model to calculate the wetted length of the exterior frame which can then be used by WINDOW, when the model is imported into the WINDOW Frame or Divider Library, for the Solar Heat Gain Coefficient (SHGC) calculation.

You can make new values in the **U-factor Names Library** so that THERM will calculate U-factors for situations other than those used for the default modeling conditions. Click on the **U-factor Surface Library** button on the **Boundary Condition Type** dialog box. Then 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**.

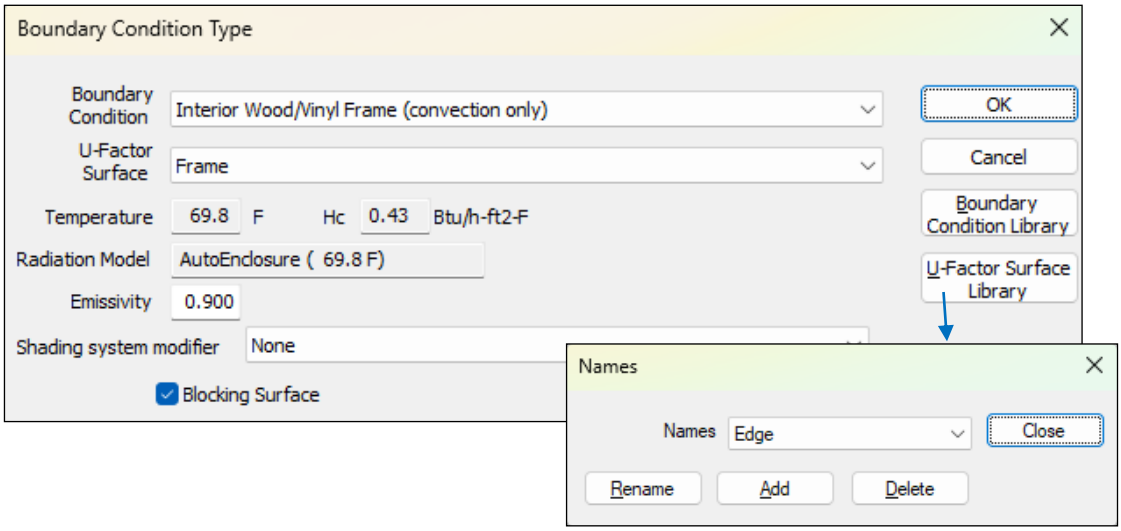


Figure 6-14. Create a new U-Factor Surface tag by clicking on the U-Factor Surface Library button on the Boundary Condition Type dialog box.

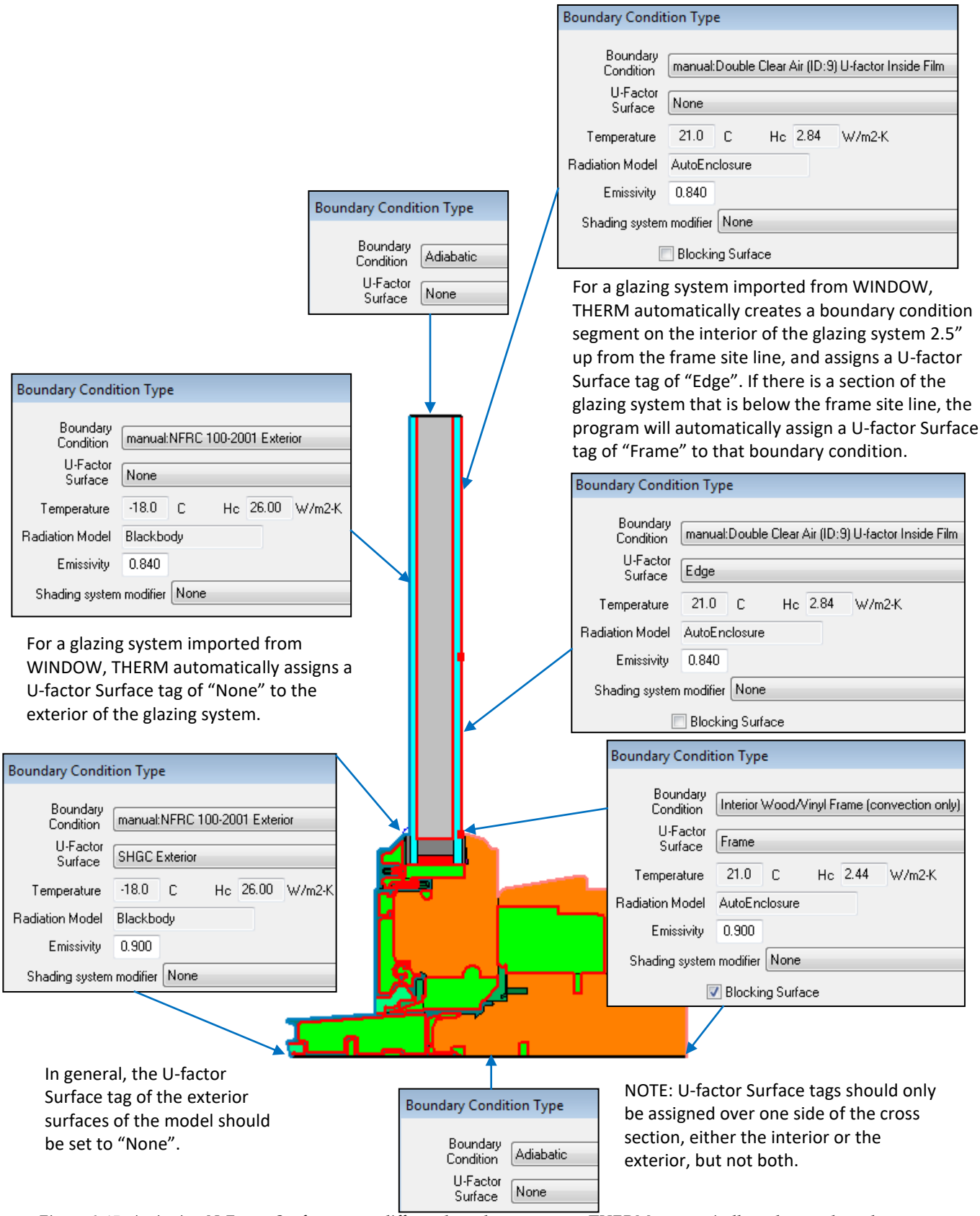


Figure 6-15. Assigning U-Factor Surface tags to different boundary segments. THERM automatically makes two boundary segments on the interior surface of the glazing system, one from the frame to the sight line (with a U-Factor Surface tag of "Frame") and one from the sight line up to 2.5" of the glass (with a U-Factor Surface tag of "Edge").

6.3. Error Detection in THERM

In order to perform a simulation, the geometry of the cross section must be correctly defined, i.e., there must not be any voids, overlapping polygons, or points that cannot be resolved by THERM.

While drawing, there are a number of features in THERM to help avoid creating voids, overlapping polygons, and bad points. The **Always Check for Overlapping Polygons** feature (in **Options/Preferences/Drawing Options**) is the main tool used to avoid drawing overlaps and voids. This option can be turned off, but this is NOT recommended for most situations. The **Show Voids / Overlaps** (from the **View** menu) feature is also a good tool for visual inspection to find voids and overlapping polygons in a cross section.

In addition to the features that attempt to prevent the creation of bad geometry during the drawing process, a number of error checks are performed when THERM calculates the boundary conditions in order to make sure the model geometry is correct.

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, Calculating Results*.

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 6-16.
- 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.

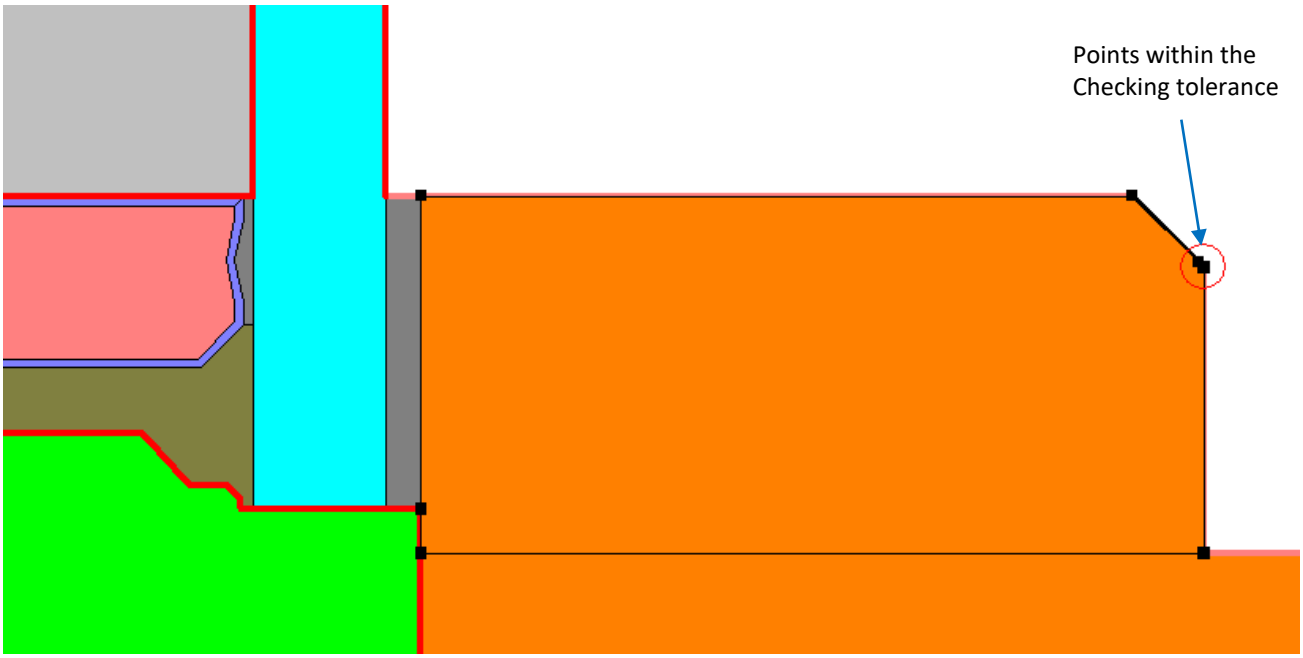


Figure 6-16. THERM will show a red circle around points that are too close together.

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:

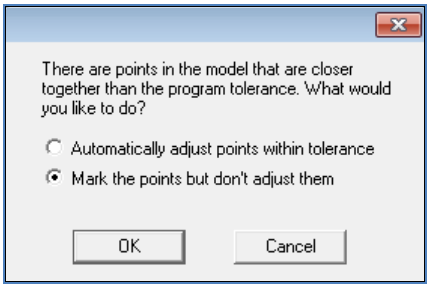


Figure 6-17. This message may appear when you create the boundary condition

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 points 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.

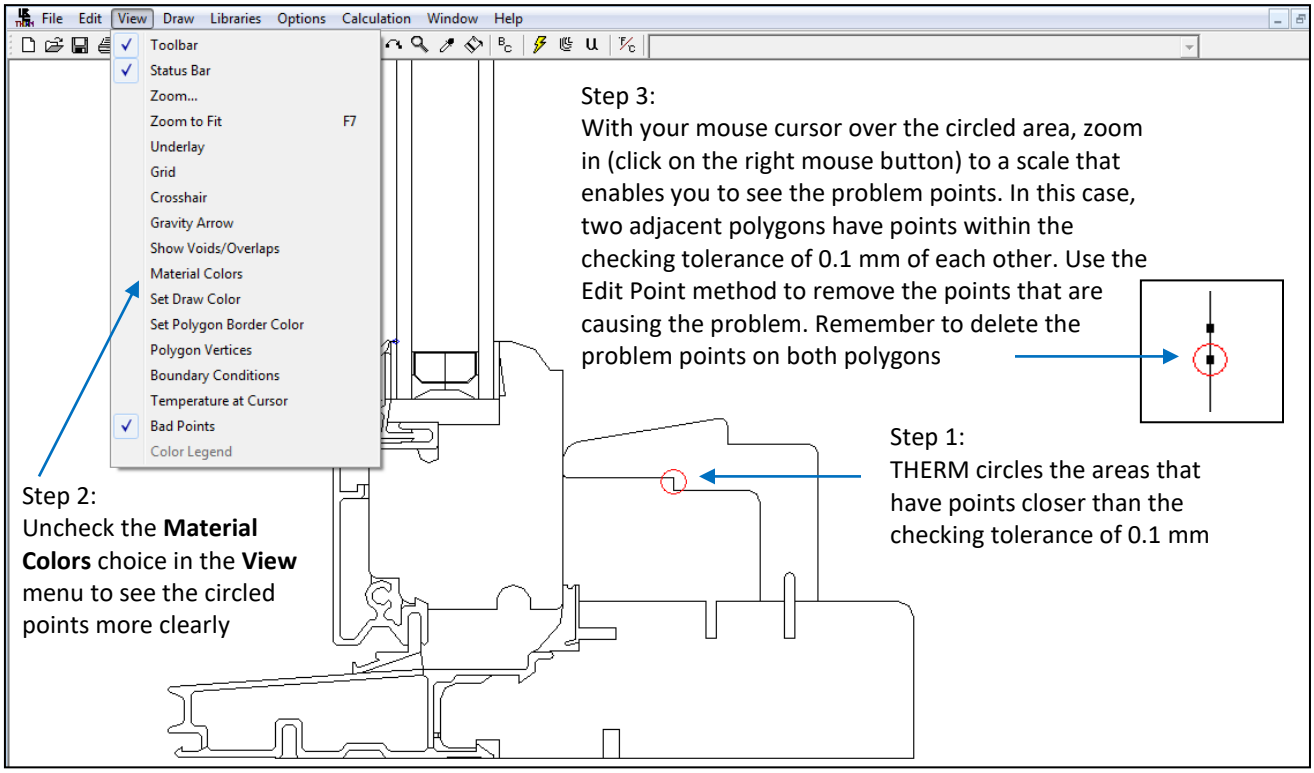


Figure 6-18. An Instance where THERM created bad points in the process of checking for matching points on adjacent polygons

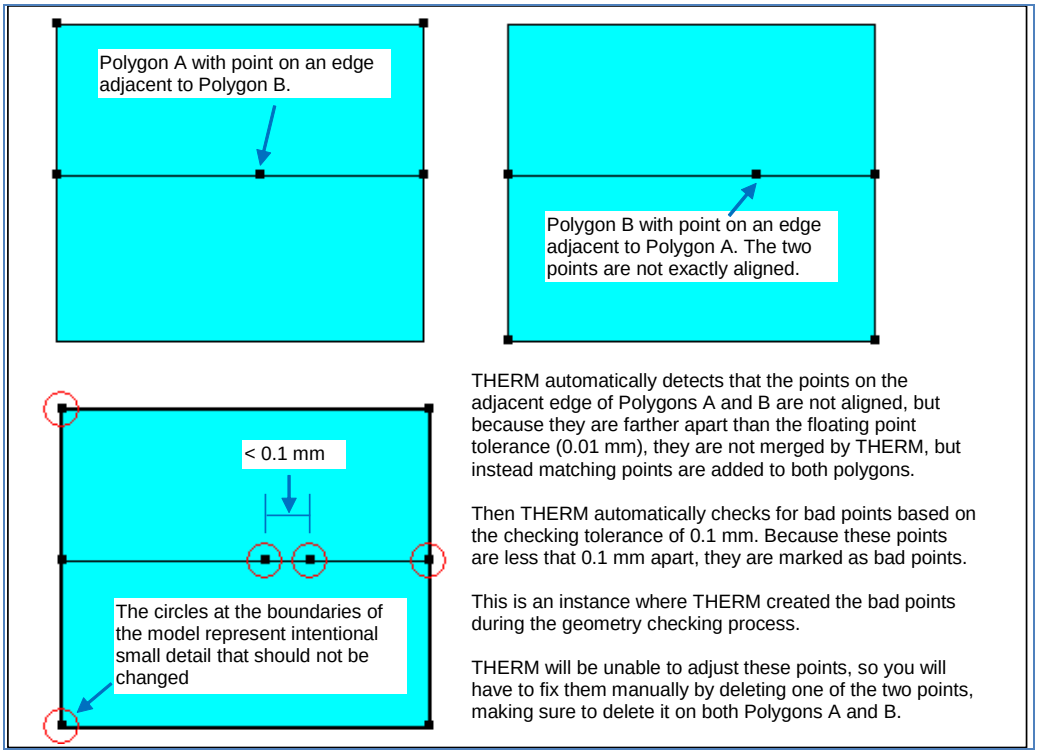


*Figure 6-19. To view the circled "bad" points more clearly, "uncheck" the Material **Colors** choice from the **View** menu.*

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 6-20. The unfixed points will be marked and you must fix them before the simulation can continue.

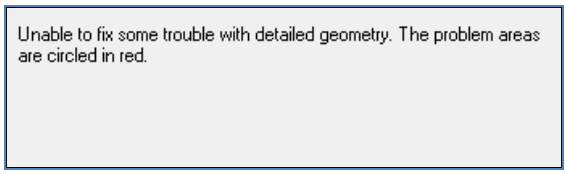


Figure 6-20. This message may appear if you select the automatic adjustment option and THERM cannot correct the problem.

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.

Preferences:

The **Options/Preferences** menu, **Preferences** tab can be used to control whether THERM asks before automatically adjusting points. The default setting is that the option is "checked", meaning that the program will always ask before adjusting the points. Turning this feature off may result in undesired results, so change this setting with caution.

The Options/Preferences menu is used to display the Preferences dialog box. The “Ask before automatically adjusting points” option is turned “on” when the box is checked, and “off” when the box is not checked.

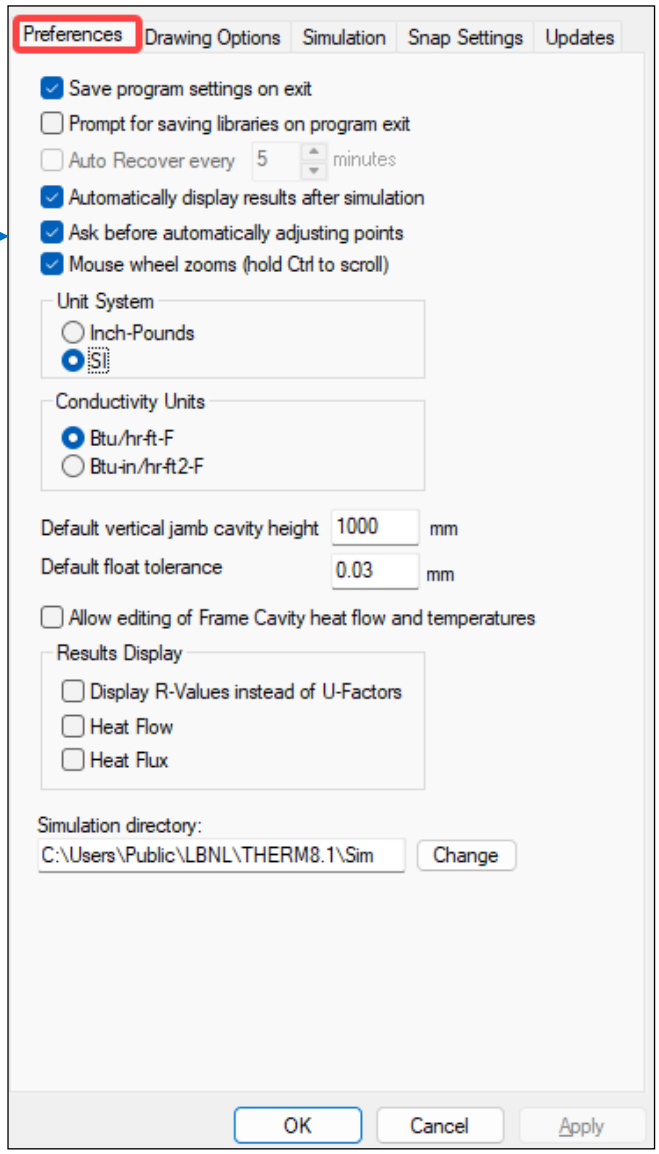


Figure 6-21. The option of having THERM ask before automatically adjusting points can be turned on and off.

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** feature 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 6-22. These areas must be fixed before the simulation process can continue.

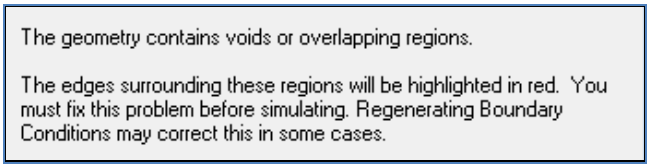


Figure 6-22. THERM issues a warning if it detects voids or overlapping regions when trying to define boundary conditions.

One way to eliminate these two conditions is by "jiggling" the points -- moving the points away from the adjacent polygon, then moving them back within the sticky distance and letting the program "snap" them to the polygon. Another way is to delete the offending polygon, and use the **Fill Void** toolbar button to create it again, which means that THERM will automatically align all points and ensure that no voids are created.

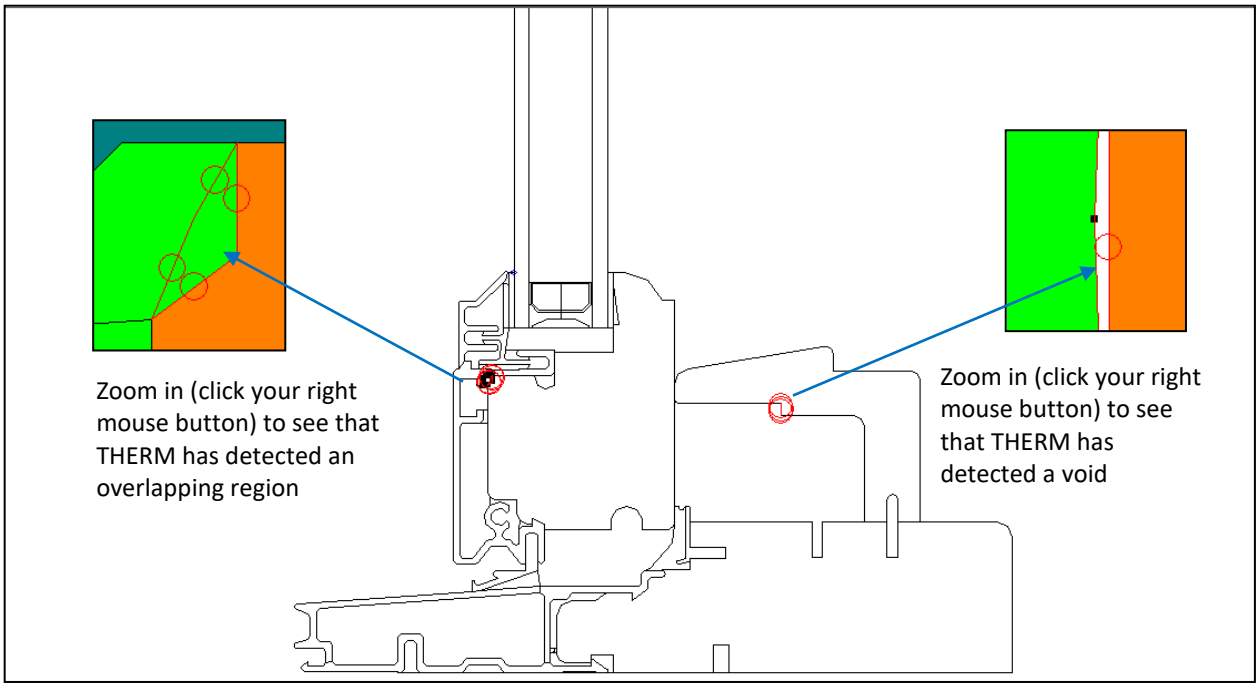


Figure 6-23. Voids and overlaps are indicated by circles once the boundary conditions have been assigned.

THERM can also find invalid polygons caused by points collapsing on each other, creating a zero-area extension. In this case both the point at the tip of the extension and the extra point at its base must be deleted.

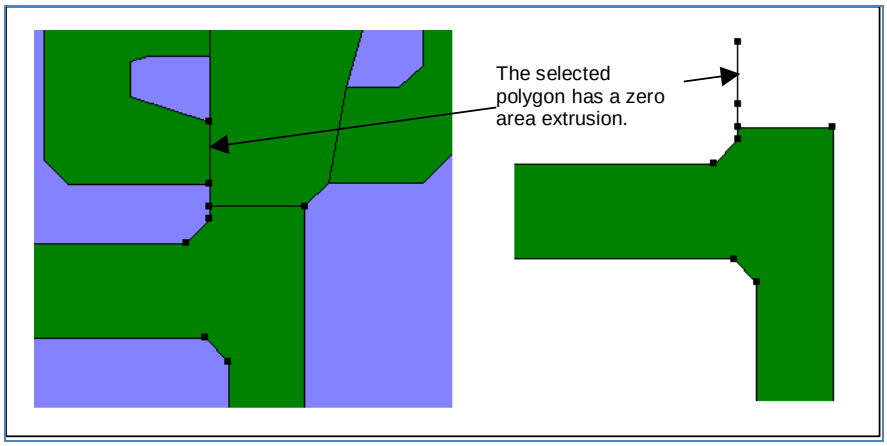


Figure 6-24. An example of a zero area extrusion which will result in an error message of "Invalid Polygon found, Stopping Simulation".

Turning off the **Material Color** and the **Boundary Conditions** from the **View** menu can sometimes help to find the problems.

THERM will simulate a problem even if "bad points" have been marked and intentionally left in the cross section, because in some cases details are drawn that give bad points but which need to be modeled.

If changes are made to polygons on the boundary of the cross section, the boundary conditions must be generated again. If all the corrections are on internal polygons, the boundary conditions will not be automatically deleted. However, in this case, it is still a good idea to press the **Boundary Conditions** toolbar button again, to ensure that all points on all surfaces have boundary segments defined between them.

6.4. Boundary Condition Library

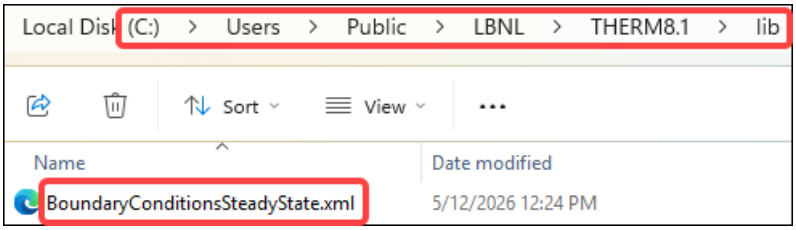
The Boundary Condition Libraries are now stored in XML files in the default location of

C:\Users\Public\LBNL\THERM8.1\lib

6.4.1. Boundary Condition Library for Steady-State Model

The Material Library for the Steady-State model is an XML file that is stored in the "lib" folder of the THERM working directory, which by default is:

C:\Users\Public\LBNL\THERM8.1\lib



6.4.1.1. Library List View

Selecting the Boundary Condition choice from the Libraries menu when the Steady-State simulation model is selected displays the Boundary Condition records in "List View". This Library has the records that have been distributed with previous versions of THERM.

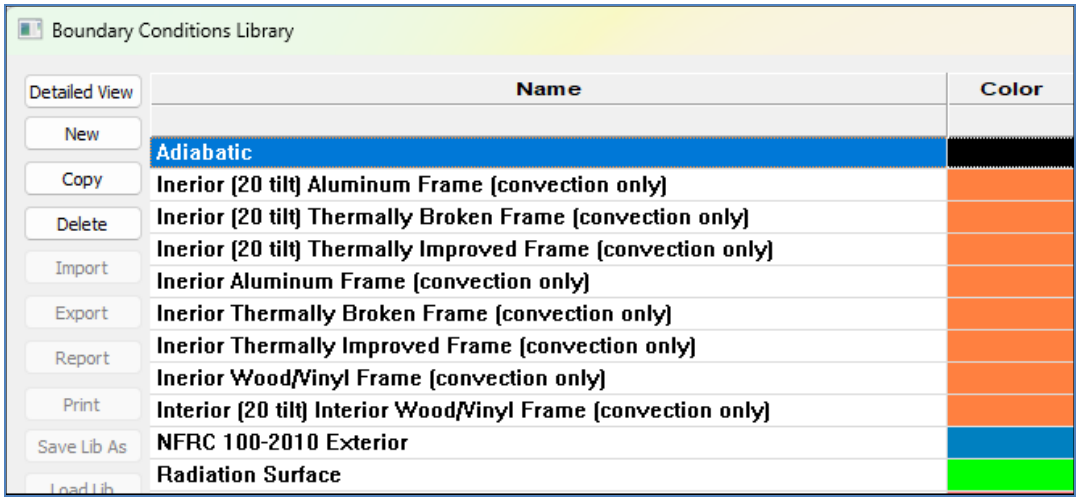


Figure 6-25. Boundary Condition List View for Steady-State model.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.
Import	*Currently disabled.*
Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the boundary condition.
Color	The color of the boundary conditions.

6.4.1.2. Boundary Condition Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each boundary condition.

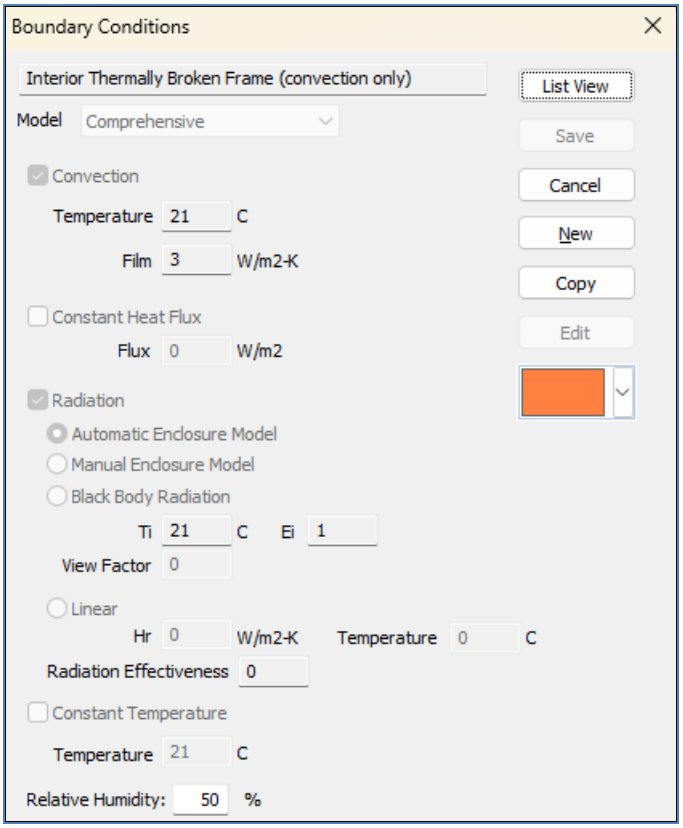
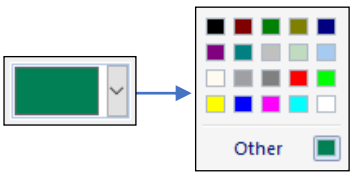


Figure 6-26. Boundary Condition Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Return to the list view of the library.
Save	Save changes to the record. Only available for user-defined records.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	*Currently not functional*
Edit	*Currently disabled*

Color	The color assigned to the boundary condition. It can be changed by clicking on the Color button  *Figure 6-27. Color Settings in the Boundary Conditions Library Detail View.* One convention is to make the interior (warm) boundary conditions in the "red" colorway, and the exterior (cold) boundary conditions in the "blue" colorway.

Boundary Condition Model = Simplified

This Boundary Condition model is defined by just Temperature and Film Coefficient

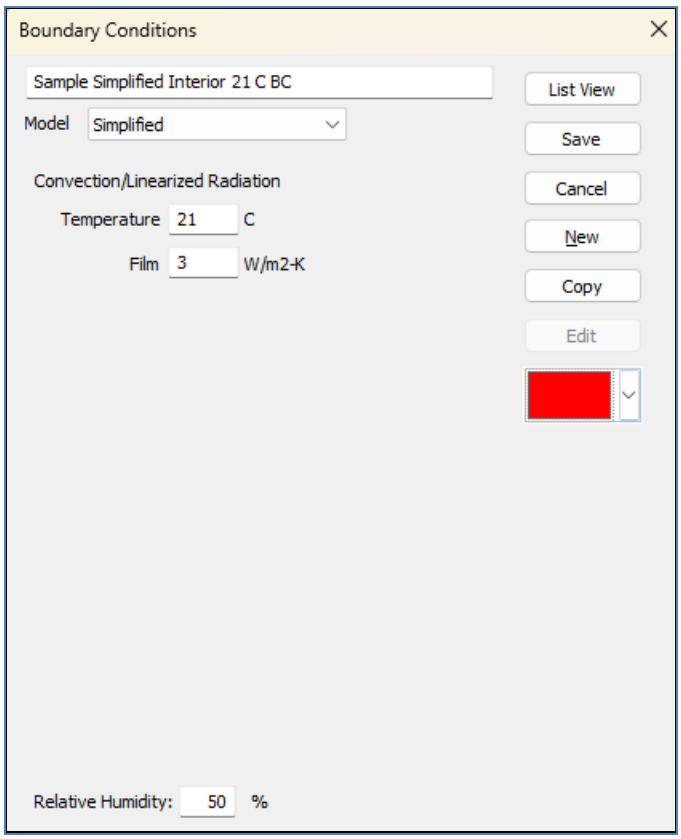


Figure 6-28. Model = Simplified for Boundary Condition definition

Model Simplified

Convection / Linearized Radiation

Temperature The temperature for the boundary condition. Units: F (IP); C (SI).

Film Coefficient The combed convection and radiation surface film coefficient to be applied to the boundary condition. **Units:** Btu/h-ft2-F (IP); W/m2-K (SI).

Boundary Condition Model = Comprehensive

The "Comprehensive" model allows more detailed boundary condition input for both convection and radiation.

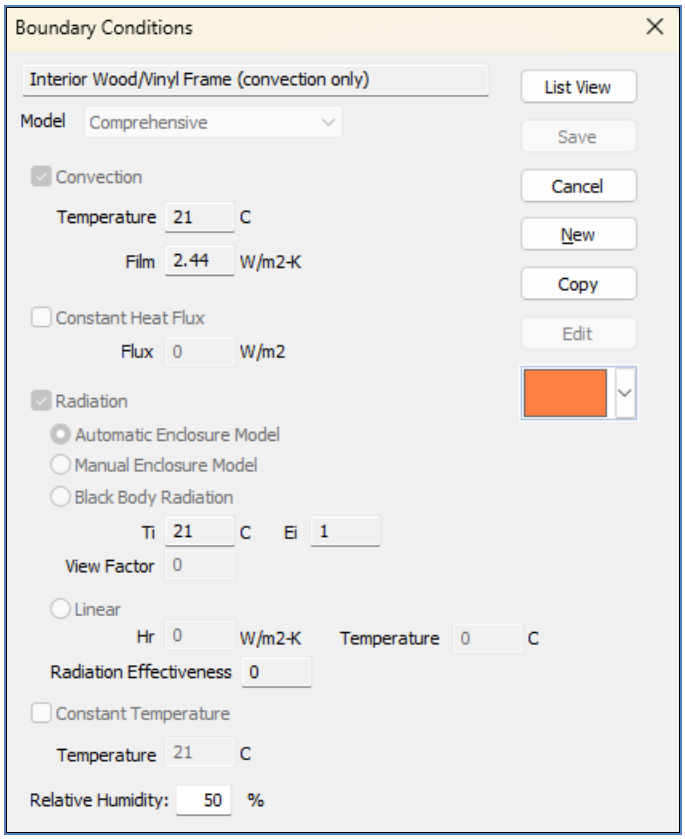


Figure 6-29. Model = Comprehensive for Boundary Condition definition

The input values for Model = Comprehensive are:

Convection Check this box to define the temperature and film coefficient for the convective heat flow.

- **Temperature:** Temperature of surrounding air, used for convective heat transfer. **Units:** °F (IP); °C (SI)
- **Film coefficient:** Film coefficients to use for convective heat transfer. The Film coefficient definition is 4 + 4v, where v = wind speed (m/sec). **Units:** Btu/h-ft²-°F (IP); W/m²-°C (SI).

Constant Heat Flux Check this box to define a constant heat flux on the boundary surface, such as solar gain.

- **Flux:** Constant flux value. **Units:** Btu/h-ft² (IP); W/m² (SI)

Radiation Check this box to define how radiation will be modeled.

- **Automatic Enclosure Model:** Select this option to have THERM automatically

define the radiation enclosure.

- **Manual Enclosure Model:** Select this option to define boundary conditions for surfaces that will be next to a Radiation Enclosure that is drawn explicitly as a polygon.
- **Black Body Radiation:** The Black Body Radiation model assumes an idealized, perfectly emitting surface, where the emittance is always 1, and there is no inter-reflection modeled. For this Radiation model the following values also must be defined
  - **Ti:** Temperature of the enclosure. Units: F (IP); C (SI)
  - **Ei:** Emittance of the enclosure, default is 1 (perfectly emitting surface)
  - **View Factor:** Fraction of radiation that leaves Radiation Enclosure surfaces that strikes boundary surfaces.
- Linear:
  - **Hr:** Radiation film Coefficient. Units: Btu/h-ft^2-^{o}F (IP); W/m^2-^{o}C (SI)
  - **Temperature:** Units: ^{o}F (IP); ^{o}C (SI)
  - **Radiation Effectiveness:** Default value is 1

Constant Temperature

Currently disabled

Used to define a boundary condition where the surface temperature is constant.

- **Temperature:** Units: ^{o}F (IP); ^{o}C (SI)

Relative Humidity

THERM will calculate the condensation potential on a surface based on the relative humidity value define with the assigned boundary condition. The condensation potential results are described in Section 7. **Units:** percentage (%)

Boundary Condition Model = Radiation Enclosure Surface

Defining an actual geometry that represents a Radiation Enclosure is not generally needed in most models, as THERM now has the option to define boundary conditions where the Radiation Model is "AutoEnclosure". However, the ability to define geometry that represents a "Radiation Enclosure" still exists in the program, as explained below, if there is a desire to use it.

If you assign the material called "Radiation Enclosure" 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) which has the boundary condition model set to "Radiation Enclosure Surface".

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.

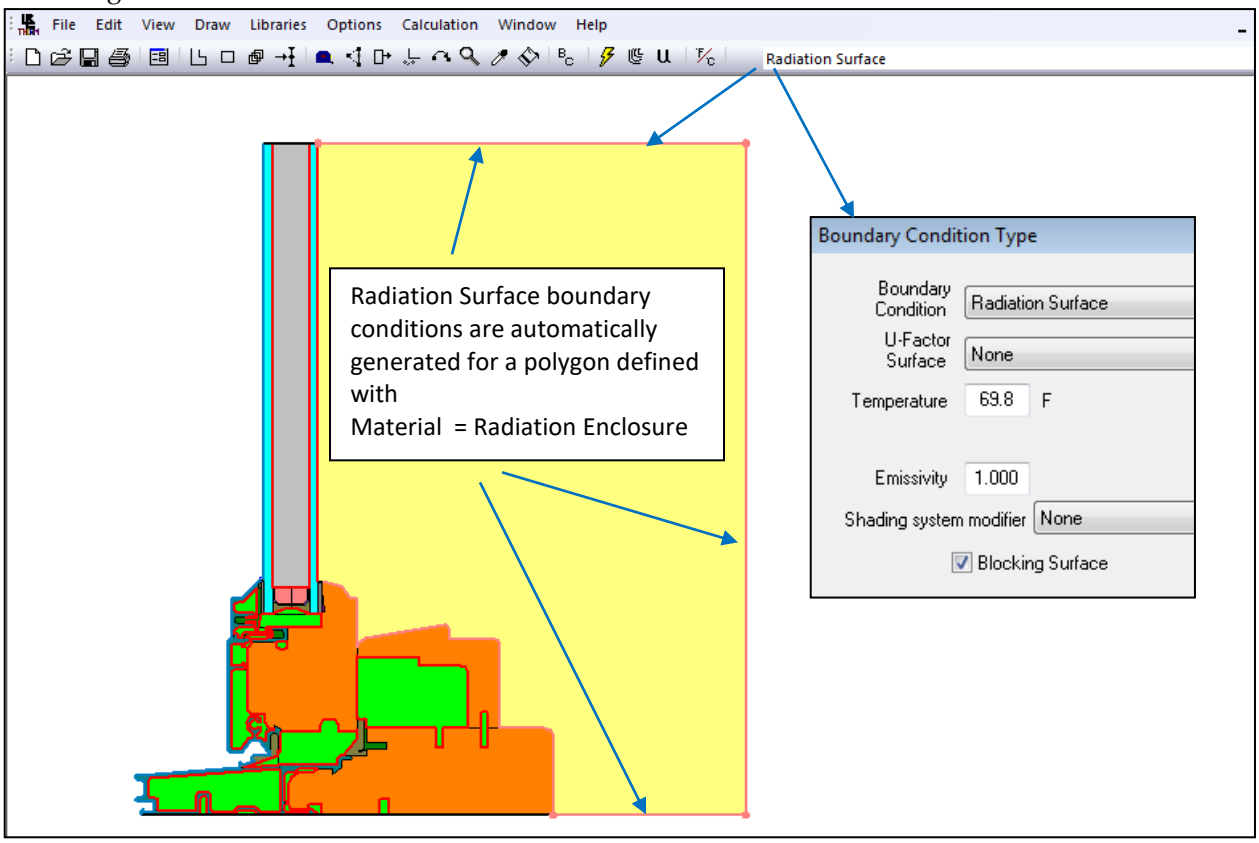


Figure 6-30. THERM automatically assigns Radiation Surface boundary conditions to the external radiation enclosure surfaces.

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 that have the following definitions:

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 **Manual Enclosure Model**.

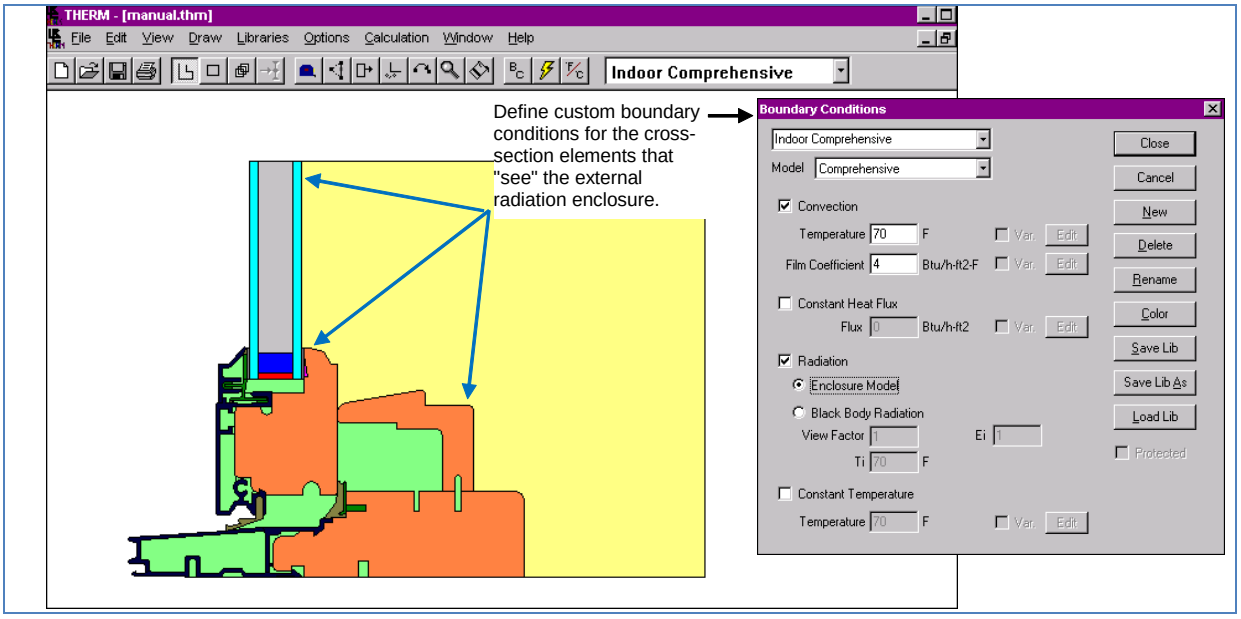


*Figure 6-31. Define new boundary conditions for the elements that "see" the radiation enclosure with the **Comprehensive** model.*

When you have defined the new boundary condition in the library, you need to assign it to the appropriate boundaries, as shown in the figure below.

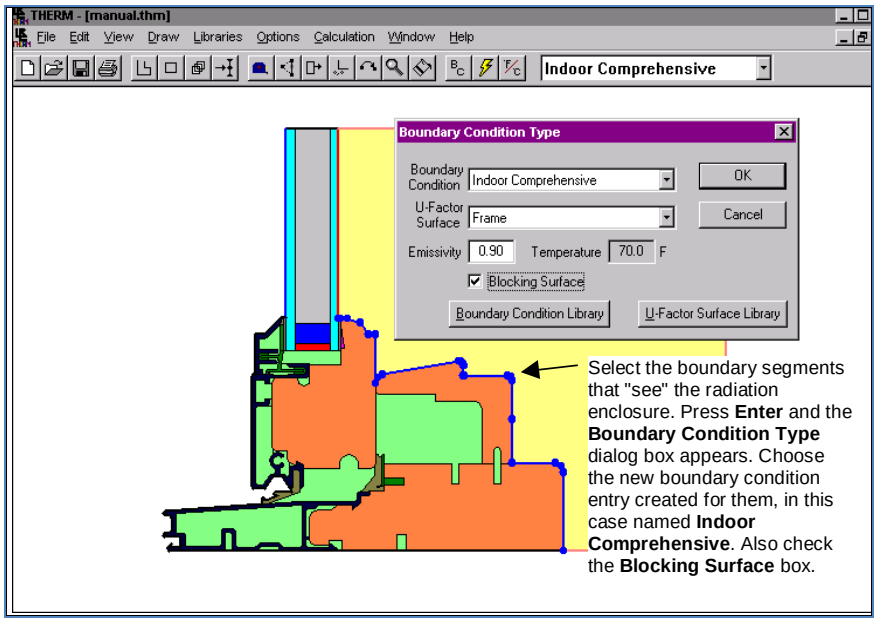


Figure 6-32. Set the boundary conditions for the surfaces that "see" the radiation enclosure.

6.4.1.3. Making New Boundary Condition Library Records

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.

- 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.

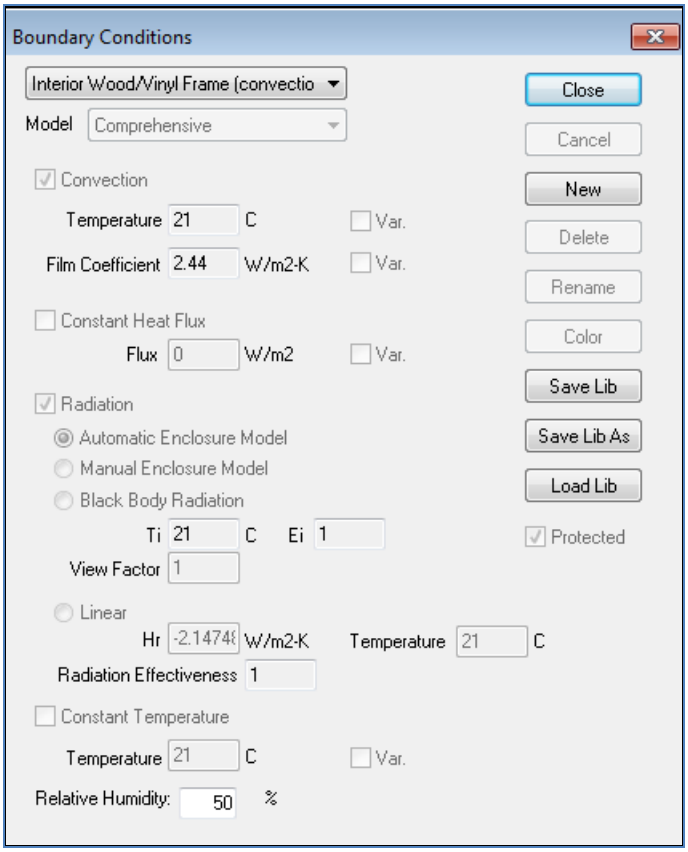


Figure 6-33. The default Boundary Condition Library that comes with THERM has boundary condition definitions that cannot be edited.

- To make a new boundary condition, click on the **New** button and a dialog box will be displayed where the new values can be added.

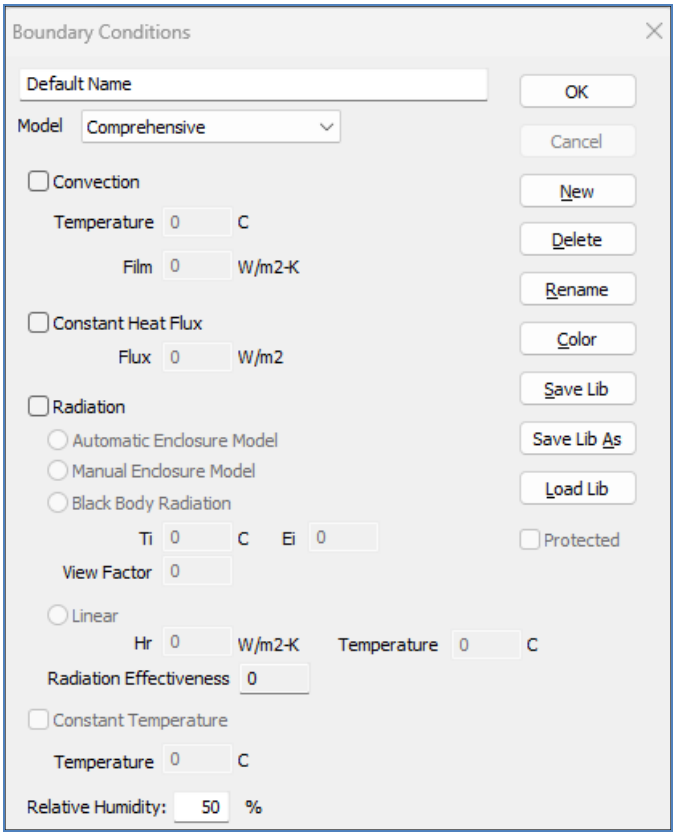


Figure 6-34. The dialog box for createing a new Boundary Condition record..

- 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.
- 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**).

6.4.1.4. Frame Cavities with Radiation Model

If frame cavities in the model are set to a material where the Material Type is set to "Frame Cavity", when THERM generates the boundary conditions for that cavity, those frame cavity boundary conditions will automatically be highlighted in red.

If you change the frame cavity to another type of material after the boundary conditions are generated, you must regenerate the boundary conditions.

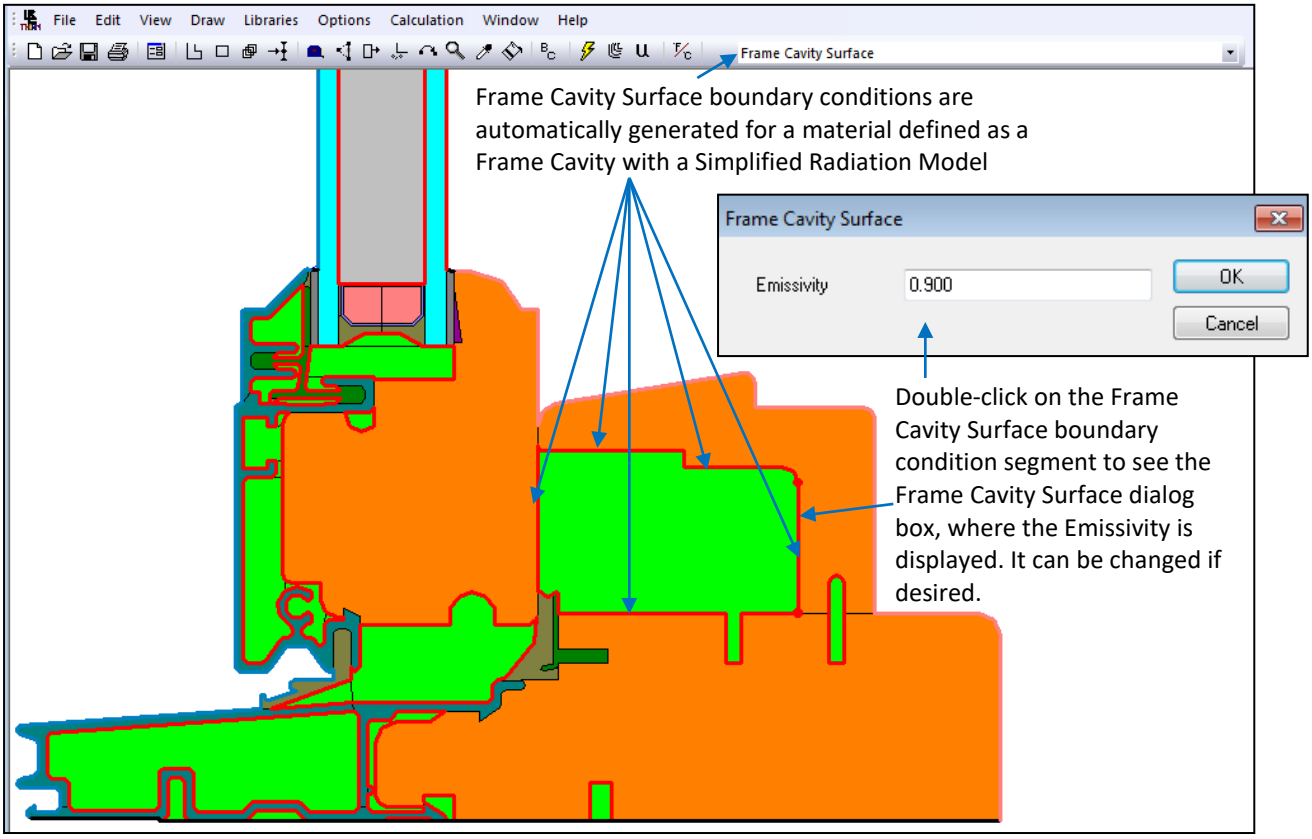


Figure 6-35. When a polygon is assigned to a Frame Cavity material, THERM automatically assigns boundary conditions to the cavity.

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 regenerated (by clicking on the **Boundary Conditions** toolbar button or pressing **F10**.)

If frame cavities are broken up into multiple cavities to better model convection heat transfer, THERM treats the sub-cavities individually for convection purposes but merges them for radiation purposes. The red boundary segments will show this.

This frame cavity illustrates the THERM feature that even when a frame cavity is broken into several polygons, to improve the thermal modeling, the program will treat these frame cavities as one when modeling radiation transfer in the cavity. This is shown by the fact that the red frame cavity boundary encloses all the separate frame cavities that are adjacent to each other

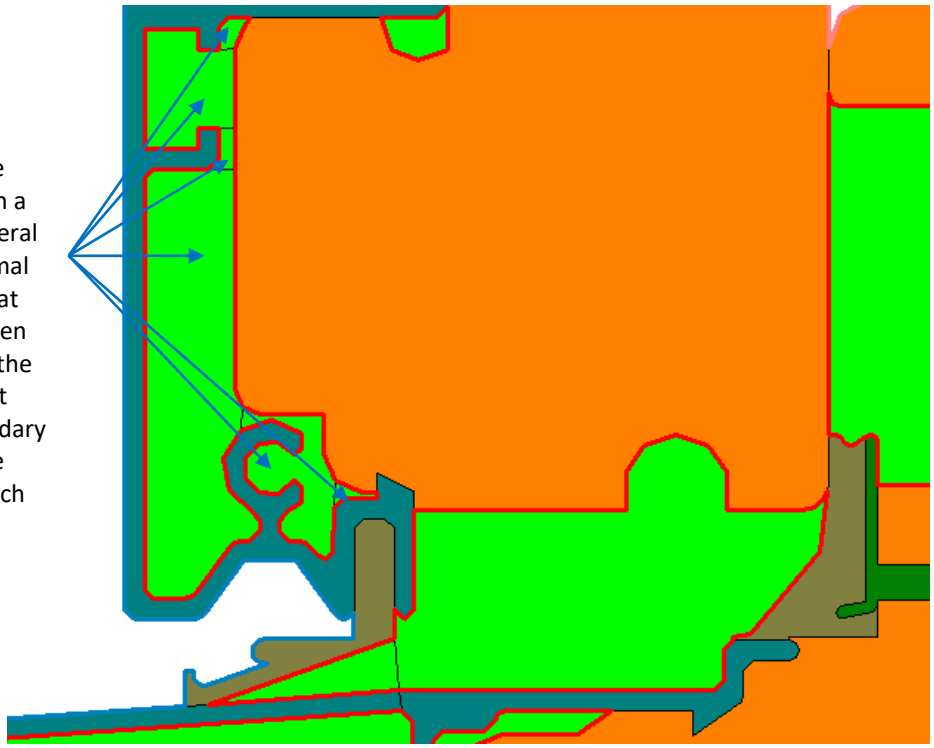


Figure 6-36. Adjacent frame cavities are treated as one when THERM models the radiation in those cavities.

THERM will automatically determine the blocking surfaces for a polygon defined with a Frame Cavity material. Blocking surface definitions are not part of a Frame Cavity boundary condition definition.

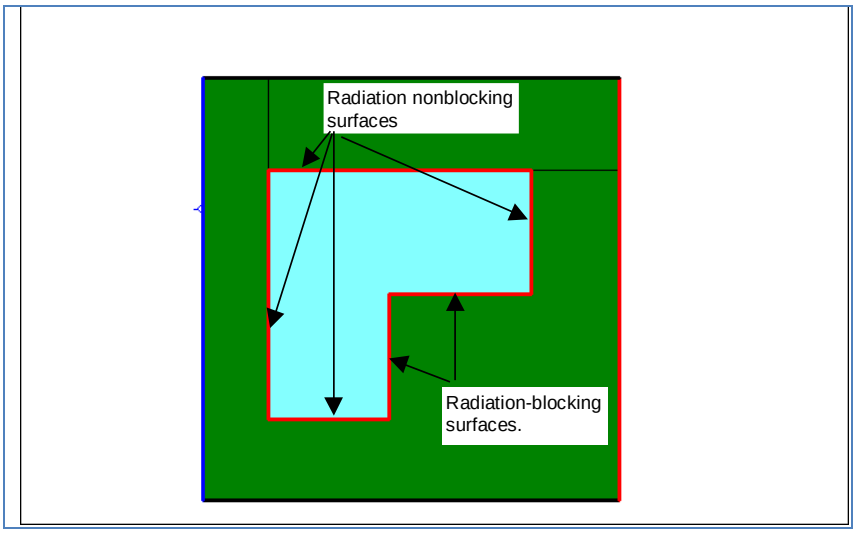


Figure 6-37. An example of radiation-blocking surfaces for a polygon defined with a Frame Cavity material

File Format and Location

The Boundary Condition Library is stored in XML files located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

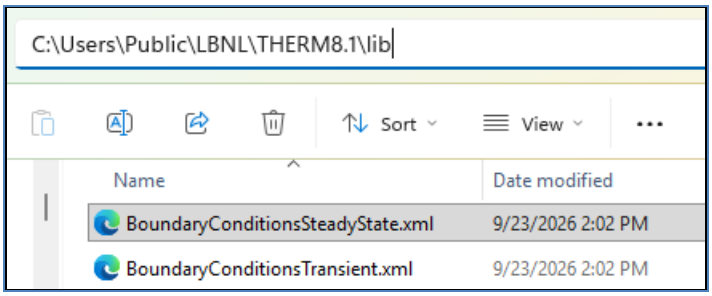


Figure 6-38. The Boundary Condition XML files are located in the "lib" subfolder of the THERM working directory.

The definitions of all the boundary conditions displayed in the THERM Boundary Condition Library are stored in this XML file.

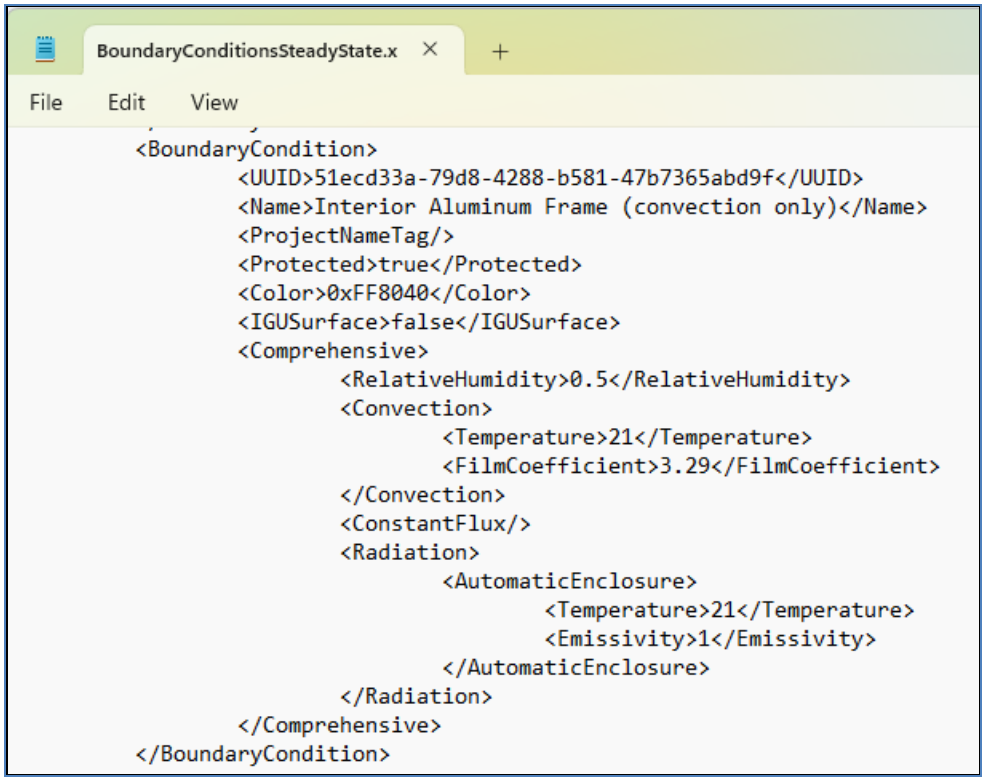


Figure 6-39. The BoundaryConditionsSteadyState.xml file contains all the values for records in the Boundary Conditions Library when the Steady State option is selected in File / Preferences.

6.4.2. Boundary Condition Library for Transient Model

6.4.2.1. Library List View

Selecting the Boundary Condition choice from the Libraries menu when the HygroTherm simulation model is selected displays the Boundary Condition records in "List View".

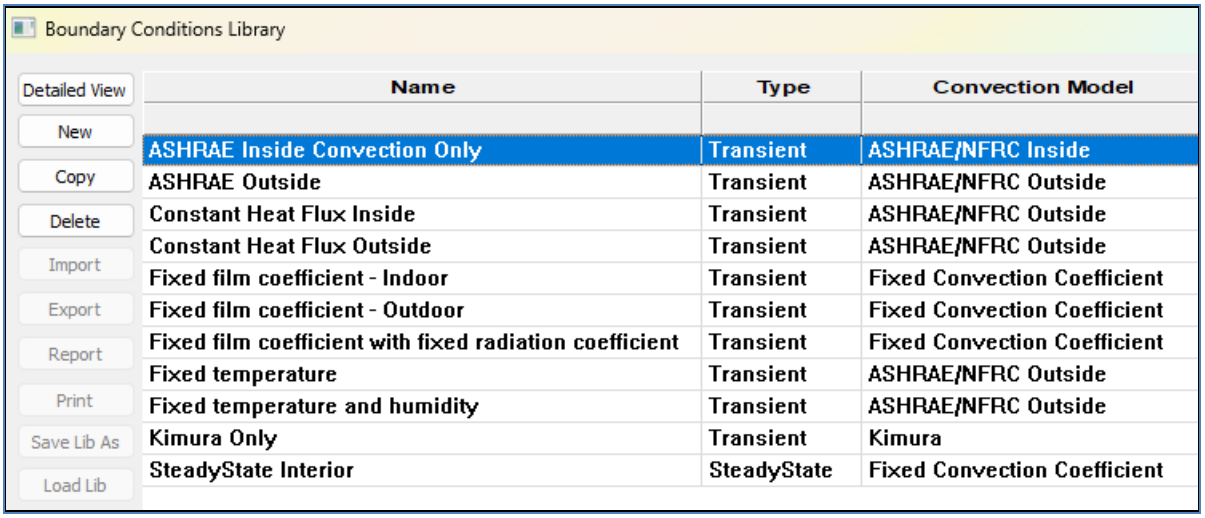


Figure 6-40. Transient Model Boundary Condition List View.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.
Import	*Currently disabled.*
Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*

Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the boundary condition.
Type	The boundary condition type, from the options of ▪ Transient ▪ Steady State (this is for the case where the transient moisture model is used with steady state algorithms, not currently implemented)
Convection Model	Options are: ▪ ASHRAE/NFRC Outside ▪ ASHRAE/NFRC Inside ▪ Fixed Convection Coefficient ▪ TARP ▪ Yazdanian Klems ▪ Kimura ▪ Montazeri
Temperature	For Type = Steady State, Convection Model Air Temperature. **Units:** F (IP); C (SI)
Film Coefficient	For Type = Steady State, Convection Model Film Coefficient. *Steady State is not currently implemented for HygroTherm.* **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Heat Flux	For Type = Steady State, Convection Model Heat Flux. *Steady State is not currently implemented for HygroTherm.* **Units:** Btu/h-ft2 (IP); W/m2 (SI)
Solar Radiation	For Type = Steady State, Convection Model Solar Radiation. *Steady State is not currently implemented for HygroTherm.* **Units:** Btu/h-ft2 (IP); W/m2 (SI)

Radiation Model	For the Neumann Boundary Condition Type, if the Radiation Checkbox is checked, the Radiation model will be one of the following: ▪ Automatic Enclosure Model ▪ Black Body Radiation ▪ Fixed Radiation Coefficient
Film Coefficient	For Type = Steady State, Radiation Model Film Coefficient. *Steady State is not currently implemented for HygroTherm.* **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Temperature	For Type = Steady State, Radiation Model Radiation Surface Temperature. *Steady State is not currently implemented for HygroTherm.* **Units:** F (IP); C (SI)
Humidity	For Type = Steady State, Radiation Model Humidity fraction. *Steady State is not currently implemented for HygroTherm.* **Legal Values: 0-1** **Units:** unitless

6.4.2.2. Boundary Condition Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each boundary condition.

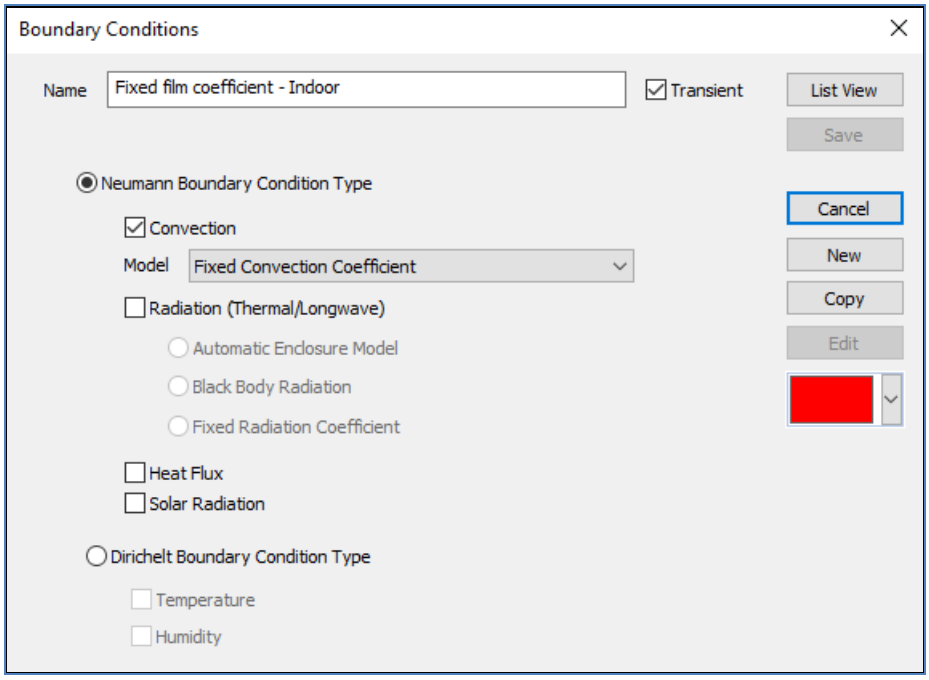
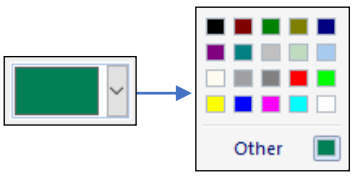


Figure 6-41. Boundary Condition Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Boundary Condition Library.
Save	Saves any changes made to the BoundaryConditions XML file. If changes are made to a field, move off the field in order for the Save button to become activated.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record and edit the name and parameters accordingly. Then click the Save button to save the changes.
Edit	*Currently not implemented.*

Color	The color assigned to the boundary condition. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.  *Figure 6-42. Color Settings in the Boundary Conditions Library Detail View.* One convention is to make the interior (warm) boundary conditions in the "red" colorway, and the exterior (cold) boundary conditions in the "blue" colorway.

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the boundary condition.
Transient	This checkbox is selected by default, and not editable in this version. The non-transient boundary conditions would be for the Steady State mode for the HygroTherm simulation engine, which has not been implemented yet.
Neumann Boundary Condition Type	
Convection	This option is checked if the boundary condition will include convection modeling.

Model	The Model pulldown list pertains to the type of convection model to be used, from the following choices:
	▪ ASHRAE/NFRC Outside
	▪ ASHRAE/NFRC Inside
	▪ Fixed Convection Coefficient
	▪ TARP
	▪ Yazdanian Klems
	▪ Kimura
	▪ Montazeri
	  ▪ For the Montazeri Convection Model, extra input values will be entered when assigning those to a specific boundary condition segment, which are
	  ▪ Building Width
	  ▪ Building Height

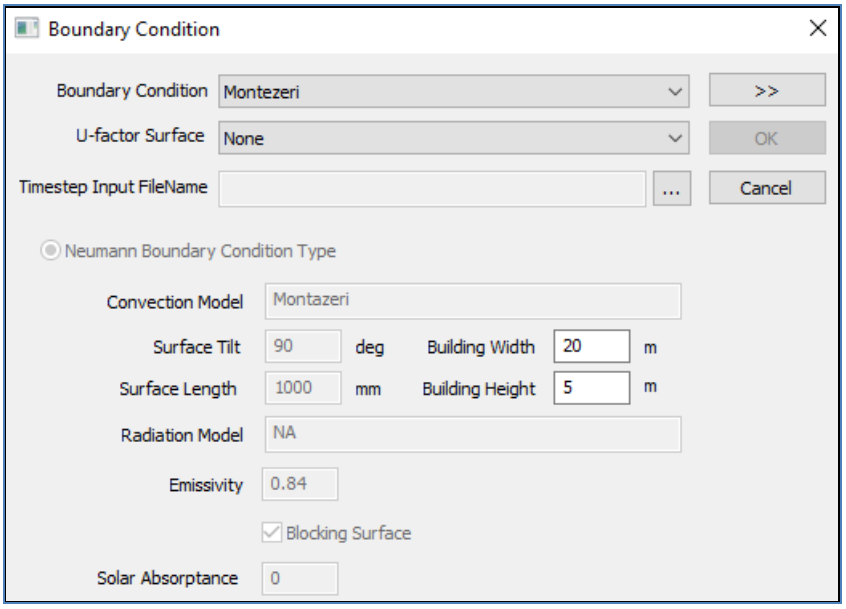


Figure 6-43. Building Width and Building Height are entered for specific boundary condition segments when they are assigned the Boundary Condition Type = Montezeri.

Radiation	This option is checked if the boundary condition will include Radiation modeling The Radiation model options are: ▪ Automatic Enclosure Model ▪ Black Body Radiation ▪ Fixed Radiation Coefficient
Heat Flux	This option is checked if the boundary condition will include Heat Flux modeling
Solar Radiation	This option is checked if the boundary condition will include Solar Radiation modeling
Dirichelt Boundary Condition Type	
Temperature	Check this option to include Temperature in this Boundary Condition Type
Humidity	Check this option to include Humidity in this Boundary Condition Type

File Format and Location

The Boundary Condition Library is stored in XML files located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

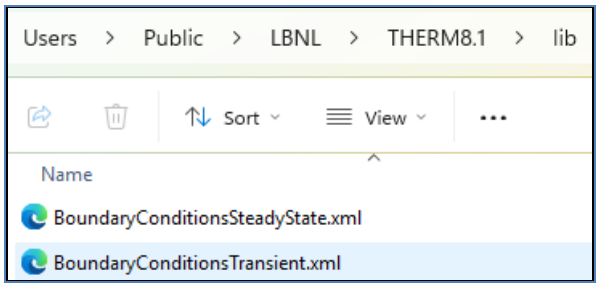


Figure 6-44. The BoundaryConditionTransient.xml file ise located in the "lib" subfolder of the THERM working directory.

The definitions of all the boundary conditions displayed in the THERM Boundary Condition Library are stored in this XML file.

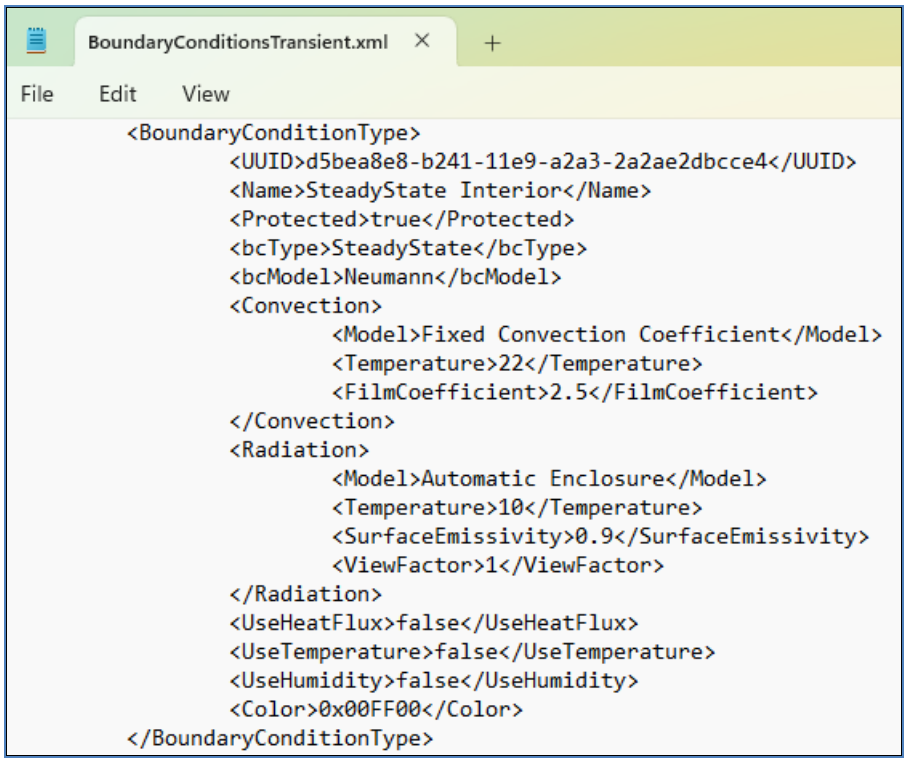


Figure 6-45. The BoundaryConditionsSteadyState.xml file contains all the values for records in the Boundary Conditions Library when the Steady State option is selected in File / Preferences.

6.4.2.3. Making New Boundary Condition Library Records

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the BoundaryCondition.xml file.

For example, if a new boundary condition record is created, all the settings for that boundary condition are saved in the BoundaryConditions.xml file.

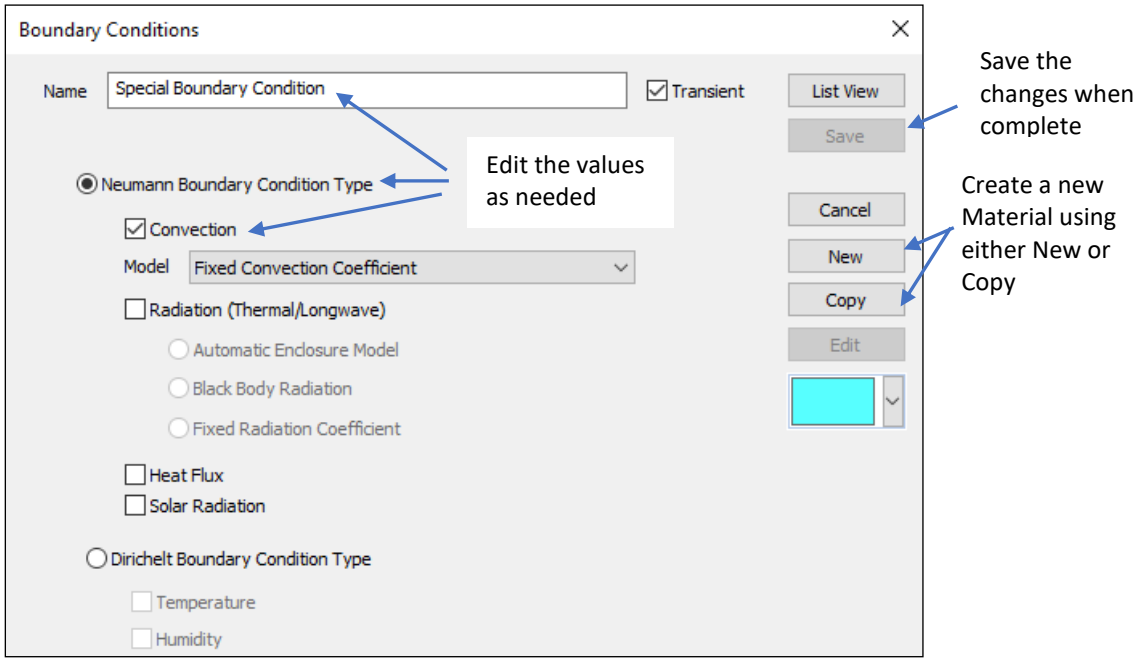


Figure 6-46. Make a new boundary condition using either New or Copy and change the input values as needed. The changes will be saved into the BoundaryConditions.xml file.

6.4.2.4. Time Step Boundary Condition Files

In addition to specifying a standard Boundary Condition for a boundary condition segment when using the transient simulation engine (HygroTherm), it is also necessary to specify a Time Step Boundary Condition, which are stored in separate XML files.

There are several examples of Time Step files that are installed with THERM 8.1, and they are found in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

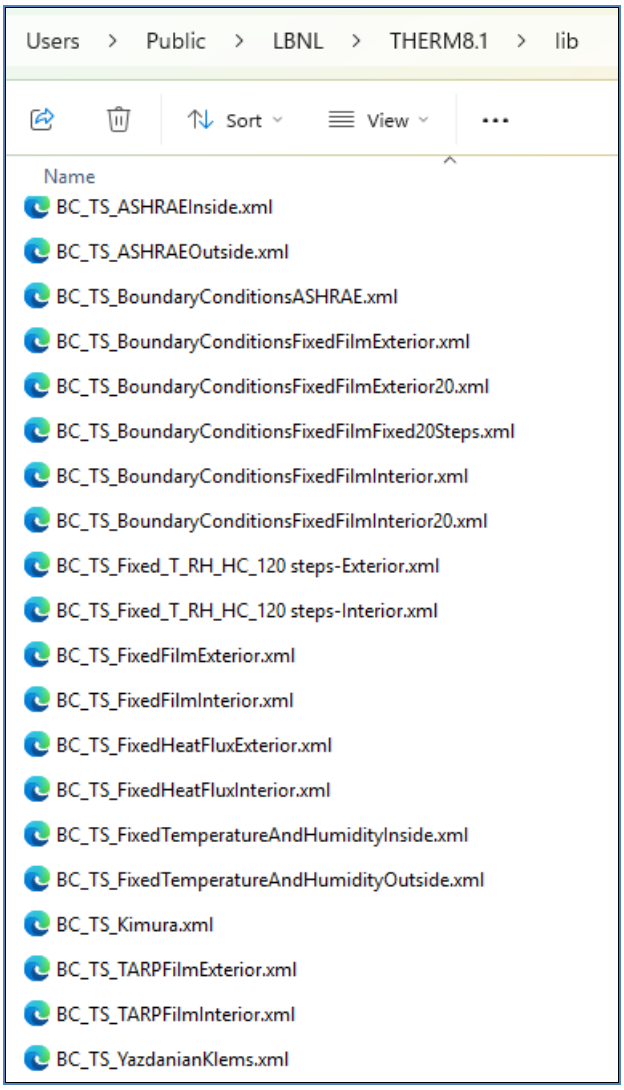


Figure 6-47. The Boundary Condition Time Step files are located in the "lib" subfolder of the THERM working directory.

These example files have a common naming convention of "BC_TS_" to indicate that they are **B**oundary **C**ondition **T**ime **S**tep files, but the files can have any name desired.

The Time Step files contain values for as many time steps as are being defined in each.

For example, the example file "BS_TS_FixedFilmInterior.xml" file has 5 timestep sections, with definitions for Temperature, Humidity and Film Coefficient for each time step.

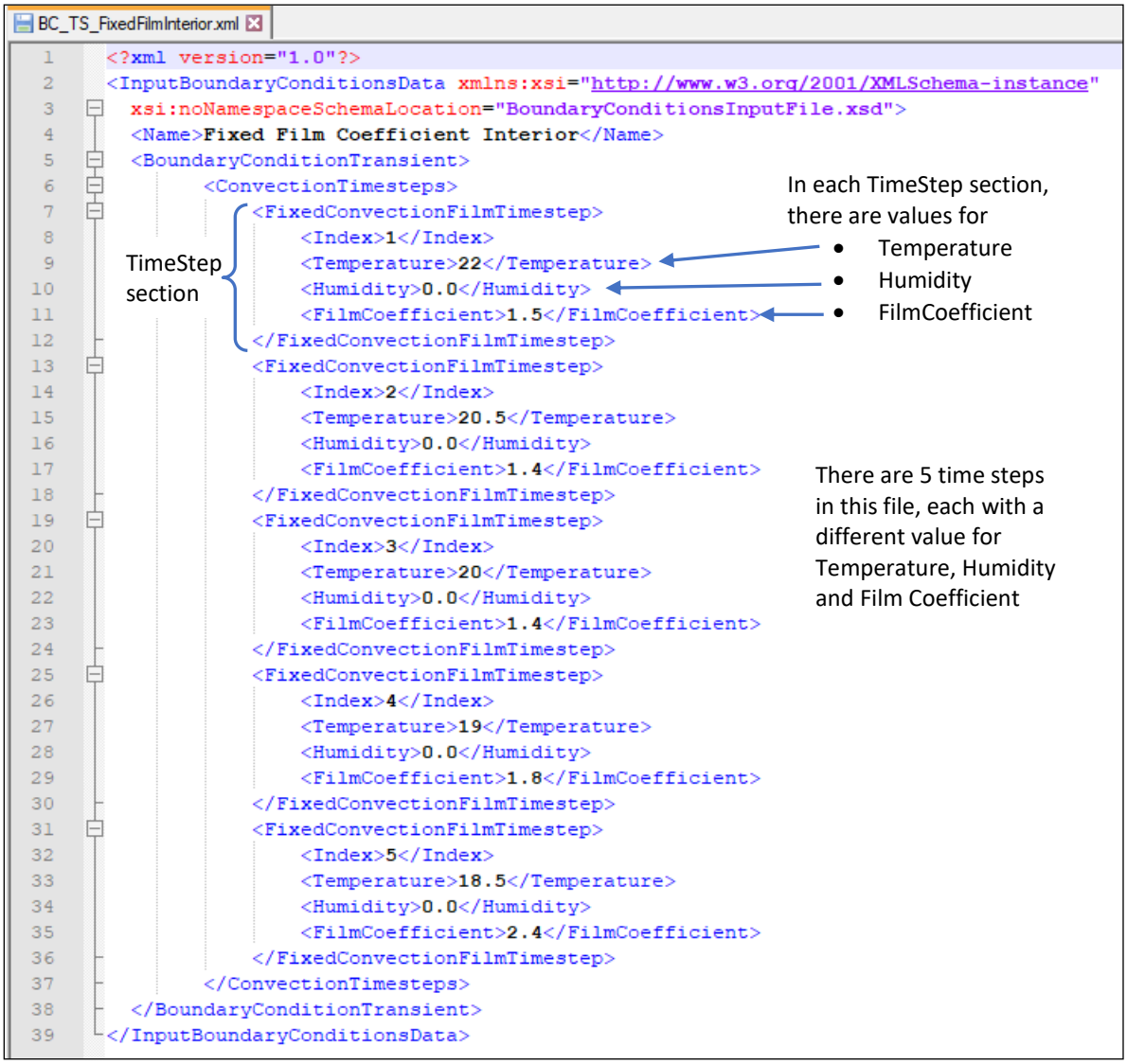


Figure 6-48. The Boundary Condition Time Step file format.

These XML files are referenced for specific boundary condition segments when the Simulation engine set to HygroTherm and Transient is checked. Under those conditions, if you double click on a specific boundary condition segment, the dialog box that is displayed will allow allow selecting a Timestamp Input File, which those XML files represent.

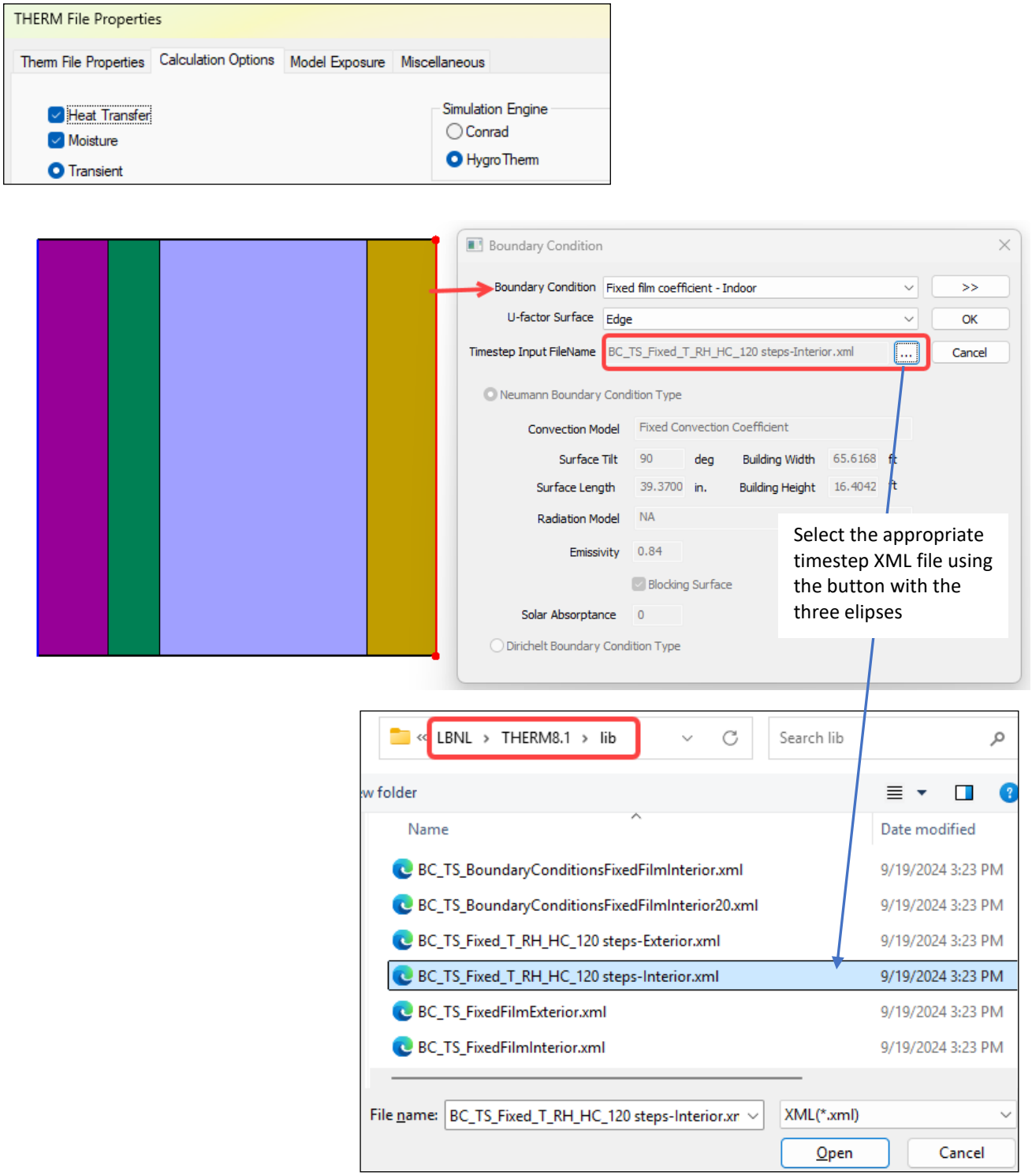


Figure 6-49. Select the appropriate Boundary Condition Timestep XML file for each boundary condition segmen

7. CALCULATING STEADY-STATE 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.

THERM includes two simulation engines, which are selected in **File/Properties/Calculation Options**:

- **CONRAD** calculates steady-state heat transfer.
- **HygroTherm** calculates heat transfer and, as an option, moisture transfer. It can run both steady-state and transient simulations.

Both engines evaluate conduction and radiation from first principles. Convective heat transfer is approximated through the use of film coefficients obtained from engineering references.

This chapter covers steady-state simulations using either engine. HygroTherm transient simulations are covered in *Section 9*.

	Conrad	HygroTherm
Steady-state heat transfer	Yes	Yes
Moisture transfer	No	Yes
Transient simulations	No	Yes
CMA calculations	Yes	No

7.1.1. Conrad Simulation Engine

Conrad, THERM's original steady-state conduction algorithm, is a derivative of the public-domain computer program TOPAZ2D. CONRAD uses the radiation view-factor algorithm VIEWER, which is a derivative of the public-domain computer program FACET.

7.1.2. HygroTherm Simulation Engine

HygroTherm was developed at LBNL to calculate heat and moisture transfer. It does not use CONRAD or VIEWER, and it calculates radiation view factors with its own routines. When a steady-state simulation includes moisture, all materials in the model must have moisture properties.

7.1.3. Mesher

THERM generates the mesh automatically, with either the Quad Tree mesher or the Simmetrix mesher. Both engines can use either mesh generator. The Quad Tree mesher uses the Finite Quadtree algorithm. With either engine, 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.

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, Stop Current Calculation** and **Glazing Options** menu choices will be inactive (grayed out) until you calculate current results.

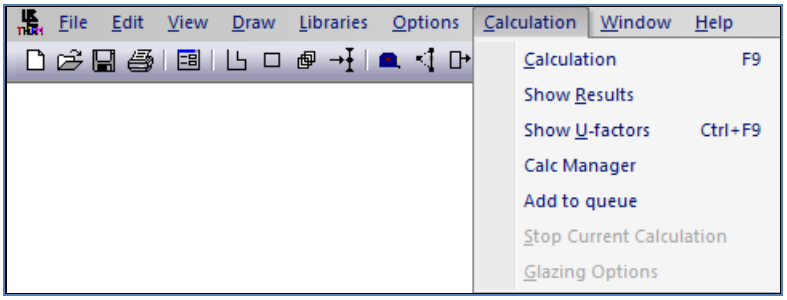


*Figure 7-1. The **Calculation** menu choices*

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.

Show Results Use this menu choice to display the results of the calculation. This is equivalent to clicking on the **Show Results** toolbar button.

Show U-factors (Ctrl-F9) Use this choice or the **Ctrl-F9** keyboard short cut to display the **U-Factors** results dialog box.

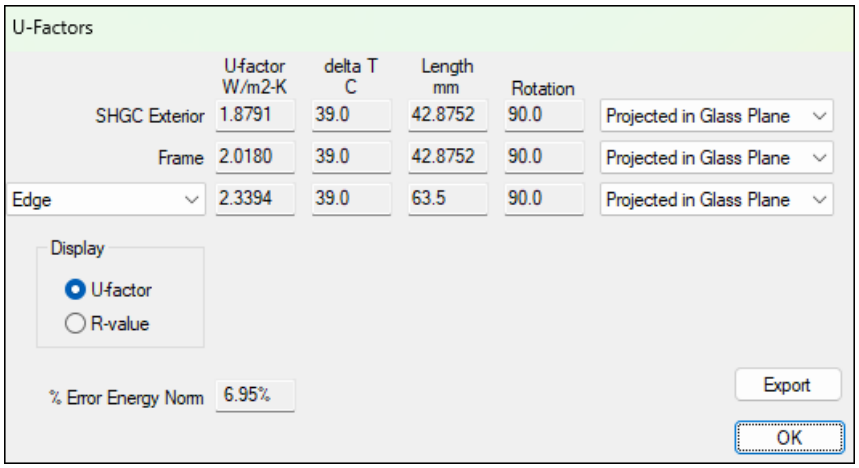


*Figure 7-2. The **U-factor** dialog box*

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. Select the **Calc Manager** menu choice to open the **Calculation Manager** dialog box, shown in the figure below.

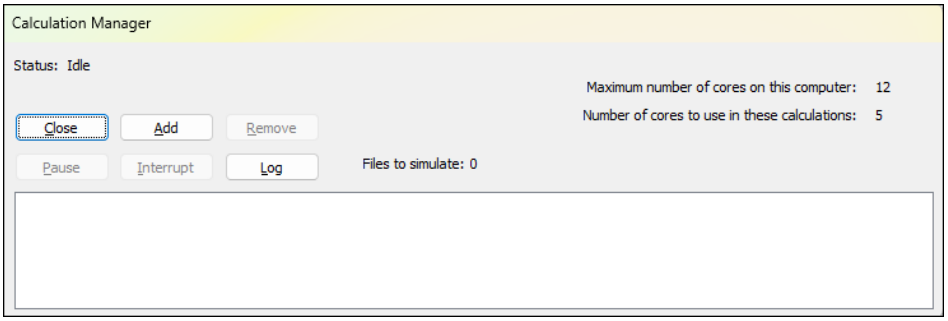


*Figure 7-3. Selecting the **Calculation/Calc Manager** menu choice opens the **Calculation Manager** dialog box.*

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).
- **Remove:** *Currently disabled.*
- **Start or Pause:** *Currently disabled.*
- **Interrupt:** *Currently disabled.*
- **Log:** The Log button displays a list of the filenames that have been run as well as the run time, run status, and any error messages generated during the calculation. This log can be cleared manually using the Clear Log button. The data in the log can be sorted by column by clicking once on the column name.

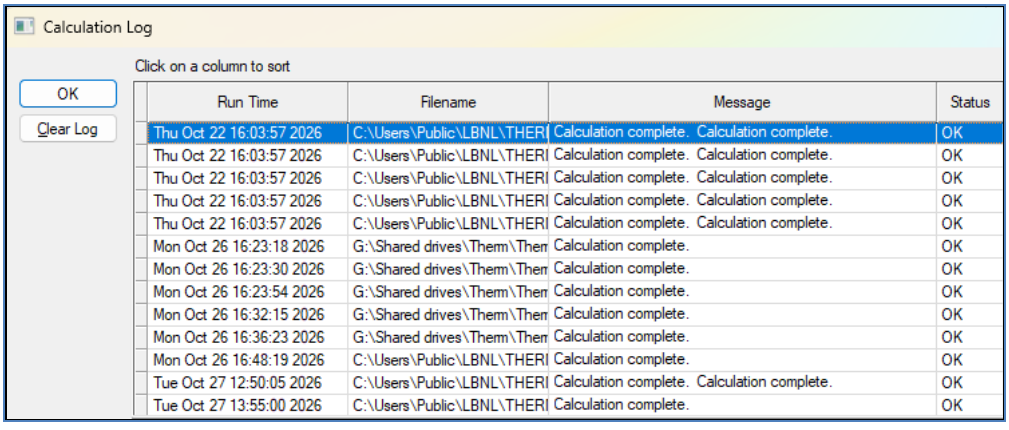


Figure 7-4. Calculation Log accessed from Calc/Calc Manager.

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.

Glazing Options

If a glazing system is selected in the THERM model, the **Glazing Options** menu choice displays the **Glazing System Options** dialog box, where the currently selected glazing options can be viewed, as well as added and removed.

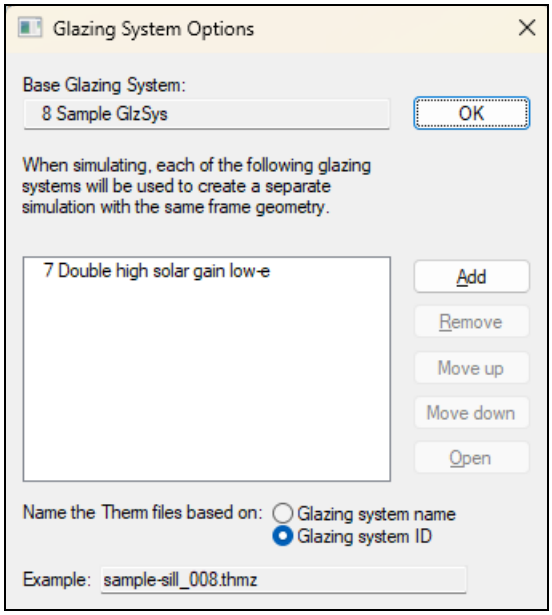


Figure 7-5. Use Calculation/Glazing Options menu to display the Glazing System Options dialog box.

7.2.2. Calculating Results for a THERM file

To start calculating a THERM file, do the following:

1. Set the appropriate **File/Properties/Calculation Options** settings (see *Section 7.2.3. Calculation Options* for a description of these values).
2. Press the **Calculation** toolbar button, use the **Calculation/Calculation** menu item, or press **F9** to start the calculation.



3. If a base case file has been created with multiple glazing, there will be extra options when the file is simulated using the **Calc** toolbar button, the **Calculation/Calculation** menu option, or the **F9** shortcut key. Using these methods, a **Glazing Option Simulation** dialog box will appear, allowing the following three options, as shown in the figure below:

   - **Create the glazing option files and perform all simulations:** This choice is the default, and it creates and runs all the THERM files for the multiple glazing options defined in the base case file.
   - **Simulate the current file (base case) only:** This choice is useful when setting up the base case file to make sure there are no geometry errors before running the entire multiple glazing options.
   - **Create the glazing option files but do not perform the simulations:** This choice will cause the program to make individual THERM files for each of the glazing options associated with the base case file, with either the Glazing System ID or Name appended on to the base case filename, but it will not simulate the files. These files can then be opened and simulated individually, or simulated through the Calc Manager.

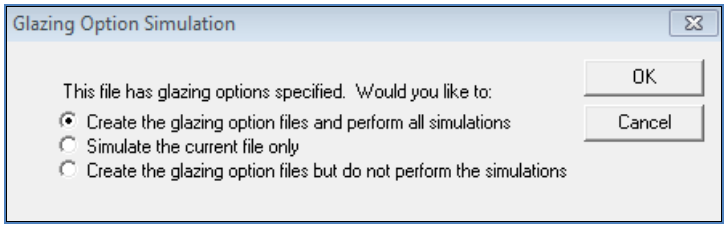


Figure 7-6. If multiple glazing options have been defined in the file, the Glazing Options Simulation dialog box will appear when the Calculation is started.

4. When the calculation begins, the program will show the calculation steps in the Status Bar at the bottom of the screen. Depending on the complexity of the cross section and the detail of the mesh, as well as the CPU speed and amount of memory of the computer, calculation steps can take several seconds to a few minutes.

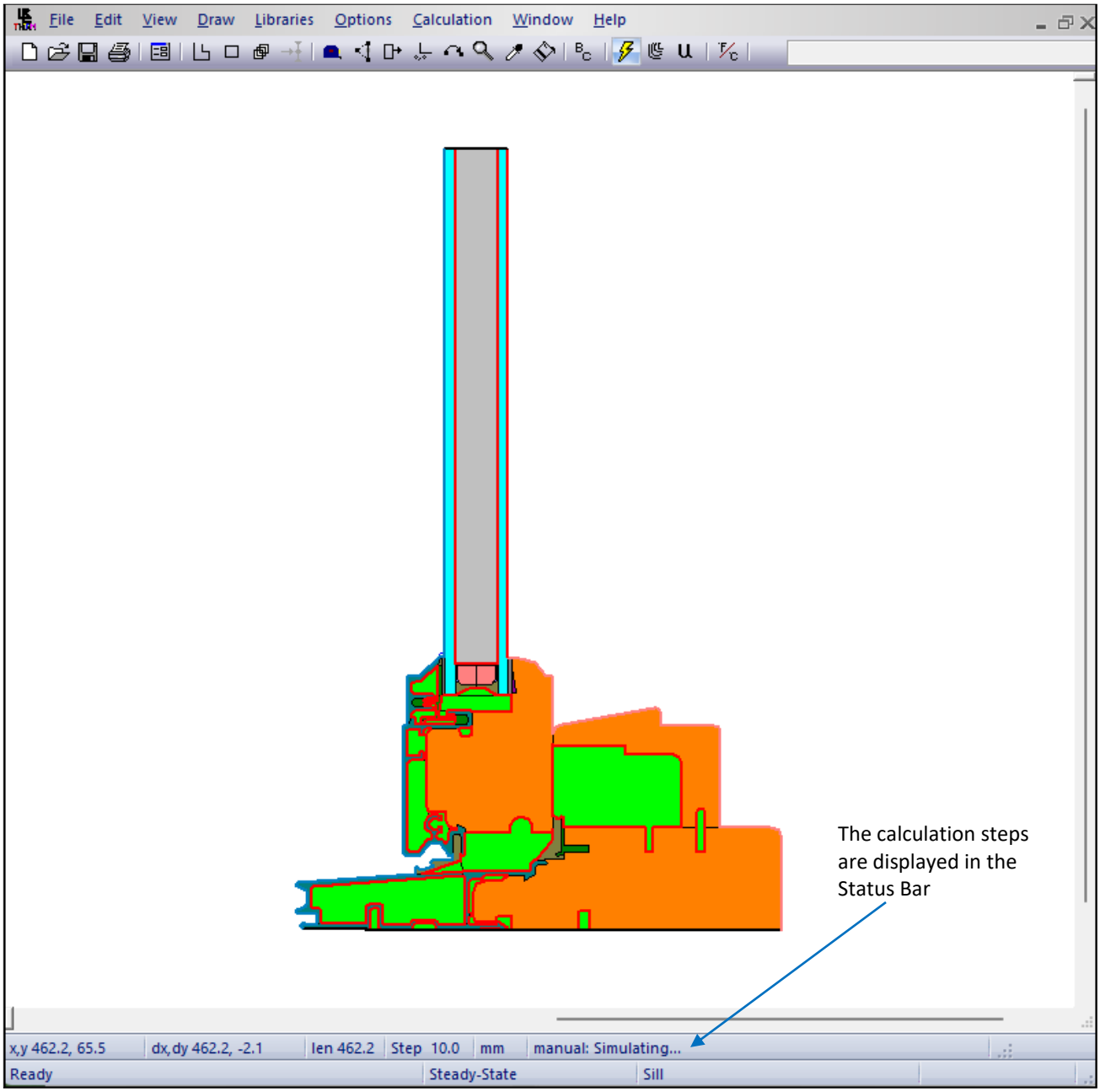


Figure 7-7. THERM displays the steps in the calculation in the Status Bar.

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.

These options are accessed from the **File** menu / **Properties** option / **Calculation Options** tab

The Simulation Engine options control what other options are available to be set.

Simulation Engine: Conrad

If the **Simulation Engine** is set to **Conrad**, the options shown in the figure and table below can be defined.

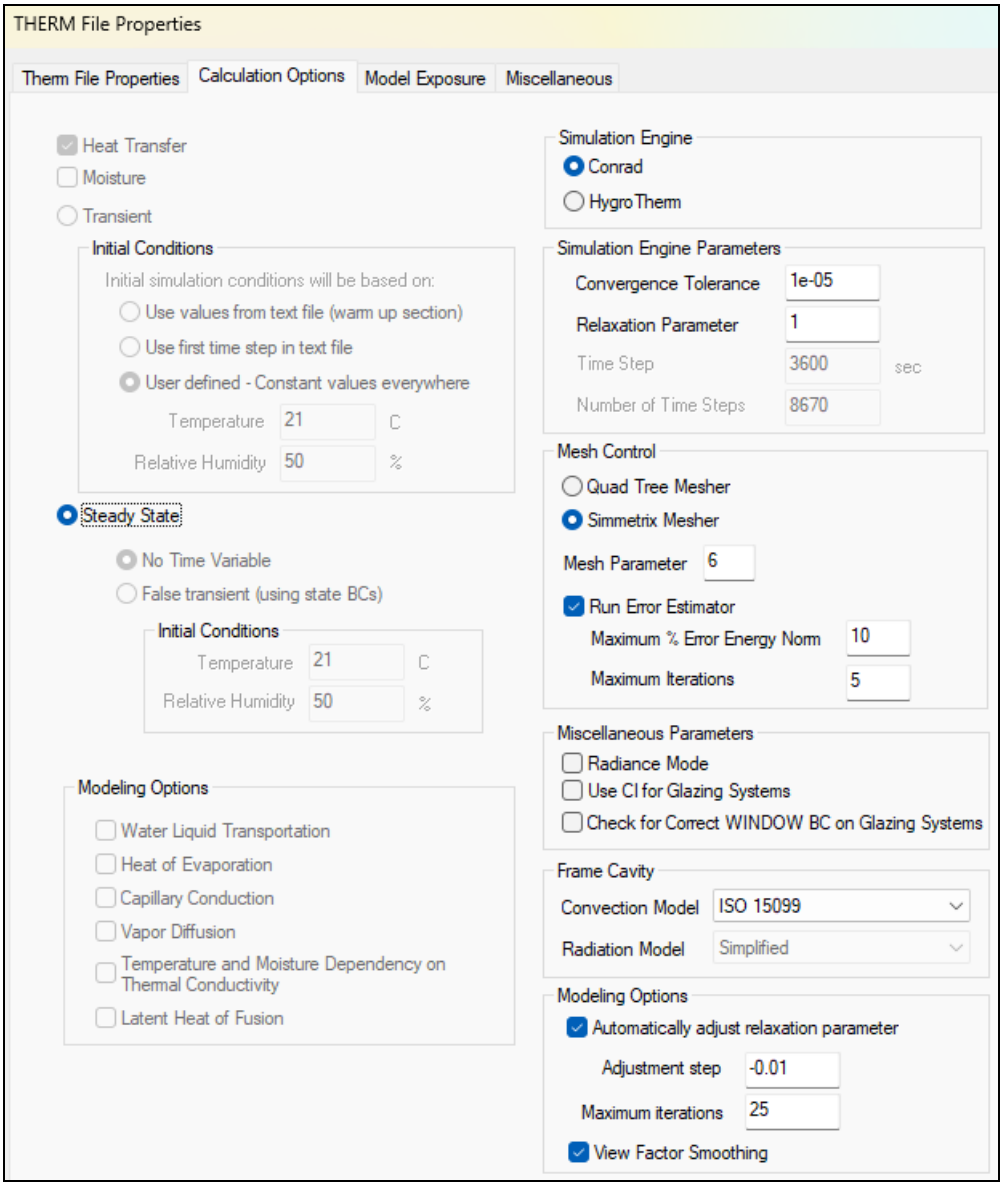


Figure 7-8. ***THERM File Options*** *affect how THERM calculates results.*

Simulation Engine

Conrad or HygroTherm

The **Conrad** option is checked.

When the Conrad simulation engine is selected, the "Steady State" radio button on the left hand part of the dialog box will automatically be selected, and all other options on that side of the screen are grayed out.

Simulation Engine Parameters

This section only applies to the HygroTherm simulation engine. For the Conrad simulation engine, these settings are found in Options / Preferences / Simulation tab.

Convergence Tolerance

This feature is used for the detailed radiation model, which is the default radiation model in THERM. 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:** 0.0000001

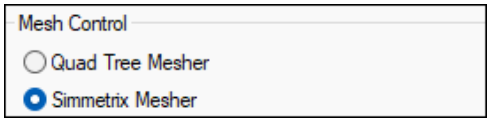
In general, the default values should be used.

Relaxation Parameter

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

In general, the default values should be used.

Mesh Control



Quad Tree Mesher

The old mesh engine which was not able to mesh complext models.

Simmetrix Mesher

The new mesh engine which can mesh much more complicated models – this is the default setting and the recommended option

Mesh Parameter

The mesh parameter determines the maximum element size in the mesh. The larger the mesh parameter, the smaller the maximum size of a mesh element. 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.)

There are different values for mesh parameter depending on the selected mesher.

Default:

- **Quad Tree mesher:** 6. If the mesh parameter is over 8 or 9, the program may display a message saying "Array Overflow" and suggest reducing the Mesh Parameter.
- **Simmetrix:** 8. This value can be set to at least 50 if needed.

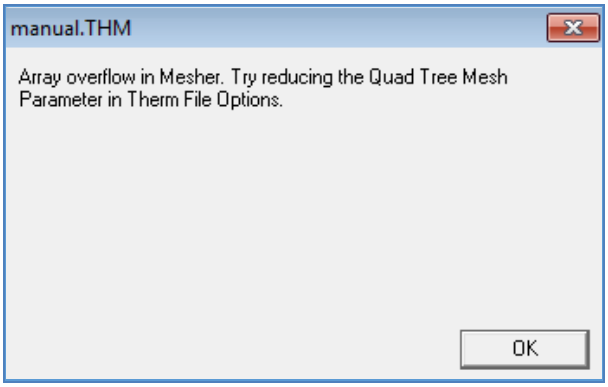


Figure 7-9. If the Quad Tree mesh parameter is too high, this message will appear, and you will need to reduce the mesh parameter in order to run the file.

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]. If this box is not checked, this percentage is 100% (see "Maximum % Error Energy Norm" below), which means THERM will not refine any areas of the mesh. The value of the percent error energy norm for your simulation will be displayed under **Calculation/Show U-factors**.

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**: 10%

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 if the target value for the percent error energy norm has not been reached. The last set of results will be displayed. **Default**: 5.

Miscellaneous Parameters

Radiance Mode

Check this option to run shading systems defined with THERM files, such as cellular shades, louverd and roller shutters, etc. See the *WINDOW User Manual* for instruction about modeling these types of shading systems.

Use CI Model for Glazing Systems

A check box indicates that you want to use the **Condensation Index Model** when WINDOW glazing systems are imported. **Default:** unchecked

Check for correct WINDOW BC on glazing systems

This feature checks to make sure that the boundary conditions on a glazing system imported from WINDOW have not changed. If the boundary conditions for the glazing system are not the ones imported with the system, the program displays the message below.

If for whatever reason the glazing system boundary conditions have been deliberately changed, this option should be unchecked so that the message doesn't appear.

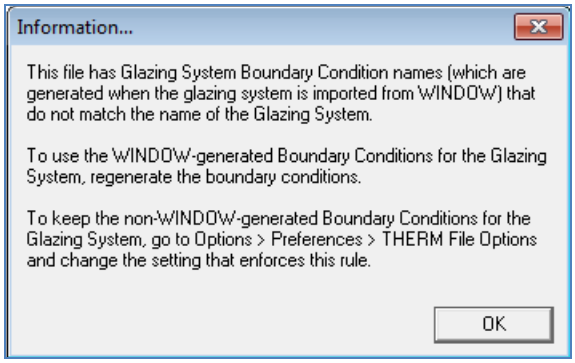


Figure 7-10. The error message displayed if "Check for correct WINDOW boundary conditions on glazing systems" is checked and the glazing system boundary conditions have been changed in THERM.

Frame Cavity

Convection Model Choices are

- ISO 15099
- EN 1007

Radiation Model Choices are

- Simplified: This is the default setting, and cannot be edited at this time

Modeling Options

Automatically adjust relaxation parameter

Currently not used by either simulation engine.

Conrad uses the setting in Options/Peferences/Simulation tab.

Adjustment step

Currently not used by either simulation engine.

Conrad uses the setting in Options/Peferences/Simulation tab.

Maximum iterations

Only used by the HygroTherm simulation engine. The Conrad simulation engine uses the value in Options/Preferences/Simulation tab.

Limits the solver during each one of the Mesh Control Maximum Iterations calculations. If the solution does not converge, this value can be increased. If the solver can't settle on a solution within this limit, THERM reports a "failed to converge" error. **Default:** 25

View Factor Smoothing

Currently not used by either simulation engine.

Conrad uses the setting in Options/Peferences/Simulation tab.

Simulation Engine: HygroTherm – Steady State / Heat Transfer model

The HygroTherm simulation engine can also calculate steady-state heat transfer. The section below explains how to set the parameters for this type of calculation

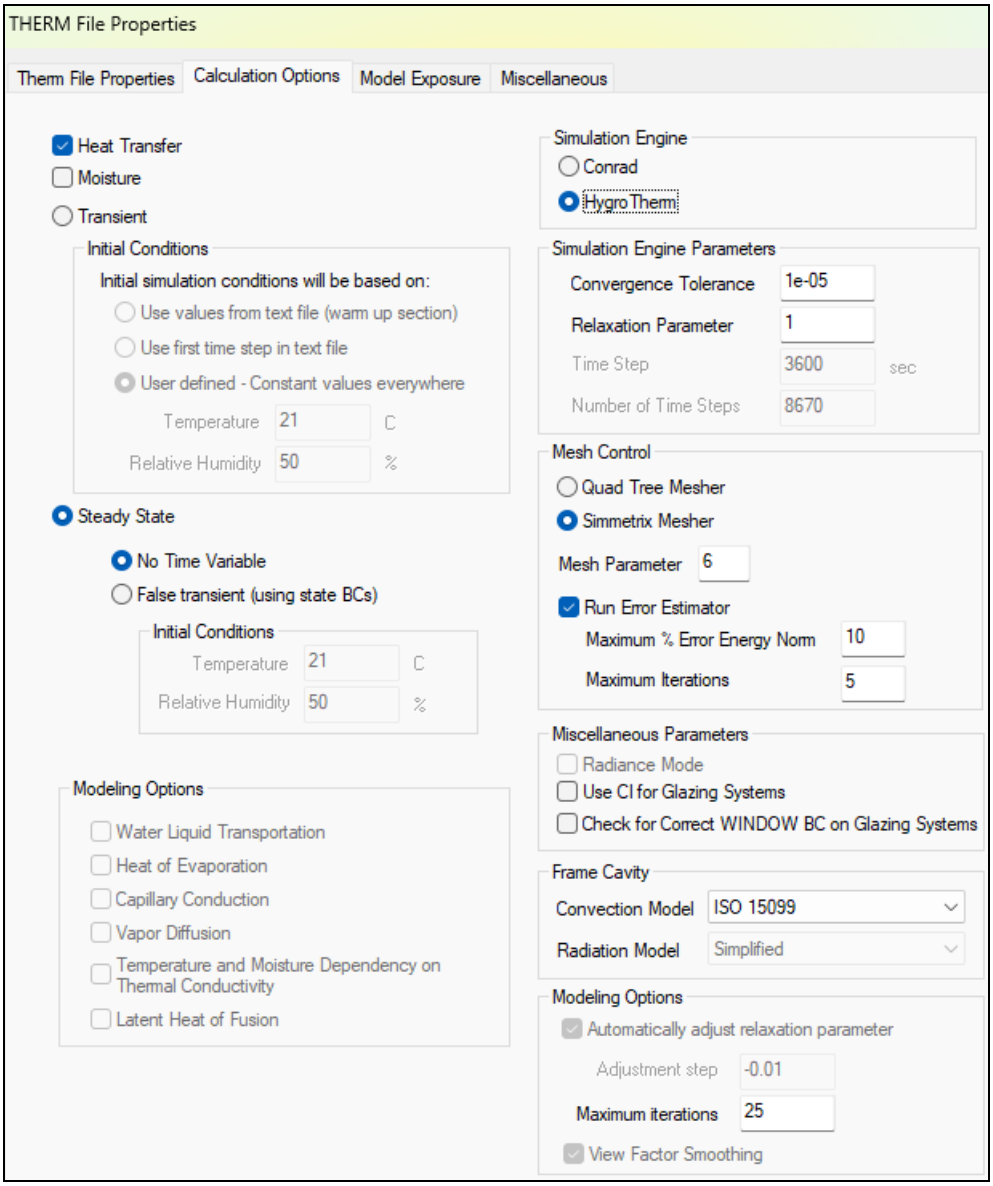


Figure 7-11. THERM File Calculation Options for steady-state thermal modeling using the HygroTherm simulation engine.

Simulation Engine

Conrad or HygroTherm

The **HygroTherm** option is checked.

When the HygroTherm simulation engine is selected, the "Heat Transfer" checkbox is automatically selected, as well as the "Steady State" radio button with the "No Time Variable" radio button also automatically selected.

These are the options that allow comparable steady-state calculations to the Conrad simulation engine.

All the other options are the same as if the Conrad simulation engine is selected.

Simulation Engine Parameters

This section only applies to the HygroTherm simulation engine. For the Conrad simulation engine, these settings are found in Options / Preferences / Simulation tab.

Convergence Tolerance

This feature is used for the detailed radiation model, which is the default radiation model in THERM. 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:** 0.0000001

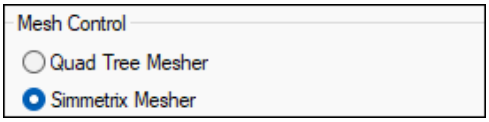
In general, the default values should be used.

Relaxation Parameter

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

In general, the default values should be used.

Mesh Control



Quad Tree Mesher

The old mesh engine which was not able to mesh complext models.

Simmetrix Mesher

The new mesh engine which can mesh much more complicated models – this is the default setting and the recommended option

Mesh Parameter

The mesh parameter determines the maximum element size in the mesh. The larger the mesh parameter, the smaller the maximum size of a mesh element. 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.)

There are different values for mesh parameter depending on the selected mesher.

Default:

- **Quad Tree mesher:** 6. If the mesh parameter is over 8 or 9, the program may display a message saying "Array Overflow" and suggest reducing the Mesh Parameter.
- **Simmetrix:** 8. This value can be set to at least 50 if needed.

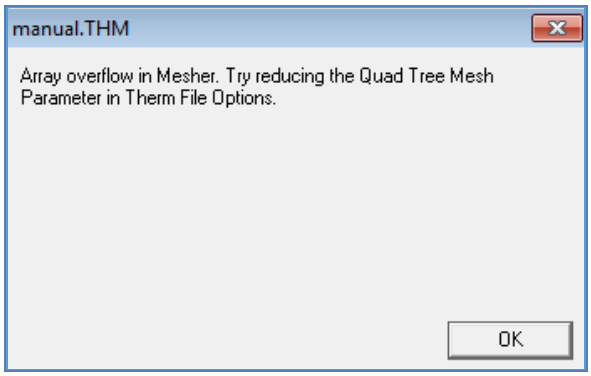


Figure 7-12. If the Quad Tree mesh parameter is too high, this message will appear, and you will need to reduce the mesh parameter in order to run the file.

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]. If this box is not checked, this percentage is 100% (see "Maximum % Error Energy Norm" below), which means THERM will not refine any areas of the mesh. The value of the percent error energy norm for your simulation will be displayed under **Calculation/Show U-factors**.

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**: 10%

Maximum Iterations

Only used by the HygroTherm simulation engine. The Conrad simulation engine uses the value in Options/Preferences/Simulation tab.

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 if the target value for the percent error energy norm has not been reached. The last set of results will be displayed. **Default**: 5.

Miscellaneous Parameters

Radiance Mode

Check this option to run shading systems defined with THERM files, such as cellular shades, louverd and roller shutters, etc. See the *WINDOW User Manual* for instruction about modeling these types of shading systems.

Use CI Model for Glazing Systems

A check box indicates that you want to use the **Condensation Index Model** when WINDOW glazing systems are imported. **Default:** unchecked

Check for correct WINDOW BC on glazing systems

This feature checks to make sure that the boundary conditions on a glazing system imported from WINDOW have not changed. If the boundary conditions for the glazing system are not the ones imported with the system, the program displays the message below.

If for whatever reason the glazing system boundary conditions have been deliberately changed, this option should be unchecked so that the message doesn't appear.

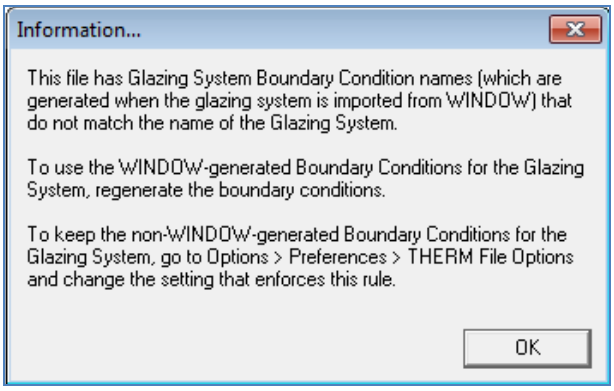


Figure 7-13. The error message displayed if "Check for correct WINDOW boundary conditions on glazing systems" is checked and the glazing system boundary conditions have been changed in THERM.

Frame Cavity

Convection Model Choices are

- ISO 15099
- EN 1007

Radiation Model Choices are

- Simplified: This is the default setting, and cannot be edited at this time

Modeling Options

Automatically adjust relaxation parameter *Not used in the HygroTherm simulation engine.*

Adjustment step *Not used in the HygroTherm simulation engine.*

Maximum iterations Limits the solver during each one of the Mesh Control Maximum Iterations calculations. If the solution does not converge, this value can be increased. If the solver can't settle on a solution within this limit, THERM reports a "failed to converge" error. **Default:** 25

View Factor Smoothing *Not used in the HygroTherm simulation engine, which always smooths view factors.*

Options / Preferences

The **Options** menu / **Preferences** choice displays several tabs, including a **Preferences tab** and a **Simulation tab**, which are described in detail below.

Preferences tab

The options on this tab apply to both Conrad and HygroTherm simulation engines.

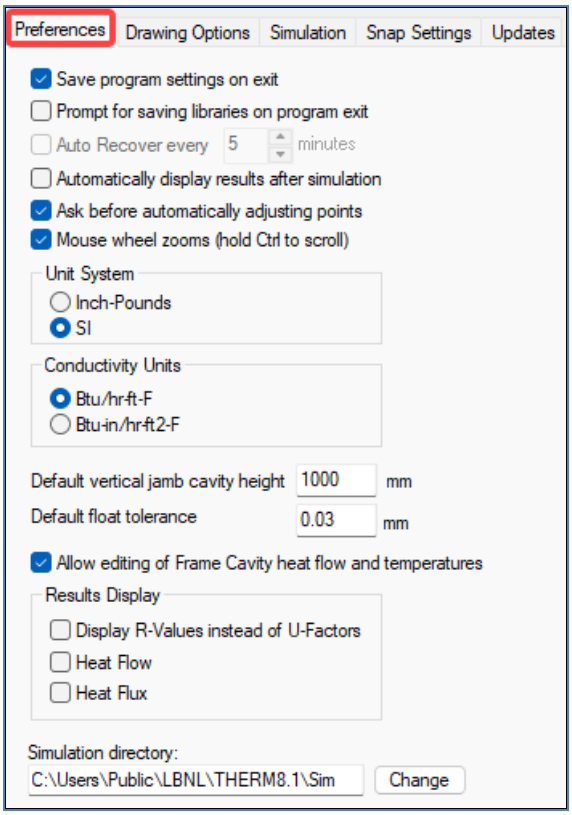


Figure 7-14. Options / Preferences / Simulation tab. The "ConRad Simulation" group of options are only for the Conrad simulation engine. The rest of the options apply to both the Conrad and HygroTherm simulations.

Save Program Settings on exit

This option will save any program level settings (not File level settings) that were set during the session.

Prompt for saving libraries on program exit

This option will generate a prompt to the user to save any changes that were made to libraries, if they weren't already saved, when exiting the program.

Auto Recover every "x" minutes

Not currently implemented.

Automatically display results after simulation

This option will cause the simulation results to be displayed automatically at the end of the simulation.

Ask before automatically adjusting points

When the mesher determines that points are too close together, if this option is selected, the program will ask the user whether the points should be automatically adjusted by the program, or whether the user wants to adjust them manually.

The recommendation is to not have the program automatically adjust the points until the user has inspected them. The simulation can be started again and the option to automatically adjust them can be selected.

Default: checked

Mouse wheel zooms (hold Ctrl to scroll)

This option allows the mouse to zoom.

Unit System

Options are:

- Inch-Pounds (IP)
- SI

Conductivity Units

Options are:

- Btu/hr-ft-F
- Btu/he-ft2-F

Default vertical jamb cavity height

The vertical jamb cavity height used when simulating a jamb profile.

Default: 1000 mm (SI); 39.27 inches (IP)

Default float tolerance

Float Tolerance is the geometric snap distance THERM uses when comparing or merging points and edges in a cross-section. Two points or two edge endpoints separated by a distance less than or equal to Float Tolerance are treated as coincident; an edge whose length is below Float Tolerance is treated as zero-length and discarded.

The value is a length, expressed in the active unit system (mm in SI, inches in IP). The default is `0.03 mm` and the allowed range in the preferences UI is `0` to `1` (display units).

The setting in this tab is the "program level" value, which is used for all newly created files. However, this value does not change the tolerances set in files that already exist. The "file level" value, that is saved with the individual file, is set in **File/Properties/Miscellaneous tab** in the **Float Tolerance** input.

Default: 0.03 mm (SI), 0.001181 inche (IP)

Effects of changing the default value:

- **Lowering Float Tolerance (e.g., 0.001 mm)** preserves fine geometric detail. Vertices that you intentionally placed close together stay distinct. The cost is that geometry imported from CAD or pasted from another file is more likely to leave near-but-not-exactly-coincident vertices, which can produce slivers, unshared edges between adjoining polygons, and meshing problems.
- **Raising Float Tolerance (e.g., 0.1 mm or higher)** is more aggressive about merging. Sloppy geometry is automatically cleaned up so that adjoining polygons share edges, but real detail finer than the tolerance is collapsed away. If the tolerance becomes a noticeable fraction of the smallest feature in the cross-section, the error estimator may report a failure (and ask the user to reduce the value) and the simulation results should not be trusted.
- **Default (0.03 mm)** is a balance chosen so that typical drafting precision in millimeters cleans up correctly while sub-millimeter features (slat thicknesses, films, gaskets) are preserved. The recommendation is to not change the value.

A separate field, **Checking Tolerance (found in File/Properties/Miscellaneous tab**, must be greater than Float Tolerance; THERM displays a warning if Float Tolerance ≥ Checking Tolerance.

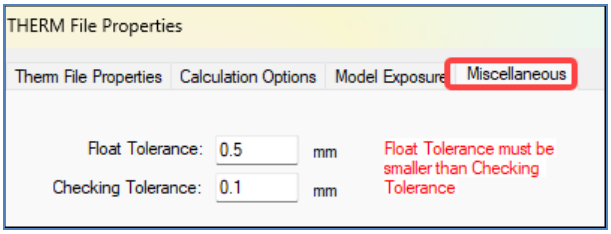


Figure 7-15. The Float Tolerance must be < the Checking Tolerance, and THERM will produce an error message if this is not the case.

Allow editing of Frame Cavity heat flow and temperatures

This option allows the user to edit the frame cavity heat flow and temperatures, when an individual frame cavity is selected.

Default: unchecked

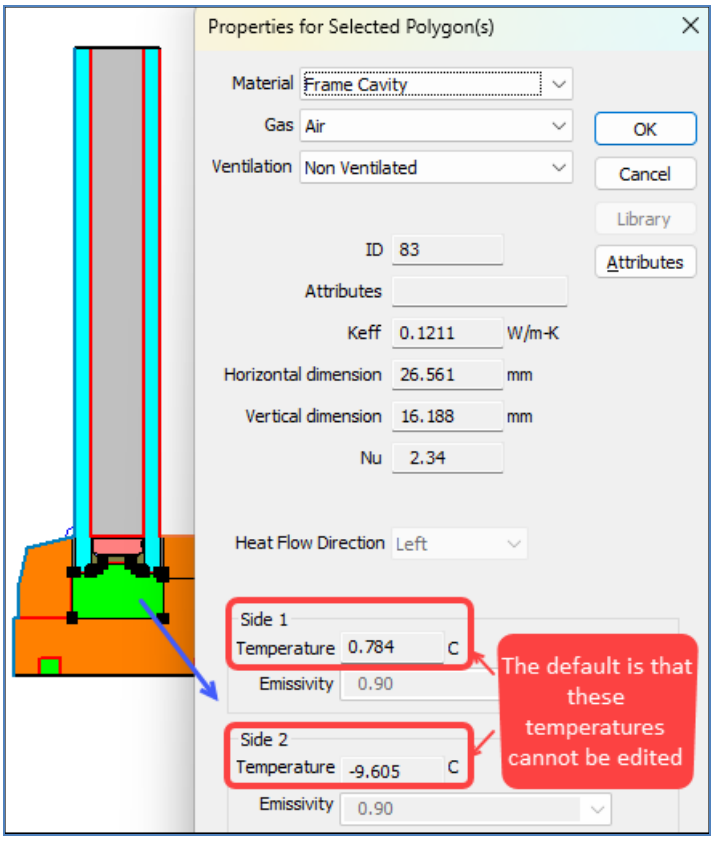


Figure 7-16. The default setting is to NOT allow editing of the frame cavity temperatures. If the option is checked, then those temperatures can be edited.

Results Display

Display R-values instead of U-factors

This option sets the results in the U-factor dialog box to display R-values instead of U-factors.

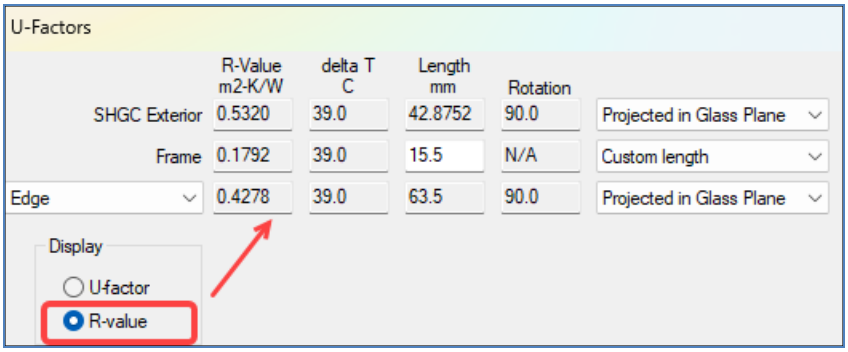


Figure 7-17. This option sets the results in the U-factor dialog box to display R-values

Heat Flow

If this box is checked, a column is added to the U-factor results dialog box with Heat Flow calculated values.

Default: unchecked. Units: W (SI); Btu/h (IP)

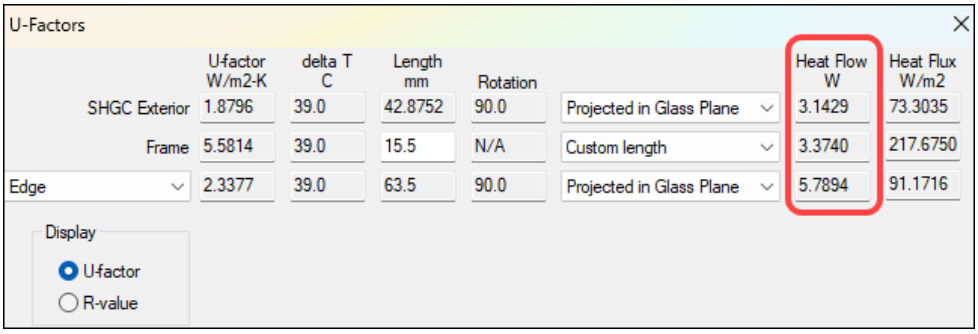


Figure 7-18. Heat Flow results in the U-factor dialog box

Heat Flux

If this box is checked, a column is added to the U-factor results dialog box with Heat Flux calculated values.

Default: unchecked. Units: W/m2 (SI); Btu/h-ft2 (IP)

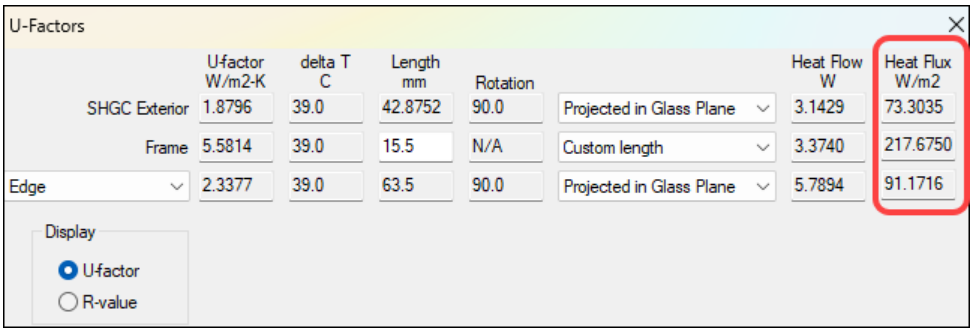


Figure 7-19. Heat Flux results in the U-factor dialog box

Simulation directory

The path for the simulation folder, which is THERM's working folder for calculations. THERM writes temporary files there while it performs the calculations. The results thermselves are not kept there, but are stored in the THMZ file itself.

Default: C:\Users\Public\LBNL\THERM8.1\Sim

THERM creates the folder if it does not exist. If THERM cannot write to the folder set in Preferences (for example, a folder under Program Files or on a drive that is no longer available), it switches back to the default location at startup.

What THERM writes to the folder

- **Intermediate simulation files:** files named after the model (for example .con, .msh and .sol). They are deleted after each calculation unless "Save simulation intermediate files" is checked on the **Options/Preferences/Simulation tab**.

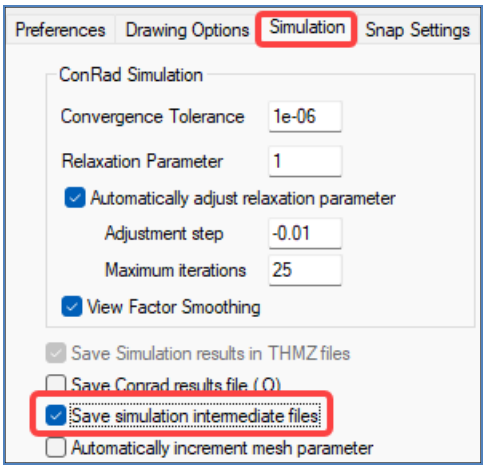


Figure 7-20. The intermediate simluation files will be saved if the "Save simulation intermediate files" option is checked in the Options/Preferences/Simulation tab,

- **Calculation Manager:** while the Calculation Manager runs, one subfolder per thread (T1, T2, and so on). Each holds temporary copies of the settings and libraries used by that thread.
- **Logs:** the simulation log (ThermLog.txt) and the warnings log (ThermWarningsLog.txt).

The simulation folder is set only in Preferences and does not depend on where the THMZ file is stored. A model in any folder is calculated in the same simulation folder.

There is one exception. With the Conrad solver, the Conrad results file (.O) is written to the folder that contains the THMZ file, not to the simulation folder. THERM deletes it after reading the results, unless "Save Conrad results file (.O)" is checked. The folder that contains the model must therefore allow THERM to create files.

THERM works with files through normal Windows paths. A model in a cloud-synced folder that appears as a local folder (for example OneDrive) can be opened and calculated like any other file. We recommend keeping the simulation folder itself on a local drive, because THERM creates and deletes many temporary files there during each calculation.

Simulation tab

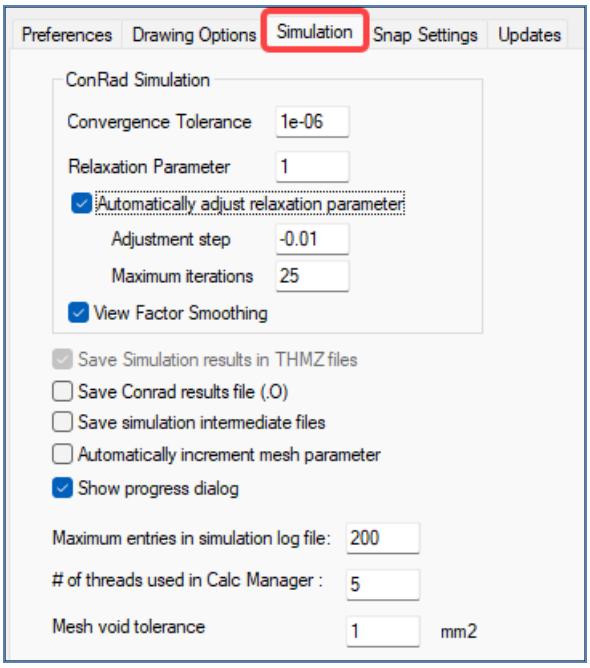


Figure 7-21. Options / Preferences / Simulation tab. The "ConRad Simulation" group of options are only for the Conrad simulation engine. The rest of the options apply to both the Conrad and HygroTherm simulations.

ConRad Simulation

This only applies when the Simulation Engine = Conrad in File/Properties/Calculation Options tab

Convergence Tolerance

This feature is used for the detailed radiation model, which is the default radiation model in THERM. 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:** 0.0000001

In general, the default values should be used.

Relaxation Parameter

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

In general, the default values should be used.

Automatically adjust relaxation parameter

The default setting is that the program will automatically adjust these parameters, which is recommended.

Adjustment step

Adjustment step: If the model does not converge, based on the Relaxation Parameter above, the program will automatically increment the relaxation parameter in steps based on this Adjustment Step input. In the case of the default value of -0.01, the program would try 0.99, then 0.98, etc, until it reached convergence, as long as the Maximum iterations is not exceeded.

Default: -0.01

Maximum iterations

Limits the solver during each one of the Mesh Control Maximum Iterations calculations. If the solution does not converge, this value can be increased. If the solver can't settle on a solution within this limit, THERM reports a "failed to converge" error. **Default:** 25

View Factor Smoothing

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

Miscellaneous Options

These settings apply both to the Conrad and HygroTherm simulation engines, unless otherwise noted.

Save Simulation Results in THMZ Files

This setting saves the results of the THERM simulation in the THERM file, including the mesh, temperature, and U-factor results.

This box is checked and cannot be unchecked.

Save CONRAD Results file (.O)

This option saves the CONRAD results 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

Automatically Increment Mesh Parameter

This option will automatically increase the mesh parameter as the program is attempting to mesh the model, rather than the user having to change it incrementally if the model cannot be meshed at the given mesh parameter setting.

Show Progress dialog

Currently not implemented.

Maximum entries in simulation log file

This setting applies to the Calc Manger log file. The "entries" refer to each line in the Calc Manager log file. When the number of entries exceeds the value set here, each new calculation overwrites the oldest entry, so that the most current calculations are kept. If this value is reduced, the most recent log entries are kept.

This log is saved in the "ThermLog.txt" file in the THERM simulation folder when THERM closes.

Default: 200

# of threads used in Calc Manager

This value controls the number of parallel simulations that THERM can perform. This is useful when simulating many THERM models using the Glazing Options feature. This value should be based on then number of cores on the computer, as each core supports 2 threads.

Default: 5. **Maximum:** 7

Mesh void tolerance

This setting only applies to the Conrad simulation engine.

After a Conrad calculation, THERM adds up the areas of all mesh elements and compares the sum with the total area of the material polygons in the model. Radiation enclosures are not included. In a CR (condensation resistance) model, frame cavities are also not included. If the mesh area is smaller than the polygon area by more than the mesh void tolerance, part of the model was not meshed. THERM then shows the warning "The mesh appears to have gaps" and does not show results for that calculation.

The tolerance is an area: mm^2 in SI units and in^2 in IP units. The default is 0.5 mm^2. A small additional allowance of 0.005% of the mesh area is always added to the tolerance to cover round-off.

Default: 1 mm^2 (SI); 0.00155 in^2 (IP)

7.3. Feedback and Troubleshooting During the Calculation Process

During the calculation process, the feedback bar at the bottom of the THERM application window will identify which stage of the automated calculation procedure THERM is working on.

Conrad Simulation Engine

The stages of the calculation procedure for the Conrad Simulation engine are:

- Premeshing
- Reading Geometry
- Meshing
- Generating Input
- View Factors (optional)
- Simulating
- Error Estimator (optional)
- Saving Results

HygroTherm Simulation Engine for steady-state calculations

The stages of the calculation procedure for the HygroTherm simulation engine for steady-state calculations are fewer because the engine handles the grometry, the calculation input and the view factors itself.

- Meshing
- Simulating
- Error Estimator (when Run Error Estimator is checked)
- Refining mesh
- Saving Results

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, and the information attached to it (materials, boundary conditions, U-factor tags, and mesh settings) to make sure that there are no problems in the geometry that the mesher cannot handle. This happens before every simulation, for both mesh generators (Quad Tree and Simmetrix) and for both simulation engines (Conrad and HygroTherm).

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 the figure below. THERM may create zero area extensions in two ways.

- 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.

- 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.

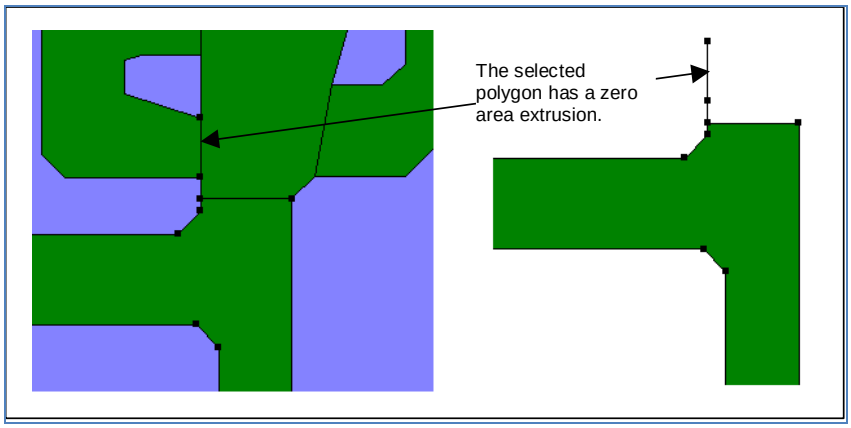


Figure 7-22. An example of a zero area extrusion which will result in an error message of "Invalid Polygon found, Stopping Simulation".

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 **Fill** tool or the **Draw/Fill Void** option.

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 the program warns if no U-factor surface tags have been defined. 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

This step only applies to the Quad Tree mesher. 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.

The Simmetrix mesher reads the geometry directly so this is not a stage that THERM shows for that mesher.

7.3.3. Meshing

In this stage, the mesh is being generated. This is the most time consuming portion of the simulation process, depending on the mesher chosen. The Simmetrix mesher generates the mesh much faster than the Quad Tree mesher.

For complex geometry, such as building components with multiple cross sections such as greenhouse windows, or single cross sections with very fine detail, the recommendation is to use the Simmetrix mesher. The Quad Tree mesher for these complex models may have a problem creating a mesh. In this case you will get a message saying “Mesh Generation Error”.

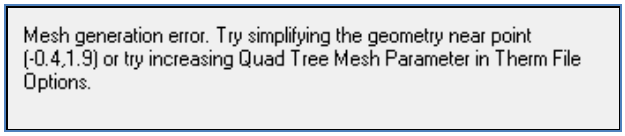


Figure 7-23. THERM displays this message when there is a Mesh Generation Error.

If you get this error message, you should probably use the Simmetrix mesher.

If you want to continue to use the Quad Tree mesher, try simplifying the geometry near the identified point (the x and y coordinates are given in the error message) and usually the program will draw a circle in the area where the mesh error is located. You can also try increasing the **Quad Tree Mesh Parameter** value in the **Options/THERM File Options** menu choice. 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. 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 higher value. This process may have to be repeated several times before the mesh can be generated, and there is an upper limit of approximately 9 for the **Quad Tree Mesh Parameter**. If the model does not mesh with a Mesh Parameter of 8 or 9, simplifying it will be the only option.

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. View Factors (optional)

This stage indicates that the model has radiation enclosure surfaces, such as a frame caity with the detailed radiation model, a radiation enclosure, or a Condensation Index calculation. The view factors between the surfaces are calculated in this stage.

7.3.6. Simulating

The finite element calculation takes place during this stage. It the model does not converge, THERM displays the message "Solution failed to converge. Try changing relaxation parameter or decrease convergence tolerance." The Convergence Tolerance and Relaxation Parameter values are found in **File/Properties/Calculation Options tab**.

7.3.7. Error Estimator (optional)

The error estimator is run if the "Run Error Estimator" is checked in the Mesh Control section of the **File/Properties/Calculation Options tab**.

If the estimated error is above the Maximum % Error Energy Norm (set in **File/Properties/Calculation Options tab)**, THERM refines the mesh and repeats the Generating Input, View Factors (when present),

Simulating and Error estimator steps. It stops when the error is below the limit or when the number of passes reaches Maximum Iterations. In the second case THERM shows "Warning: The maximum number of meshing iterations reached before the error target was reached." and displays the results of the last pass. To reach the error target, increase Maximum Iterations, or start from a finer mesh by changing the Mesh Parameter.

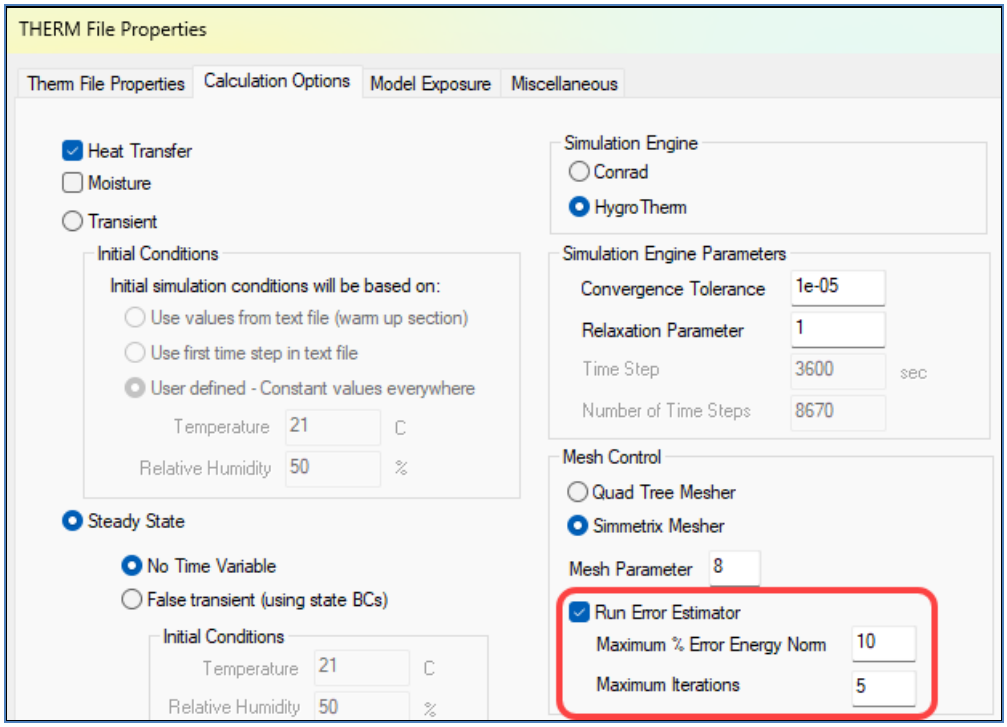


Figure 7-24. The settings for the Error Estimator are set in the File/Properties/Calculation Options tab.

7.4. Viewing the results

When the program has finished the simulation (the calculation steps in the status bar or the hourglass have disappeared), the results are automatically displayed, unless the "Automatically display results after simulation" is unchecked (in Options/Preferences/Preferences tab).

The results can be displayed by clicking the Show Results toolbar button, or the Calculation menu / Show Results option.

7.4.1. Graphic Results

The graphic results that are displayed after a simulation are shown below, and described in details in this section.

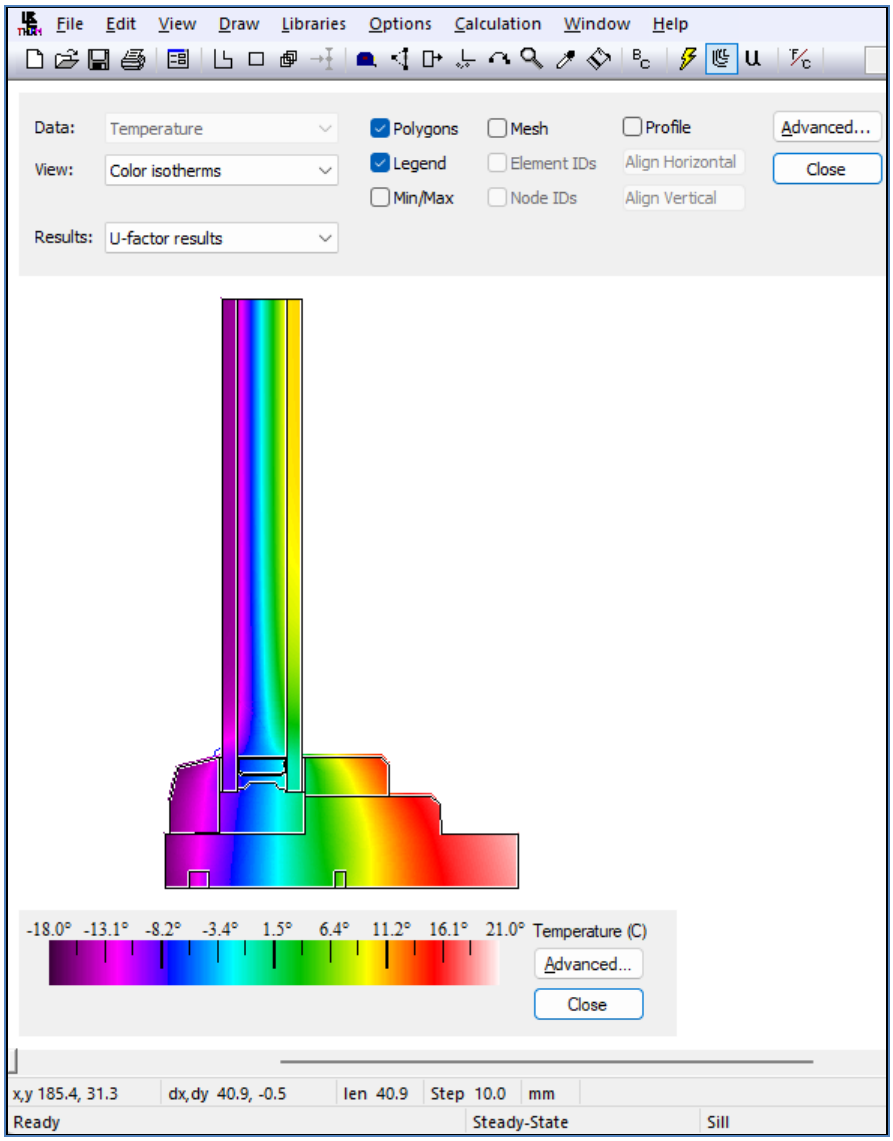


Figure 7-25. The results viewer.

Display Options

Data

For steady-state calculations using either the Conrad or HygroTherm simulation engine, the only values that are displayed are temperature based.

Default for steady-state calculations: Temperature.

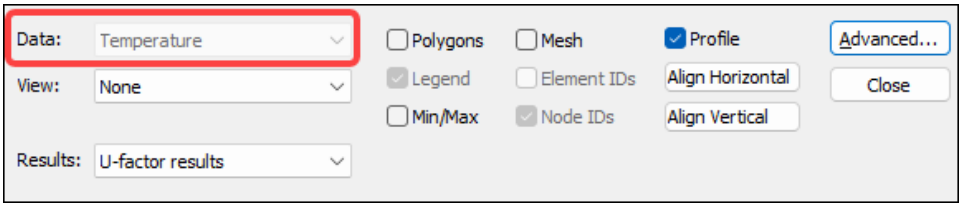


Figure 7-26. The Data pullown list is set to Temperature and cannot be changed to any other option for steady-state simulation results.

View

This pulldown list controls the type of results that are displayed for the Data selection, which in the case of steady-state calculations is Temperature.

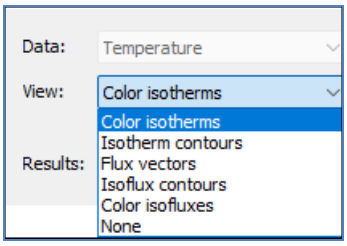


Figure 7-27. The type of results that can be displayed based on the Data selection of Temperature.

Color Isotherms

The color isotherm 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.

Checking the **Legend** checkbox will display the temperature scale.

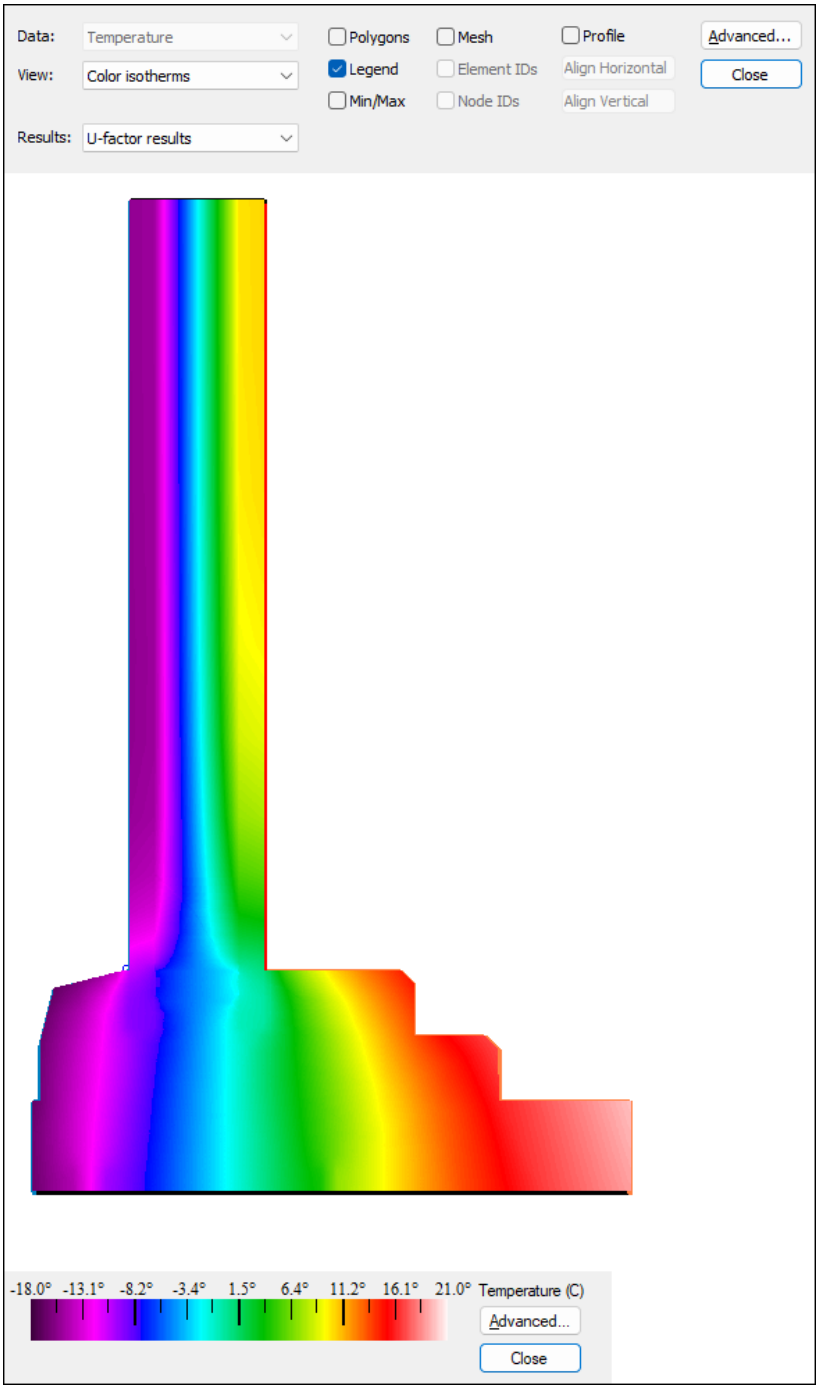


Figure 7-28. Color Isotherm Results with Legend selected.

Isotherm contours

When the heat-transfer analysis is complete, THERM can display the lines of isotherms (representing contant temperatures) 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.

Selecting this option will show the isotherms through the model. The program displays the temperatures on the isotherms approximately every other isotherm.

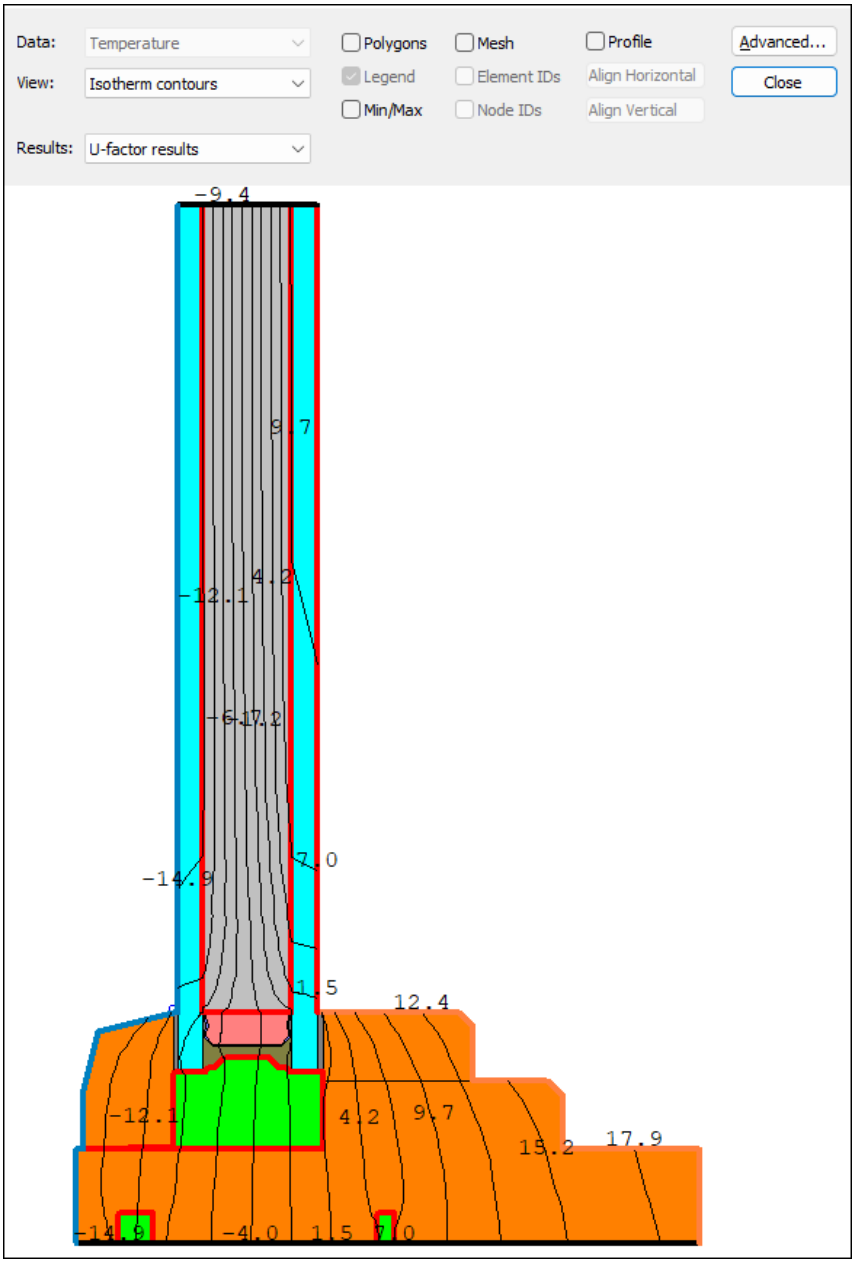


Figure 7-29. Isotherm Contour results.

Flux vectors

The flux vector results indicate the magnitude and direction of heat flow through the cross section. There is one flux vector for each mesh element. 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, and the magnitude is represented by the length of the arrow. This representation of the data is most useful for determining the direction of heat flux.

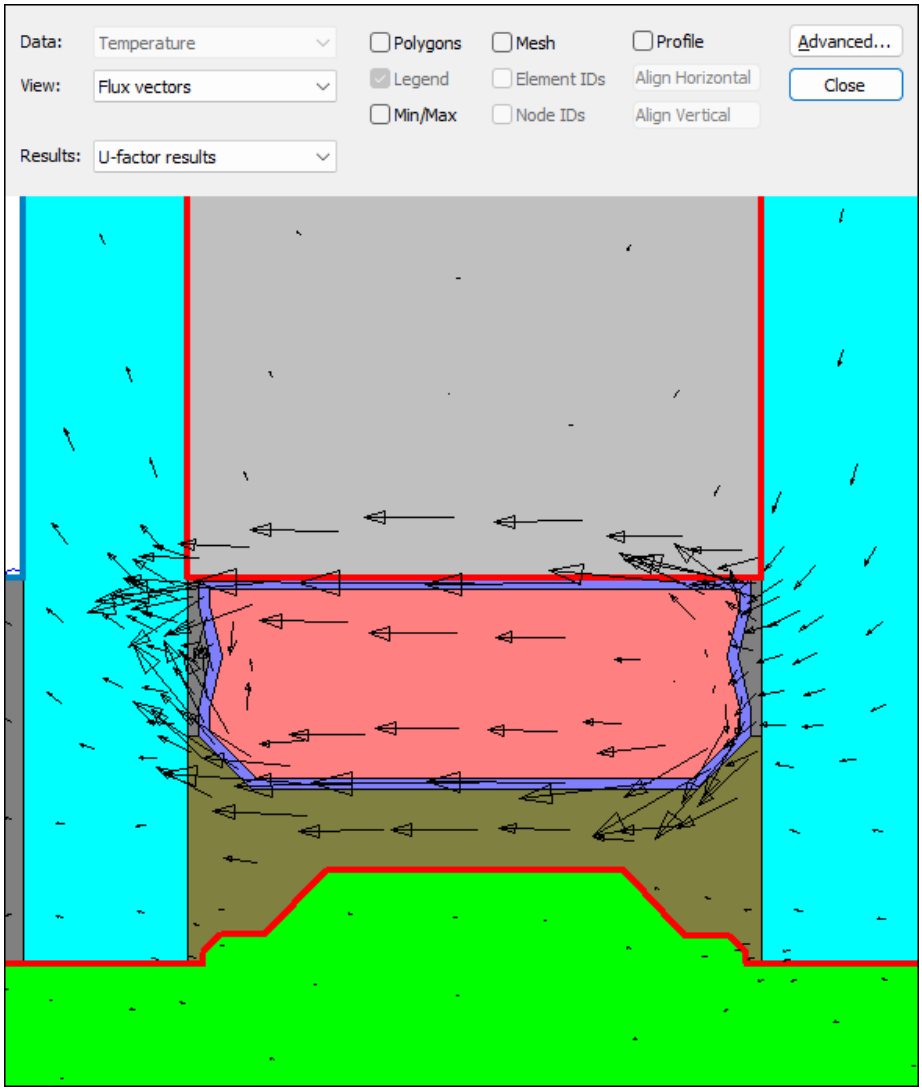


Figure 7-30. Flux Vector results.

Isoflux contours

The constant flux lines delineate equal heat flux magnitude through the model. The flux (W/m2 (SI); IP: Btu/h-ft2) will be shown approximately every other flux line.

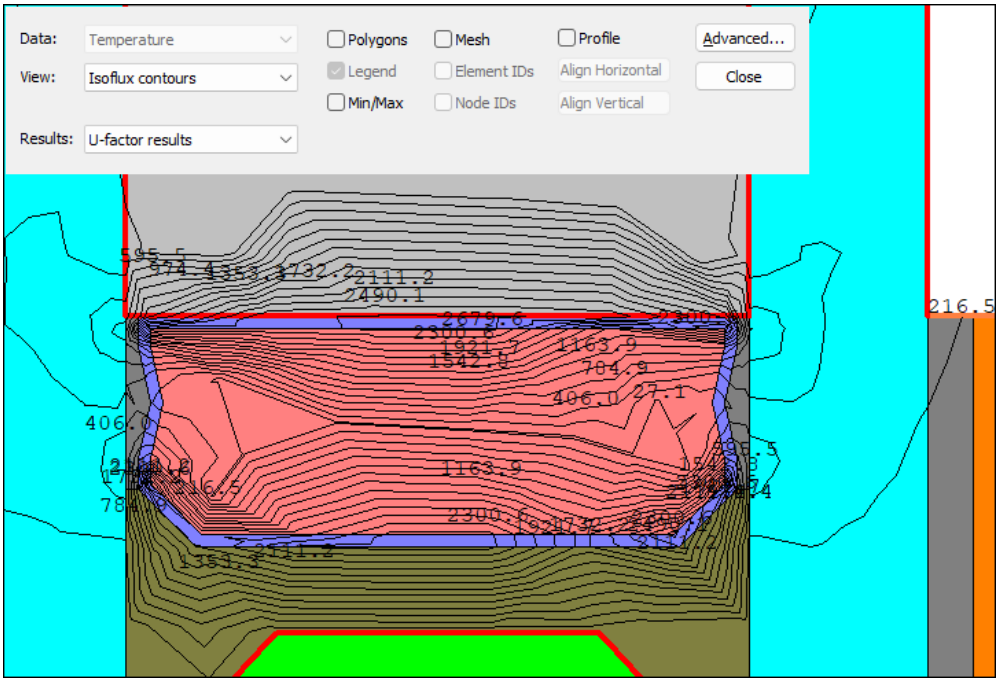


Figure 7-31. Isoflux vector results.

Color isofluxes

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.

Checking the **Legend** checkbox will display the flux magnitude scale.

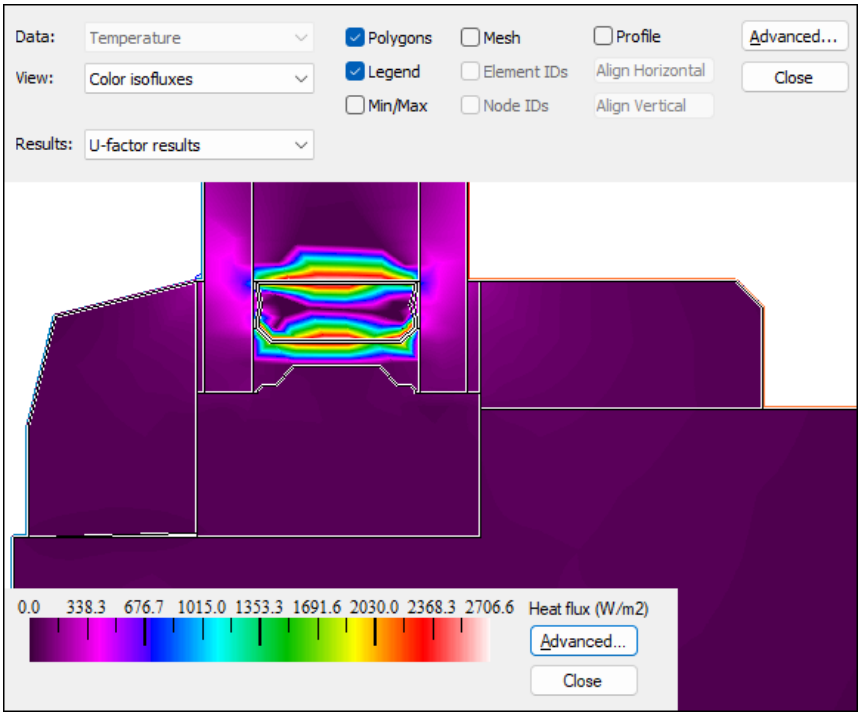


Figure 7-32. Color isofluxes.

None

If None is selected, it is still possible to check the boxes for other display options.

Results

The Results pulldown list allows the results values displayed to be based on the following calculated results

- U-factor results
- CI results

Default for steady-state calculations: U-factor results.

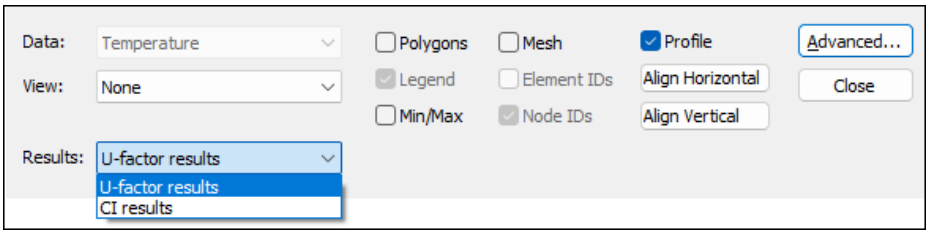


Figure 7-33. The Results pullown list allows the results being displayed to be based on either U-factor or CI results.

Checkbox results options

Polygons This option shows the outline of all the cross-section polygons for the selected View Option.

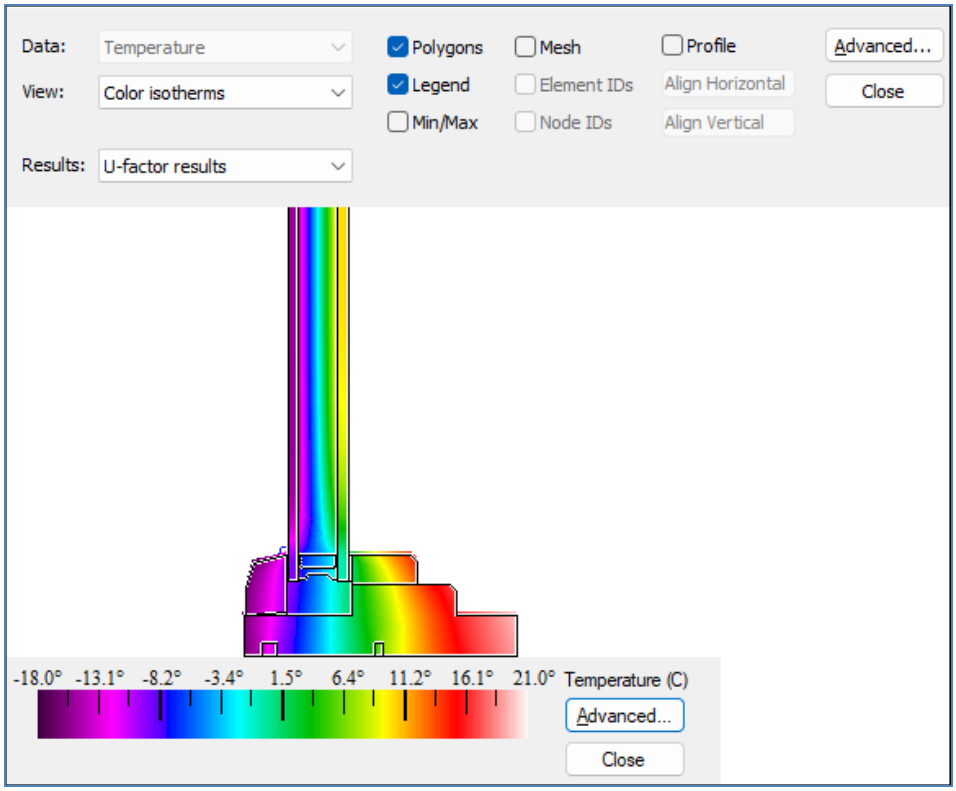


Figure 7-34. With the Polygons option checked, the polygon outlines are displayed with the results selected in the View pulldown list.

Legend This checkbox displays the color legend for the two "color-based" results,

- Color isotherms
- Color isofluxes.

Min/Max This checkbox displays the minimum and maximum temperatures in the simulation for all the options in the View pulldown list, including "None". It is possible to see the Mesh and the Min/Max temperatures by setting View = None, and checking the Min/Max box.

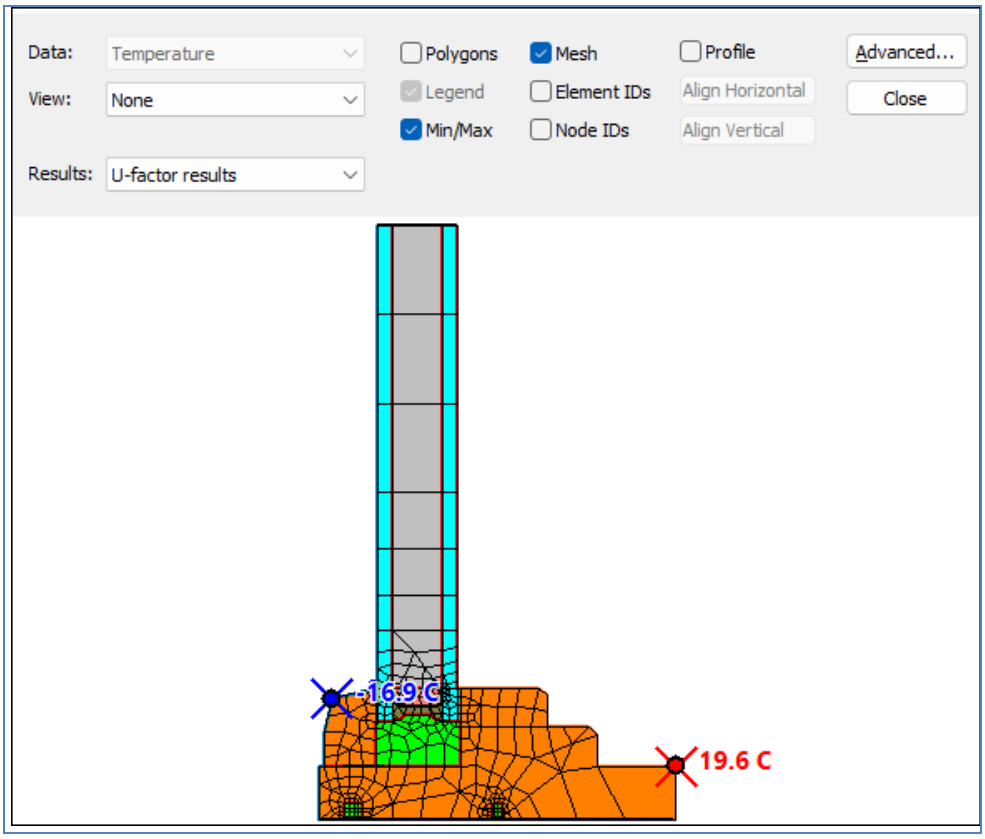


Figure 7-35. With View set to None, and the Min/Max and Mesh boxes checked, the program will display the mesh with the minimum and maximum temperatures displayed.

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.

Checking the Mesh box (with View = None) will show the mesh that the program generated in order to perform the simulation. The program will generate a finer mesh in areas with fine detail or highly conductive materials.

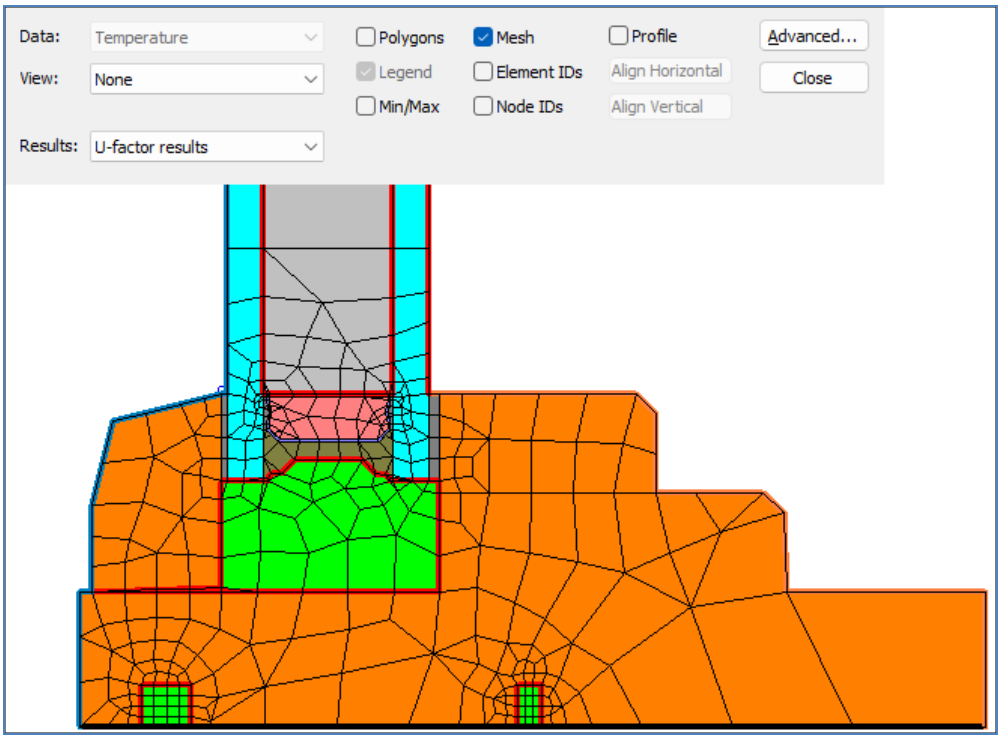


Figure 7-36. Set View=None, and check Mesh, to view the mesh for the model.

Element IDs

If the Mesh box is checked, the Elemend IDs box is available to also check. This will display the numerical ID values that have been assigned to each mesh element. **Default**: unchecked.

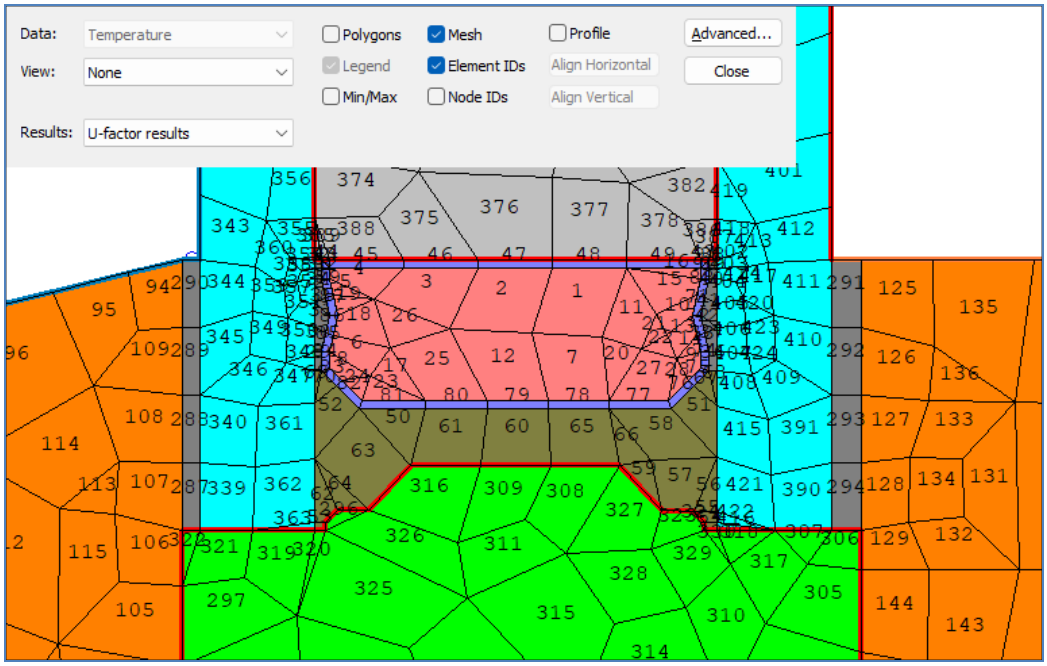


Figure 7-37. If the Mesh box is checked, the Element IDs box can be checked to display the mesh element numerical IDs.

Node IDs Check this box to see the numerical ID values that have been assigned to each mesh node. **Default**: unchecked.

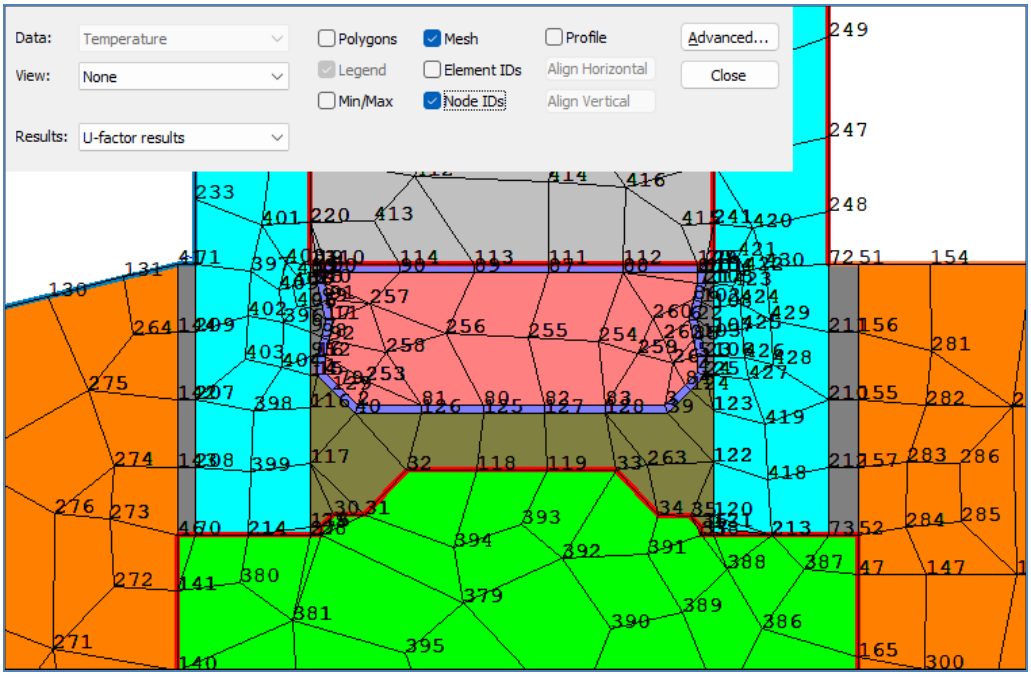


Figure 7-38. If the Mesh box is checked, the Node IDs box can be checked to display the mesh node numerical IDs.

Profile

The temperature profile along the horizontal line is displayed to the right of the model. The horizontal line can be moved up and down to view the temperature profile at different points in the model.

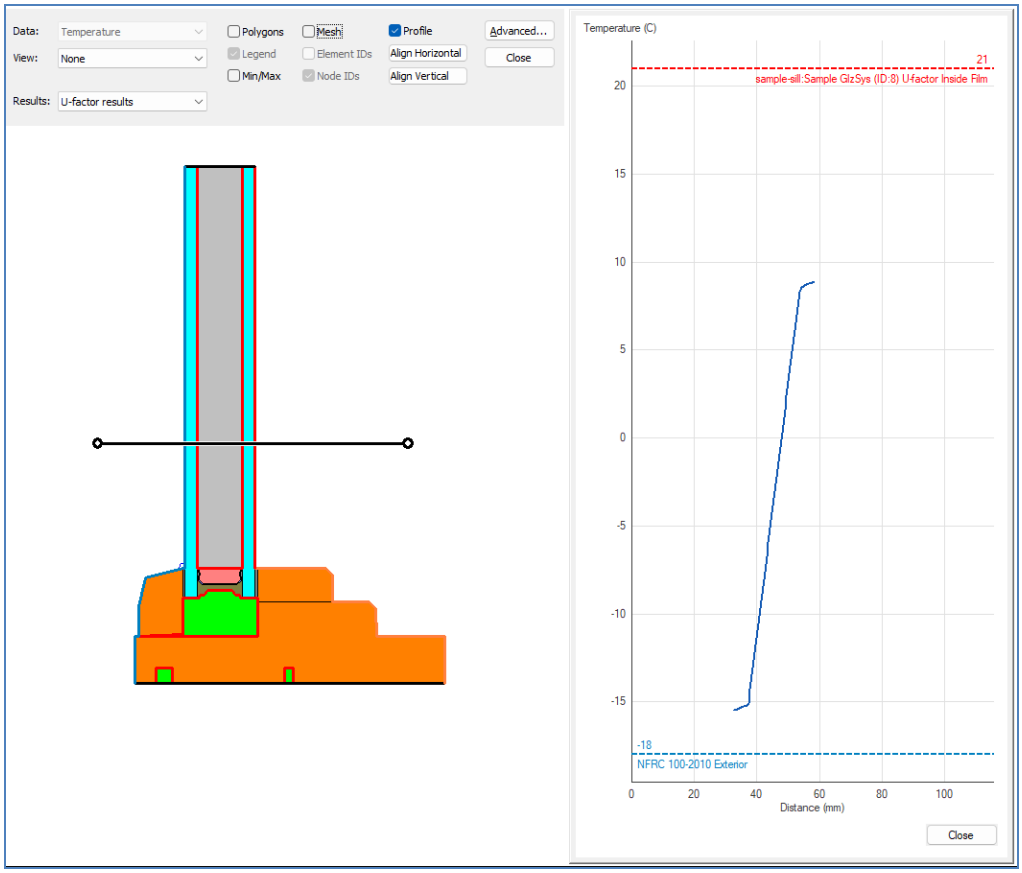


Figure 7-39. The temperature profile through the model at the horizonatal line, with Align Horizontal selected.

There are also options to change the line to generate the profile from horizontal (the default) to vertical.

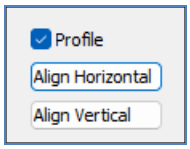


Figure 7-40. The temperature profile can be "cut" either horizontally (the default) or vertically.

Advanced

The **Advanced** button allows setting the **Minimum** and **Maximum** ranges, as well as the **Interval**, for the results available in THERM.

It is also possible to make changes to those values and click the "Make these program defaults" button to save the changed values across THERM sessions.

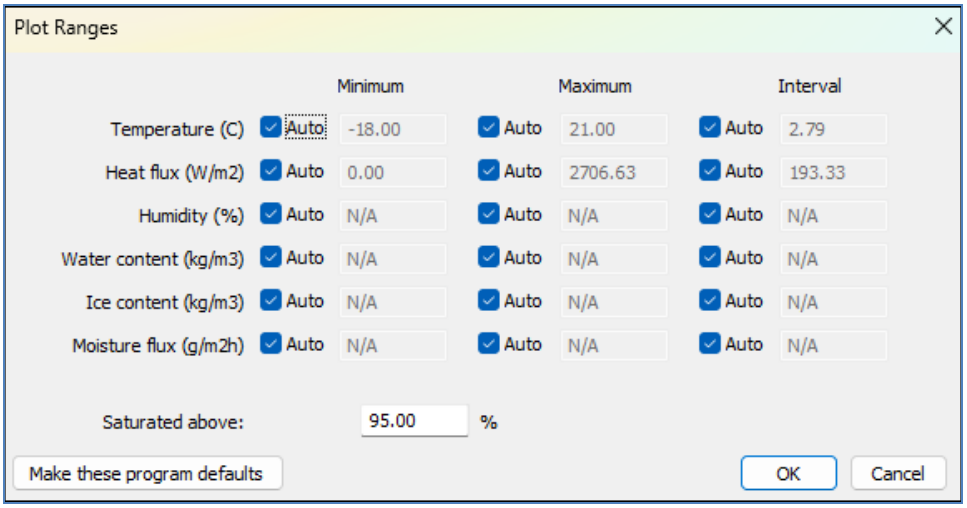


Figure 7-41. The Advanced button allows control over the minimum, maximum and interval settings for the graphic simulation results.

Surface Condensation Potential

This option shows a thick yellow line on the surfaces of the model where condensation may be a problem, based on the Relative Humidity value and temperature of the Boundary Conditions. This condensation potential is for surfaces, not internal condensation. This is the "potential" for condensation, although it does not mean there will be condensation there.

In order to display this result, the **Results** pulldown option must be "CI results". With that selection, the checkbox "Surface condensation potential" will appear

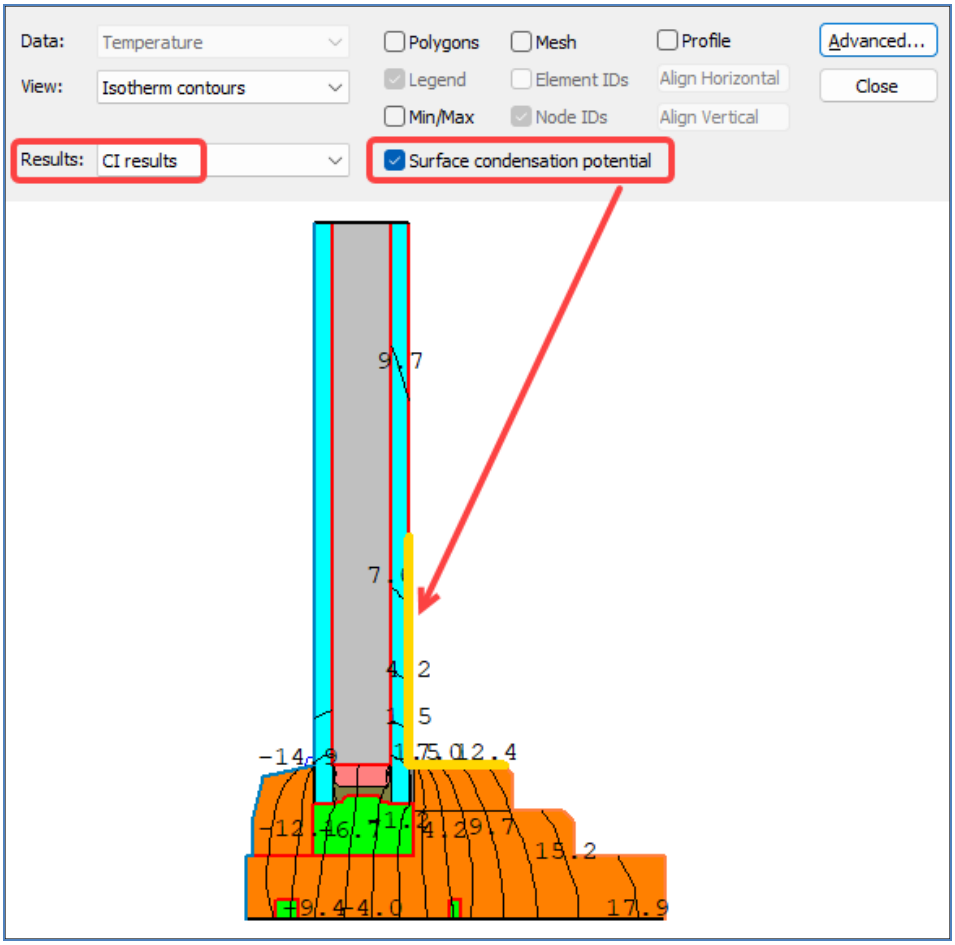


Figure 7-42. If Results is set to CI results, the Surface ondensation potential can be displayed.

The program calculates the dewpoint temperature based on the temperature and relative humidity defined in the boundary conditions for the surfaces, and then displays a thick yellow line along the surface that has that temperature. The Relative Humidity defined in the Boundary Condition represents the humidity of the bulk air next to the boundary.

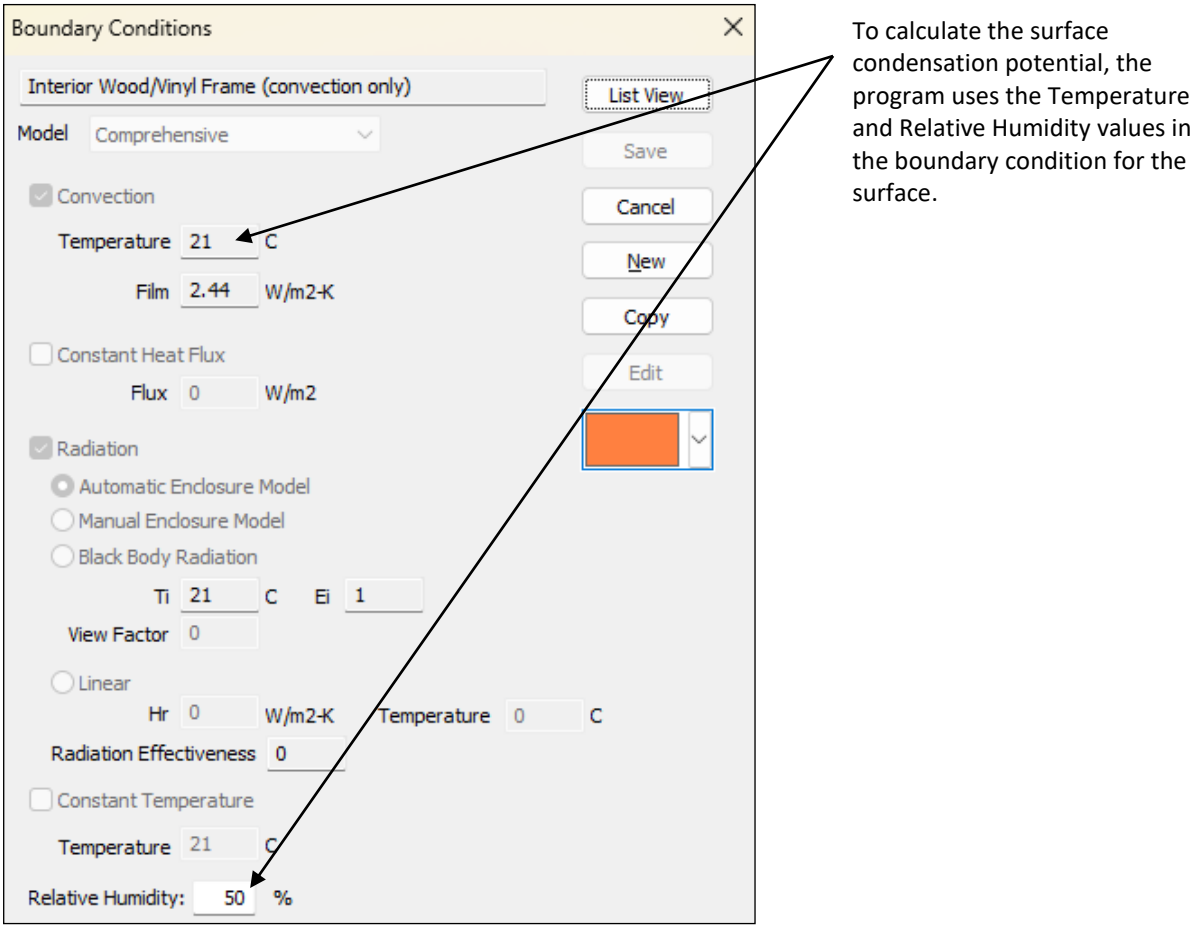


Figure 7-43. The Surface Condensation potential is calculated based on the Temperature and Relative Humidity define in the boundary condition for the surface.

7.4.2. U-factors

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

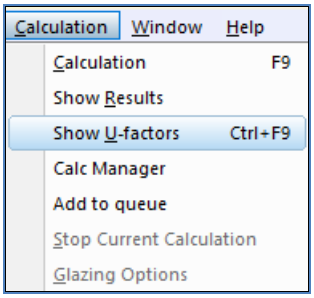


Figure 7-44. Select the Calculation/Show U-factors menu to see the U-Factor results.

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. 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.) When more than two U-factors are defined in a model, the additional labels and results appear in a pull-down list.

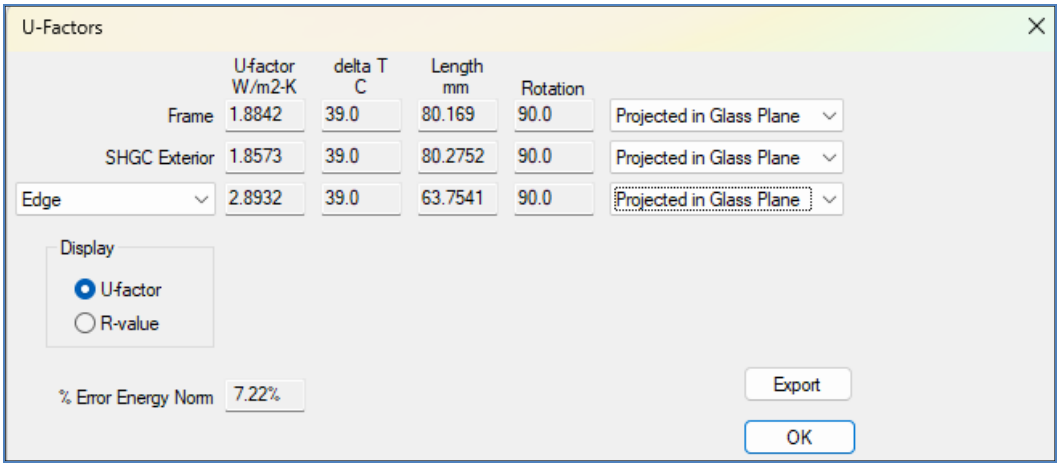


Figure 7-45. The U-factor results dialog box

The U-factors dialog box has the following information and settings.

U-factor name — Name of the U-factor, based on the U-factor tags (defined in **Libraries / Ufactor Names**) applied when assigning Boundary Conditions. The standard names for models with glazing systems are:

- Frame
- Edge
- SHGC Exterior

U-factor — The U-factor calculated in the U-factors dialog box based on the simulations results and the Length input value.

Units: W/m²-K (SI); Btu/h-ft²-F (IP)

Delta T — The temperature difference, between boundary conditions, used to calculate the U-factor.

Units: °C (SI); °F (IP)

Length — The length of the model across which the U-factor is calculated. This is dependent on how the length is defined in the U-factor dialog box, accessed from the Calculation menu, Show U-factors menu option.

Units: mm (SI); inches (IP)

Rotation — The degrees of rotation of the THERM model, set in the Draw menu, the Rotate menu choice.

Units: degrees from horizontal in a clockwise direction.

Example: If a THERM file is rotated 15 degrees clockwise using the **Draw/Rotate** menu option, the program is rotating the model off the vertical axis. In the U-factor dialog box, the program is reporting the rotation counterclockwise off the horizontal axis, which in this case would be 75 degrees.

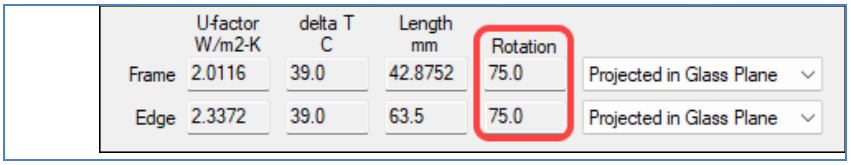


Figure 7-46. U-factor dialog box reports the 15 degree rotation from vertical as 75 degrees from horizontal.

Basis

The dimension used to calculate the U-factor, from the pulldown options in the U-factors dialog box (see *Figure 7-45* for more details). The options are:

- Total Length
- Projected X
- Projected Y
- Custom Length
- Projected in Glass Plane

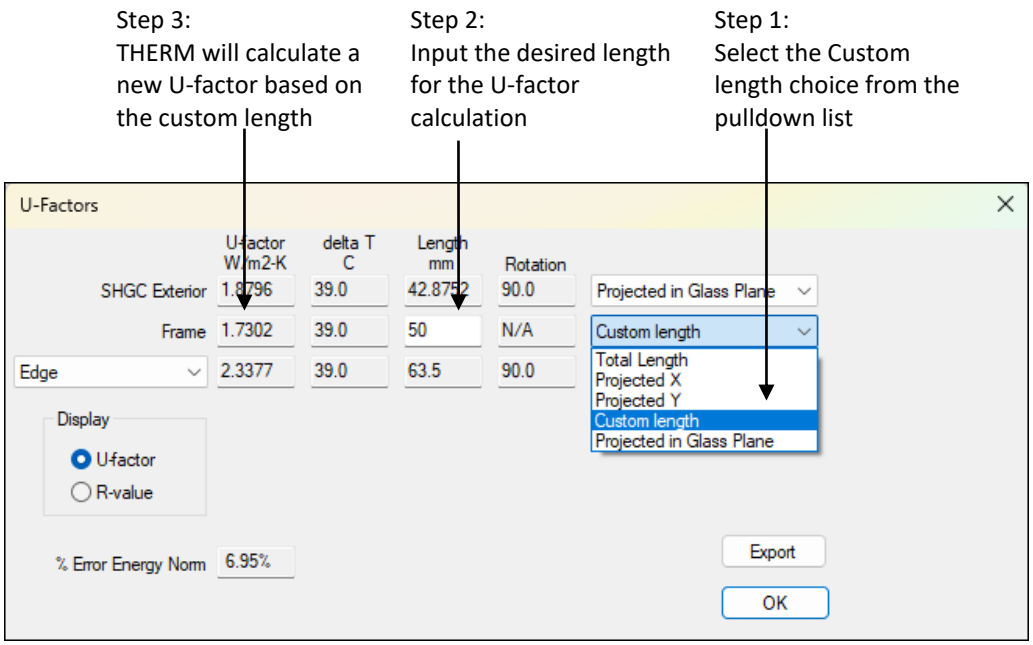


Figure 7-47. A Custom frame length may be specified and the U-factor will automatically recalculate.

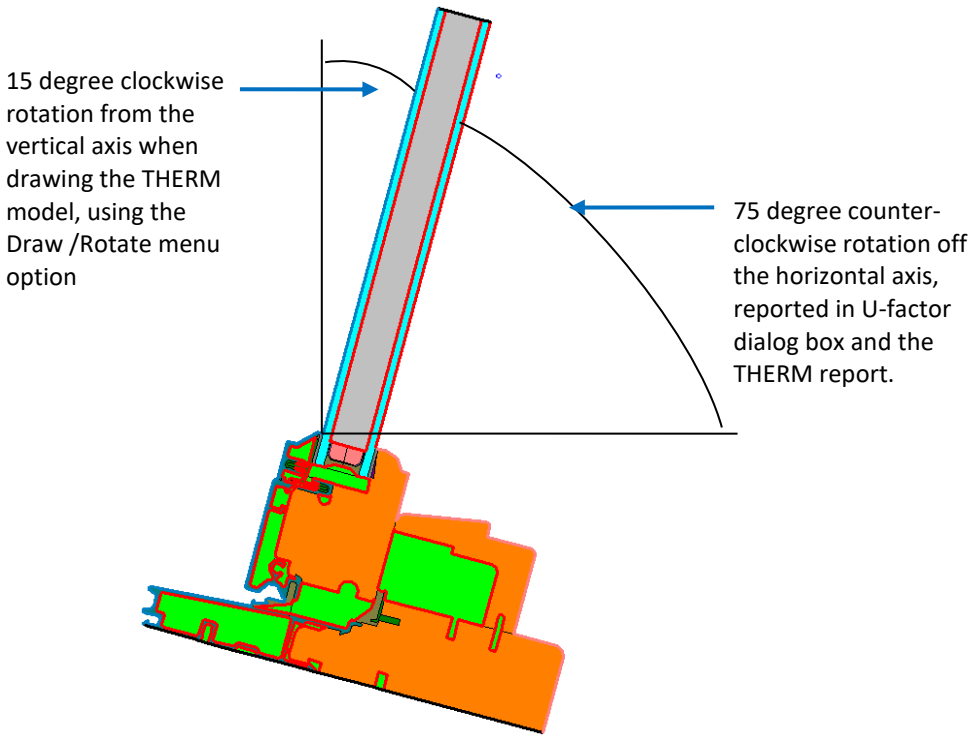


Figure 7-48. Model rotation drawn in THERM and as it is reported in the U-factor dialog box and the THERM report.

The figure below illustrates the definition of Projected X and Projected Y.

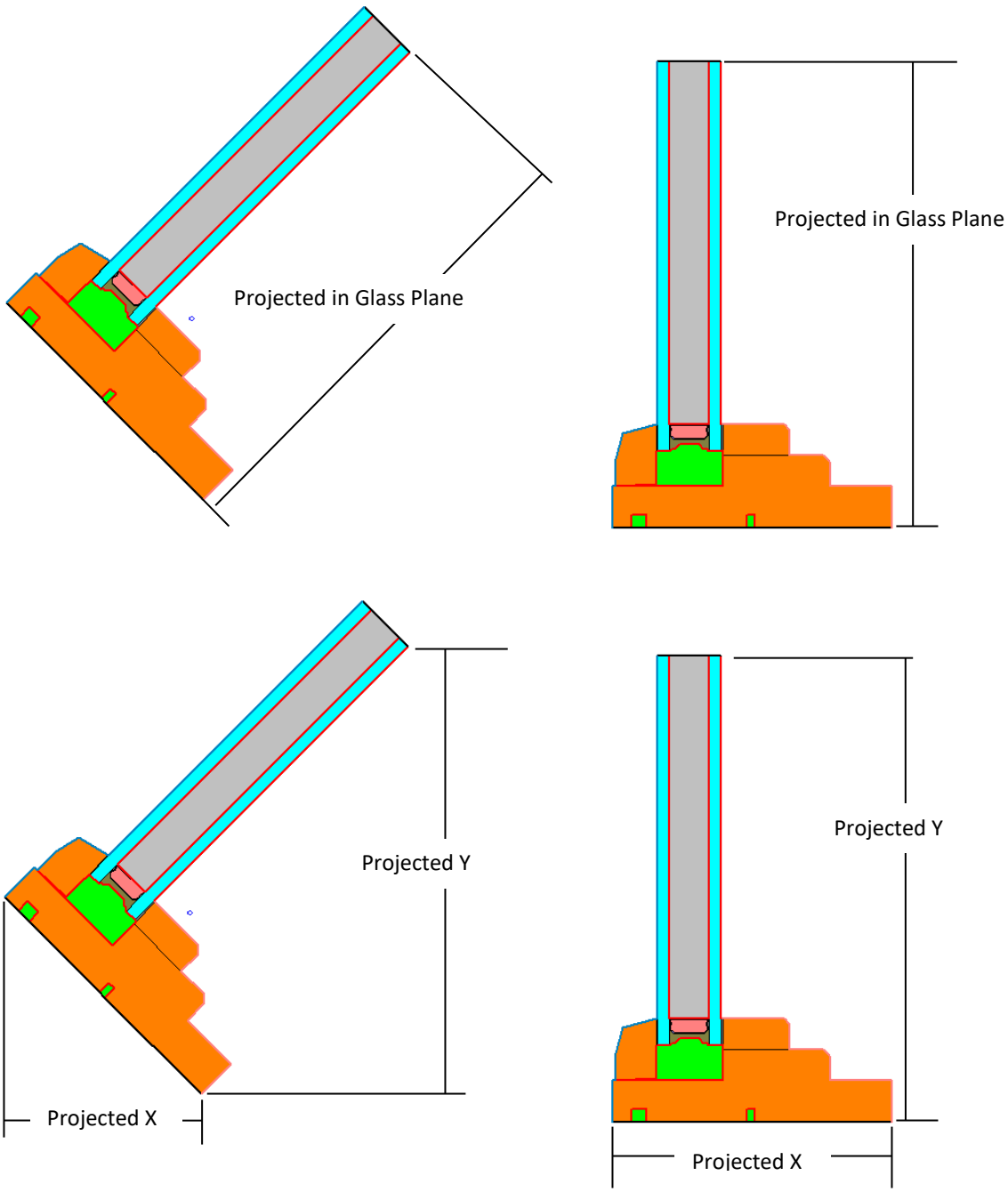


Figure 7-49. Projected X and Projected Y

Display

The radio buttons in the Display group allow the display of the results as either U-factors or R-values.

- **U-factor:** The calculated U-factor based on the Length and Basis input values. **Units:** W/m2-K (SI); Btu/h-ft2-F (IP)
- **R-value:** The inverse of the U-factor. **Units:** m2-K/W; h-ft2-F/Btu (IP)

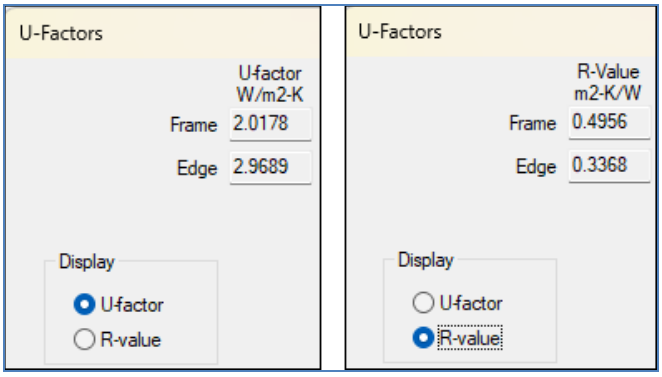


Figure 7-50. The results can be displayed as either U-factors or R-values% Error Energy

% Error Energy Norm

The program will calculate this value if the "Run Error Estimator" checkbox is checked in the File/Properties/Calculation Options tab.

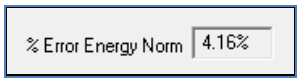


Figure 7-51. The % Error Energy Norm will be displayed if the "Run Error Estimator" is checked in Therm File Options.

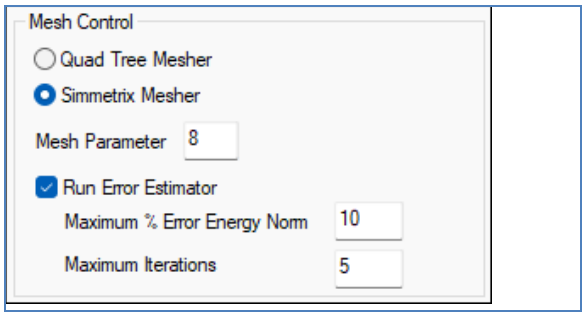


Figure 7-52. Check the "Run Error Esimator" in Therm File Options for the program to calculate the % Errro Energy Norm

Export

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. See *Figure 7-53* below for instructions for exporting the U-factor results.

OK Clicking the OK button closes the U-factors dialog box.

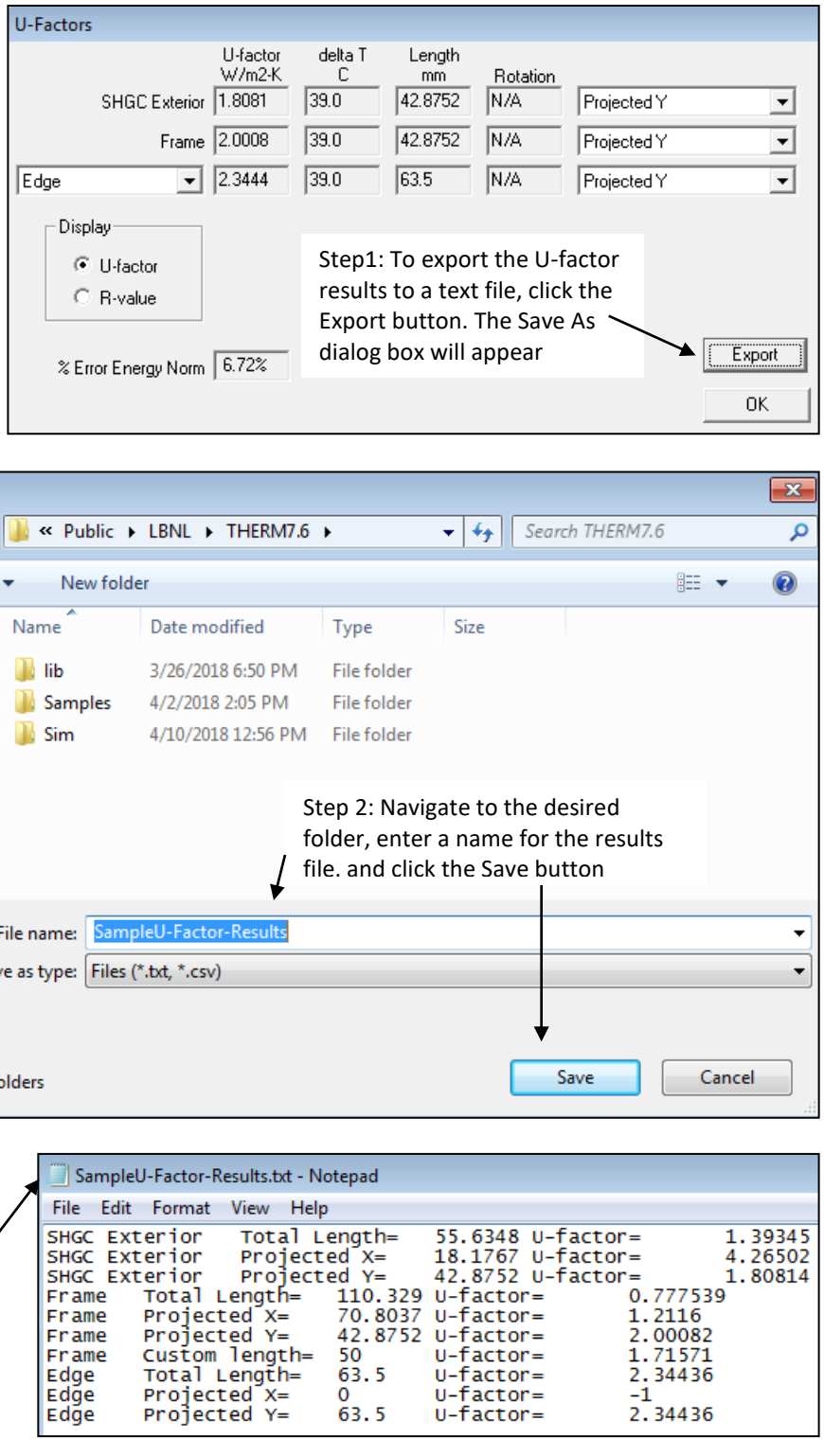


Figure 7-53. You can export the U-factor results to a text file using the Export button on the U-factors dialog box.

7.4.3. Value 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 **Value 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.
- Display the results using the **Calculation / Show Result** menu option, or clicking on the **Show Results** toolbar icon.
- If you want to know the temperature at a specific location in the cross section, you can add that point to the appropriate polygon, which will force the mesher to put a node there. Then you can use the Temperature at Cursor feature to view the temperature at that point after you have calculated the results.

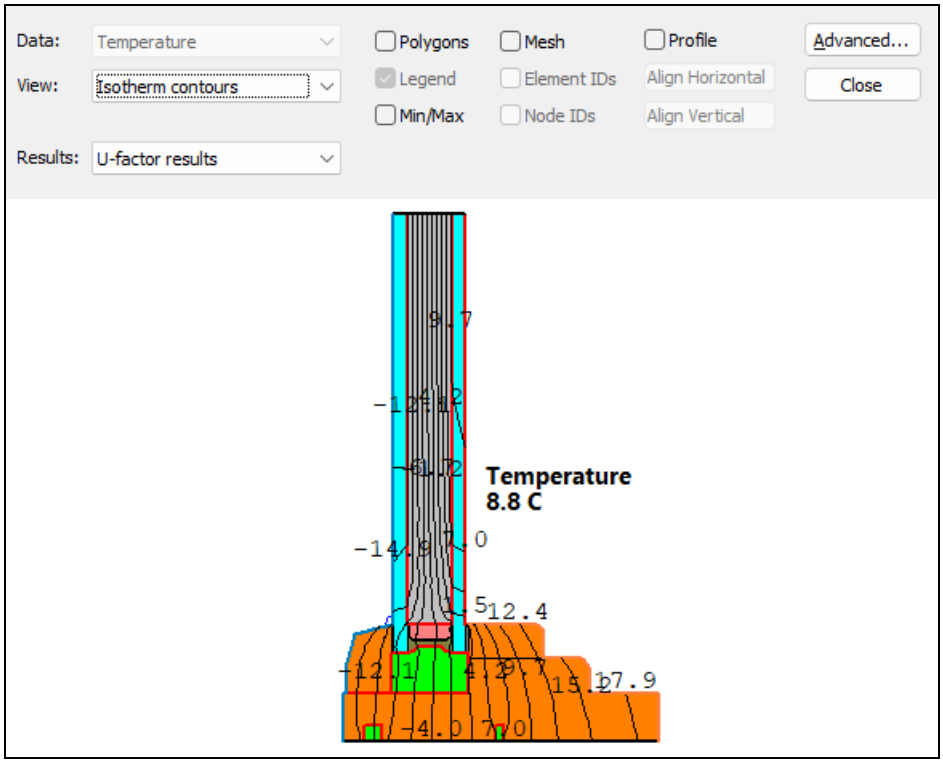


Figure 7-54. Display of the temperature at the cursor location

It is also possible to see the temperatures at specific mesh node locations by doing the following:

- Set the View = None
- Check the Mesh checkbox
- Move the cursor to a specific node and the termperature at that node will be displayed

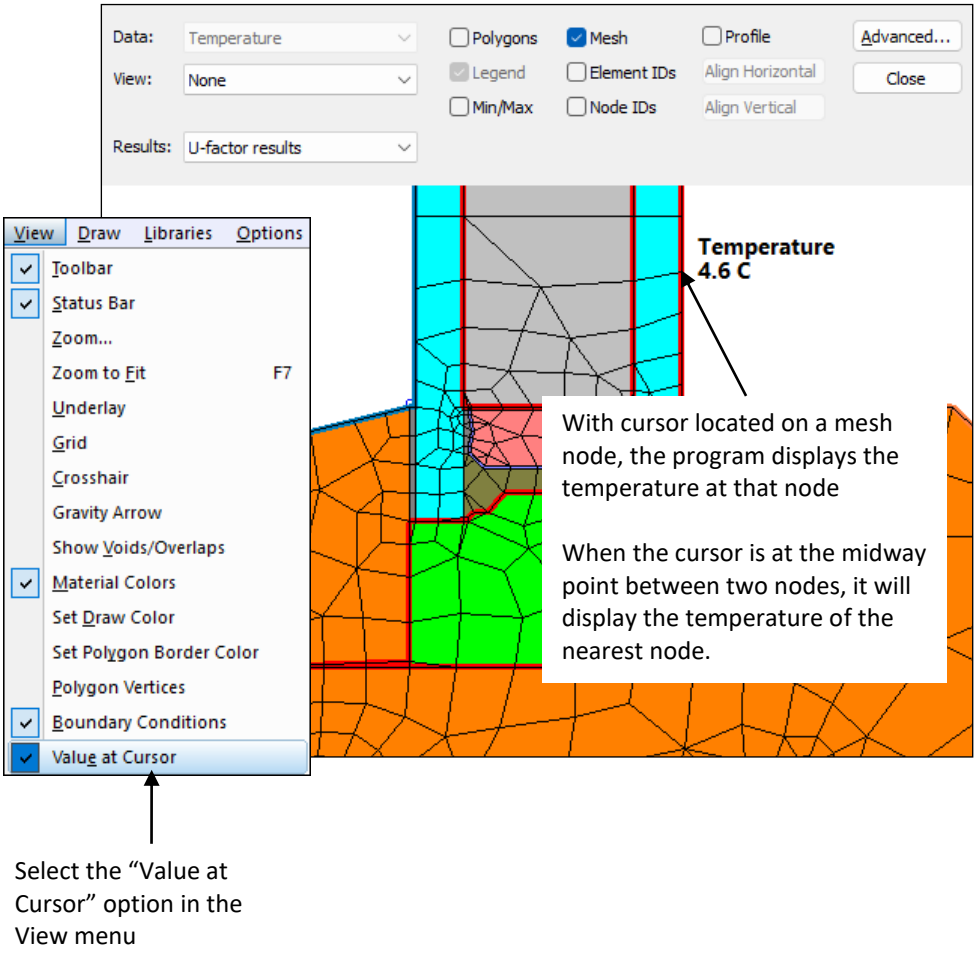


Figure 7-55. Display of the temperature when the cursor is on a mesh node

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.

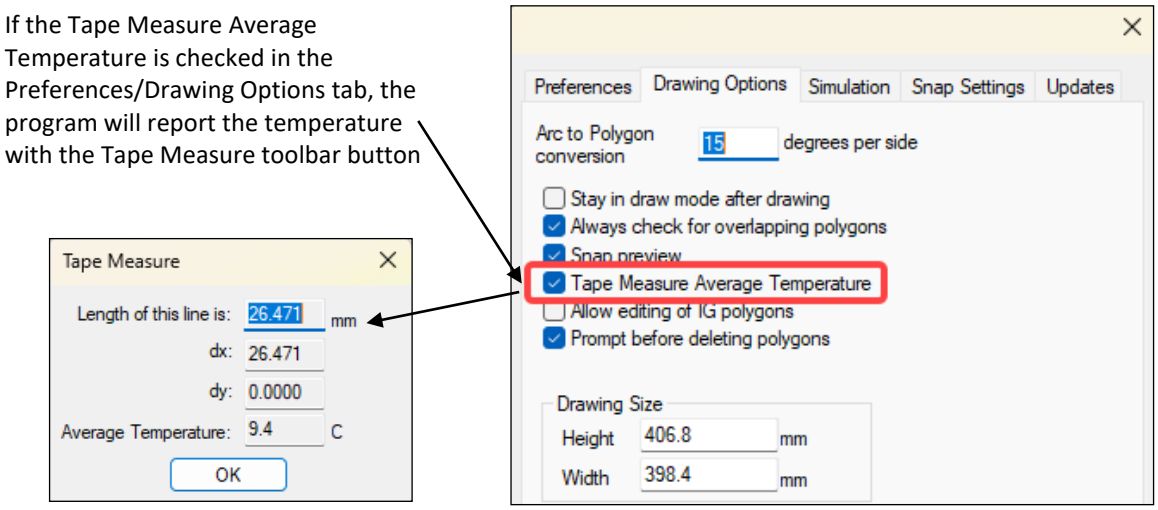


*Figure 7-56. Turn on the **Tape** Measure **Average Temperature** feature from the **Options/Preferences** menu in order to display the temperature at a specific point.*

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.

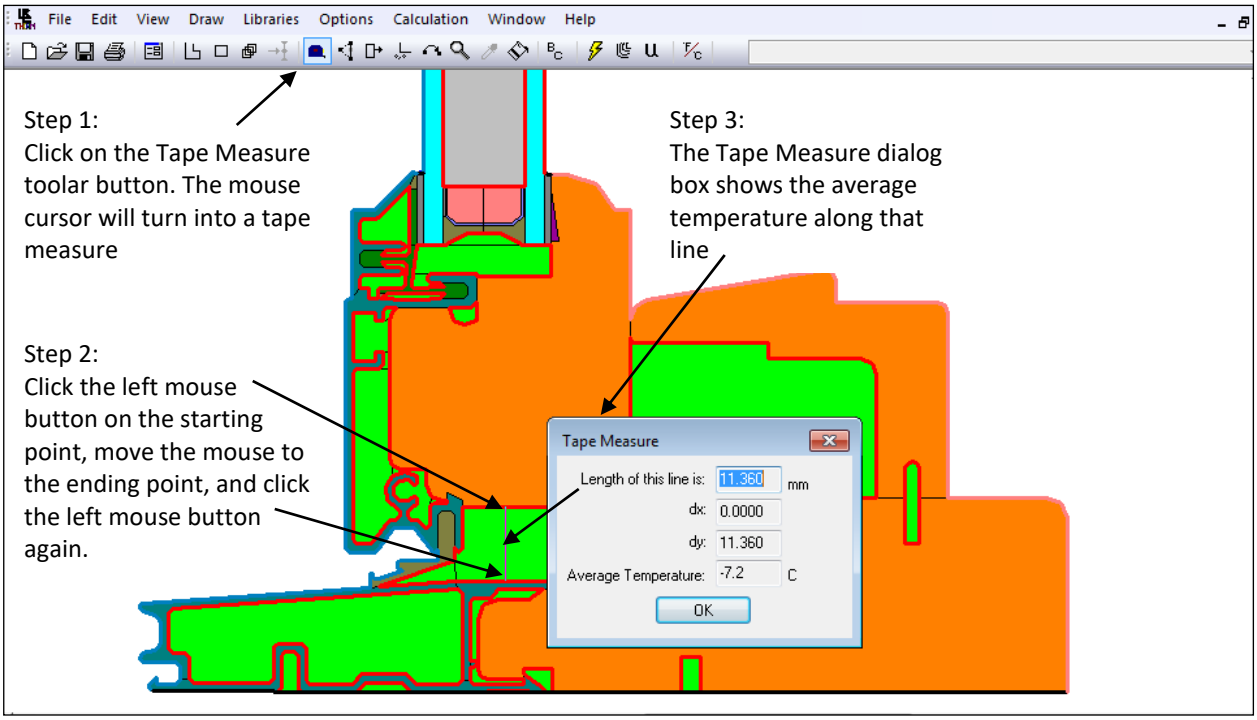


Figure 7-57. Use the Tape Measure to determine the average temperature across a particular surface.

7.4.5. Report

The current version of THERM has many bugs related to the Report feature, so it is not recommended to use that feature. A new version will have a redesigned report that will cover both steady state and transient models.

7.4.6. THERM File Properties

The **THERM File Properties** dialog box is accessed using the **File** menu, **Properties** choice. The values defined apply only to the THERM file currently being edited, and are not global THERM parameters.

There are four tabs in THERM File Properties:

- Therm File Properties
- Calculation Options
- Model Exposure
- Miscellaneous

Therm File Properties tab

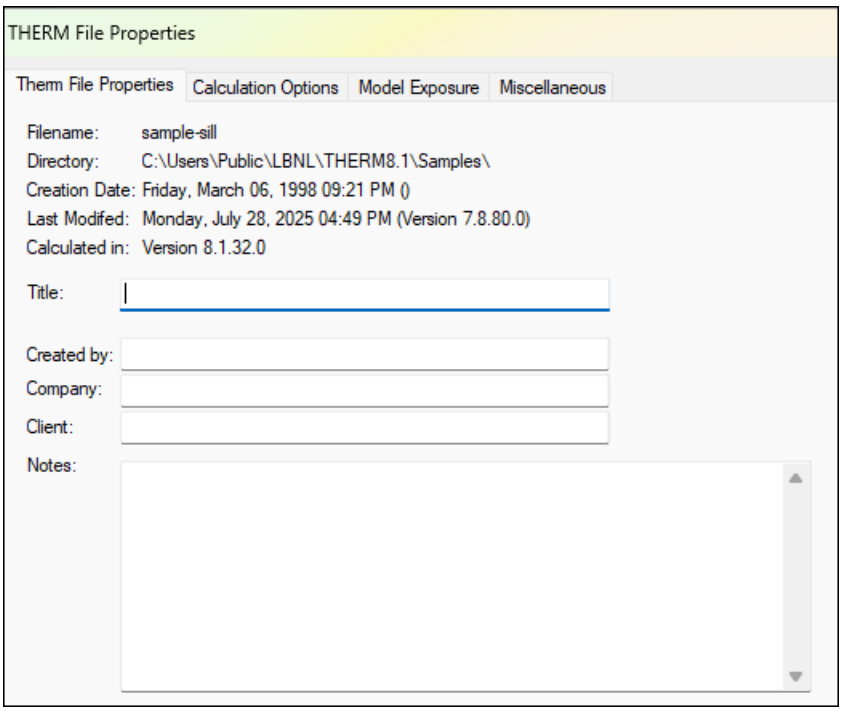


Figure 7-58. Therm File Properties tab allows input of information about the THERM file for the report.

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.
Calculated in	Shows the version of THERM used in the calculation.
Title	Enter a name for the model (optional) which is more descriptive than the THERM file name. For example, the serial number and product name could be included here.
Created by	Enter the name of who created the model (optional).
Company	Enter the name of the company (optional).
Client	Enter the name of the client (optional).
Notes	Enter any other information about the model.

Calculation Options tab

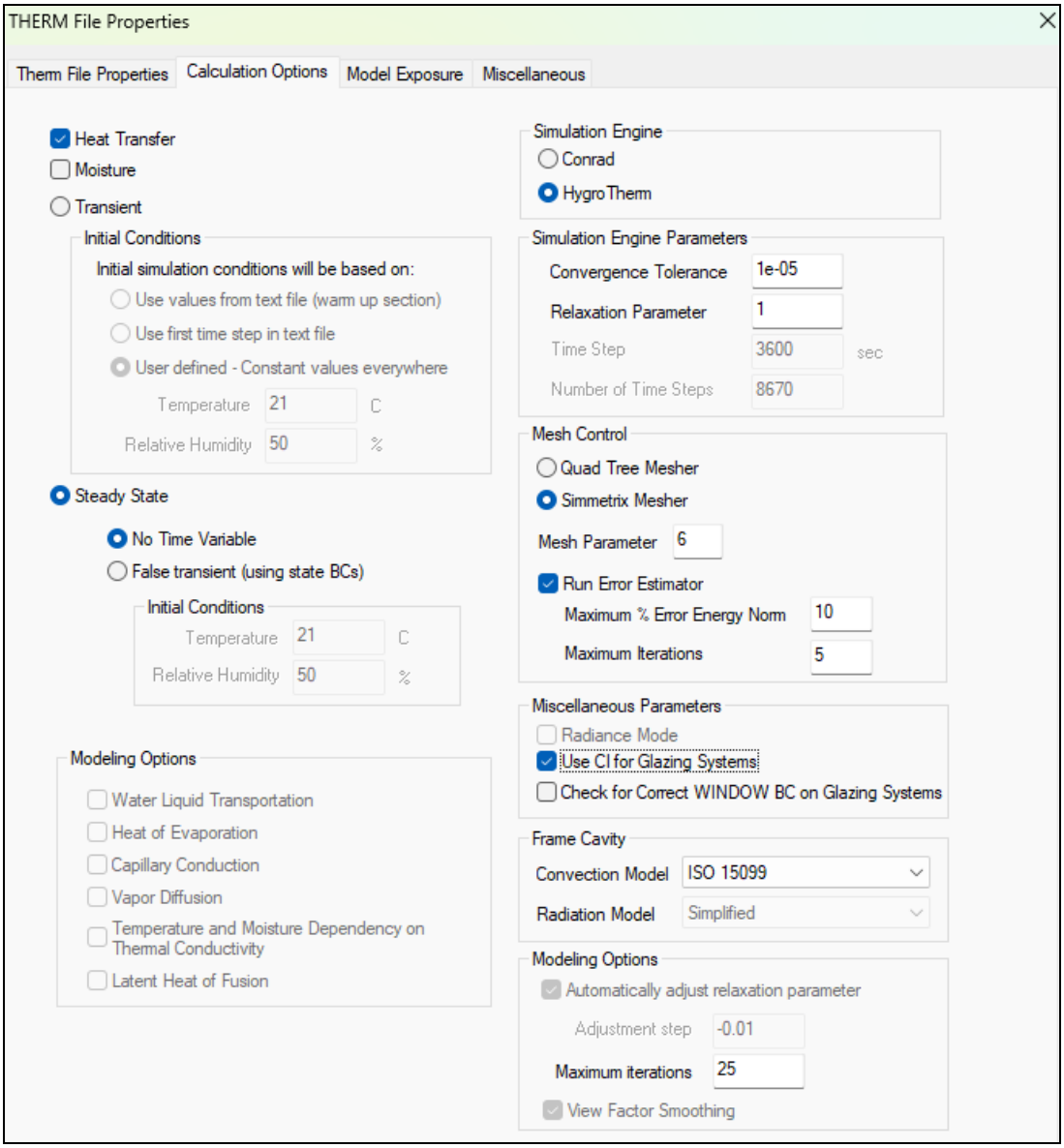


Figure 7-59. Therm File Properties / Calculation Options tab controls many important aspects of a simulation.

Heat Transfer	*This only applies to the HygroTherm simulation engine.* Check this box to model heat transfer with the HygroTherm Simulation Engine
Moisture	*This only applies to the HygroTherm simulation engine.* Check this box to model moisture with the HygroThermSimulation Engine

Transient *This only applies to the HygroTherm simulation engine.*	
Initial Conditions	Initial simulation conditions will be based on either: ▪ User values from text file (warm up section): *currently not implemented* ▪ Use first time step in text file: *currently not implemented* ▪ User defined – Constant values everywhere ▪ Temperature: Default: 21 C. ▪ Relative Humidity: Default: 10. Units: percentage (%)
Steady State *This applies to both the Conrad and HygroTherm simulation engines.*	
No Time Variable	This uses the steady state equation to calculation heat transfer and there is no time variable. See technical documentation
False transient (using state BCs)	This runs a transient simulation until it reaches a steady state solution. Initial Conditions: this sets the temperature and relative humidity of the entire model to start the simulation. It is also necessary to use transient Boundary Conditions that have contant temperature and relative humidity values over the timesteps.
Modeling Options *This only applies to the Transiet Thermal + Moisture simulation engine.*	
	These are modeling options to set when modeling moisture using the HygroTherm simulation engine. *The default state for all these options is unchecked (off):* • **Water Liquid Transportation:** Models transport of water in a liquid state. If this is not checked, then only water vapor is modeled. Available when Moisture is checked • **Heat of Evaporation:** Models phase change of water. Includes the heat absorbed when water evaporates inside a material and released when vapor condenses. Available when both Heat Transfer and Moisture are checked. Used in transient calculations only. • **Capilary Conduction:** Includes the heat carried along by liquid water as it moves through a material. Has an effect only when Water Liquid Transportation is also checked. Available when both Heat Transfer and Moisture are checked. Used in transient calculations only. • **Vapor Diffusion Conduction:** Models diffusion of vapor thorugh material. • **Temperature and Moisture Dependency on Thermal Conductivity:** *Currently not available* • **Latent Heat of Fusion:** Includes the heat released when water inside a material freezes and the heat absorbed when ice thaws. With this option checked, a material holds near 0 C until all of its water has frozen or thawed, and the results include the ice content in each material. Available for transient calculations with Heat Transfer checked. The water in the material comes from the initial conditions or from the moisture calculation.
Simulation Engine	
ConRad	Steady-state thermal simulation
HygroTherm	Transient thermal and moisture simulation engine

Simulation Engine Parameters	
Convergence Tolerance	This controls the simulation error norm tolerance. If the error norm calculation is greater than this value, the program will display a "non-convergence" error message. If using the transient moisture model and it doesn't converge, this value could be reduced to 1e-02. Used in both the ConRad and HygroThermFEM simulation engines. **Default:** 1e-05.
Relaxation Parameter	This is the initial value for relaxation parameter used in iterations. Used in both the ConRad and HygroThermFEM simulation engines. **Default:** 1.
Time Step	*This only applies to the HygroTherm simulation engine, Transient calculation* This parameter is used by the HygroTherm transient model to set the length of the time step. The default value of 3600 seconds equals 1 hour. **Default:** 3600. **Units:** seconds.
Number of Time Steps	*This only applies to the HygroTherm simulation engine, Transient calculation* This parameter determines how many times steps are calculated when the HygroTherm simulation engine is selected. It works in conjunction with the Time Step value. The default Time Step value of 3600 seconds (1 hour) used with the default value of the Number of Time Steps (8670) would result in a simulation at 1 hour intervals for a year. **Default:** 8670. **Units:** unitless
Mesh Control	
Quad Tree Mesher	The mesh engine used in previous versions of THERM (7.8 and lower).
Simmetrix Mesher	A new mesher that will mesh much more complex model geometries than the mesher in THERM 7 and lower.
Mesh parameter	The relative size of the finite element mesh created for the model. The larger the number the finer the mesh. The upper limit of the mesh parameter is usually 9. Increasing this value may help mesh files that return "mesh errors". Default: 6. D**efault:** ▪ Conrad: 6 ▪ Simmetrix: 8

Run Error Estimator	*This only applies to the Steady State options of both Conrad and HygroTherm simulation engines.* If this option is checked, the program will perform the calculation of the error energy norm. If the simulation does not meet the value in Maximum % Error Energy Norm, the program will increment the Quadtree Mesh Parameter (if Automatically increment mesh parameter is checked on the Simulation tab), until the Maximum Iteration value is reached. **Default:** checked.
Maximum % Error Energy Norm	*This only applies to the Steady State simulation engine.* THERM incorporates a local error estimator and subsequent local mesh refinement. The error estimator is based on a published and well regarded methodology based on the estimate of Error Energy Norm (EEN). In this technique, only regions with an EEN larger than the threshold value are refined, resulting in optimum mesh size. An EEN value of 10% is chosen for the default because it results in overall computational accuracy of well below 1%, which is the ISO 10211 requirement. **Default:** 10%
Maximum Iterations	*This only applies to the Steady State simulation engine.* The number of iterations for the program to run in an attempt to meet the Maximum % Error Energy Norm specified. **Default:** 5
Miscellanceous Parameters	
Radiance Mode	*This option only applies to the Conrad simulation engine.* If this option is checked, THERM will use Radiance to model the optical properties of a THERM model. This is used for modeling shading systems whose geometry is modeled in THERM. The THERM background will turn gray when this option is checked, to indicate that this option is selected, because this is not an option that would normally be used. Default: unchecked.
Use CI for Glazing Systems	If this option is checked, THERM will perform a Condensation Index simulation as well as the standard U-factor calculation. Default: unchecked.
Check for Correct WINDOW BC on Glazing	If this option is checked, THERM will check to see that the Boundary Conditions applied to an imported WINDOW glazing system match those that were imported with that glazing system.

Systems	Default: unchecked.
Frame Cavity	
Convection Model	The convection model used when modeling frame cavities: ▪ ISO 15099 ▪ EN10077 Default: ISO 15099
Radiation Model	The radiation model used in the frame cavity calculations. The only option is Simplified, which is grayed out and can't be changed
Modeling Options *These options only applies to the Conrad simulation engine*	
Automatically adjust relaxation parameter	When equations are not converging, the relaxation parameter modifies previous iteration result in order to better converge. The values are from 0 to 1. The solver starts from a value of 1 and reduces it from there. Default: checked
Adjustment step	Incremental reduction of the relaxation parameter from 1 to 0. Default: -0.01
Maximum iterations	Number of iterations to apply the automatic relaxation parameter adjustment to. *For both the HygorThermFEM and Steady-State simulation engines* Default: 25
View Factor Smoothing	This is used for gray body radiation calculation. If the view factors (surfaces being viewed) are very different, such as blocking surfaces, the view factor smoothing helps accuracy of results. See TARCOG technical documentation on the THERM documentation webpage.

Model Exposure tab

The Model Exposure tab defines the cross section type, orientation and gravity vector of the model

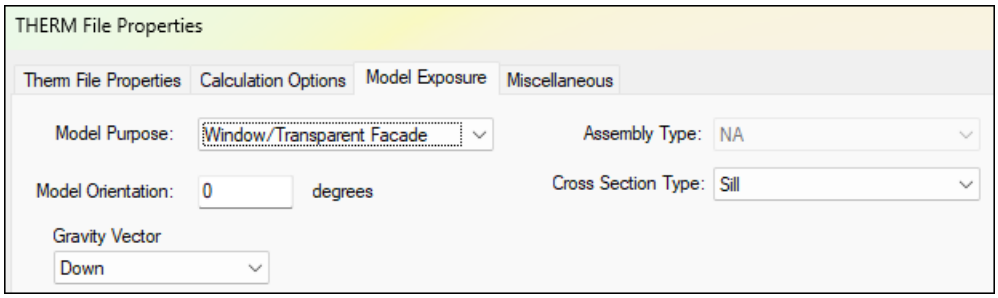
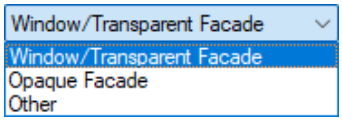


Figure 7-60. Therm File Properties / Model Exposure tab controls many important aspects of a simulation.

Model purpose

This pulldown list consists of the following options

- Window / Transparent Façade
- Opaque Façade
- Other



Default: Window / Transparent Facade

Assembly Type

If **Model Purpose** is set to **Opaque Façade**, this pulldown list becomes active with the following choices:

- Wall
- Roof

If **Model Purpose** is set to either **Window/Transparent Façade** or **Other**, this value is set to "N/A".

Model Orientation

The direction the exterior surface of the model faces. This orientation of the exterior surface is used to determine the wind direction for the simulation.

Below are the standard Cardinal orientations, but any value from 0 – 360 may be entered.

- North = 0
- East = 90
- South = 180
- West = 270

Legal values: 0 – 360.

Units: degrees

Cross Section Type

The values in this pulldown depend on the Model Purpose selection.

- If Model Purpose = Window / Transparent Façade, the choices are

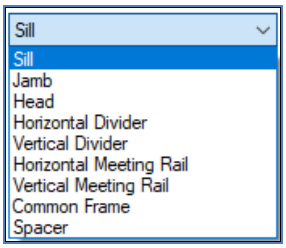


Figure 7-61. Cross-Section Type if Model Purpose = Window/Transparent Facade.

Note: this setting was located under the Therm File Properties tab in previous versions of the program.

- If Model Purpose = Opaque Façade, the choices are

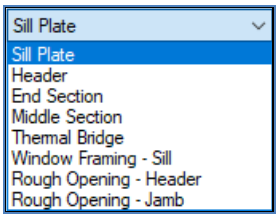


Figure 7-62. Cross-Section Type if Model Purpose = Opaque Facade.

- If Model Purpose = Other, the choices are

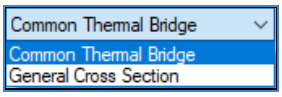


Figure 7-63. Cross-Section Type if Model Purpose = Other.

Gravity Vector

The choices for the gravity vector are

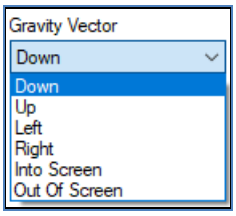


Figure 7-64. Gravity Vector options.

Note: *Setting the gravity vector was done from the View menu in previous versions of THERM*

Note: *This setting will change the View/Gravity Arrow to match. Don't change gravity vector from the View menu in this version of THERM.*

Miscellaneous tab

Below is a description of the fields in the File / Properties / Miscellaneous tab.

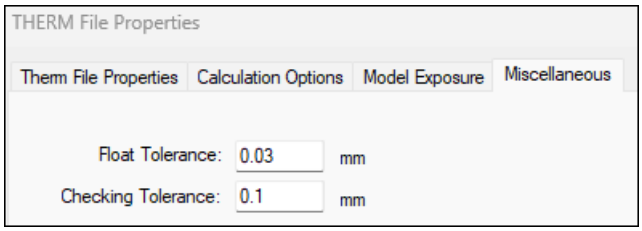


Figure 7-65. Therm File Properties / Model Exposure tab controls many important aspects of a simulation.

Float Tolerance	Default: 0.02 mm
Checking Tolerance	Default: 0.10 mm

8. MODELING FRAME AND EDGE HEAT TRANSFER

8.1. Overview

In order to calculate the total fenestration product U-factor, the U-factor is determined for the following components, and then these values are area-weighted for the whole product:

- Center-of-glazing values from WINDOW
- Frame and edge-of-glazing values from THERM

8.2. Cross-Sections to be Modeled

The fenestration product being modeled should be divided into the cross-sections which completely define the heat transfer through the product. THERM is used to calculate the U-factor values for the edge-of-glazing as well as the frame components (sill, head, jamb, meeting rail, and divider).

The operator type of the fenestration product, as well as the configuration of each component of the frame, will determine the number of cross sections that are modeled. Table 8-1 lists the minimum and maximum cross sections that should be modeled for each operator type, and Figure 8-1 illustrates where the cross sections should be taken. The maximum assumes that each frame cross section is unique and therefore should be modeled. If there are some cross sections of the frame which are identical, such as a fixed window with the same cross section for the jambs, then only one cross section needs to be modeled for the jamb. However, the new modeling assumptions for cavities when modeling Condensation, based on ISO 15099, require that different cross sections be modeled for head, sill, and jambs if those cross sections contain frame cavities. If only U-factor is being modeled, the head and the sill can be combined into one calculation if the geometries for each are the same.

Table 8-3. Cross sections to be modeled

Operator Type	Number of Cross Sections to Model			Cross Sections
	Minimum		Maximum	
Single Lite (Fixed, Picture, Transom, Casement, Awning, etc)	**CR/CI & U Calc**	**U only Calc**	4	▪ Head ▪ Jamb (1 or 2) ▪ Sill
	3 If Jambs are the same	2 If Head and Sill are the same		
Two Lite Vertical (Vertical Slider)	5 If Left and Right Upper Jambs are the same, and Left and Right Lower Jambs are the same	4 If Head and Sill are the same	7	▪ Head Fixed or Vent ▪ Jamb Fixed & Vent ▪ Sill Fixed & Vent ▪ Meeting Rail
Two Lite Horizontal (Horizontal Slider, Sliding Patio Door)	7	5 If Head and Sill are the same	7	▪ Head Fixed & Vent ▪ Jamb Fixed & Vent ▪ Sill Fixed & Vent ▪ Meeting Rail
Two-lite casement, (Window Wall, Curtain Wall)	4 If Left and Right Head are the same, Left and Right Sills are the same, and Left and Right Jambs are the same.	4	7	▪ Head Fixed & Vent ▪ Jamb Fixed & Vent ▪ Sill Fixed & Vent ▪ Meeting Rail

.

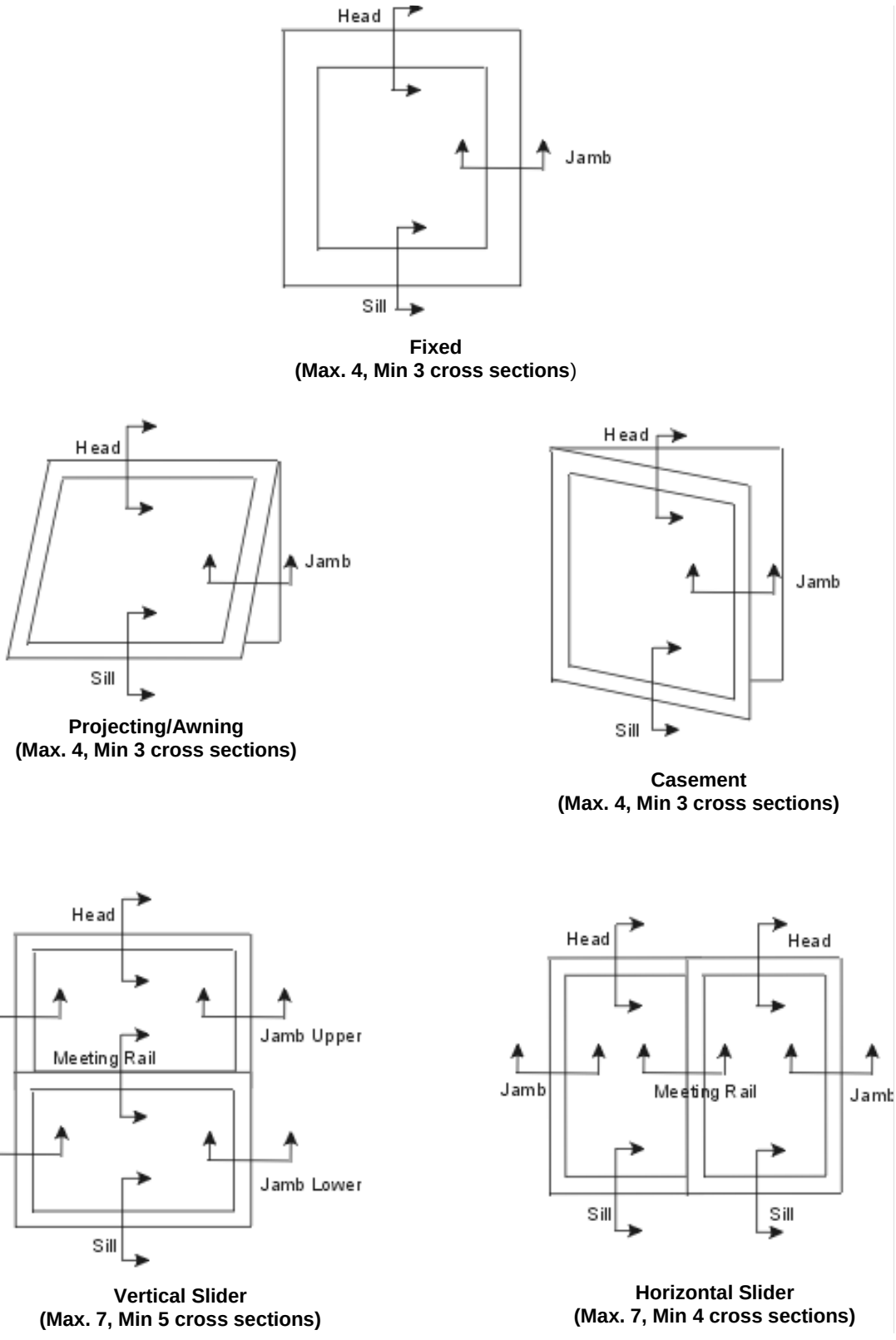


Figure 8-1. Minimum and maximum cross sections to be modeled based on operator type.

8.3. Draw the Cross Sections

After the number of cross sections to be modeled has been determined, draw them in THERM. "Drawing Cross-Section Geometry" in this manual contains detailed instructions about how to draw cross sections.

8.3.7. Getting Started – Drawings and DXF Files

In order to reproduce accurate cross sections in THERM, it is necessary to use dimensioned drawings and/or DXF files. See "Importing a DXF or Bitmap File as an Underlay" in this manual.

If only dimensioned drawings are available (assembly drawings and die drawings), and there are no DXF files, it is still possible to create a cross-section in THERM. See "Drawing Using the Keyboard (Numeric Cursor Positioning)" in this manual.

8.3.8. File Properties – Therm File Properties tab

The File/Properties/Therm File Properties tab allows the user to enter descriptive information about the file.

The THERM version the program was calculated in is displayed.

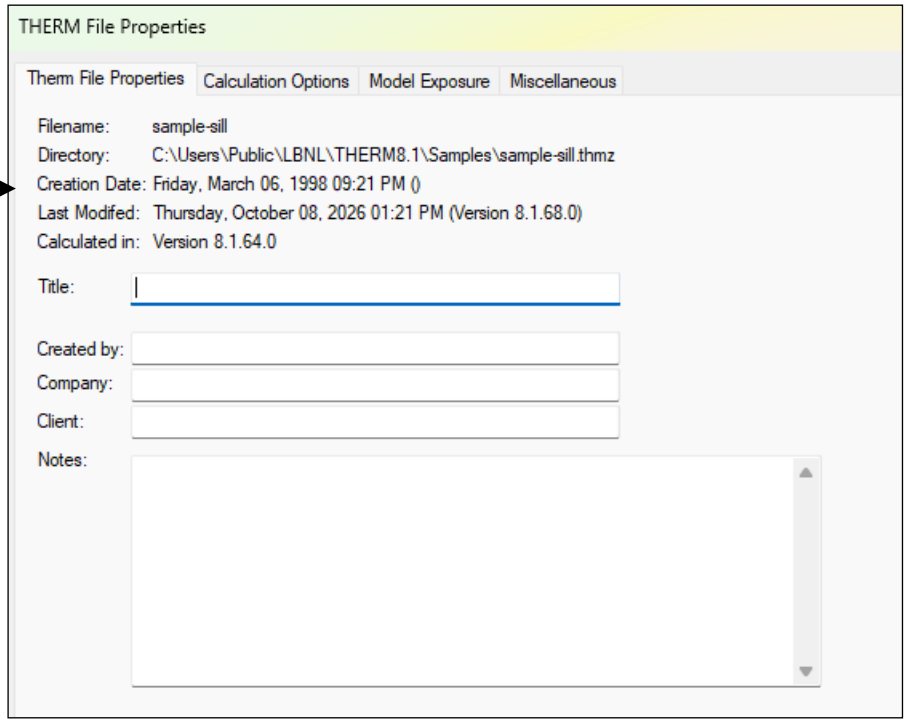


Figure 8-2. The File / Properties / Therm File Properties tab.

8.3.9. Cross-Section Orientation

It is extremely important to specify the **Cross Section Type** in the **THERM File Properties / Model Exposure tab** (accessed from the **File/Properties** menu). The **Cross Section Type**, in combination with the glazing system orientation, will determine how many of the parameters in the the ISO 15099 simulation models are implemented, such as the frame cavity gravity vector. This setting will appear in the WINDOW **Frame Library** in the **Type** field, and it also appears in the lower right-hand corner of the main THERM screen.

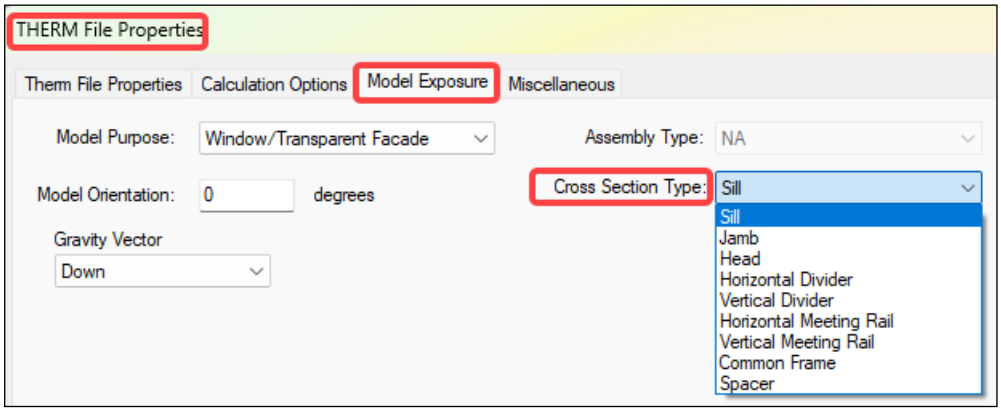


Figure 8-3. The File / Properties / Therm File Properties tab.

Cross sections are usually oriented in a vertical direction, i.e., all the glazing systems should be pointing either up or down, not horizontally. (The exception is for skylights, which are modeled at a 20 degree tilt). When creating the cross section, the frame components may be drawn horizontally if that is easiest (i.e., if the DXF file is drawn that way), but *before importing the glazing system*, rotate the cross section so that the glazing will come in either up or down.

So, for example, jamb cross sections can be drawn horizontally, but they can be rotated into a vertical position for modeling. The figure below shows how a DXF file might be drawn. The horizontal jamb and meeting rail sections can be drawn horizontally as in the DXF file until the point of importing the glazing system, at which time they can be rotated 90 degrees. For heads, the glass should face down, for sills and jambs the glass should face up, with the inside surfaces of the model facing to the right and the exterior surfaces facing to the left.

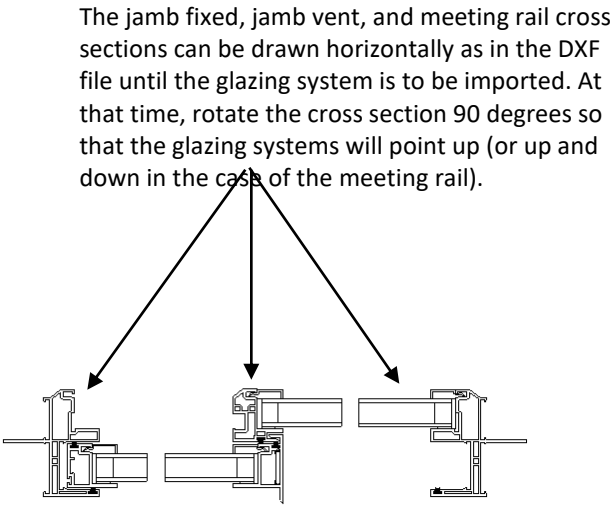


Figure 8-4. DXF file with horizontal cross sections that can be rotated 90 degrees before importing glazing systems and simulating.

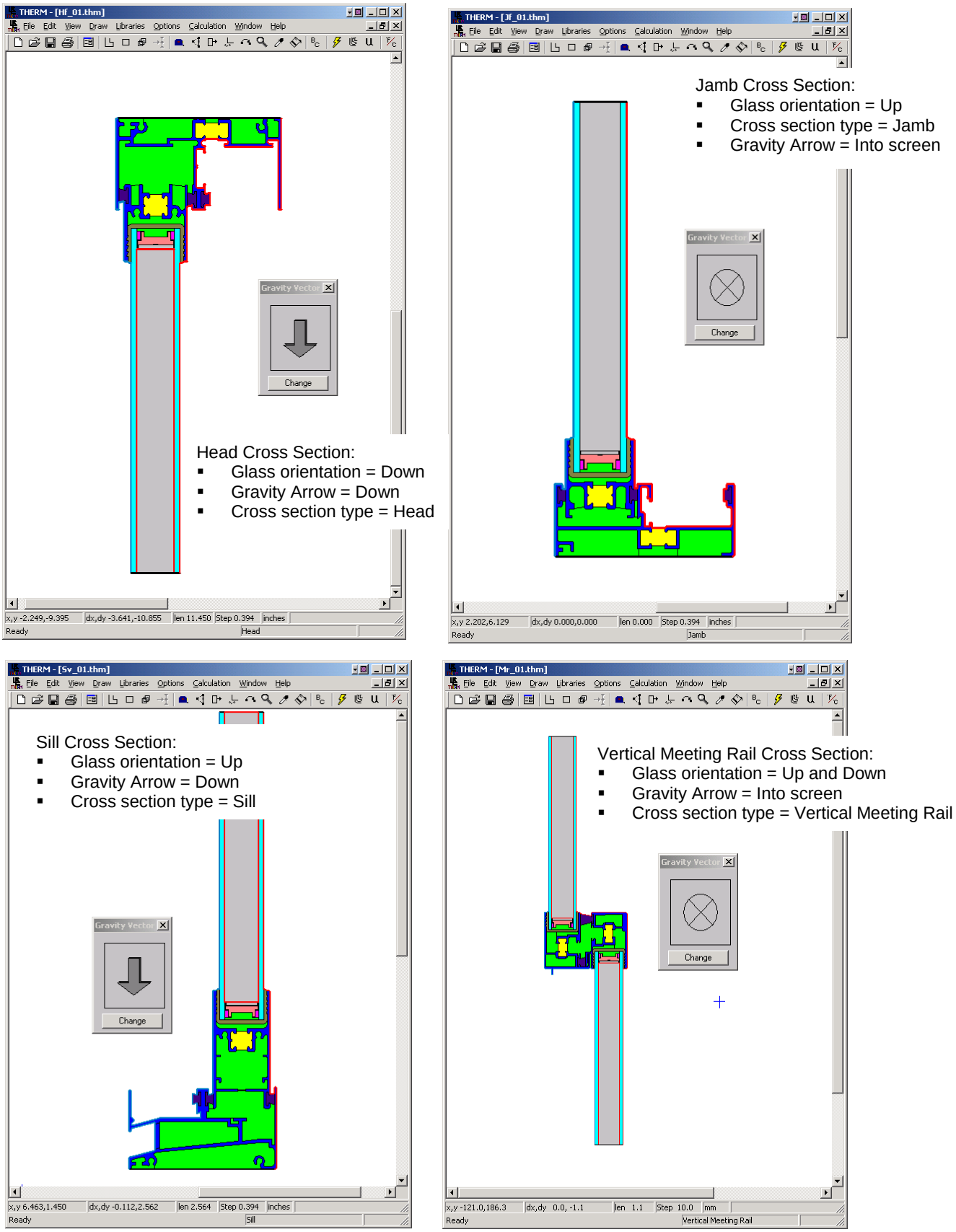


Figure 8-5. Sample cross sections from a horizontal slider showing the cross section orientation, cross section type label in the lower right corner of the screen, and gravity arrow orientation..

8.3.10. What is Not Modeled

Generally, non-contiguous components of a fenestration product are not not modeled.

- Optional Interior trim
- Optional jamb, head, and sill extensions
- Nailing flanges which can be removed from a fenestration product, and which are removed for testing. These may be vinyl flanges on wood windows, or flanges on vinyl or aluminum windows designed to be removed for some installations. Permanent nailing flanges that would be in place during a test should be modeled.
- Hinges
- Locks
- Balances
- Non-continuous Operator Hardware
- Weep Holes
- Setting Blocks
- Shear Blocks
- Corner Keys

8.3.11. Deformable Parts

Because deformable parts, such as glazing clips, weatherstripping and other snap-in parts, are often drawn in the undeformed states in DXF files and assembly drawings (many times in the DXF files they overlap other parts), simulator judgement is still required to ensure that these parts are modeled in a way that results in accurate heat transfer results. For example, there are several instances when small air gaps can be replaced with solid materials, such as in sweeps and seals.

8.3.12. Frame Cavities

There is one default Frame Cavity defined in THERM. It is not defined in the Material Library, but can be selected from the tool bar pull down list as a material to define a specific polygon.

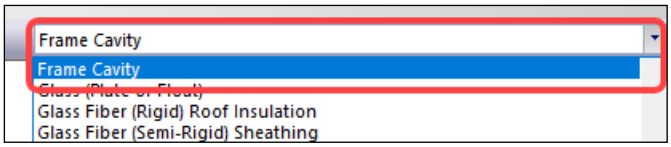


Figure 8-6. A polygon can be defined as a Frame Cavity from the toolbar pulldown list.

For each polygon defined as a Frame Cavity, the Gas and Ventilation type can be specified by double clicking on the polygon to see the Properties for Selected Polygon(s) dialog box. It is also possible to use the Edit / Select Special to select all the polygons defined as Frame Cavities and change the properties in the dialog box for all of the selected polygons at the same time.

- **Gas:** The Gas pulldown list shows the records in the Gas Library
- **Ventilation:** The Ventilation pulldown list contains two options
  - Non ventilated
  - Slightly Ventilated

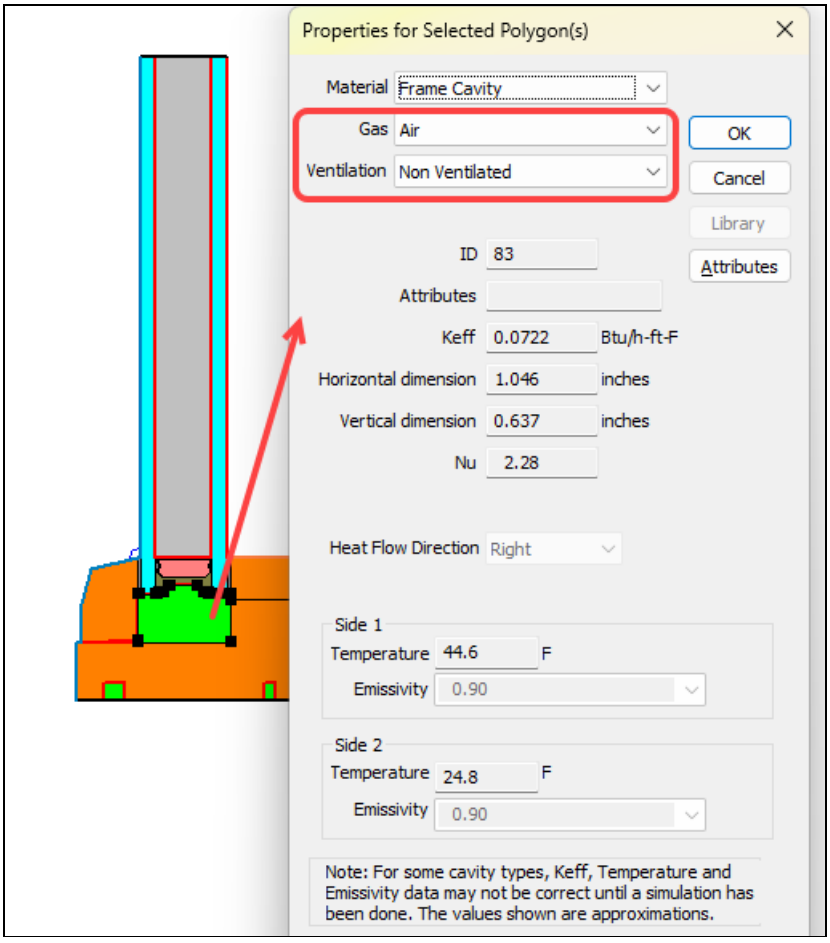


Figure 8-7. For each polygon defined as a Frame Cavity, the Gas and Ventilation characteristics can be selected.

THERM also makes the following assumptions about the frame cavity:

- **Default frame cavity height:** 1 meter. This default is set in Options / Preferences / Preferences tab

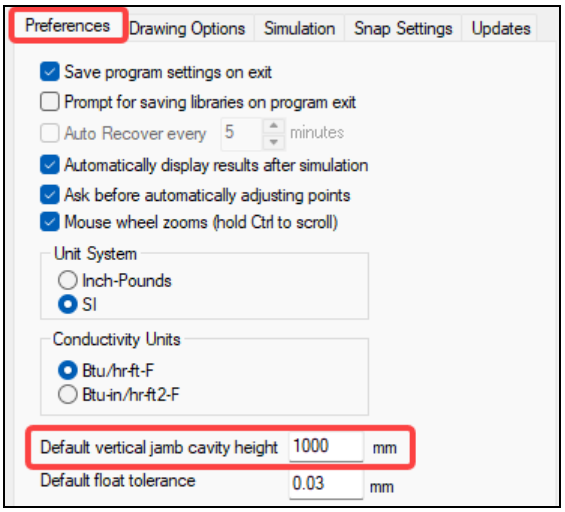


Figure 8-8. The Default vertical jamb cavity height is set in Options / Preferences

If the frame cavity convection model (set in File / Properties / Calculation Options tab) is set to "ISO 15099", which is the default for Frame Cavities, the *emissivities, temperatures and heat flow direction will be automatically calculated by the program during simulation*.



Figure 8-9. The Convection Model is set in File / Properties / Calulation Options tab.

Therefore, the default values that appear for individual frame cavities before the simulation do not need to be edited. These automatic calculations are computed as follows:

- **Heat Flow Direction:** The calculated heat flow direction will depend on the Cross Section Type setting, accessed from the File / Properties / Model Exposure tab. It also depends on the orientation of the glazing system, ie, whether it is imported up, down, left or right. The "gravity vector" can be displayed using the **View/Gravity Arrow** menu option. The assumptions for each type of cross section are the following:
  - **Head:** Cross Section Type = Head, glass imported facing down, gravity vector is pointing down
  - **Sill:** Cross Section Type = Sill, glass imported facing up, gravity vector is pointing down
  - **Jamb:** Cross Section Type = Jamb, glass imported facing up, gravity vector is pointing into the screen
  - **Meeting Rail:** Cross Section Type = Meeting Rail, glass imported facing up and down, gravity vector is pointing into the screen for a horizontal slider and gravity vector is pointing down for a vertical slider.
- **Temperature:** The program calculates the temperatures of the cavity walls based on the surrounding material temperatures, and determines the heat flow direction based on the temperature differences.

It is not necessary to change the initial default cavity temperatures before the simulation. After the simulation, the calculated temperatures can be viewed by double clicking on a frame cavity.

- **Emissivity:** The default emissivities that are displayed in a frame cavity before it is calculated are essentially irrelevant because THERM will recalculate and override them during the simulation of the cross section. However, the frame cavity surfaces of the Frame Cavity polygon will be outlined in red. Double-clicking on one of these red surface segments will display a dialog box that can be edited. This allows the emissivity values assigned by the program to be overridden. The edited values will then be what the program uses for the simulation.
- The program determines the emissivities of the frame cavity walls based on the emissivity values of adjacent materials. If there are materials with different emissivities in the walls perpendicular to the heat flow direction, the program will area-weight the emissivity values. After the simulation, the calculated emissivities can be viewed by double clicking on a frame cavity.

  It is important to remember that the emissivity of a metal surface, such as aluminum and steel, will depend on the surface finish, i.e., painted or unpainted. Many metal cross sections, particularly extrusions, will be painted on the outside, but unpainted on the inside. Remember to change emisstances for surfaces that have a different emissivity than the default emissivity for the material.

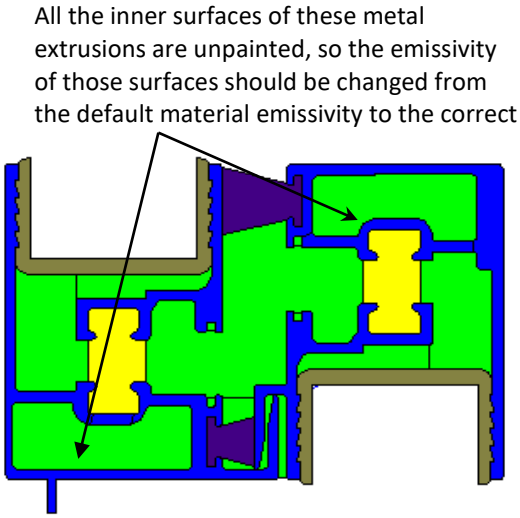
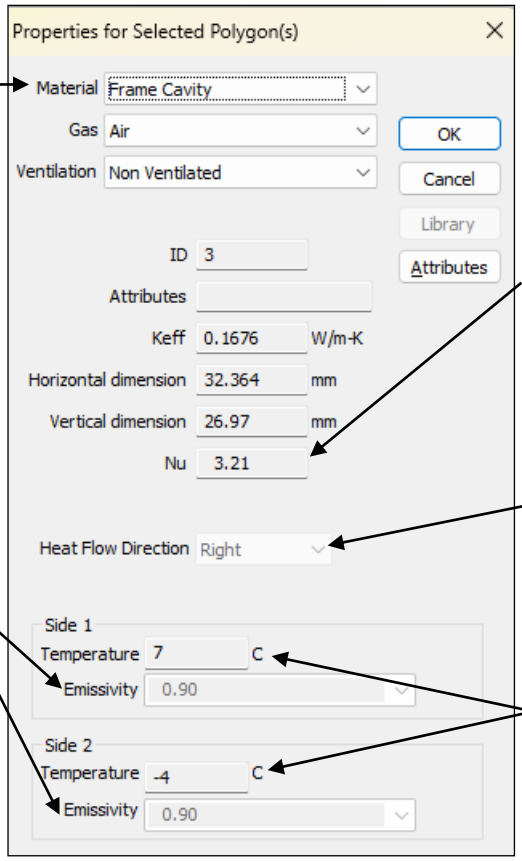


Figure 8-10. Set the emissivity of unpainted metal surfaces, such as the inside surfaces of extrusions, from the material default to the appropriate emissivity for each surface.

When the cross section has been simulated, double click on a frame cavity to view the resulting heat flow direction, temperatures, and emissivities automatically calculated by the program, as shown in the figure below.

The information in this dialog box is from a frame cavity in a THERM file that has been simulated, so the values for Nusselt (Nu), Heat Flow Direction, Temperature and Emissivity (for Side 1 and Side 2) are not the default values, but have been automatically calculated by the program.

Select the "Frame Cavity" choice from the pulldown list. Because the **Cavity Model** for that material is set to "ISO 15099", the heat flow direction, emissivities, and temperatureswill be



The **Nusselt number** (Nu) is calculated *before* the simulation based on default temperatures, but will be *recalculated* based on the actual cavity temperatures during the

The **Heat Flow Direction** is automatically calculated by the program; therefore it is not necessary to check it for every cavity before the simulation

The **Temperatures** are automatically calculated by the program.

The emissivity values are associated with the materials adjacent to the cavities, and they are automatically calculated by the program during the simulation. If there are materials adjacent to the cavity which have different emissivities, the program will area-weight the final values, as in this example.

Therefore, it is not necessary to check the emissivity of every frame cavity before the simulation because what is displayed are default values that are not particularly

Figure 8-11. Select the Frame Cavity choice from the Material Library that matches the emissivity for both sides of each frame.

THERM bases the convection in the frame cavity on rectangularizton of the cavity according to ISO 15099 specifications. The current modeling recommendation is to break up cavities that are separated by a constriction, or "throat", less than or equal to 5 mm (0.20 inches).

- A throat is defined as any place in a frame cavity where two walls, two points, or a wall and a point in the cavity are at or within 5 mm (0.20 inches) of each other.

However, if a frame cavity has a Nusselt number less than or equal to 1.20 before simulating, it is not necessary to break it up into smaller cavities. The Nusselt number can be displayed by double clicking on the frame cavity, which displays the **Properties** dialog box. The horizontal and vertical dimensions of the bounding rectangle, which determine the Nusselt number, are also shown in this dialog box. When there is a constriction in a cavity, there will, in effect, be two separate air flows, separated by the constriction. This is the logic behind breaking up a cavity at the 5 mm throat constrictions. Certain cavities, even when there is a constriction in the cavity, will already have a small amount of convective heat transfer, which results in a Nusselt number close to 1.0. These cavities do not need to be separated into multiple cavities because the constriction is not altering the air flow.

Note: Before a file is calculated, the Nusselt number is based on the default temperatures and emissivities for the frame cavity. There is a slight chance that the Nusselt number of a frame cavity before a calculation is less than or equal to 1.20, and after the calculation is greater than 1.20. In this case, the frame cavity does not need to be broken up.

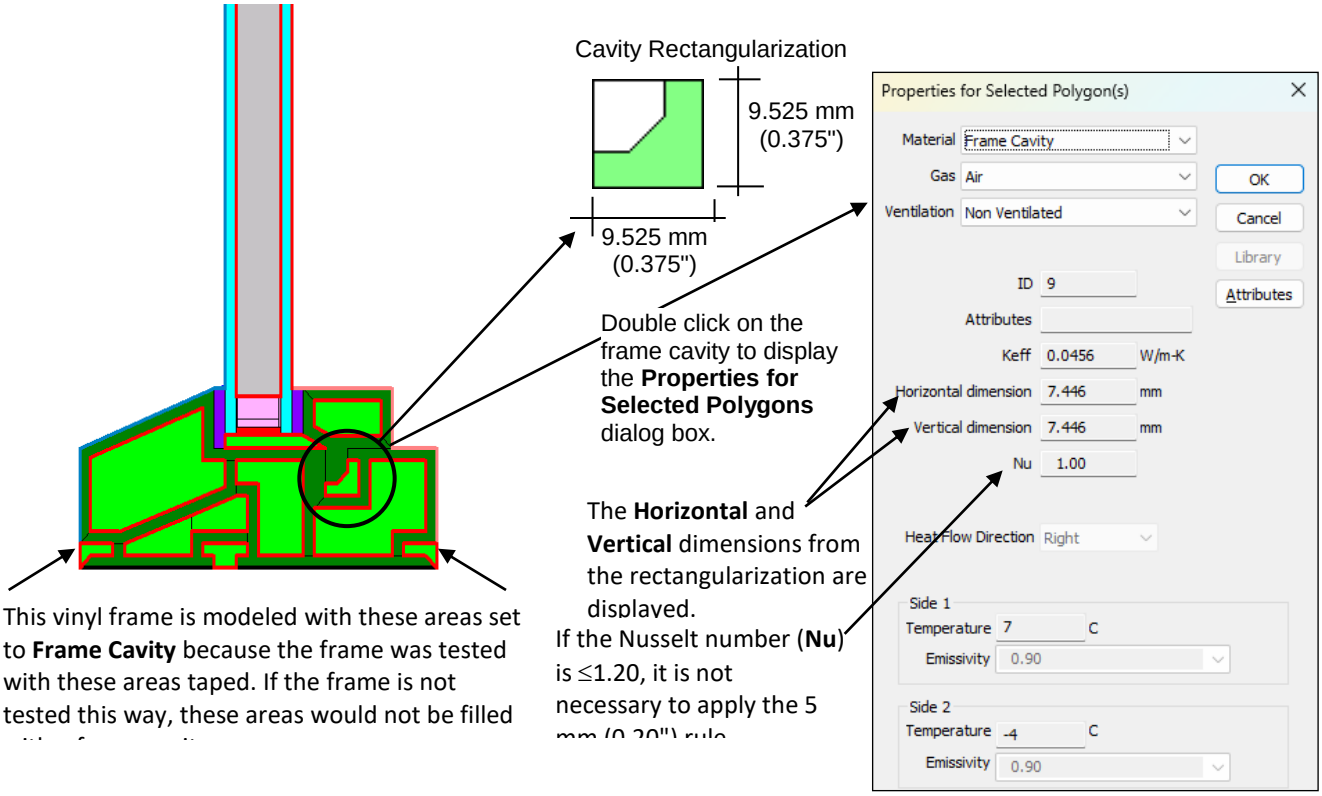


Figure 8-12. THERM uses the ISO 15099 rectangularization specification to model the thermal characteristics of cavities.

The best method for determining whether a frame cavity with a 5 mm (0.20") throat will have to be divided up, and how to divide it if necessary, is the following:

1. Verify that there are throats (constrictions) in the frame cavity less than or equal to 5 mm (0.20 inches). If there are no throats, do not break up the frame cavity. If throats are present, check the Nusselt Number of the undivided frame cavity. If the Nusselt Number is less than or equal to 1.20, the simulator should not break the frame cavity into smaller polygons.
2. If the Nusselt number of the undivided frame cavity is greater than 1.20:
    a. Working from the largest part of the frame cavity to the smallest:
        i. Identify all horizontal throats (constrictions) in the frame cavity that are less than or equal to 5 mm (0.20 inches) and break-up the larger frame cavity into smaller frame cavities.
        ii. Check the Nusselt number of the remaining large cavity.
            a) If the Nusselt number is <= 1.20, no further frame cavity division is necessary.
            b) If the Nusselt number is > 1.20, go to (iii)
        iii. Repeat step (a) for vertical / diagonal throats.

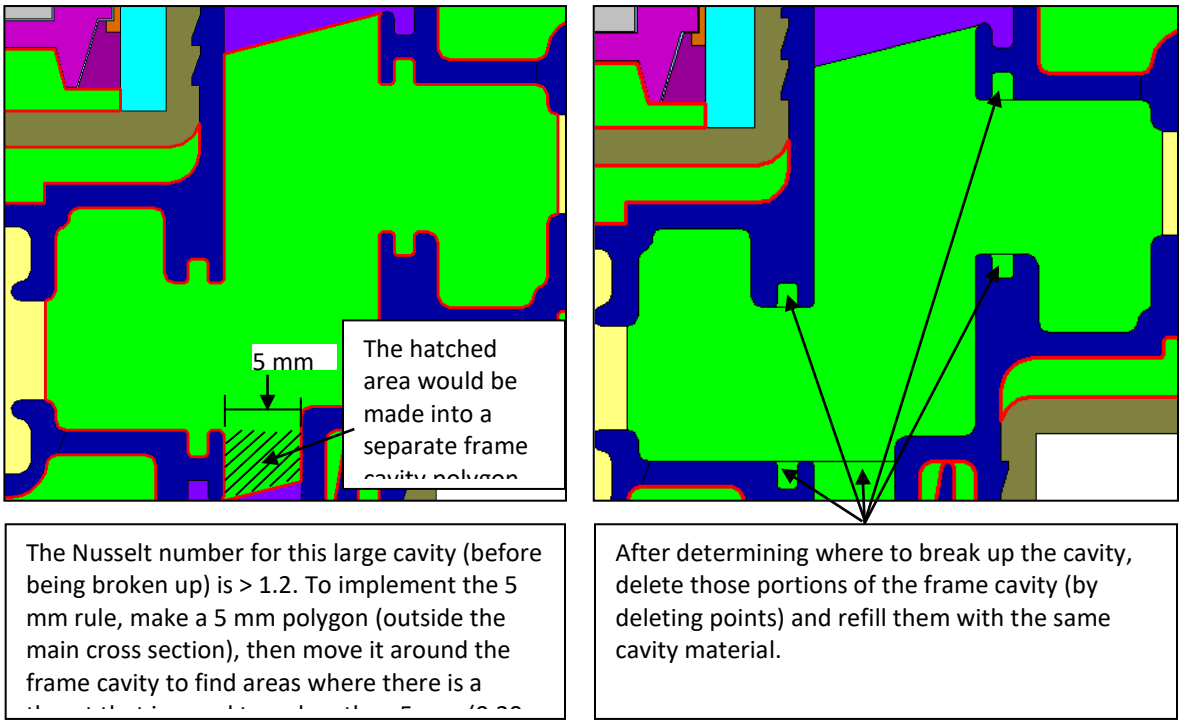


Figure 8-13. Breaking up frame cavities.

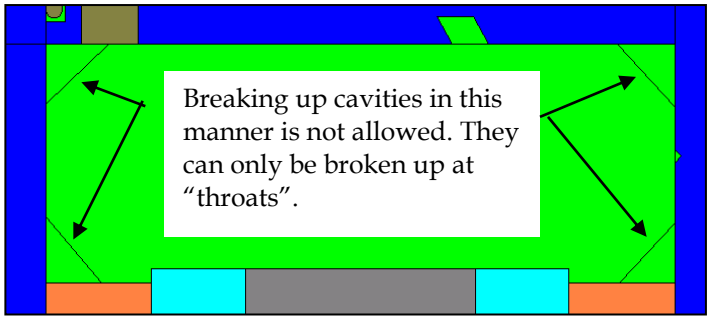


Figure 8-14. Breaking up frame cavities in this manner is not recommended.

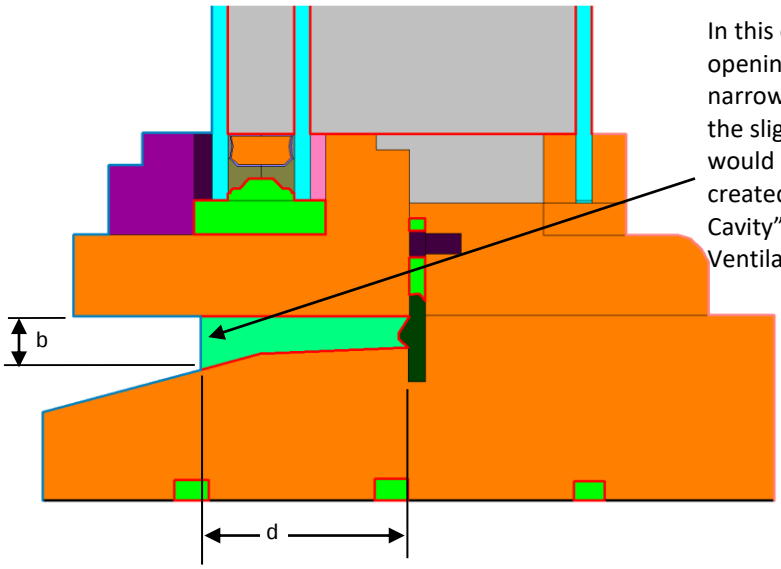
8.3.13. Slightly Ventilated Exterior Cavities

Air cavities that are open to the exterior within a frame section, such as the air cavity under the sash of a casement window that is vented to the exterior, are modeled according to *ISO 15099* which states that cavities greater than 2mm but equal to or less than 10 mm are modeled as slightly ventilated air cavities. The figure below illustrates this.

A cavity open to the exterior environment should be modeled (creating a polygon) only when d ≥ b.

If 2 mm < b ≤ 10 mm, the cavity should be modeled as "Frame Cavity" with the "Slightly Ventilated" choice selected

For cavities ≤ 2 mm, use "Frame Cavity"



In this example, where the opening to the outside narrows to 10 mm is where the slightly ventilated cavity would start – a polygon is created and defined "Frame Cavity" with the "Slightly Ventilated" choice selected.

Figure 8-15. Slightly Ventilated exterior cavities.

If there are openings ≤ 2 mm that open into a frame cavity defined as "Frame Cavity" with the "Slightly Ventilated" choice selected, these small cavities are defined as the standard "Frame Cavity".

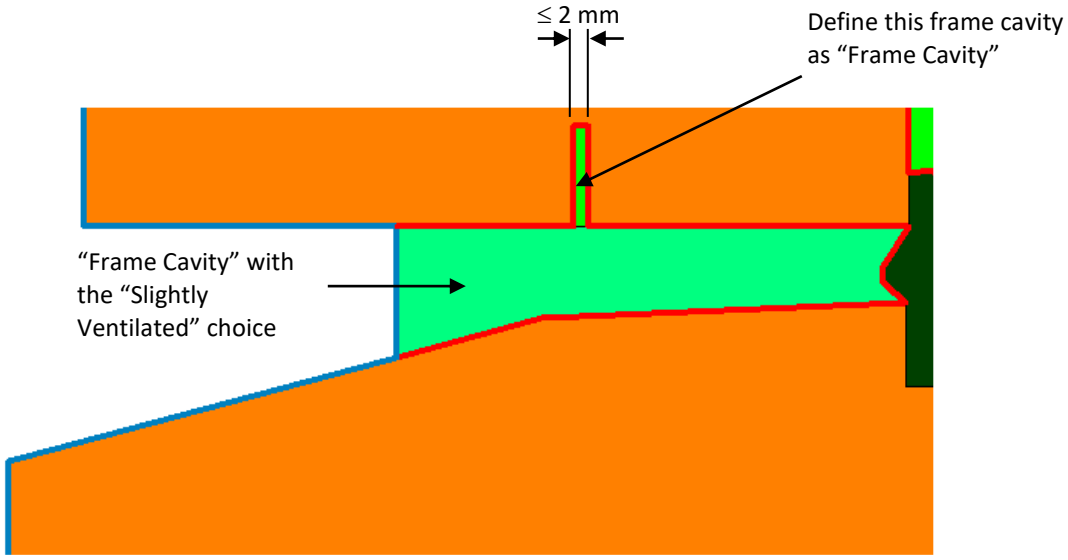


Figure 8-16. Small cavities adjoining slightly ventilated cavities.

If during testing, the product is sealed at the inside surface with tape, those cavities should be simulated with the standard "Frame Cavity" material.

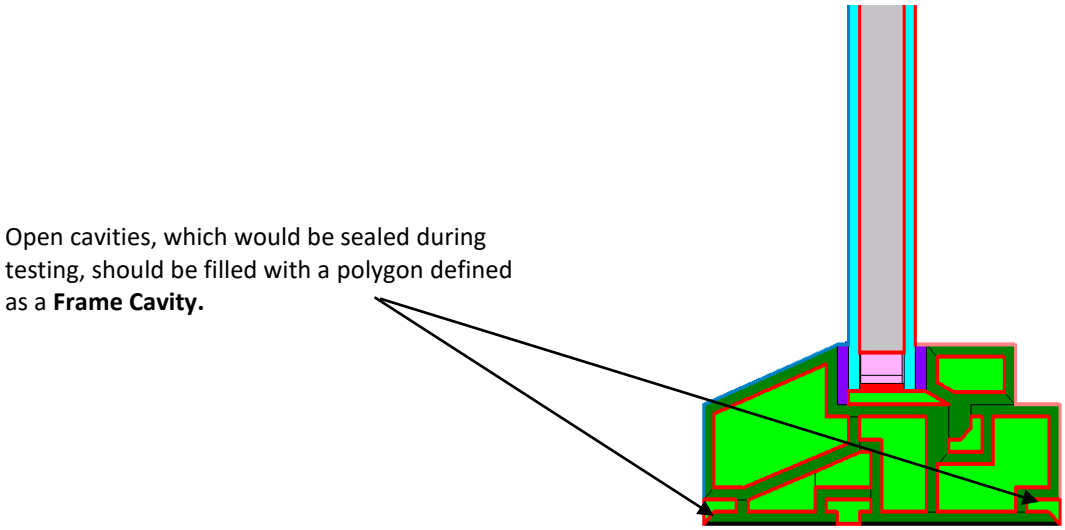


Figure 8-17. Cavities open to the interior are assigned the Frame Cavity option in the toolbar pulldown list.

8.3.14. Modeling Sloped and Curved Surfaces

Because the exact geometry of sloped surfaces can be modeled in THERM, there is no need to change the boundary conditions for sloped surfaces.

For curved surfaces, THERM approximates curves with line segments. The number of line segments used in the curve will determine how close the final model is to the original geometry of the product. When importing a DXF file, the number of line segments is set using the **Arc to Polygon** setting in the **Options** menu. See Section 5 of this manual for a detailed discussion of this procedure. In general, an **Arc to Polygon** setting of 45 degrees for small rounds and fillets in extrusions will give great enough accuracy without introducing unnecessary detail. For larger curved details, all points on the represented line should be within 5 mm of the actual line or curve. The averaged distance (for all points) between the represented line and the actual line or curve should not be greater than 2.5 mm (ISO 15099 Standard).

8.3.15. Modeling Sloped Sills

A product with a sloped sill should be modeled in the same way that it was tested (if it is tested). When a vertical sliding or fixed window is tested, the air space underneath the sloped portion of the sill is filled with expanded polystyrene insulating material. For the simulation of the sill section this window should have the insulating material modeled underneath the sloped portion of the sill as well, as shown in the figure below. The bottom surface of the insulating material will have an adiabatic boundary condition applied. The surface of the insulating material that faces to the interior will have a boundary condition applied that is the same as the condition applied to the frame above the insulating material. The U-factor surface tag for the interior facing surface of the insulating material will be defined as **Frame**, which ensures the correct projected frame dimension will be used in the area weighting of the total window U-factor.

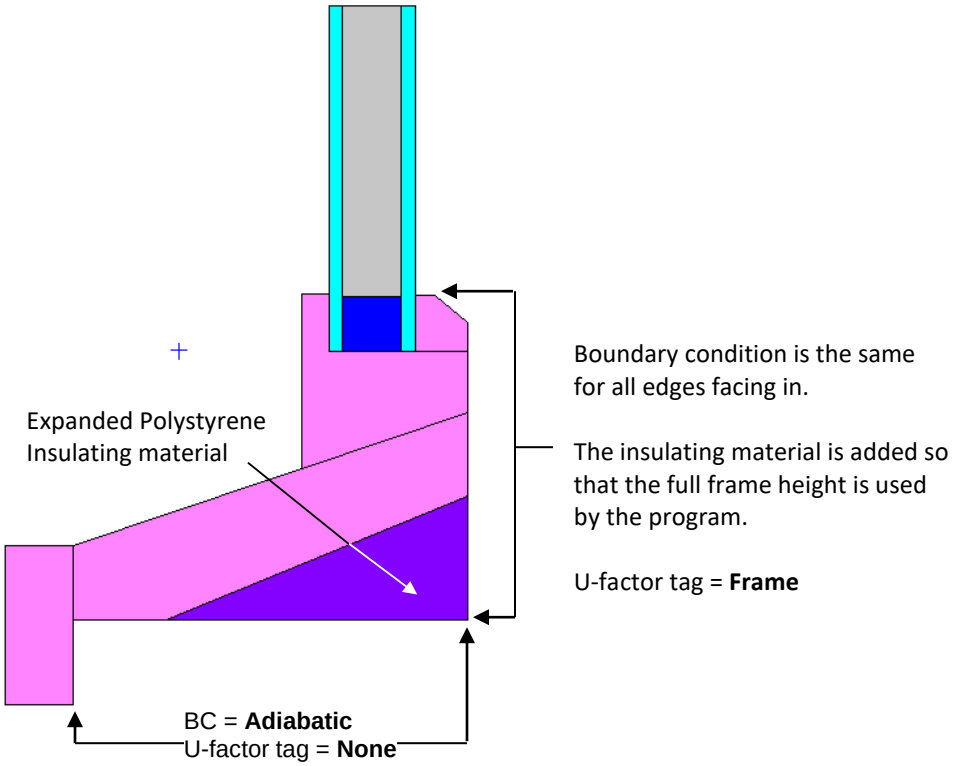


Figure 8-18. Model the sloped sill as it was tested, with insulating material under the sill.

8.4. Importing Glazing Systems

When the frame cross section has been drawn, and rotated into a vertical orientation if necessary, the next step is to import a glazing system. Section 5.10, "Inserting a Glazing System" in this manual contains detailed information about this step. The discussion in this manual is added information about glazing systems.

8.4.1. Overview

For whole product simulations, a glazing system can be imported into THERM from the **Glazing System Library** in the WINDOW program. When THERM imports the glazing system from the WINDOW library, it obtains the following information from the glazing system:

- emittance of the glazing surfaces
- the effective conductivity of the glazing cavity
- the interior and exterior boundary conditions of the glazing system

Glazing systems should always be imported into the cross section in a vertical (either up or down) orientation (it may be necessary to rotate the cross section in order to do this). The figure below shows where the **Locator** should be positioned based on the orientation of the glazing system when it is imported. *The orientation of the glazing system determines in part the gravity vector orientation so it is extremely important to model it correctly.*

NOTE: Although in general it is best to rotate and flip cross sections before the glazing systems are imported and boundary conditions are defined, the exception is skylights, which are modeled at a tilt. In that case, import the glazing system in a vertical orientation, define the boundary conditions, and then rotate the entire cross section.

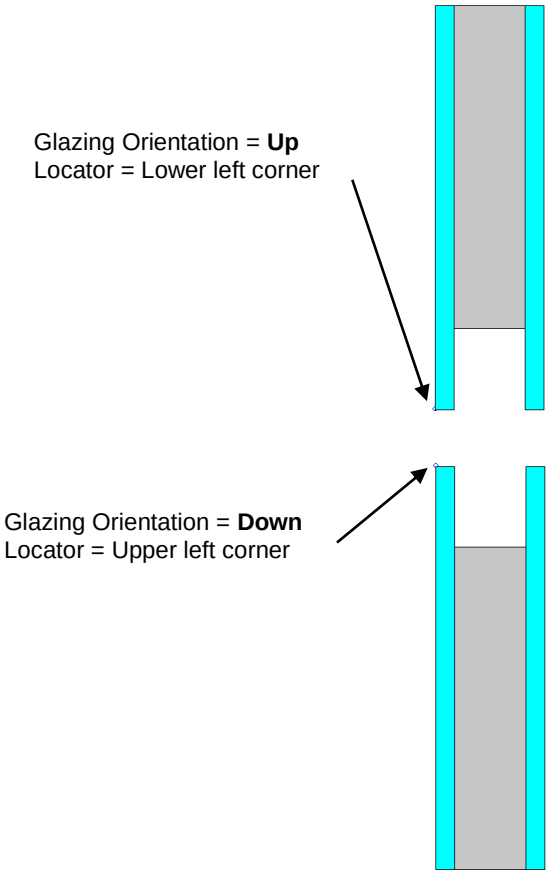


Figure 8-19. The Locator position when importing glazing systems in different orientations.

8.4.2. Inserting Glazing Systems

Section 5 of this manual contains a detailed description about how to insert glazing systems into a THERM file for U-factor and Condensation Resistance calculations. The figure below is a brief overview of the steps.

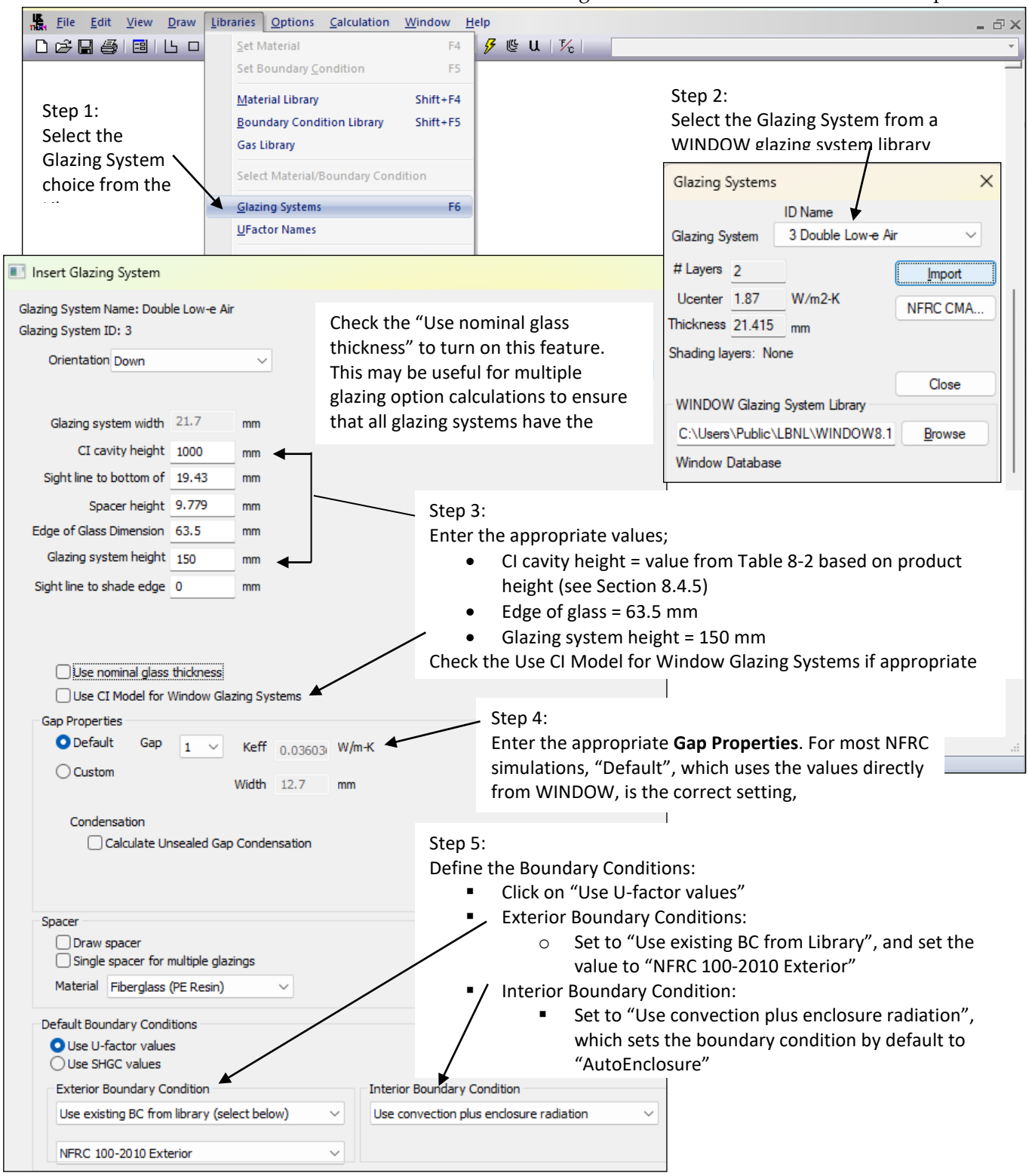


Figure 8-20. Inserting a Glazing System into a cross section.

General modeling assumptions when inserting a glazing system are:

- **CI cavity height:** See Table 8-2, Section 8.4.5 for values. Only used if the Condensation Resistance model is activated.
- **Edge of Glass:** 63.5 mm (2.5 inches)
- **Glazing System Height:** 150 mm (6.0 inches) for all products
- **Exterior Boundary Condition:** Set to "Use existing BC from library" and select "NFRC 100-2010 Exterior". This will be applied automatically by the program on the exterior surfaces of the glazing system and frame.
- **Interior Boundary Condition:** Set to "Use convection plus enclosure radiation". This will cause the program to automatically calculate the boundary conditions for the glazing system to the "Autoenclosure" radiation model, which does not require the Radiation Enclosure geometry to be drawn.

The nominal thickness feature, turned on by checking the "Use nominal glass thickness" when importing a glazing system, can be useful for multiple glazing options, where THERM expects all the glazing system options to have identical thicknesses. This feature is explained in more detail in the *THERM User's Manual*.

8.4.3. Multiple Glazing Options

THERM allows multiple glazing options to be associated with a glazing system. This feature can be useful for simulations where many different glazing systems are to be modeled in the same frame cross section.

To use this feature, follow these steps:

1. **Determine the frame cross sections to be modeled:** The glazing options for a frame cross section should be identical in their geometry, including overall thickness, cavity thickness, spacer height, and sight line dimension. Therefore, the first step is to determine the number of frame cross sections, including dividers and meeting rails, that should be defined in THERM for each set of glazing options

2. **Create and Simulate the Base Case File: For each** frame/glazing option set, create a THERM file with the complete frame cross section and one of the glazing options. Create the boundary conditions for this model, making sure to check the "Use convection plus enclosure radiation" when importing the glazing system, and simulate it to make sure that the geometry is correct. This will be the "base cas**e" file, and will contain no results on**ce the multiple glazing options have been defined and simulated.

3. **Define the glazing options:** There are two ways to define the glazing options in a file:

   - From the **Calculation** menu, select the **Glazing Options** menu choice.

     OR

   - Double click on the glazing system geometry, and click on the **Glazing Options** button in the **Glazing System Info** dialog box.

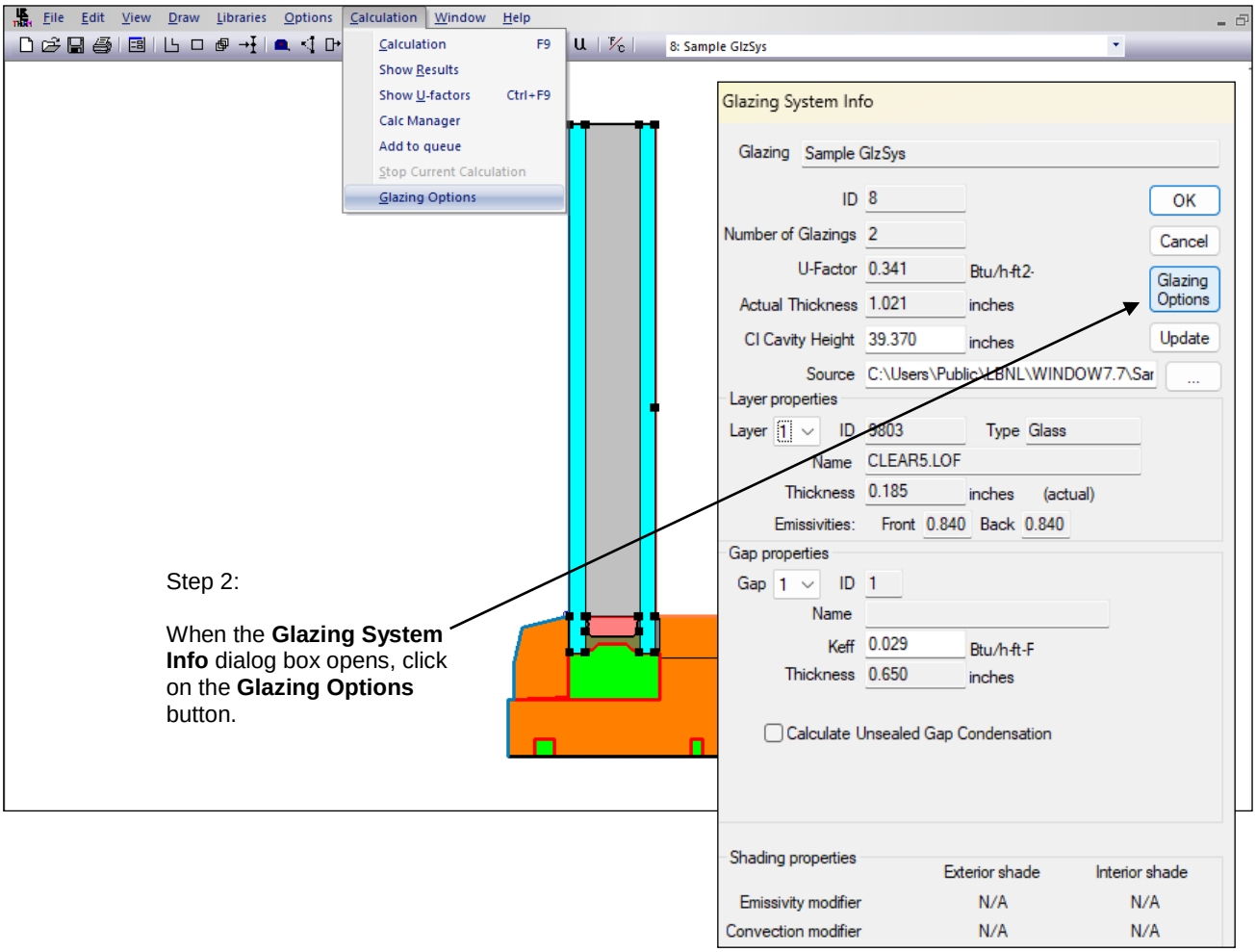


Figure 8-21. Define multiple glazing options.

- **Add glazing options:** From the **Glazing System Options** dialog box, click on the **Add** button to see a list of all the glazing systems in the currently selected **Glazing System Library** that have the same overall thickness and glass layer thickness as the base case glazing system. (Note: Use the nominal glass feature if necessary to make all the glazing systems a uniform thickness.) Click on the glazing systems in the **Add Glazing Options** dialog box that are to be associated with this cross section (use **Shift-click** or **Ctrl-click** to select multiple glazing systems) Click **OK** when all are selected.
  4. THERM will make a separate file for each glazing option, and in the **Glazing System Options** dialog box, click on the radio button choice to determine how the program will automatically name each file. The options are either that the program will append the **Glazing System ID** (as a 3 character number, such as 001, 002) or **Name** to the base case filename.
  5. **Boundary Conditions:** THERM will automatically calculate the boundary conditions for each separate glazing system option.

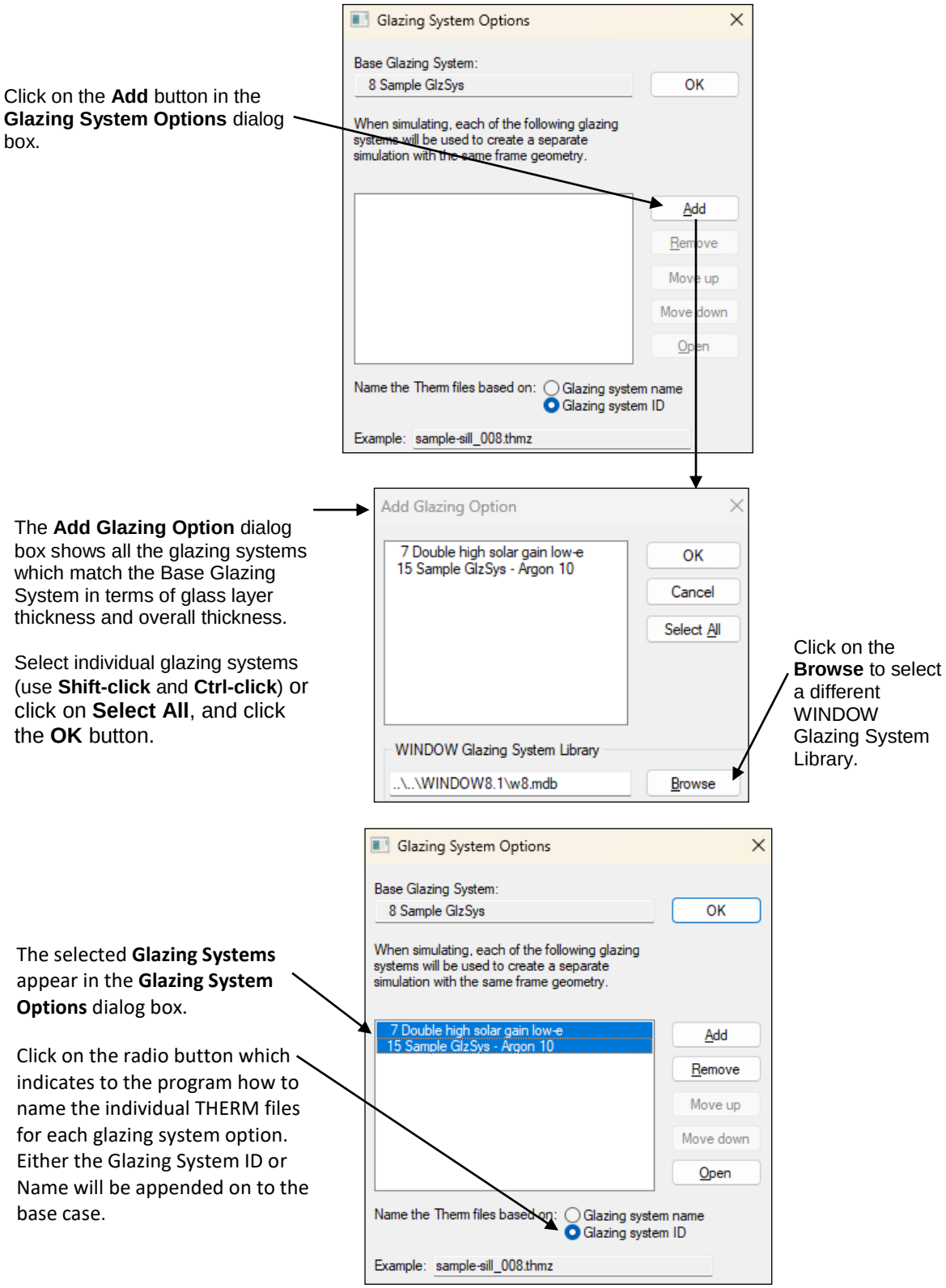


Figure 8-22. Select the glazing systems to be associated with the base case file.

6. **Calculate the Results:** the file can now be simulated, or more files can be created and all the files simulated through the **Calc Manager**. This is explained in more detail in *Section 8.6.4*, "Calculating the Results".

8.4.4. Linking Glazing Cavities

When a glazing system cavity touches another cavity, there are two options in terms of modeling.

1. Link the glazing system cavity to the other cavity (rather than modeling the second cavity as a frame cavity), or
2. Model the non-glazing system cavity as a frame cavity.

The method used to determine how to model this case depends on the geometric relationship of the glazing system cavity to the other cavity. Here are the guidelines:

Un-sealed glazing units
(such as storm panels and Venetian blind panels)

- Apply the 5 mm rule to the "throat" of the cavity next to the glazing system cavity. If it is less than 5 mm, make it into a frame cavity.
- If a cavity touches two glazing systems and the "throat" at each glazing system is > 5 mm, link to the glazing system with the highest Keff (double click on the glazing sytem to see the Keff value for each gap).

Sealed glazing units (IGs):

- The 5 mm rule is not applied to sealed IG units – all the cavities that touch the glazing cavity (such as U-shaped spacer cavities) are linked, even if the throat is less than 5 mm.

Internal Dividers (Grills):

- These rules do not apply to internal dividers, which are only modeled if the distance between the edge of the internal divider and the glass layer is < 3 mm. See Section *8.8. Dividers*.

The figure below shows another example of linking cavities based on whether the "throat" connecting the cavity to the glazing system cavity is less than 5 mm.

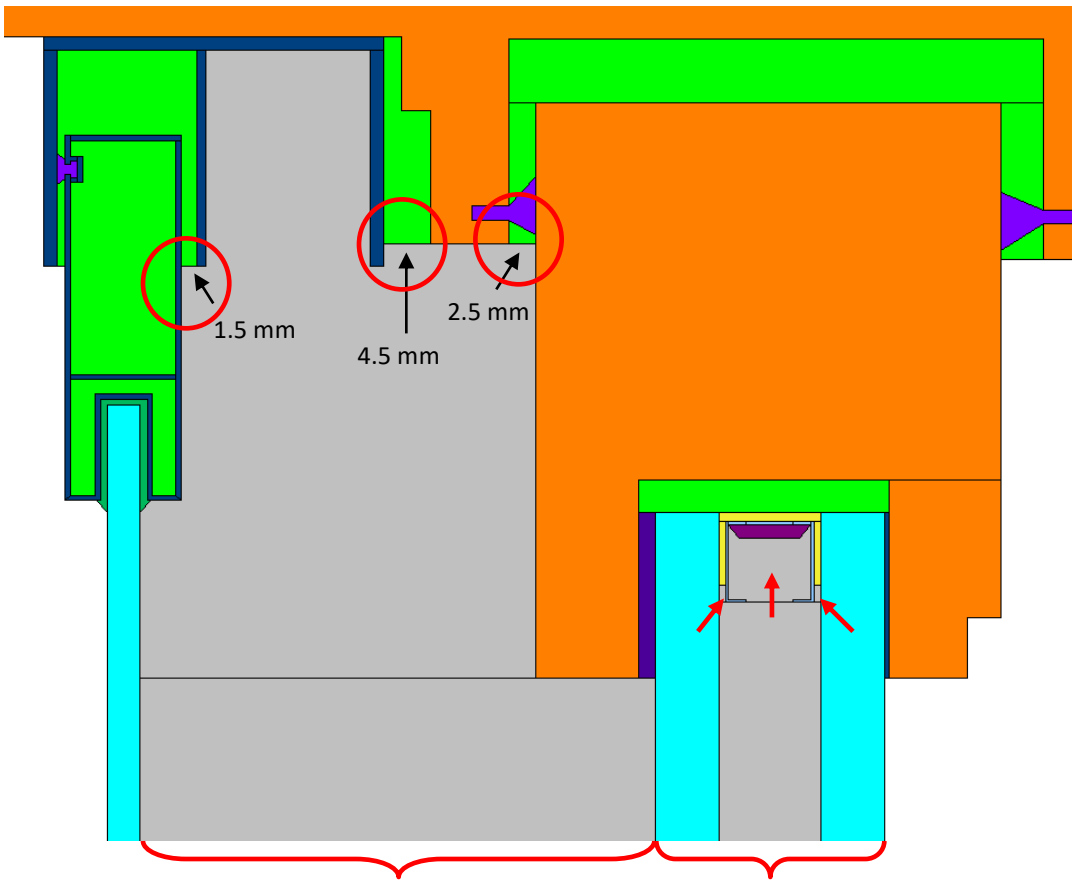


Figure 8-23. Example of a model where the cavities are not linked to the glazing system cavity but are instead modeled as frame cavities.

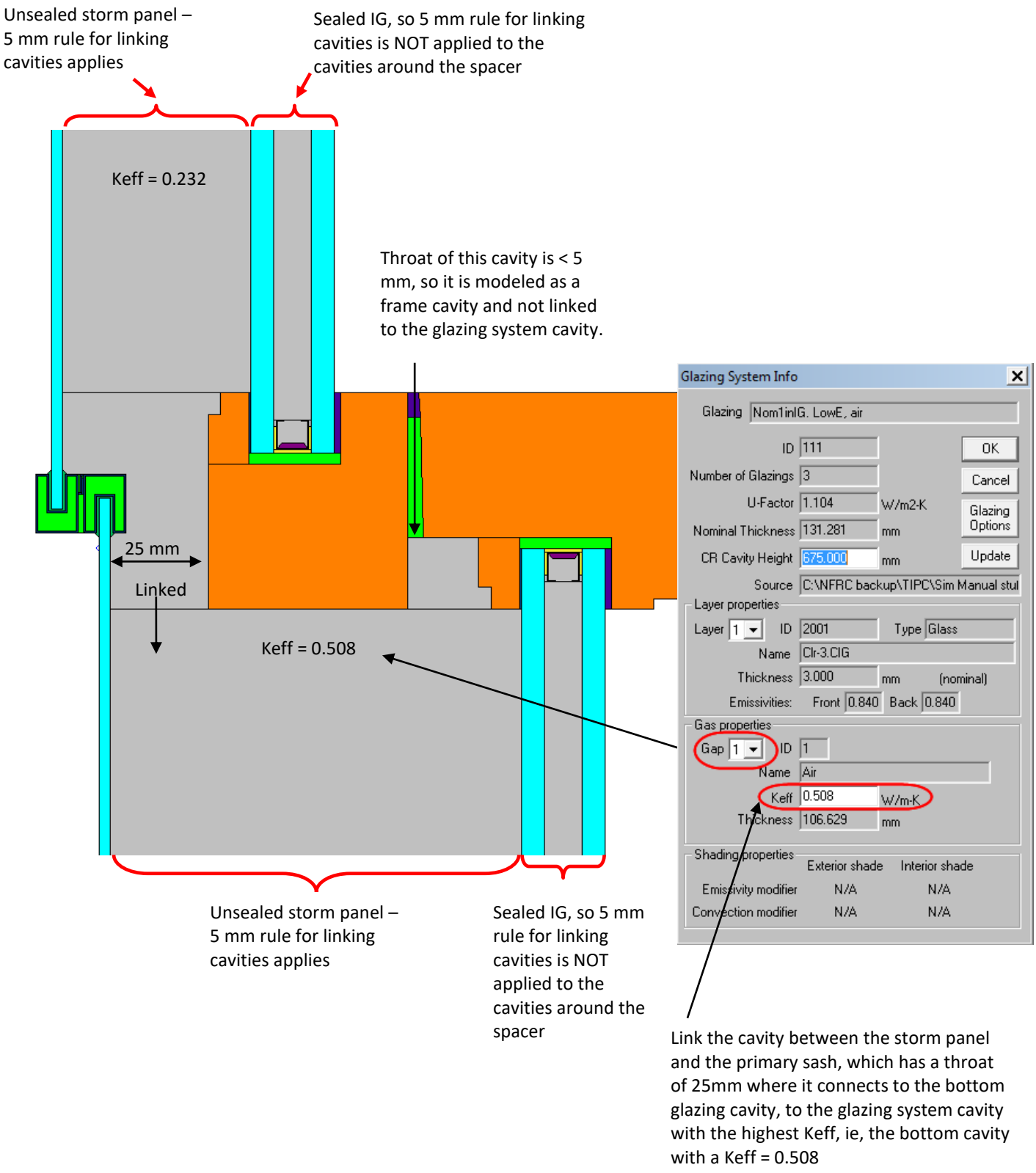


Figure 8-24 Link the cavity between the storm panel and the primary sash, which is > 5 mm, to the glazing system with the highest Keff value.

8.4.5. Condensation Calculation

There are two ways to activate the Condensation model in THERM:

- When importing the glazing system, check the "Use CI Model for Window Glazing System" checkbox.
- In the **File** menu, **Properties** choice, **Calculation Options** tab, check the "Use CI for Glazing Systems"

When inserting the glazing system into the model, as discussed in Section 8.4.2, the table below indicates values that can be used for the "CI cavity height".

If the real product height is not listed in the table below, subtract 100 mm from the product height to use for the default glazing height for condensation modeling. For example, if a product has a height of 2090 mm, the the Actual Glazing Height (CI cavity height in THERM) would be 1990 mm.

Table 8-4. Default Values for Actual Glazing Height Modeling for Condensation Resistance

Real Product Height	Default Glazing Height for Condensation Modeling (input as the "CI cavity height" in THERM)
2000 mm	1900 mm
1500 mm (vertical slider)	675 mm per sash
1500 mm (non-vertical slider)	1400 mm
1200 mm	1100 mm
600 mm	500 mm

Activating the condensation model in THERM will cause boundary conditions to be drawn inside the glazing system cavity, as shown in the figure below. When this model is simulated, THERM will automatically calculate both the U-factor results and condensation temperature data that will be be used, when this profile is imported into WINDOW, to generate the overall condensation value for the whole product. During the simulation, two simulations will appear at the bottom of the screen, the first for the U-factor results and the second for the condensation results.

The condensation model is not used for vertical cross sections, such as a Jamb, a Vertical meeting rail, or a Vertical Divider. Even if the condensation model is activated (i.e., the "Use CI Model for Window Glazing System" is checked) for these Cross Section Types, THERM will not perform the condensation simulation for that file.

THERM generates temperature data from both the condensation and U-factor simulations which WINDOW uses to calculate the frame CI/CR values (shown in the Frame Library Detailed View) and also the whole product condensation values (shown in the Window Library Detailed View). When the condensation model is not appropriate for the whole product condensation calculations, WINDOW will use the temperatures from the THERM U-factor calculation instead, as shown below.

Table 8-5. Temperatures used by WINDOW based on Product Type and Tilt

Product Type (From THERM)	WINDOW 5 Whole Product Tilt (input in Window Library Detailed View)	Temperatures Used by WINDOW for whole product CR/CI calculations
Horizontal elements: Sill, Head, Horizontal Meeting Rail, Horizontal Divider	90	CI/CR
Horizontal and Vertical elements: ▪ Sill, Head, Horizontal Meeting Rail, Horizontal Divider; ▪ Jamb, Vertical Meeting Rail, Vertical Divider	≥ 0, ≤ 20	U-factor
Vertical elements: Jamb, Vertical Meeting Rail, Vertical Divider	90	U-factor
Horizontal and Vertical elements: ▪ Sill, Head, Horizontal Meeting Rail, Horizontal Divider; ▪ Jamb, Vertical Meeting Rail, Vertical Divider	> 20, < 90	CI/CR not calculated

The application of these rules happens when WINDOW performs a calculation in the Window Library. It is up to the simulator to have provided WINDOW with the appropriate simulations in THERM in order for WINDOW to be able to calculate the result. For example, if there are no condensation results for a Sill or Head THERM file modeled in a window with a 90° tilt, then WINDOW will not calculate a whole product condensation value because the needed condensation temperature data does not exist. A message, shown below, will indicate which cross sections are missing the condensation data.

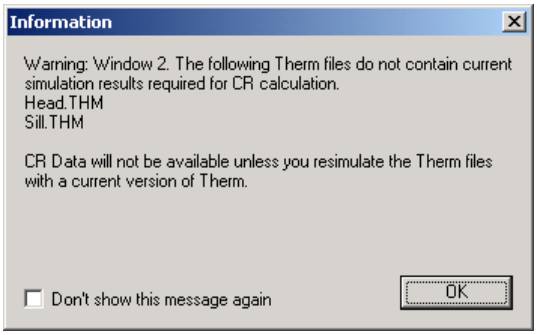


Figure 8-25. WINDOW will display a message if it determines that condensation temperature data is not available from the THERM file.

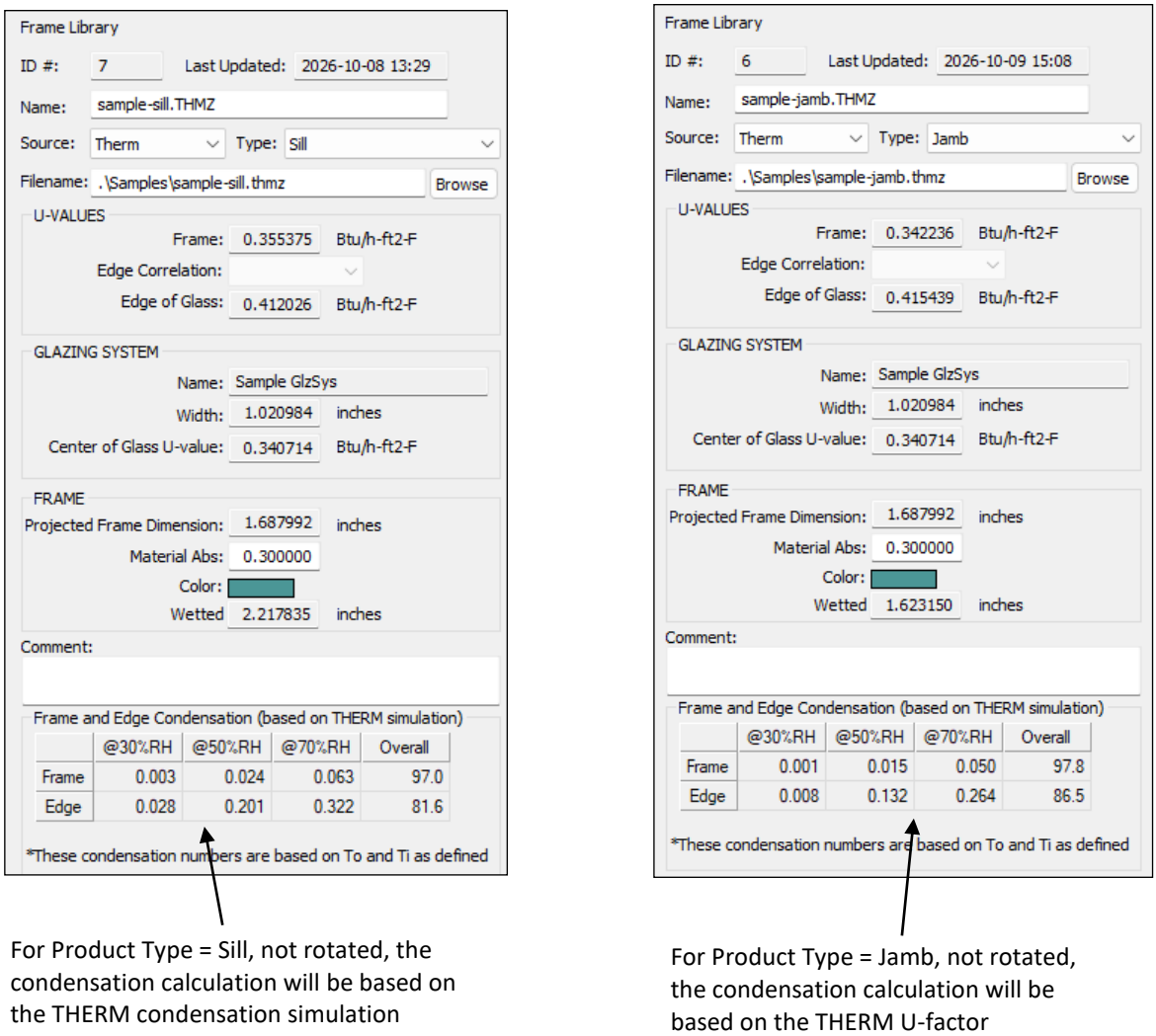


Figure 8-26. WINDOW Frame Library Detailed View displays the type of THERM temperature data to be used with the condensation calculation, either U-fator or condensation.

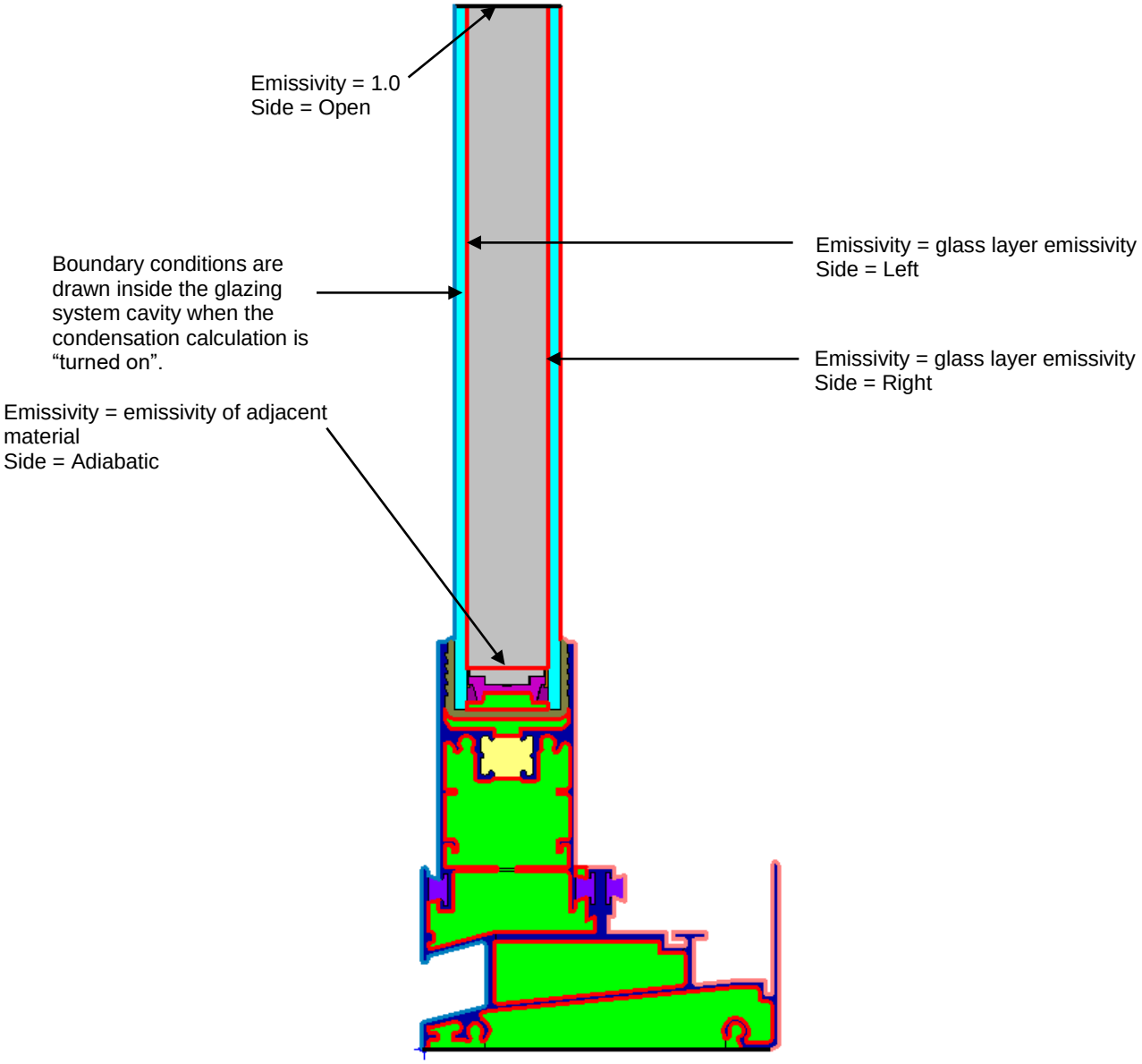


Figure 8-27. Modeling the profile with the condensation model turned on.

8.5. Defining Boundary Conditions

Chapter 6 of this manual contains a detailed explanation of how to define boundary conditions in a model. The information found in this section is supplemental to that discussion.

8.5.1. Overview

Boundary conditions should be defined for all the surfaces on the perimeter of the models, as well as at the surfaces adjacent to a radiation enclosure. These boundary conditions define the temperatures and film coefficients for each element of the perimeter. Different boundary conditions are defined for the surfaces on the interior and the exterior of the cross section. Surfaces which are assumed to have no heat transfer are assigned **Adiabatic** boundary conditions.

There are three main categories of boundary conditions:

- **Interior:** An interior boundary condition is used for all interior surfaces, and assumes that the surface is exposed to natural convection, and the heat transfer coefficient used depends on the temperature of the surface, which is a function of the material.
  - **For Glazing Systems:** The interior glazing system boundary conditions should be set to the boundary condition that is associated with the glazing system imported from WINDOW. (WINDOW calculates the center-of-glazing surface temperatures, and THERM uses these values to automatically calculate the edge-of-glazing boundary conditions for glazing systems imported from WINDOW). At the time the glazing system is imported from WINDOW, set the Interior Boundary Condition pulldown to "Use Convection plus enclosure radiation", which will cause the program to set the glazing system interior boundary condition **Radiation Model** to "AutoEnclosure". THERM will then model the cross section with the radiation enclosure feature without the geometry of Radiation Enclosure being drawn. The boundary condition is named "<filename>:<glazing system name> U-factor Inside Film". Double click on an interior boundary conditions to see its characteristics, as shown in the figure below.

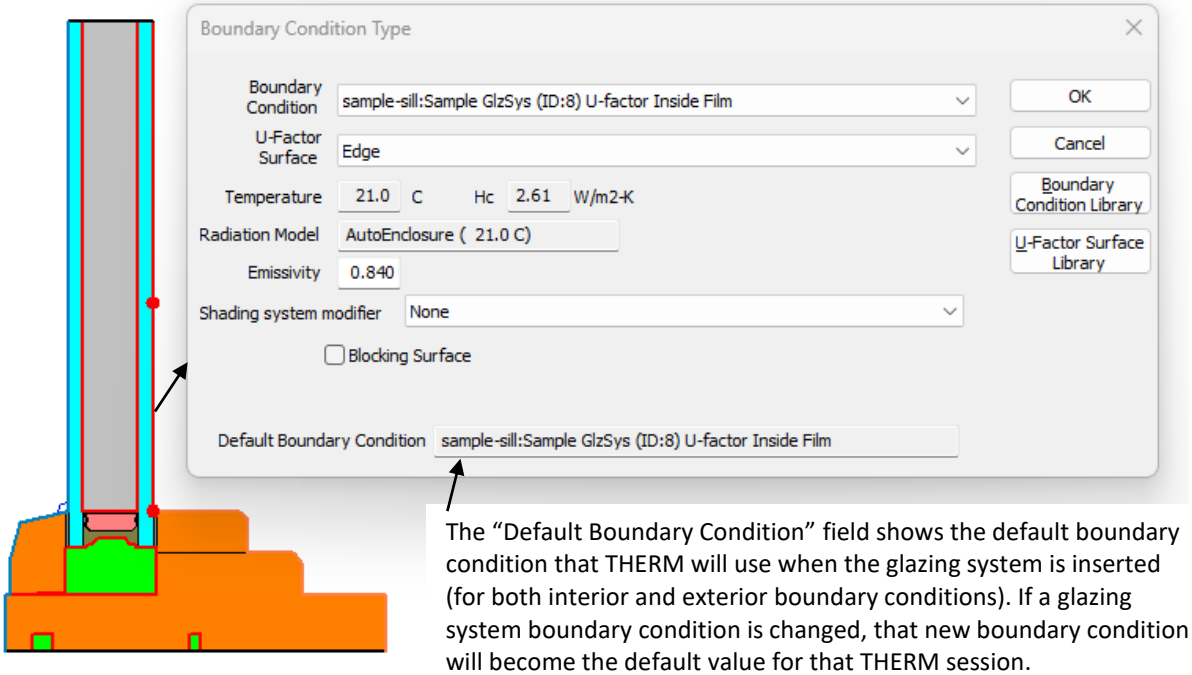


Figure 8-28. Double click on a glazing system interio boundary condition segment to see its characteristics.

- **For Frame Elements:** Set all non-glazing system interior boundary conditions according to the material, from the Boundary Conditions Library. The list below is the default set of boundary conditions (which all have the **Radiation Model** set to "Autoenclosure"), one set for cross sections at a 90° tilt and another set for cross sections at a 20° tilt.**Interior Aluminum F**rame (Convection only). Users can also create their own boundary conditions as needed for specific models.
  - Interior Thermally Broken Frame (Convection only)
  - Interior Thermally Improved Frame (Convection only)
  - Interior Wood/Vinyl Frame (Convection only)
  - Interior (20 tilt) Aluminum Frame (Convection only)
  - Interior (20 tilt) Thermally Broken Frame (Convection only)
  - Interior (20 tilt) Thermally Improved Frame (Convection only)
  - Interior (20 tilt) Wood/Vinyl Frame (Convection only)

    If there are frame materials that fall into more than one category, a single boundary condition can be applied to the entire interior surface of a given frame section, even if it contains different material components (i.e. vinyl sash and aluminum frame). The surface condition applied in this case would be the component with the greatest interior exposed area of that section.

- **Exterior:** The "NFRC 100-2010 Exterior " exterior boundary condition can be used for all exterior surfaces, including the glazing system, and assumes that the surface is exposed to a 5.5 m/sec (12.3 mph) air velocity, which corresponds to a convective film coefficient of 26 W/m^2-°C (4.58 Btu/h-ft^2-°F). Users can create their own boundary conditions as needed for their modeling conditions.
- **Adiabatic:** The adiabatic boundary condition is used for any surface assumed to have no heat flow. This is used for the top of the glazing system at the boundary between the edge-of-glazing and the center-of-glazing, because the assumption is that the heat transfer between the two sections are independent of each other. Adiabatic is also used for the bottom of the frame that would sit in the mask wall of the thermal chamber during testing.

THERM has a **Boundary Condition Library** (accessed from the **Library/Boundary Condition Library** menu, or by double clicking on a boundary condition) which has the standard boundary conditions defined by NFRC, shown in Table 8-6, as well as **Adiabatic**. By default, the exterior boundary conditions are blue, interior boundary conditions are red, and the adiabatic boundary condition is black. Users can create their own boundary conditions in this library as needed to model their specific conditions.

Table 8-6. Boundary condition definitions

Boundary Condition	Radiation Model	Convective Film Coefficient			
		Tilt = 90°		Tilt = 20°	
		W/m^2-°K	Btu/h-ft^2-°F	W/m^2-°K	Btu/h-ft^2-°F
NFRC 100-2010 Exterior	Blackbody	26	4.578	26	4.578
Interior Aluminum Frame (convection only)	Automatic Enclosure Model	3.29	0.579	4.65	0.819
Interior Thermally Broken Frame (convection only)	Automatic Enclosure Model	3.00	0.528	4.09	0.72
Interior Thermally Improved Frame (convection only)	Automatic Enclosure Model	3.12	0.549	4.32	0.761
Interior Wood/Vinyl Frame (convection only)	Automatic Enclosure Model	2.44	0.429	3.09	0.544
WINDOW Glazing System boundary condition <filename>:<glazing system name> U-factor Inside Film	Automatic Enclosure Model	Depends on the WINDOW calculations for the imported glazing system			

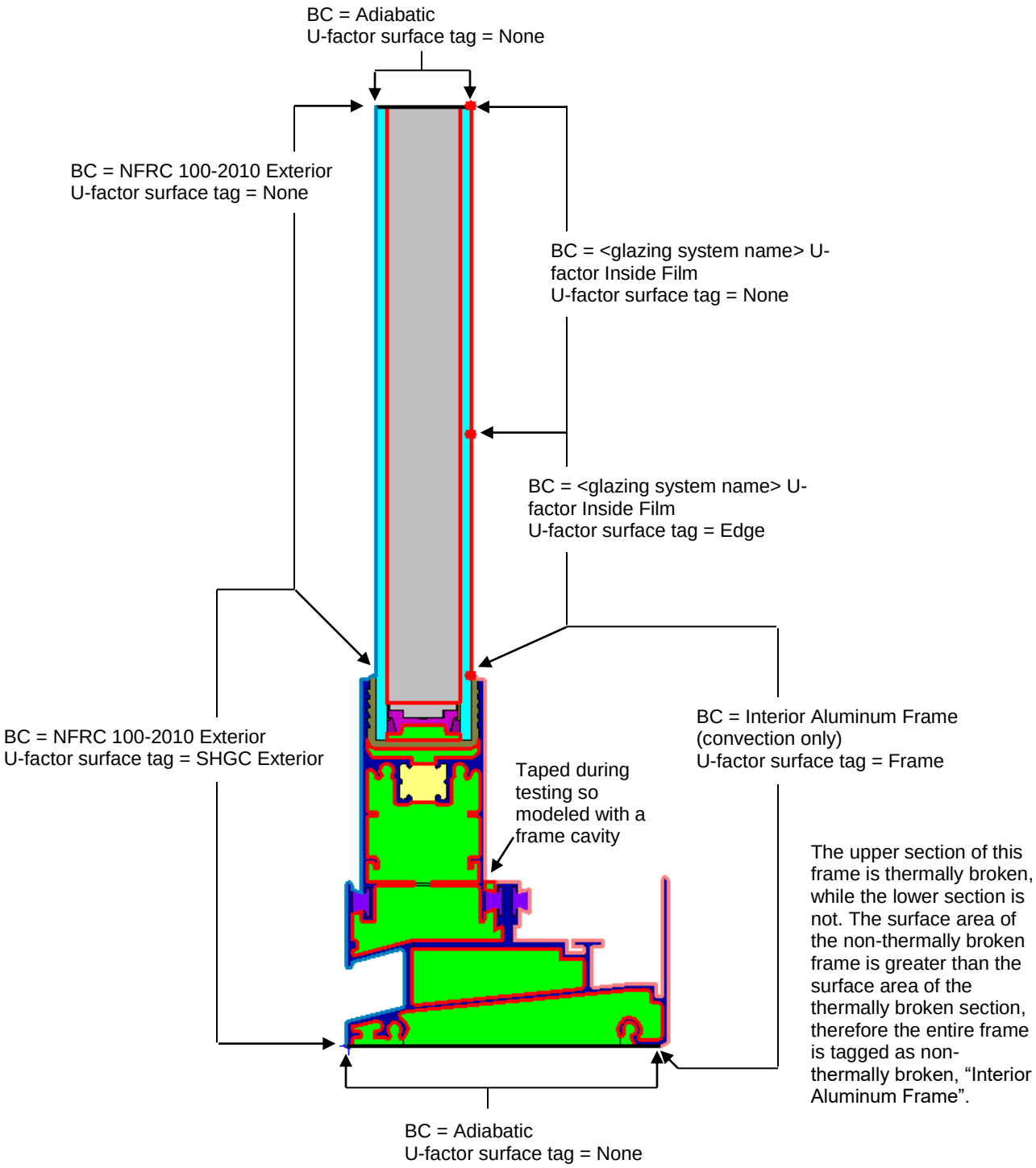


Figure 8-29. Defining the boundary conditions for a cross section.

8.5.2. Assigning Boundary Conditions and U-factor Surface Tags

Boundary conditions for a cross section are created in THERM by pressing the **Boundary Condition** toolbar button, clicking on the **Draw/Boundary Conditions** menu option, or pressing the **F10** key.

THERM will automatically assign boundary conditions, both interior and exterior, to a glazing system imported from WINDOW, as discussed above. However, the frame boundary conditions should be assigned in most cases.

- **NFRC 100-2010 Exterior** for the entire exterior surface of the glazing system and frame
- **Adiabatic** for the bottom of the frame
- **Interior Frame Components:** The appropriate interior convection-only boundary condition for all other interior surfaces from the THERM Boundary Conditions Library.

The technique for doing this is described in detail in Chapter 6, "Defining Boundary Conditions" of this manual.

In THERM, in addition to assigning boundary conditions to a boundary segment, the **U-factor Surface Tags** can also be assigned for each boundary condition. The **U-factor Surface Tags**, which are selected from the **U-factor Names** library, are used by THERM to calculate the component U-factors. For a THERM U-factor calculation that will be used in WINDOW, it is necessary to use the following **U-factor Surface Tags**, with the same capitalization (WINDOW will not recognize any other values):

- **Frame:** Use this tag for all interior boundary conditions that are part of the frame U-factor calculation, including the boundaries of the glazing system below the sight line. THERM will automatically assign the U-factor tag of **Frame** to the portion of the glazing system that is below the sight line, based on the **Site Line to Bottom of Glass** value entered when the glazing system is inserted from WINDOW.
- **Edge:** Use this tag for all interior boundary conditions that are to be used in the edge-of-glazing calculation. THERM will automatically assign the U-factor tag of **Edge** based on the dimension entered in the **Edge of Glass Dimension** when importing the glazing system, which for NFRC modeling should be 63.5 mm (2.5 inches). This dimension will be added above the sight line, defined by the **Sight line to bottom of glass dimension** when importing the glazing system.
- **None:** Use this tag for the 86.5 mm (3.5 inches) of glazing system that is modeled above the Edge-of-glazing, and for the glazing system exterior boundary condition. This is the default U-factor tag automatically assigned by THERM for all surfaces except the glazing system.
- **SHGC Exterior:** Use this tag for all exterior non-glazing surfaces. This tag is used to calculate the wetted length of the exterior frame to be used in WINDOW for the Solar Heat Gain Coefficient (SHGC) calculation.

The **Blocking Surface** checkbox should be checked for frames, but not for glazing systems.

To change the boundary condition for one boundary segment, **double click** on the segment, or **single click** and press **Enter**, and the **Boundary Condition Type** dialog box will appear. This dialog box allows specification of both the boundary condition and the U-factor tag at the same time. To change the boundary condition for multiple boundary segments, click on the first segment and **double click** (or **single click** and press **Enter**) on the last segment in a *counterclockwise* direction. The choices made in the **Boundary Condition Type** dialog box will then be applied to all the selected boundary segments.

The Radiation Model of all interior boundary conditions should be set to “Automatic Enclosure Model”. If for some reason the interior boundary conditions do not have the **Radiation Model** set to “AutoEnclosure” (double click on the boundary condition to open the **Boundary Condition Type** dialog box which displays the setting), they can be easily changed to that model. To change the **Radiation Model**, go to the **Boundary Conditions Library**, either through the **Library** menu, **Boundary Condition Library** choice, or by double

clicking on a boundary condition segment and clicking on the **Boundary Condition Library** button. In the **Boundary Condition Library**, under the **Radiation Model**, click on the "Automatic Enclosure Model" radio button.

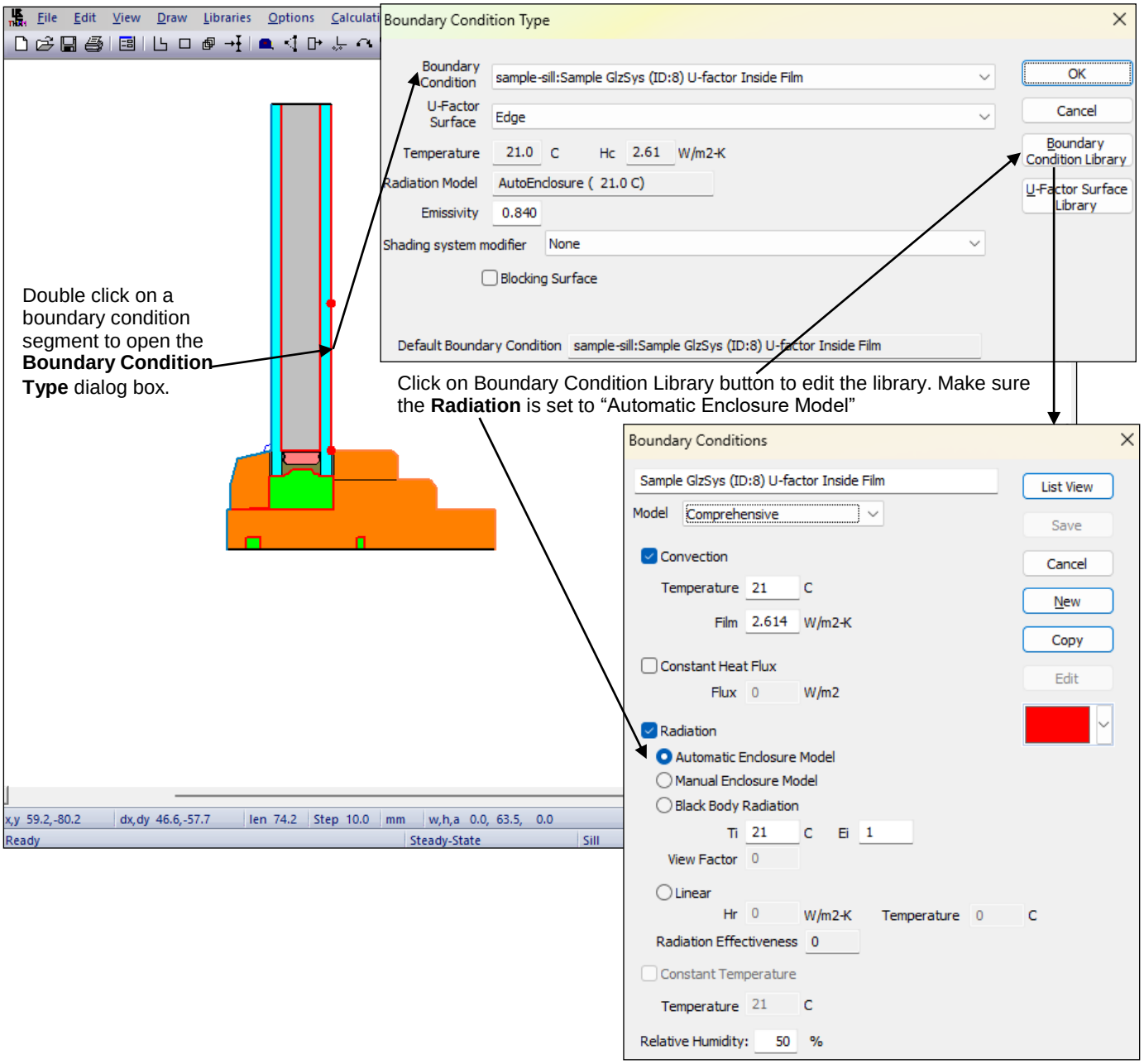


Figure 8-30. Make sure that the interior boundary conditions have set to "AutoEnclosure".

When boundary conditions are generated for a model, the program keeps them in the model until they are regnerated. If the boundary conditions are regenerated, the program will display a dialog box with three choices for how the program should generate them, as shown below.

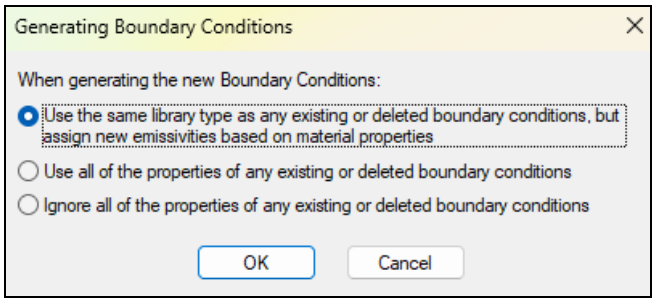


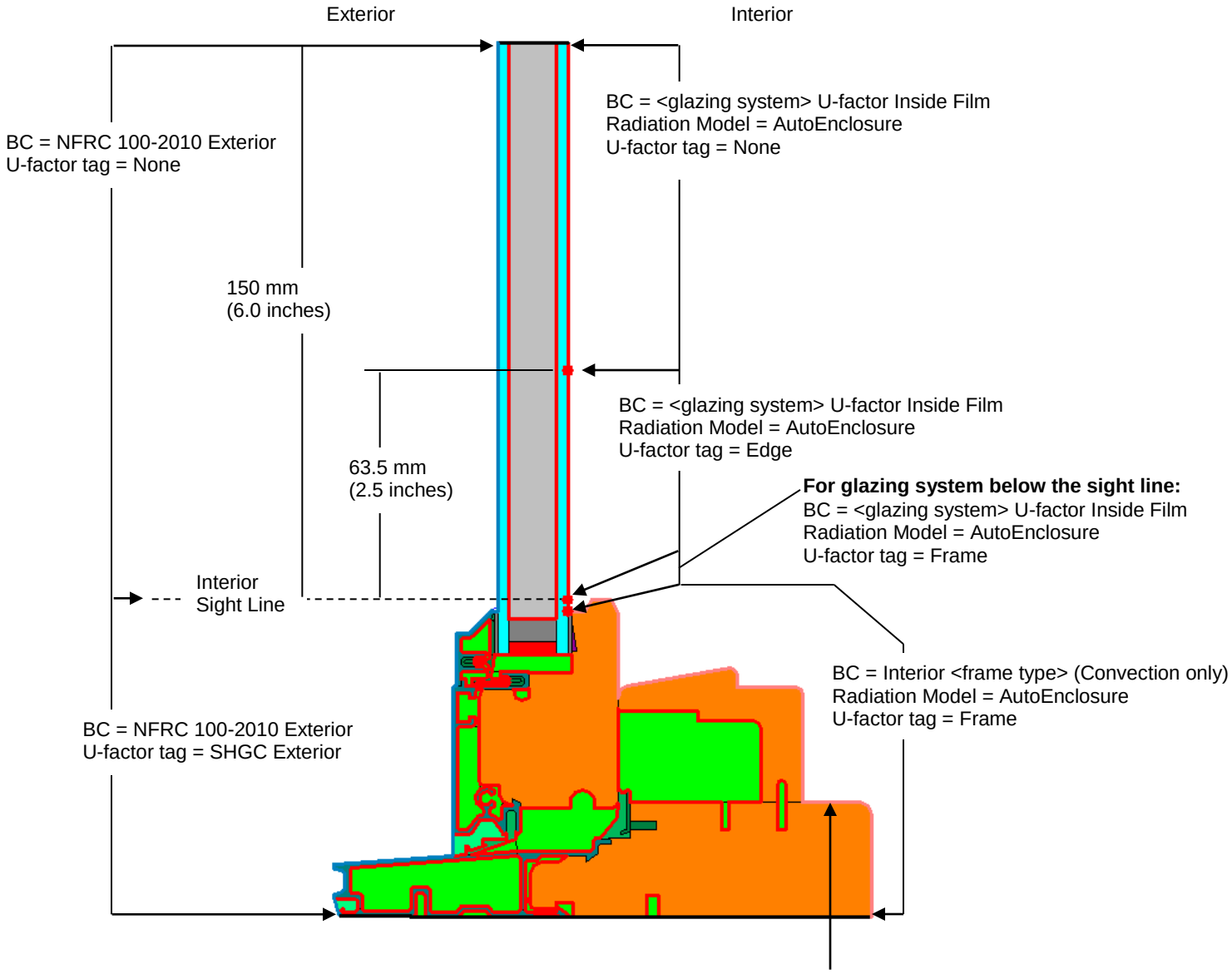
Figure 8-31. The dialog box displayed when boundary conditions are regenerated.

In general, pick the first option in "Generating Boundary Conditions" dialog box, which will pick up the emissivities of the materials in the model. However, do not select this option if you have edited the emissivities of material boundary conditions, such as frame cavities, by hand.

All the boundary conditions can be deleted by deleting one boundary segment.

Boundary conditions and U-factor tags are stored in libraries, and new entries can be defined for both using the **Libraries** menu choice. See *Section 6.2, Assigning Boundary Condition Definitions* in this manual for more detailed information.

Occasionally THERM loses track of the glazing system boundary conditions. If this happens they can be reassigned from the **Boundary Condition Library**.



In order to automatically model the Radiation Enclosure, make sure all interior boundary conditions have the **Radiation Model** set to "AutoEnclosure". This will cause THERM to model the Radiation Enclosure without an enclosure geometry being drawn, and the convection-only film coefficients for these boundary conditions will be calculated and assigned automatically by the program

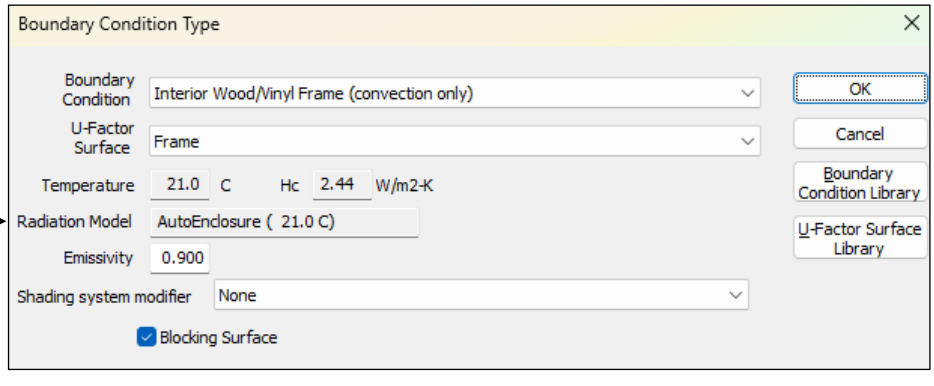


Figure 8-32. Standard boundary condition assignments for most models.

Boundary Conditions for Nailing Flanges

When modeling nailing flanges, assign the "NFRC Exterior" boundary condition, with U-factor tag set to "None", to the portion of the nailing flange that is facing the exterior condition, as shown in the figure below.

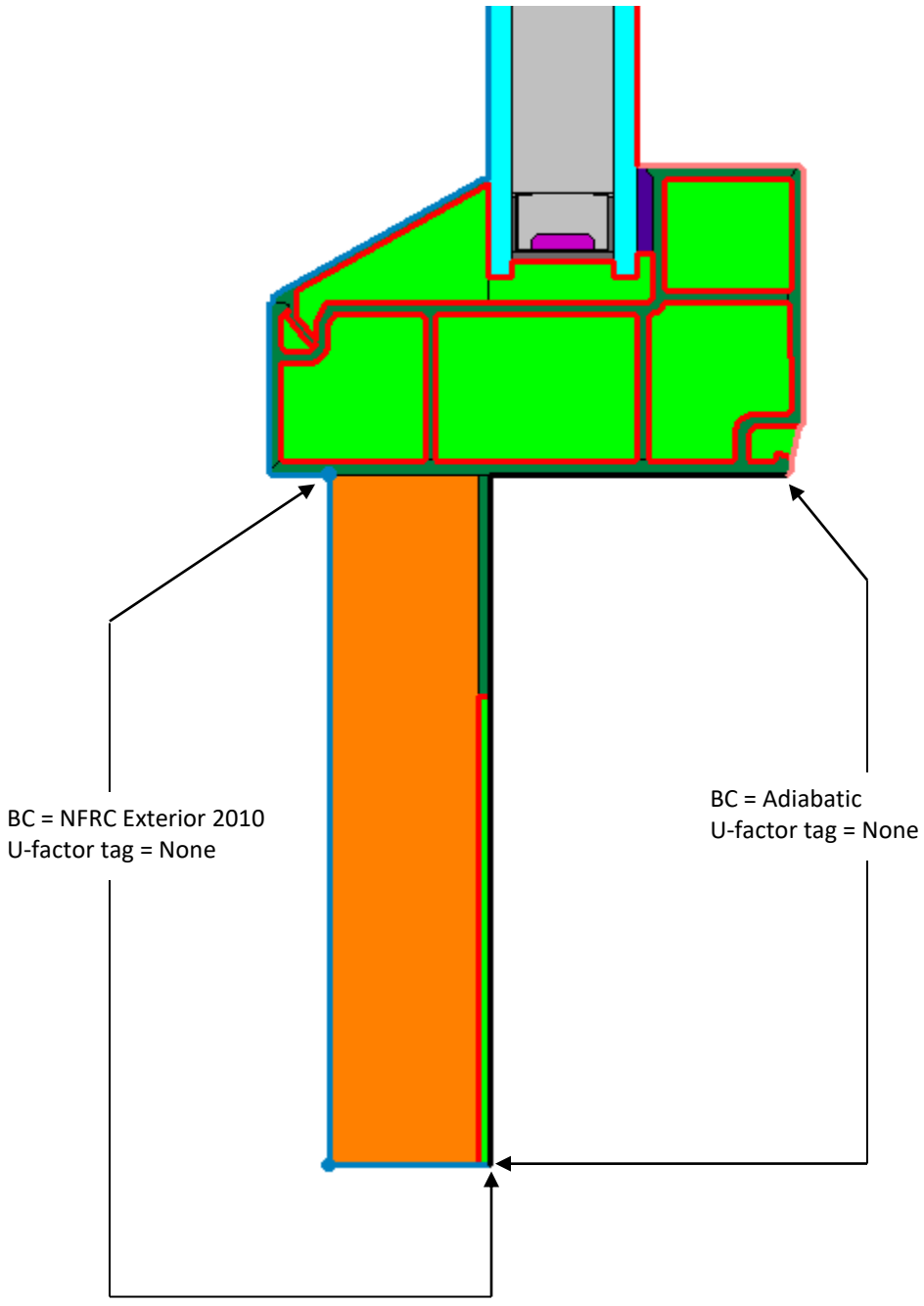


Figure 8-33. Boundary condition assignments for nailing flange.

8.5.3. Voids, Overlaps and Bad Points

In order to perform a simulation, the geometry of the cross section must be correctly defined, i.e., there must not be any voids, overlapping polygons, or points that cannot be resolved by THERM.

While drawing, there are a number of features in THERM to help avoid creating voids, overlapping polygons, and bad points. The **Always Check for Overlapping Polygons** feature (in **Options/Preferences/Drawing Options**) is the main tool used to avoid drawing overlaps and voids. This option can be turned off, but this is NOT recommended for most situations. The **Show Voids / Overlaps** (from the **View** menu) feature is also a good tool for visual inspection to find voids and overlapping polygons in a cross section.

In addition to the features that attempt to prevent the creation of bad geometry during the drawing process, a number of error checks are performed when THERM calculates the boundary conditions in order to make sure the model geometry is correct.

For example, when the boundary conditions are calculated, THERM identifies all points within 0.1 mm (0.004 inches) and 0.01 mm (0.0004 inches) of each other (points closer together than 0.01 mm are automatically merged). A message box appears saying that there are points closer than the program tolerance, as shown in the figure below. In general, select the option **Automatically adjust points within tolerance option**, in which case THERM will fix all of the points automatically. This works even for models with thin films in glazing systems. If you are unsure about whether to let the program adjust the points automatically, select the **Mark the points but don't adjust them** option, which causes THERM to draw red circles in the problem areas. Then you can examine all the circled points and fix them if appropriate, or regenerate the boundary conditions and let the program automatically adjust them. If the marked points are actual details in the drawing that need to be kept, they do not have to be fixed, although they might cause meshing problems during the simulation because of too much detail. Once all the circled areas in the drawing have been examined, press the Boundary Conditions toolbar again – if the program identifies "bad points' but you wish to keep the detail, just click OK and the boundary conditions will be generated. (Having THERM adjust the points is an "all or nothing" proposition -- which is why all the points should be examined first before having the program adjust them. Also keep in mind that the automatic fixing of points cannot be reversed by THERM.)

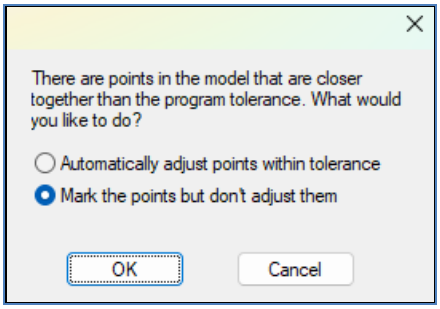


Figure 8-34. Message when trying to define boundary conditions indicating that there is a problem in the geometry.

The best way to fix points that are too close together is to delete or move the offending point. Many times, the point in question is in the adjacent polygon, so it is best to delete the point in all the polygons at the same time by selecting all the polygons using the **Shift** key, then delete the point; it will be deleted from all the selected polygons.

THERM also marks **Voids and Overlaps** by checking for "lonely edges" -- two adjacent edges with a different number of points on each edge. THERM marks voids and overlapping polygons with red circles also. One way to eliminate these two conditions is by "jiggling" the points -- moving the points away from the adjacent polygon, then moving them back within the sticky distance and letting the program "snap" them to the polygon. Another way is to delete the offending polygon, and use the **Fill Void** toolbar button to create it again, which means that THERM will automatically align all points and ensure that no voids are created.

THERM can also find invalid polygons caused by points collapsing on each other, creating a zero-area extension. In this case both the point at the tip of the extension and the extra point at its base must be deleted.

Turning off the **Material Color** and the **Boundary Conditions** from the **View** menu can sometimes help to find the problems.

THERM will simulate a problem even if "bad points" have been marked and intentionally left in the cross section, because in some cases details are drawn that give bad points but which need to be modeled.

If changes are made to polygons on the boundary of the cross section, the boundary conditions must be generated again. If all the corrections are on internal polygons, the boundary conditions will not be automatically deleted. However, in this case, it is still a good idea to press the **Boundary Conditions** toolbar button again, to ensure that all points on all surfaces have boundary segments defined between them.

8.6. Calculating Results

When the boundary conditions have been correctly defined for the cross section, the model can be simulated. Begin the calculation by either clicking on the **Lightning Bolt** toolbar button, selecting the **Calculation/Calculation** menu option, or pressing **F9**.

8.6.1. Meshing

The first step performed by THERM in the simulation is to generate a mesh from the geometry on which the thermal analysis is based. This mesh is generated automatically, and a summary of how it works is found in "Appendix C" in this manual. The latest version of THERM has a new mesher, called the "Simmetrix Mesher" which can mesh much more detailed and complex model geometry than the older "Quad Tree Mesher".

The one variable in the mesh generation that the user can control is the **Mesh Parameter**, which determines how fine the mesh is. The larger the mesh parameter value, the smaller the largest element in the mesh is. The suggested mesh parameter value to start with using the Simmetrix mesher is 10, as shown in Table 8-7.

Occasionally the mesh generator cannot create a closed mesh with the geometry that has been drawn, usually created because of extremely fine detail in the cross section. THERM circles the point(s) where the mesher failed. If this happens, first:

- **Rerun the problem with a higher mesh parameter value.** This causes the program to generate a finer mesh, and results in longer run times. The default mesh parameter value for the Simmetrix Mesher is 8, which the default for the Quad Tree mesher is 6. Mesh parameter values greater than 8 are rarely needed for single cross sections, but may be needed for full height product simulations. Options/Preferences/Simulation has a setting to "Automatically increment mesh parameter". If this option is checked the program will increase the mesh parameter until the problem meshes.

If this doesn't work:

- **Simplify any unnecessarily complicated details.** Keep in mind that the circled point often lies on the boundary between two polygons; the detail that is causing the problem could be anywhere in those two polygons and is often not at the place that is circled. One method to try is to break the problem polygon into multiple polygons, while keeping the geometry otherwise unchanged. Another option is to delete unnecessary points, such as in curves, because each point generates a mesh node.

8.6.2. Error Estimator

THERM has a built-in error estimator, and automatically refines the mesh in the areas where it is needed. The details of the error estimation algorithm are found in "Appendix C" of this manual. The error estimator returns the **Percent Error Energy Norm**, which is related to the gradient of heat flux (energy). If the returned

value is greater than the target value, THERM refines the mesh in areas with a high rate of change in the heat flux. The default value for the Maximum % Error Energy norm in THERM is 10% as shown in

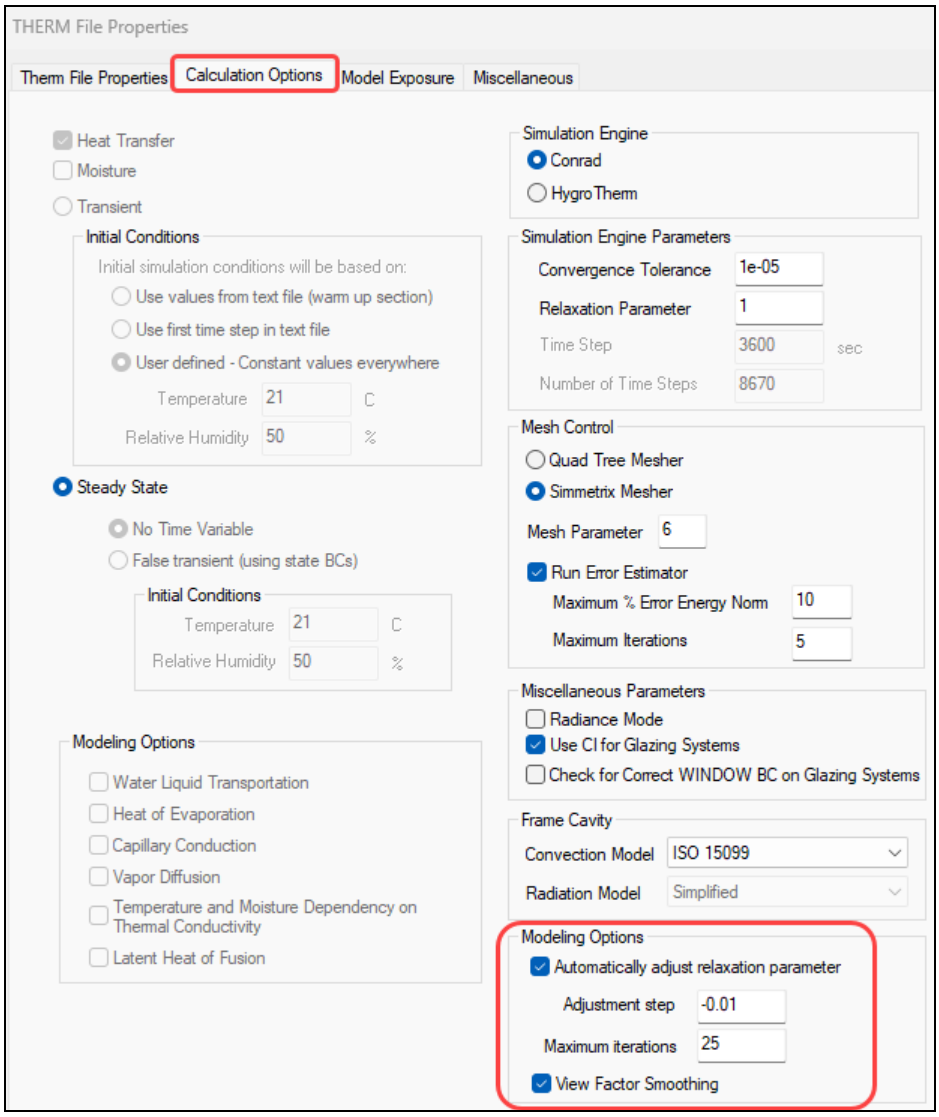


Figure 8-35. The mesh parameter and error estimator are set in THERM File Properties.

This setting forces THERM to do the error estimator calculation and return a value for the error estimate (recorded with the U-factor results) and the program may or may not refine the mesh, depending on the result of the Error Energy Norm. If an error message is displayed saying that the error energy norm cannot be reached, increase the **Maximum Iterations** value (set by default to three iterations). An Error Energy Norm of less than 10% is equivalent to an uncertainty of less than 1% in the U-factor.

8.6.3. Default Settings

In the **Calculation Options** tab found in the **File / Properties** menu, the following values can be set:

Table 8-7. Options/Preferences settings for THERM modeling

Setting	Suggested Value for Modeling
Simmetrix Mesh Parameter	8 or greater
Quad Tree Mesh Parameter	6 or greater
Maximum % Error Energy Norm	10%
Run Error Estimator	Should be checked

In the **Calculation Options** tab found in the **File / Properties / Calculation Options tab**, the Relaxation Parameter can be set to **1.0**, and the feature to "Automatically adjust relaxation parameter" should be set.

8.6.4. Calculations

In addition to simulating models one at a time, there is a **Calculation Manager**, accessed from the **Calculation/Calc Manager** menu option that allows the submittal of many files at one time to be calculated in the order they are added. The **Calc Manager Log** shows whether the jobs ran or not.

Calculating Multiple Glazing Options

If a base case file has been created with multiple glazing options (see Section 6.4.3, "Multiple Glazing Options"), when the file is simulated using the **Calc** toolbar, the **Calculation/Calculation** menu option, or the **F9** shortcut key, the **Glazing Option Simulation** dialog box will appear with the following options , as shown in the figure below:

- **Create the glazing option files and perform all simulations:** This choice is the default, and it creates and runs all the THERM files for the multiple glazing options defined in the base case file.
- **Simulate the current file (base case) only:** This choice is useful when setting up the base case file to make sure there are no geometry errors before running the entire multiple glazing options.
- **Create the glazing option files but do not perform the simulations:** This choice will cause the program to make individual THERM files for each of the glazing options associated with the base case file, with either the Glazing System ID or Name appended on to the base case filename, but it will not simulate the files. These files can then be opened and simulated individually, or simulated through the Calc Manager.

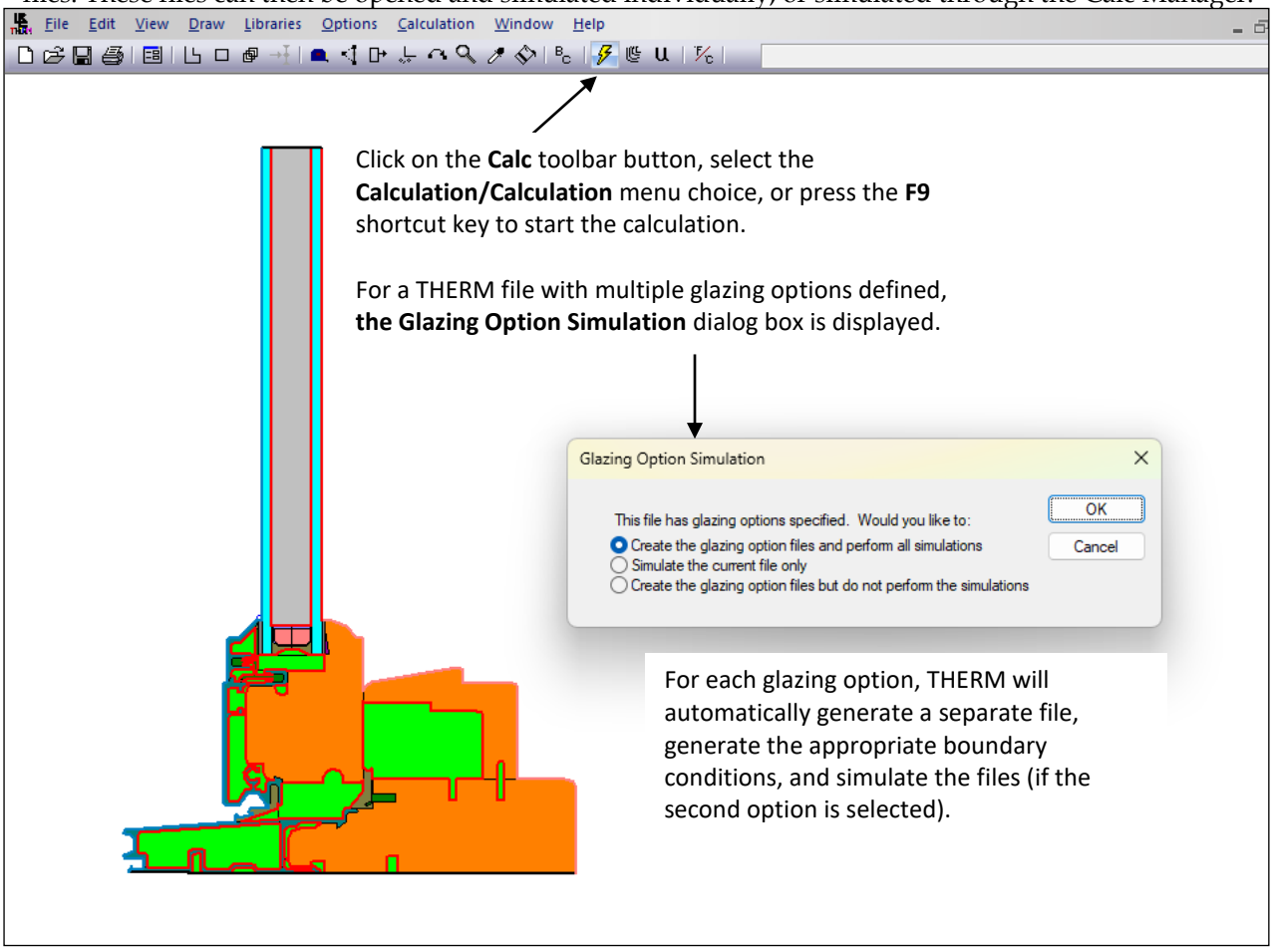


Figure 8-36. When a file w ith multiple glazing options is calculated, THERM gives several calculation options.

Multiple base case files with multiple glazing options in each file can be added to the **Calc Manager**, as shown in the figure below. The **Calc Manager** does not ask the three simulation-option questions; it implements the first choice, which is that to make all the files and then simulate them. The **Calculation Log** can be viewed to see the status of the runs. The information in the **Calculation Log** can be sorted by clicking on a column heading.

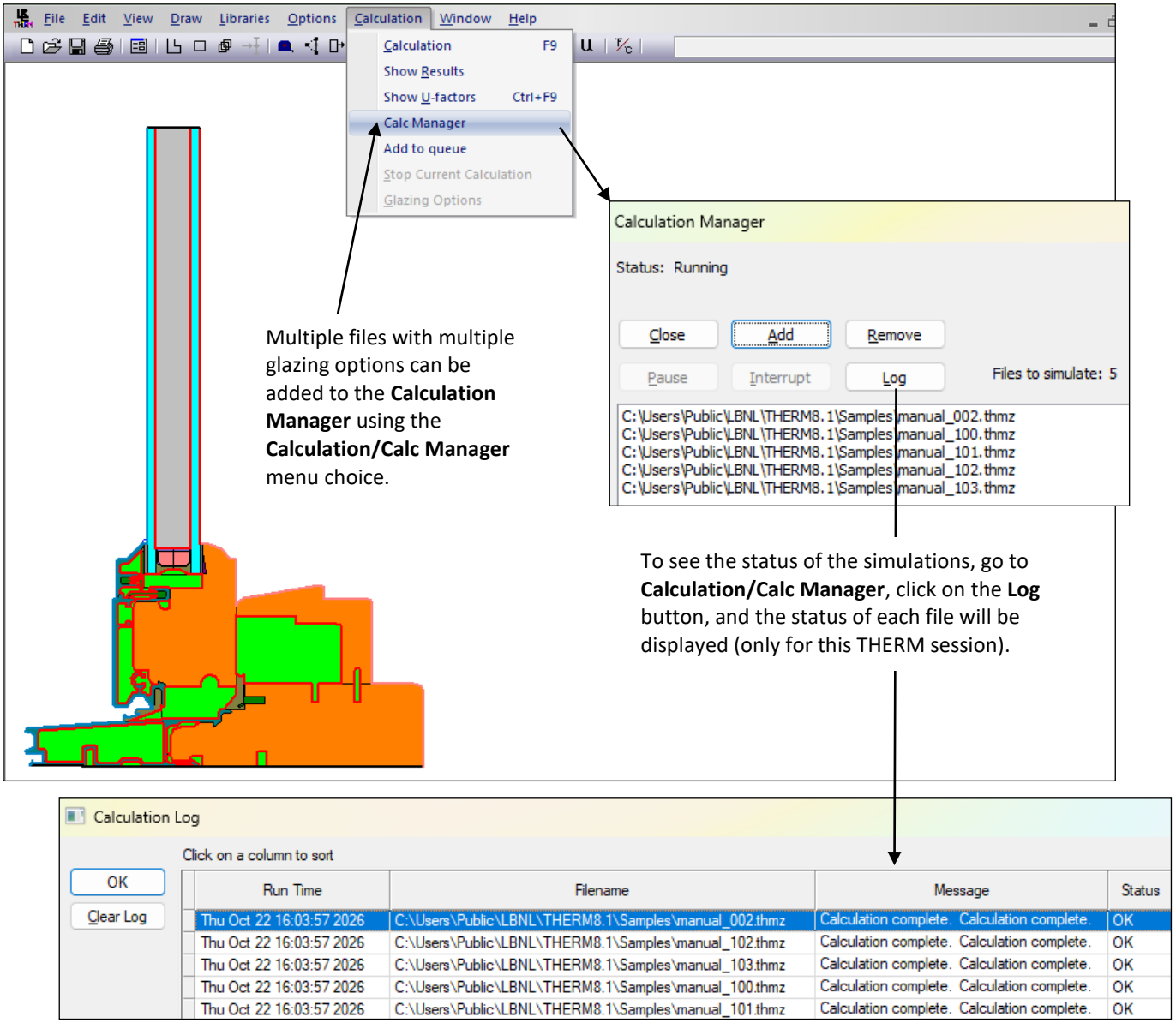


*Figure 8-37. The **Calc Manager** can be used to simulate multiple files with multiple glazing options in each file.*

8.6.5. Results

When the calculation is finished, the U-factors (shown in the figure below) can be viewed using the **Calculation/Show U-factor** menu choice or the **Show U-factor** toolbar button. u

The graphic display of the results is shown after a calculation or using the **Show Results** toolbar button.

The U-factors can be viewed for **Projected X**, **Projected Y**, **Total Length, Custom Length, Projected in Glass Plane** or **Custom Rotation**. U-factors are generally calculated as Project in Glass Plane, but there are a few special cases where the **Custom Frame Length** feature is used for specific models. (See the following discussion in *Section 6.6.6*, "Custom Frame Length" for an example using **Custom Frame Length**.)

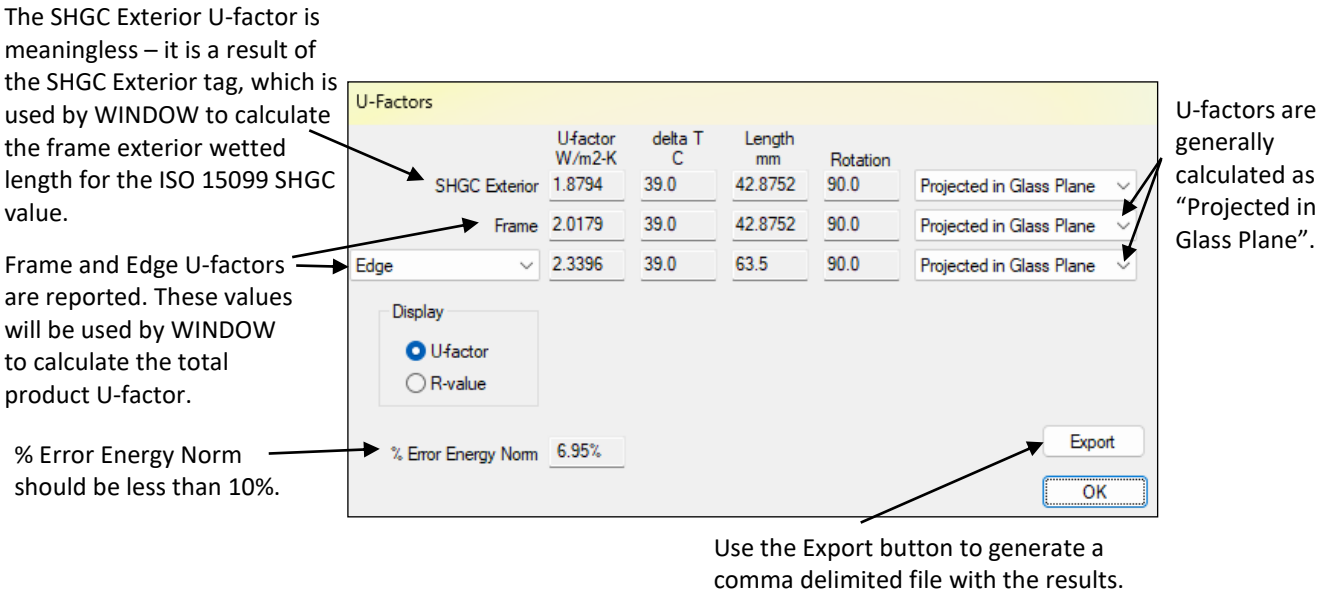


*Figure 8-38. U-factor results from the **Calculation/Show U-factors** menu choice.*

Local temperatures can be displayed at the cursor using the **View/Value at Cursor** menu choice if the Results are being displayed (by either clicking the **Show Results** toolbar button or using the **Calculation/Show Results** option). Average temperatures can be displayed with the tape measure (from the **Options/Preferences** menu, **Drawing Options** tab, check the **Tape Measure Average Temperature** checkbox).

THERM has a report which summarizes all the details of a THERM file, including the polygon ID numbers for all cavities. This report can be viewed and printed from the **File/Report** menu choice.

The THERM files (*.THMZ) are always saved with the detailed temperature data, which is used by WINDOW. The option "Save Simulation results in THMZ files", accessed from **File/Properties/Simulation tab**, is checked and grayed out, meaning the results are always saved in the files.

8.6.6. Custom Frame Length

There is a case where THERM will not calculate the project frame length correctly. This happens when a section of an adiabatic boundary "overlaps" a boundary with a U-factor tag in the projected dimension over which the length is being calculated. The figure below illustrates this. This incorrect frame length will cause the U-factor over the projected length to be calculated incorrectly. The U-factors results dialog box has a feature which allows a custom frame length to be entered, in order for THERM to calculate the correct U-factor. This situation happens in some skylight files, although it potentially could happen in any model. The figure below describes the steps required to use the Custom Frame Length feature.

For multiple glazing options, setting the custom frame length for each option can be quite cumbersome. One technique is to set the custom frame length in the base file first, and then the program will pick up for custom length values for each multiple glazing option.

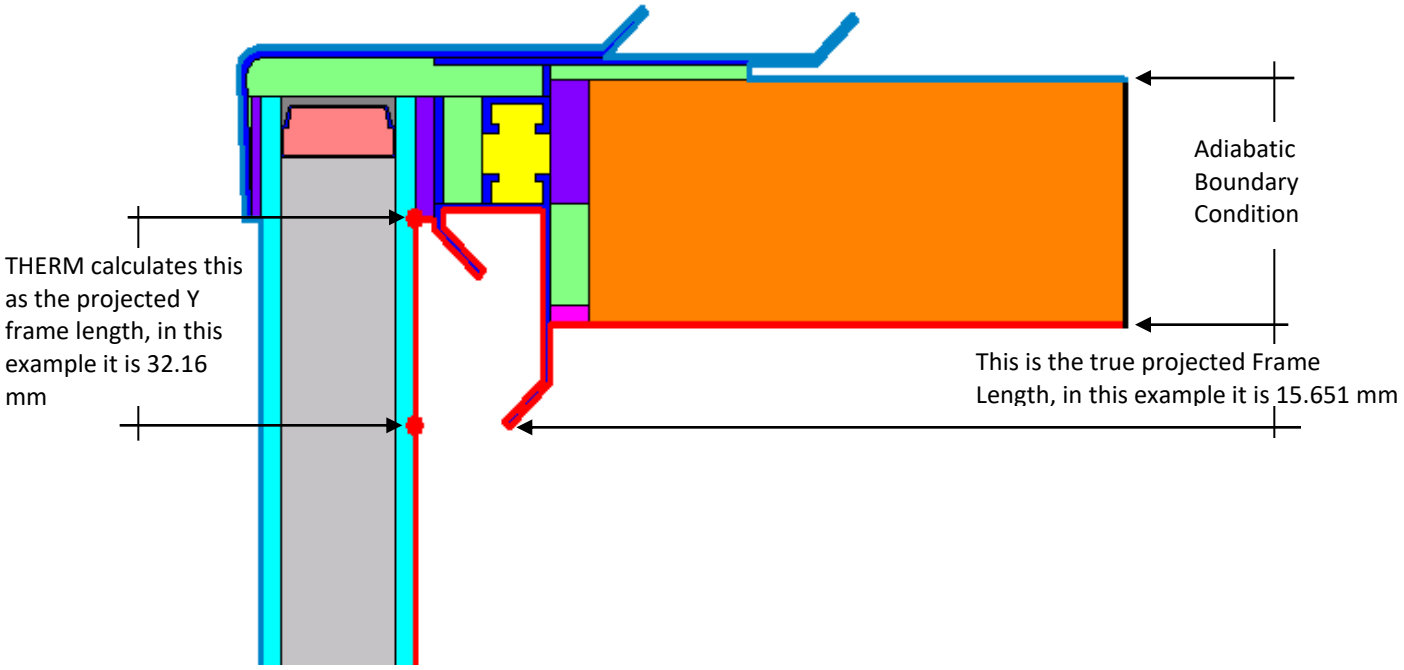


Figure 8-39 Example of a case where THERM does not calculate the projected Y frame length correctly because of an adiabatic boundary condition that "overlaps" an interior boundary condition.

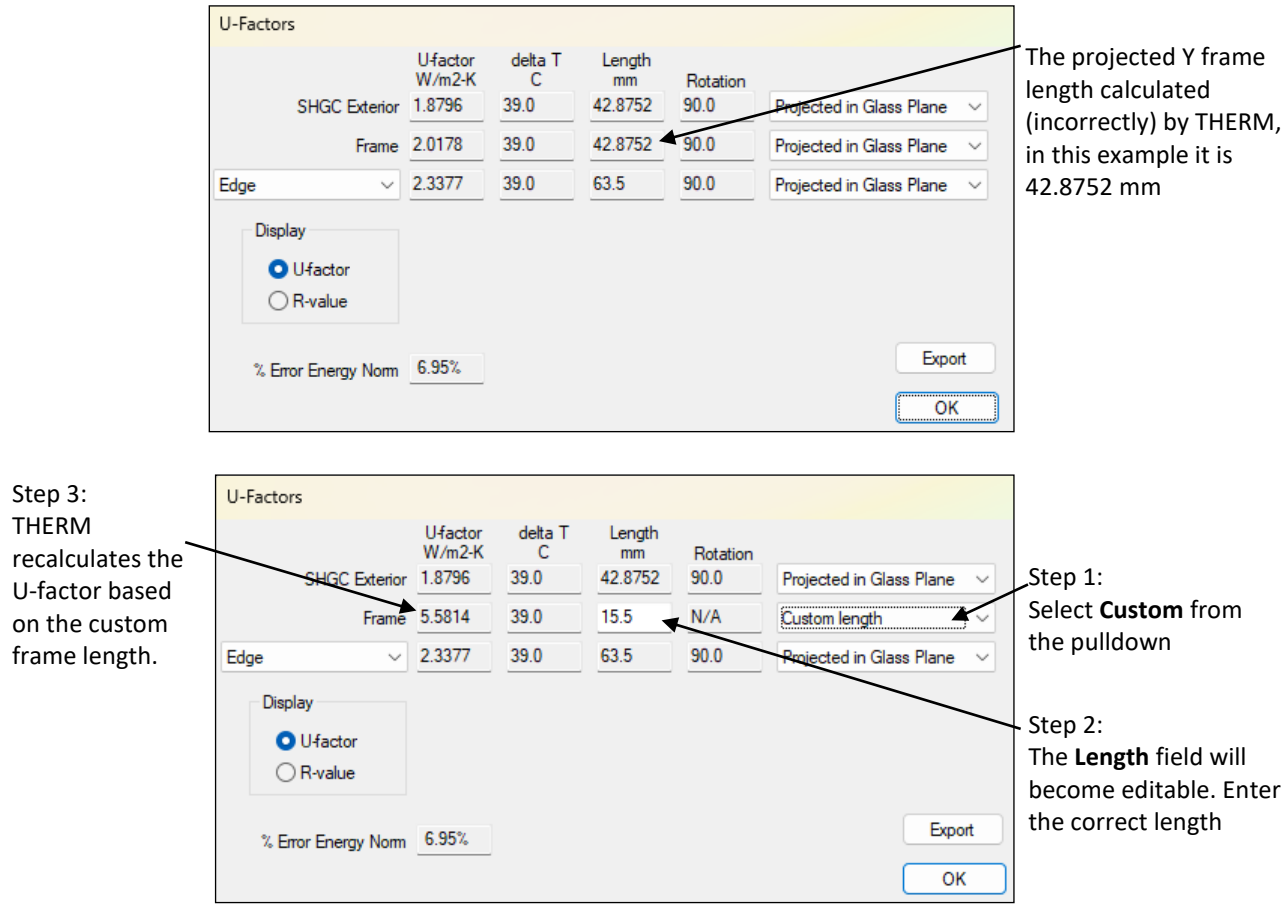
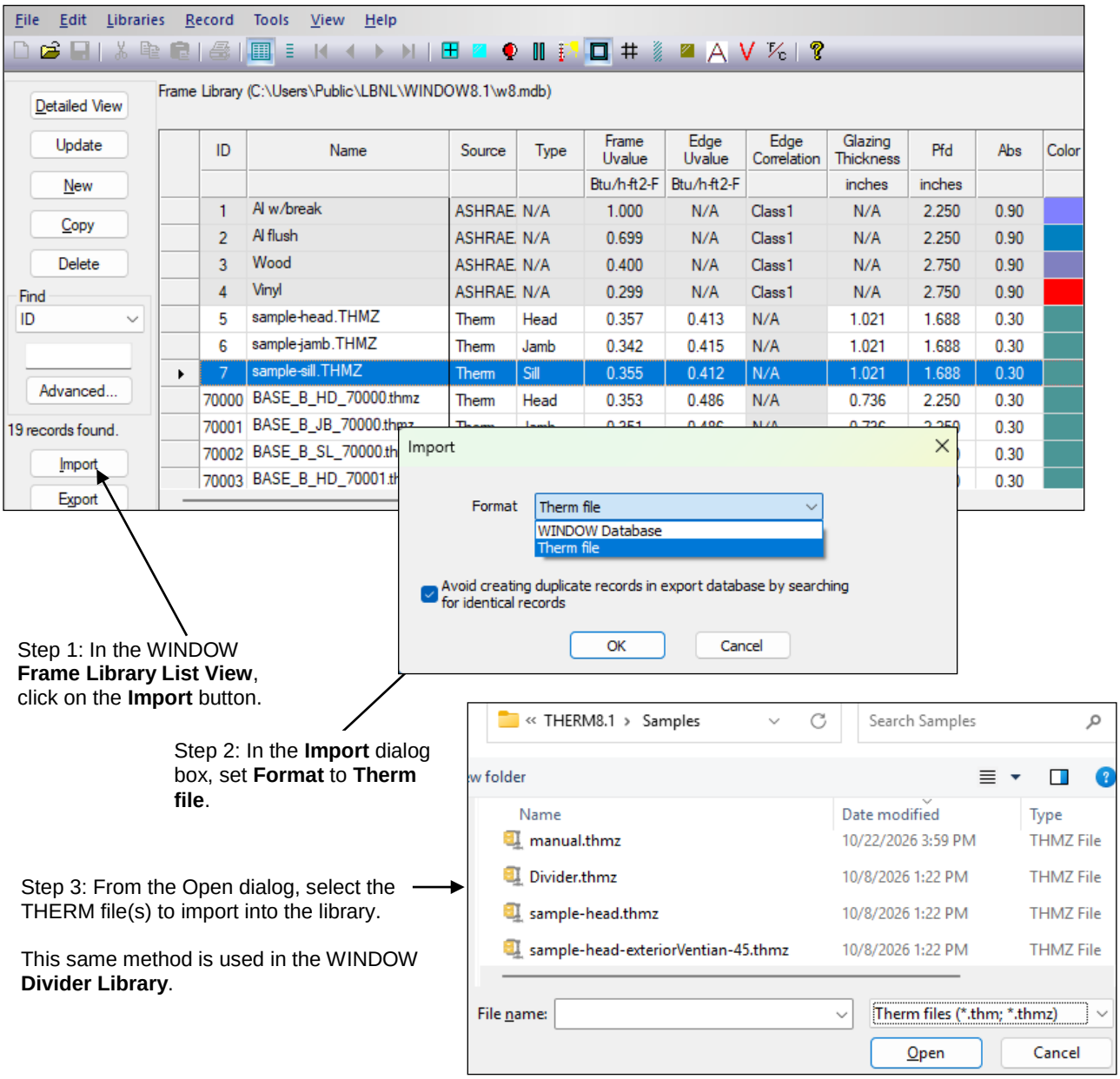


Figure 8-40 The Custom Frame Length feature allows the correct frame length to be entered.

8.6.7. Importing Results into WINDOW

Because the U-factors and temperatures from individual THERM cross sections will be used for the whole product calculations in WINDOW, it is necessary to import the THERM files with U-factor and simulation temperature results into the WINDOW program. This is explained in detail in the "Total Product Calculation" section of the *WINDOW User's Manual*. The basic steps are:

- Create the THERM models, calculate the results, and save and close the files.
- Open WINDOW and go to the **Frame Library List View**.
- Click on the **Import** button, select the "Therm Files" option from the pulldown, then THERM files to import (select multiple files using either **Shift** or **Ctrl** keys).
- WINDOW will ask how to number the new **Frame Library** records, either sequentially starting from the last existing record, or overwriting existing records.
- WINDOW will add each THERM file to the **Frame Library** as a separate record, assigning the THERM file name as the record name by default.
- If necessary, set (in WINDOW **File/Preferences/Calculation Options**) the "Default Frame Absorptance" to 0.30 or 0.50.



Step 1: In the WINDOW **Frame Library List View**, click on the **Import** button.

Step 2: In the **Import** dialog box, set **Format** to **Therm file**.

Step 3: From the Open dialog, select the THERM file(s) to import into the library.

This same method is used in the WINDOW **Divider Library**.

*Figure 8-41 Importing THERM files into the WINDOW **Frame Library**.*

8.7. Meeting Rails

Meeting rail cross sections are the stiles or rails that meet in the middle of a sliding window. In this manual, the term "meeting rail' is used generically to describe meeting rails, meeting stiles, interlock stiles, interlocking stiles, sliding stiles, check rails, and check stiles.

8.7.1. Modeling Meeting Rails

When modeling a meeting rail, both the sashes and their associated glazing systems are modeled. The figure below shows an example of the meeting rail from a horizontal aluminum slider.

Creating the cross section for a meeting rail is no different than any other model in THERM. A few things to keep in mind are:

- Two glazing systems are imported, one facing up and one facing down
  - Interior boundary conditions for each of the glazing systems are labeled with the **Edge** U-factor tag, and the program averages the values for both to derive one Edge U-factor.
  - Model the meeting rail with the glazing systems facing up and down (see "Cross Section Orientation" in this manual). If the DXF file is drawn with them in a horizontal position, draw the frame cross section, and then rotate it before inserting the glazing system.

The following discussion lists the steps for making a cross section with two glazing systems and assigning the correct boundary conditions.

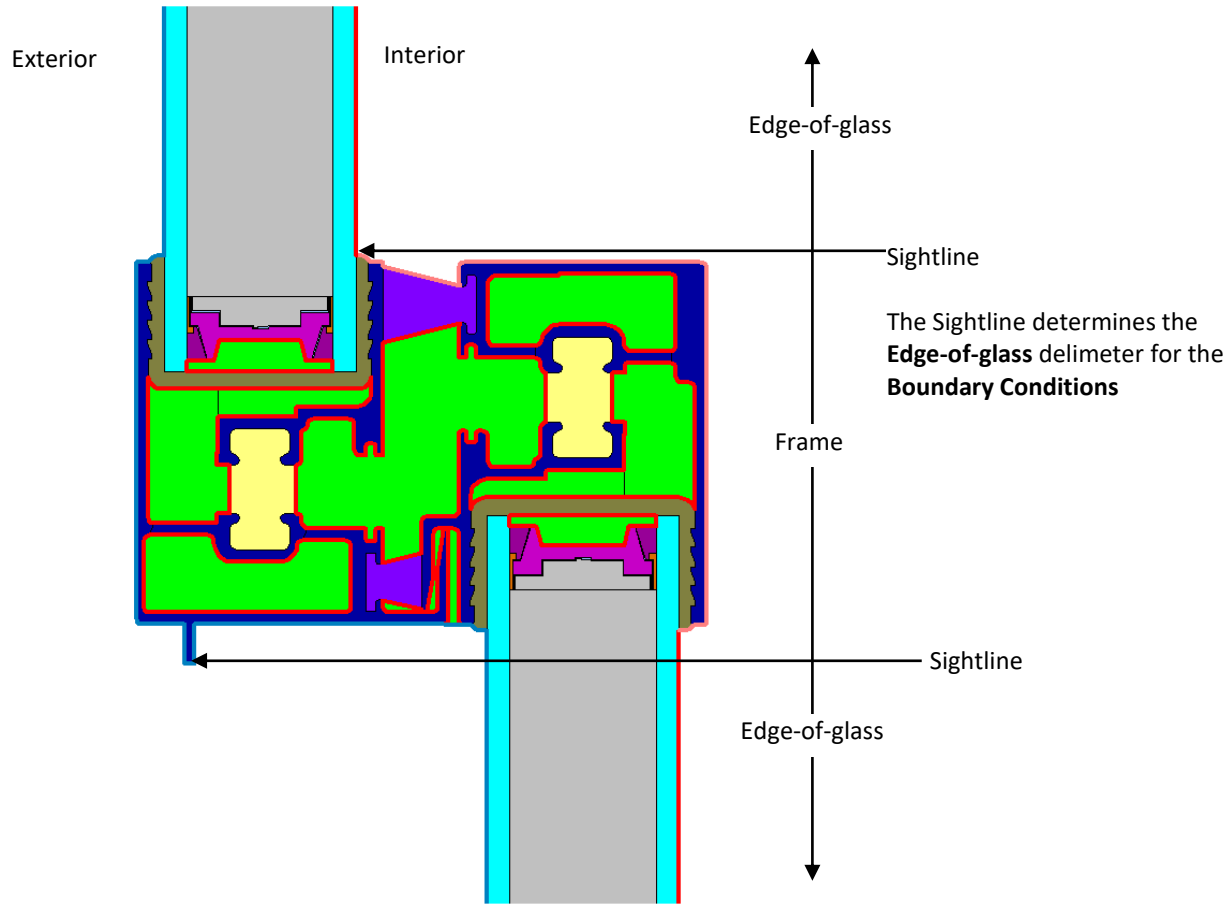


Figure 8-42. Meeting rail cross section.

8.7.2. Steps for Meeting Rail U-factor Calculation

1. Using dimensioned drawings or a DXF file, create the cross section for the frame portion of the meeting rail. In the figure below, the frame for the horizontal aluminum slider meeting rail has been created.

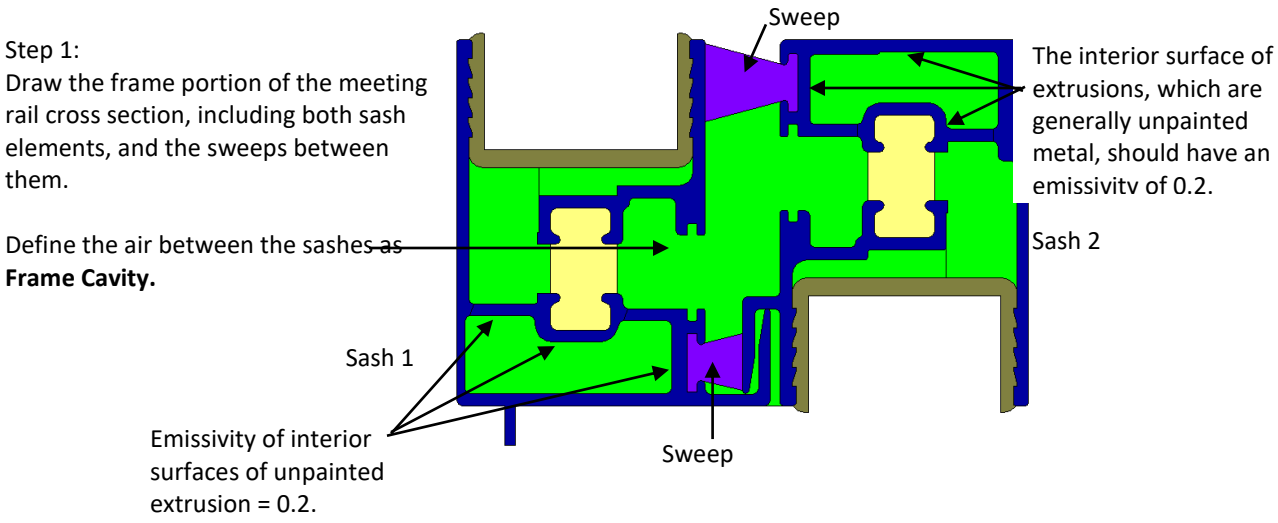


Figure 8-43. Frame portion of meeting rail cross section.

2. Position the **Locator** (using the **Draw/Locator** menu choice, or pressing **Shift F2**) in the lower left corner of the frame where the first glazing system will be inserted, as shown in Figure 8-44.

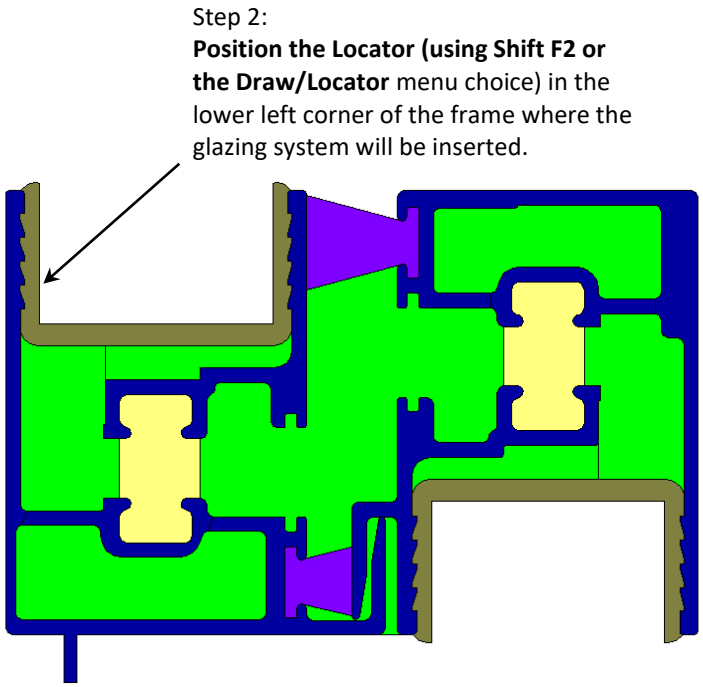


Figure 8-44. Position the Locator for the first glazing system.

3. Using the **Libraries/Glazing Systems** menu option (or the **F6** key), insert the upper glazing system, as shown in the figure below. In this example, the spacer will be copied and pasted into the cross section later. Add a spacer and use the **Material Link (Library/Create Link)** to link the glazing system cavity conductivity with adjacent cavities in a spacer which is open to the glazing system cavity, if necessary.

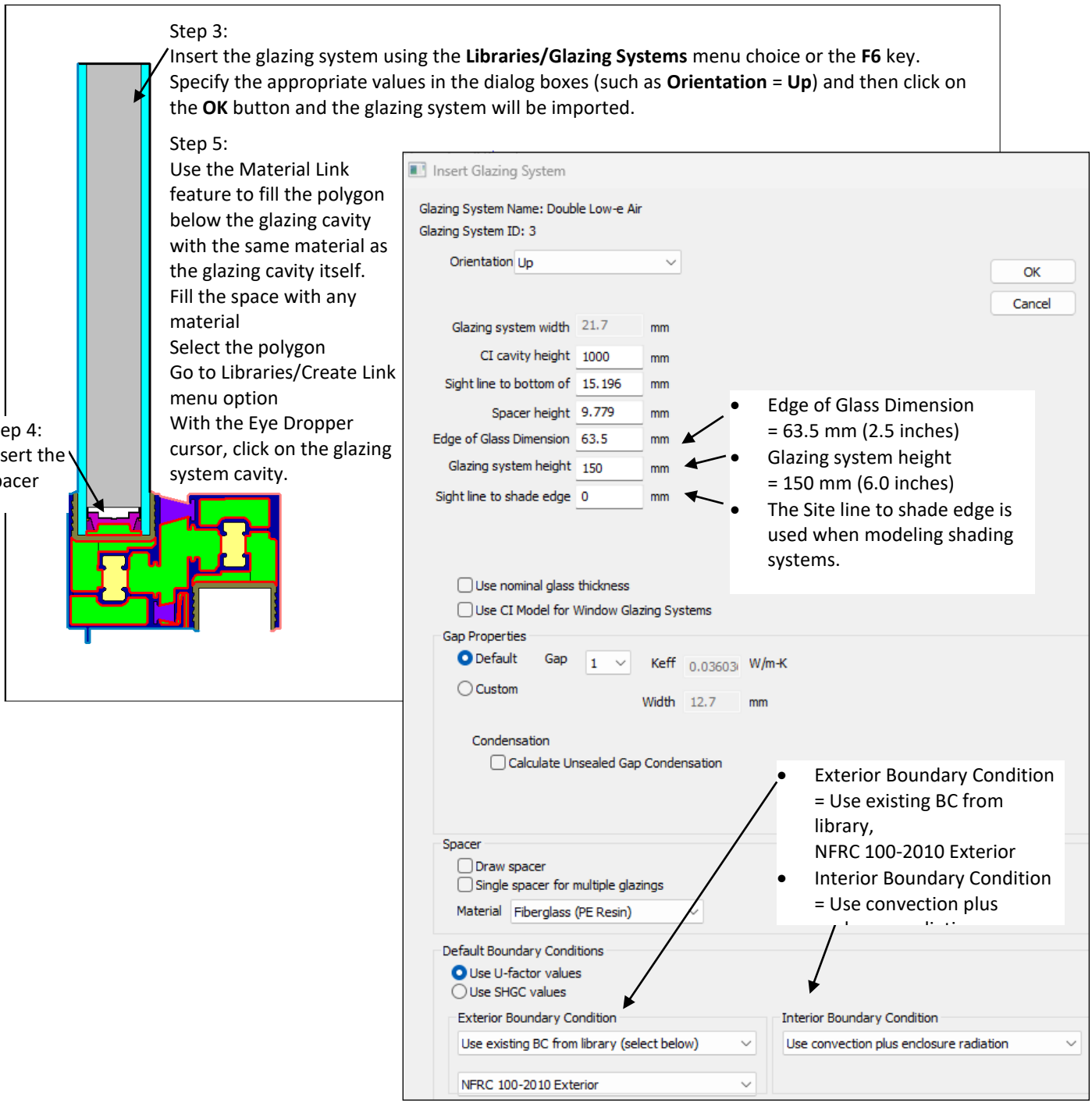
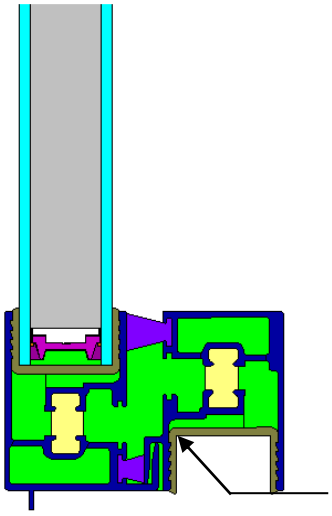


Figure 8-45. Insert the first glazing system.

4. Reposition the locator to the upper left corner for the 2nd glazing system.



Step 4:
Reposition the **Locator** in the upper left corner of the frame for the 2nd glazing

Figure 8-46. Reposition the Locator for the 2nd glazing system.

5. Insert the 2nd glazing system, setting the **Orientation** to "Down", and entering the correct values for **Sight line to bottom of glass** and **Spacer height**.

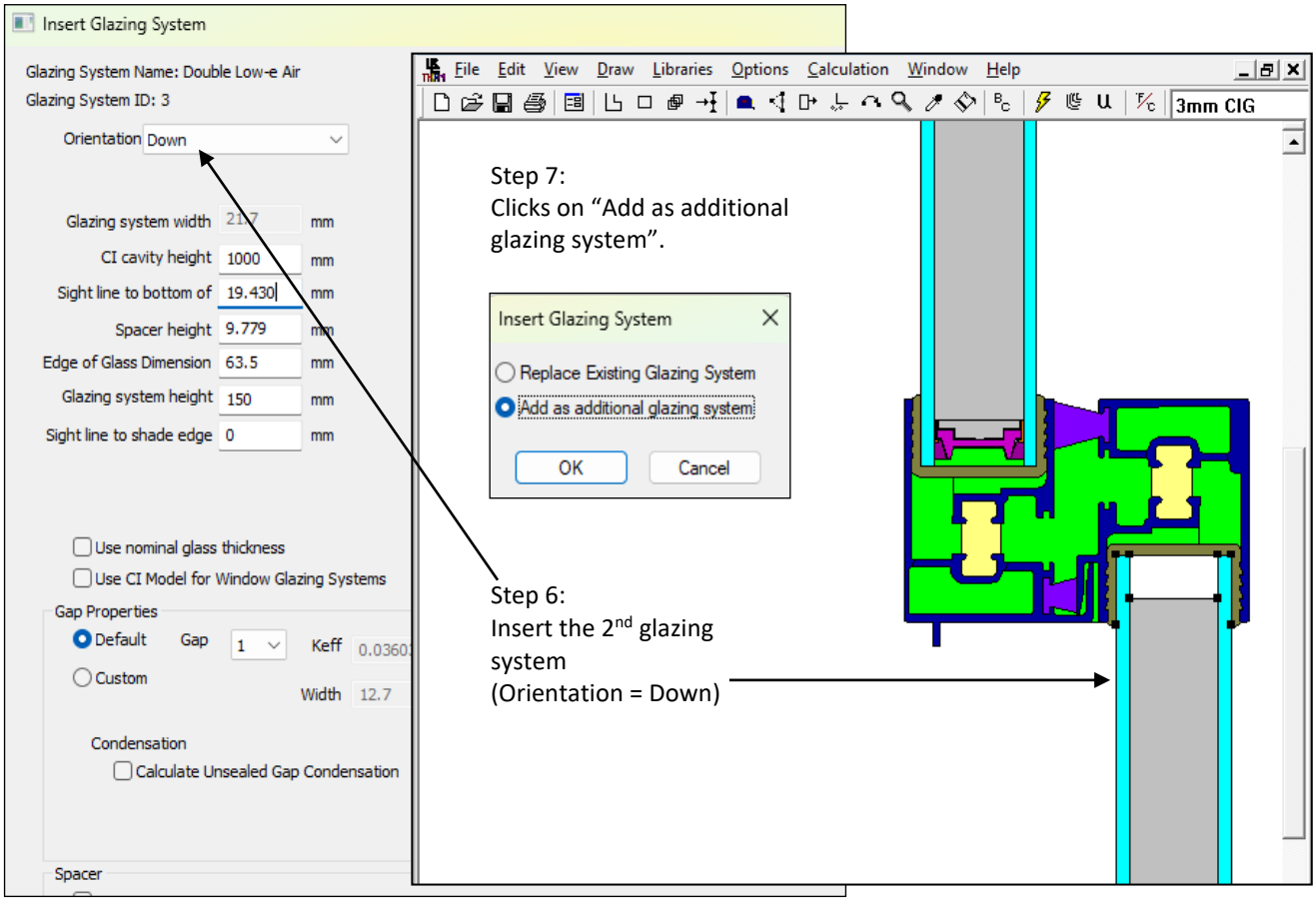


Figure 8-47. Insert the 2nd glazing system.

6. Add spacers and create materials linked to the glazing system cavity if necessary.

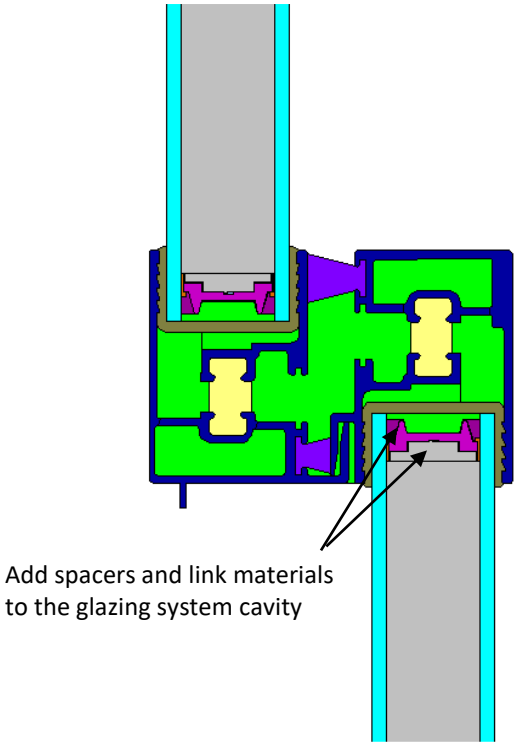


Figure 8-48. Add custom spacers.

7. Define the boundary conditions by pressing the **Boundary Conditions** toolbar button, or clicking on the **Draw/Boundary Conditions** menu choice, or pressing the **F10** key. Make sure that the interior boundary conditions are set to **Radiation Model** = "AutoEnclosure". Assign the **Edge U-factor tag** to each of the interior glazing system boundary conditions, as shown in the figure below.

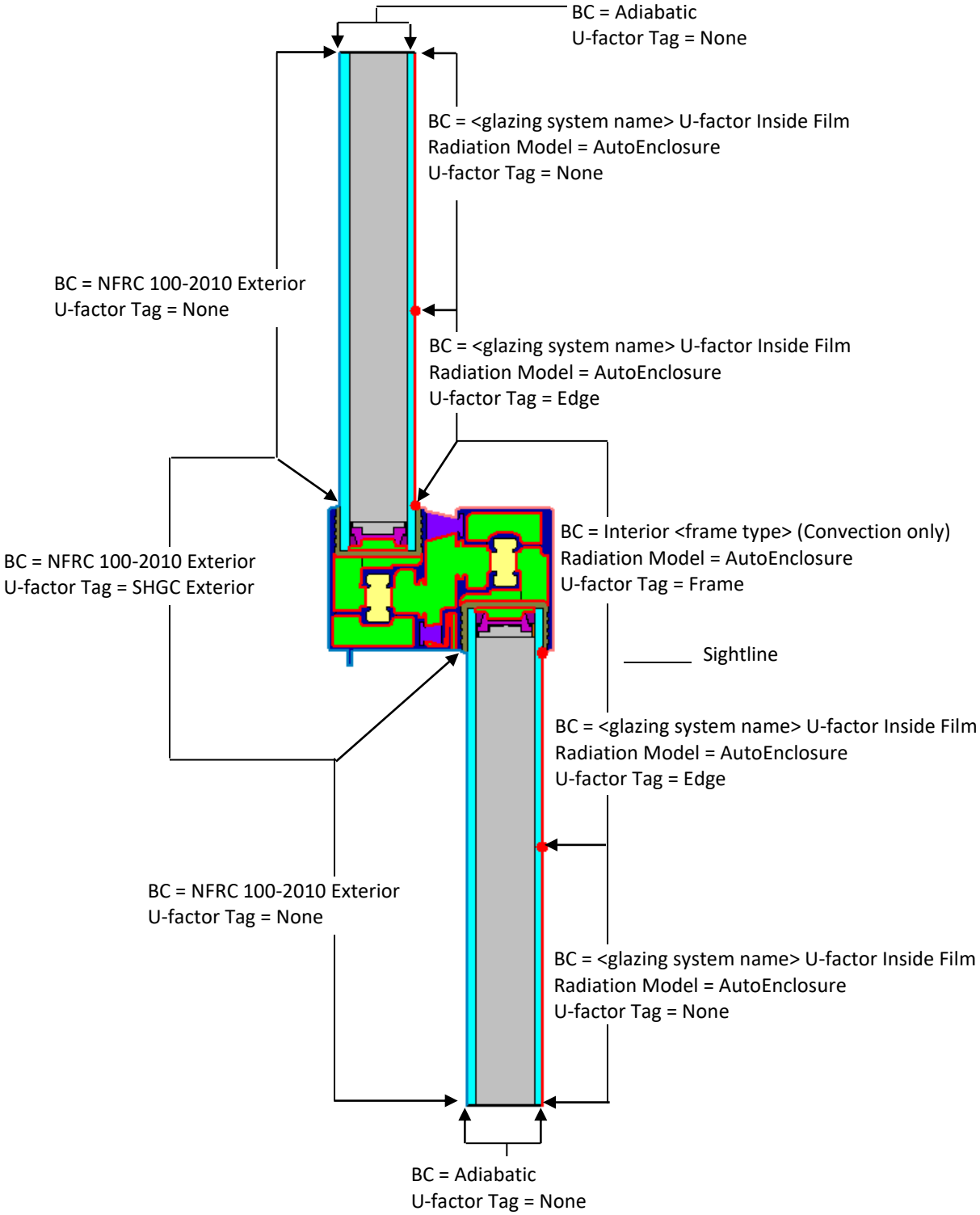


Figure 8-49. Define the Boundary Conditions for the meeting rail.

8. Run the simulation, by pressing the **Calc** toolbar button, clicking on the **Calculation/Calculation** menu choice, or pressing the **F9** key. The U-factor results are calculated for the **Frame** and **Edge U-factor tags**, as shown in the figure below.

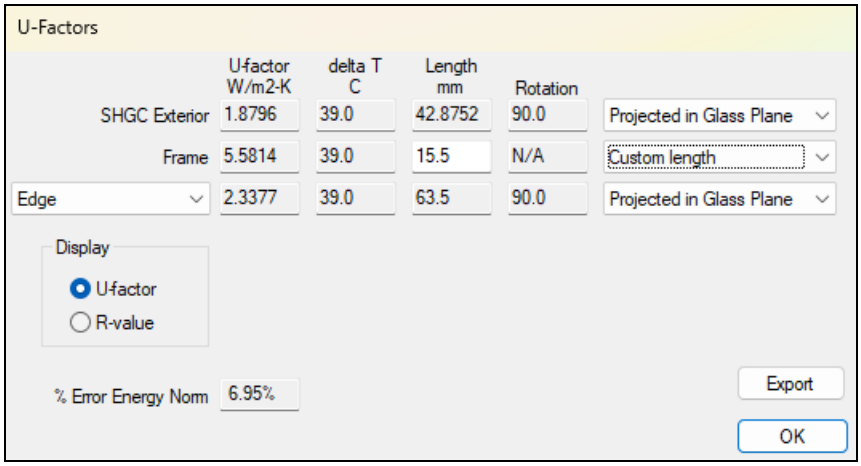


Figure 8-50. Calculate the results.

9. Import the THERM file into the WINDOW Frame Library.

8.7.3. Steps for Meeting Rail Condensation Calculation

The Condensation Resistance model is only appropriate for *horizontal meeting rails* (found in vertical sliding products) – THERM will not calculate the Condensation Resistance for a file with the **Cross Section Type** set to "Vertical Meeting Rail".

There are two methods for calculating the Condensation Resistance information in THERM, which will be used in WINDOW to calculate the total Condensation Resistance of the product:

- Check the "Use CI Model for Window Glazing System" checkbox when importing a glazing system

  OR

- In the File menu, Properties choice, Calculation Options tab, check the "Use CI Model for Glazing Systems", as shown in the figure below.

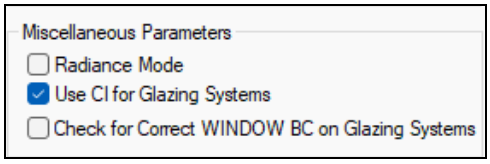


Figure 8-51. In File/Properties/Calculation Options, check the "Use CI Model for Glazing Systems" checkbox.

When the condensation model has been "turned on", red boundary conditions will appear inside the glazing system, and the following steps should be taken to simulate the file:

1. Check the emissivities of these boundary conditions. They should be the following:
   - Emissivity of the surrounding surface, such as 0.84 for standard glass, 0.90 for most frame materials, 0.20 for metal, and so forth.
   - 1.0 for the adiabatic (open end) of the glazing cavity.
   - Actual cavity height per *Table 8-2, Section 8.4.5*

2. Simulate the model. The program will calculate both U-factor results and the Condensation Resistance results.
3. Import the results into the WINDOW **Frame Library** and use the meeting rail file to create the whole product in the **Window Library** as applicable.

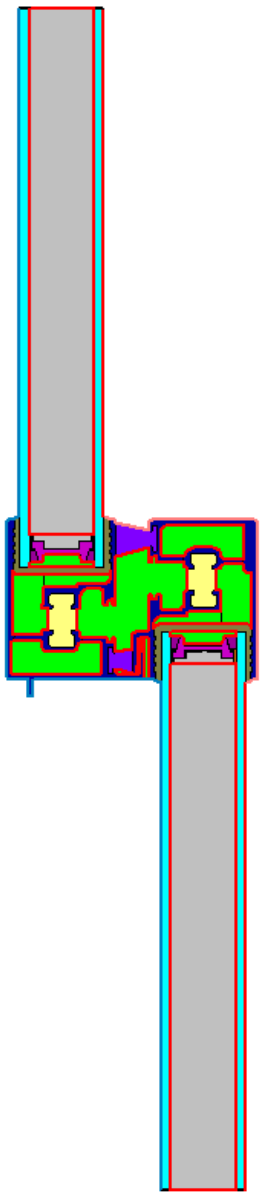


Figure 8-52. Red boundary conditions will appear inside the glazing system when the Condensation Resistance feature is activated.

4. Check the emissivities of each of these boundaries. Note that Condensation Resistance is only modeled for horizontal meeting rails (such as in a double hung window).

8.8. Dividers

8.8.1. Internal Dividers (Suspended Grilles)

The discussion below describes the methodologies in WINDOW and THERM for modeling dividers.

8.8.1.1. Modeling Steps

The modeling steps in THERM and WINDOW are the same for all divider shapes and all possible gas fills, in contrast to modeling steps in previous versions of THERM.

Linking Frame Cavities and Glazing Cavities

Internal dividers are an exception to the rules for linking frame cavities to glazing cavities. Internal dividers are only modeled if the distance between the exterior sides of the divider and the inside surface of the glazing system are less than or equal to 3 mm. Therefore, even for contoured internal dividers, a simplifying assumption is made that the 5 mm rule is not applied. The space between the divider edge and the glass are modeled as frame cavities, and the boundary of that cavity extends to the top and bottom of the divider component. See the following examples for detailed descriptions.

Modeling Steps

The modeling steps are the following:

In WINDOW:

- No new work is required, because the same glazing system that is used to model the rest of the product is used in the divider model.

In THERM:

- The new ISO 15099 modeling assumptions would theoretically warrant modeling horizontal and vertical dividers separately. However, a conservative simplification is to model all dividers, including horizontal ones, as vertical dividers. Therefore, only one divider model is created in THERM and referenced in WINDOW.
- Set the **Cross Section Type** to "Vertical Divider" for all dividers.
- Insert the glazing system twice, once facing up, with a spacer height defined as the same height as the divider height, and once facing down with the spacer height set to zero.
- *NOTE: Because all dividers are modeled as "Vertical Dividers" the condensation model is not run in THERM for these files.* However, WINDOW will still calculate a whole product CI/CR value when these dividers are used in a product, by using the U-factor temperatures for the dividers.
- Draw the true geometry of the divider in the upper glazing system, in the "spacer" area.
- Depending on the fill of the glazing system, assign the appropriate frame cavity material to the cavities between the glazing system and the divider, as well the cavity inside the divider, as follows:
  - For air-filled dividers: Assign "Frame Cavity" material
  - For gas-filled dividers: Assign "Frame Cavity" material and set the Gas value to the appropriate gas record from the pulldown list (which is a list of the records in the Gas Library).
- Assign Boundary Conditions.
- Simulate the results.

Import the file into the WINDOW Divider Library. Reference the Divider as appropriate from the Window Library when constructing the whole product.

8.8.1.2. Air Filled Glazing Systems

The modeling steps for a divider with an air-filled glazing system are explained in detail in the following pages.

In THERM:

1. Set the **Cross Section Type** to "Vertical Divider".
2. Import the glazing system for the divider, which is the same glazing system as the rest of the product, with the following settings:
   - Orientation = Up
   - **CI Cavity height** = 1000 mm (39 inches)
   - **Sight line to bottom of glass** = height of the divider (in this example it is 19.05 mm [0.75 inches])
   - **Spacer height** = height of the divider (in this example it is 19.05 mm [0.75 inches)
   - **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
   - **Glazing System Height**: 150 mm (6.0 inches)
   - **Sight line to shade edge:** 0 (not modeling shading system)
   - **Draw spacer** = Not checked

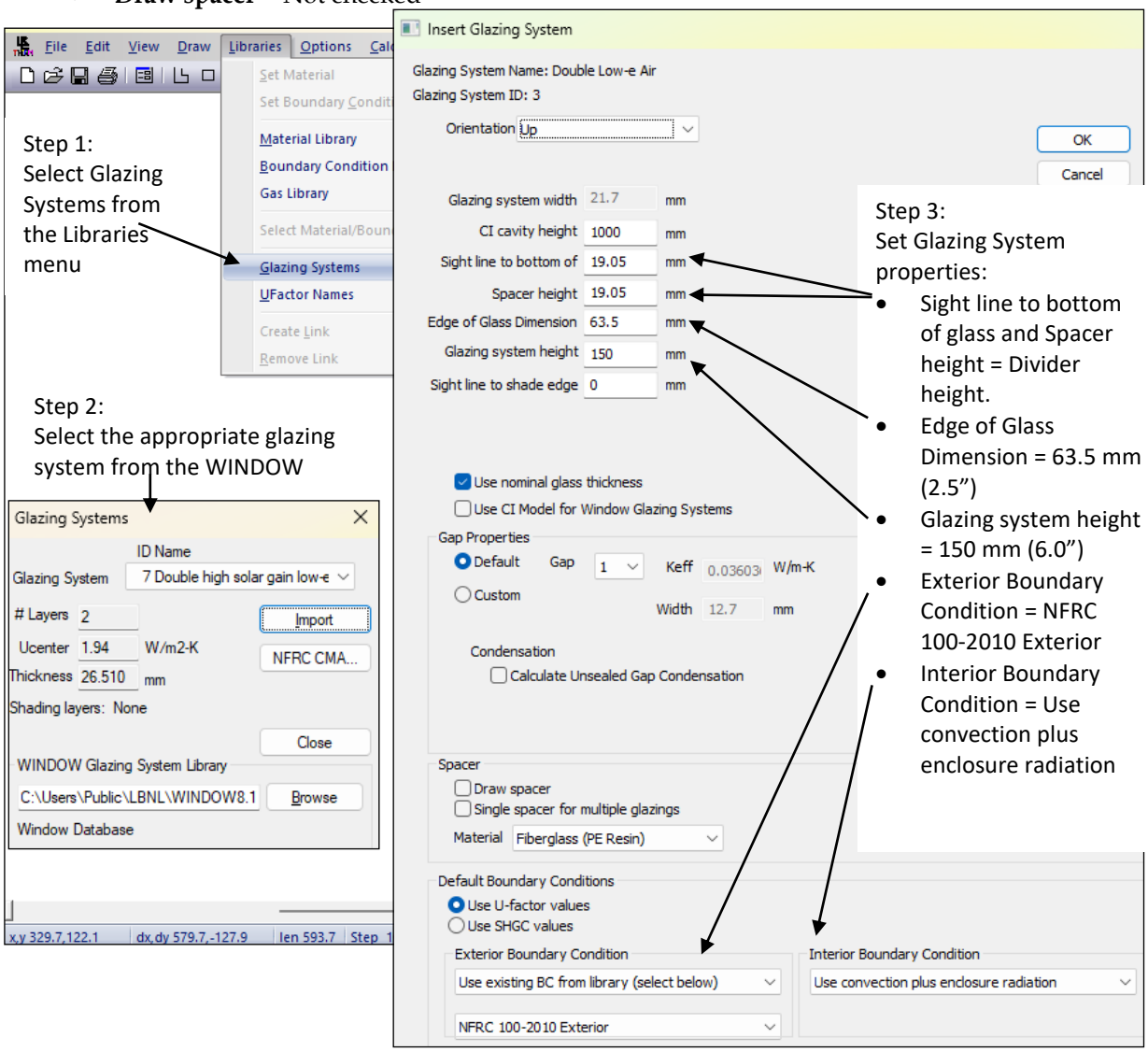


Figure 8-53. Import the first glazing system.

3. Import the glazing system again as an additional glazing system, below the first one (the locator does not have to be moved), but facing down this time. Use the following settings for this glazing system:
   - Orientation = Down
   - **CI Cavity height** = 1000 mm (39 inches)
   - Sight line to bottom of glass = 0
   - Spacer height = 0
   - **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
   - Glazing System Height: 150 mm (6.0 inches)
   - **Sight line to shade edge:** 0 (not modeling shading system)
   - **Draw spacer** = Not checked

   Insert the glazing system as an **Additional Glazing**.

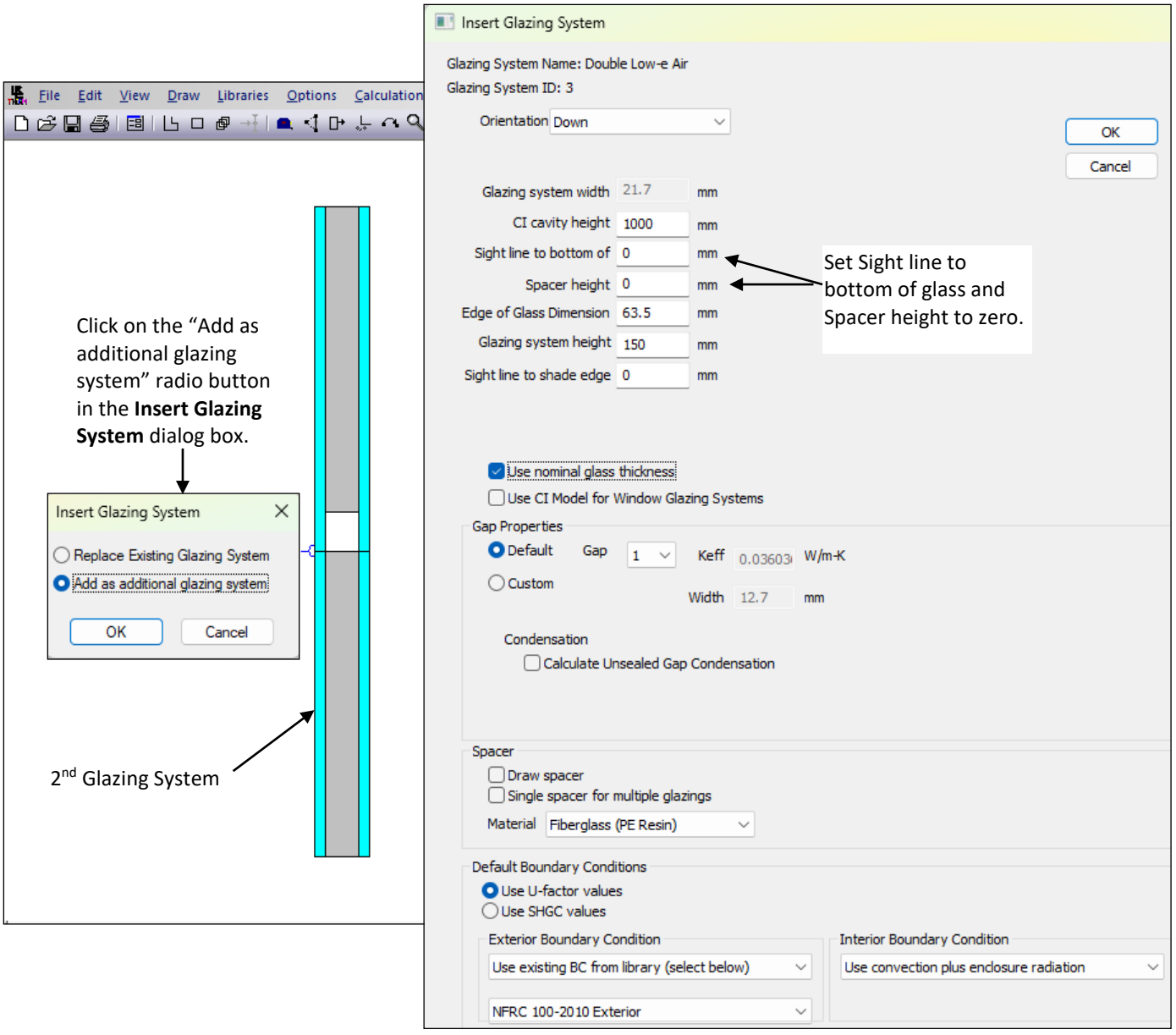


Figure 8-54. Import the second glazing system as an additional glazing system, facing down.

4. Draw (or copy and paste from another THERM file) the polygons in the cavity that represent the divider. The figure below shows the divider for this example drawn with the material set to Aluminum Alloy

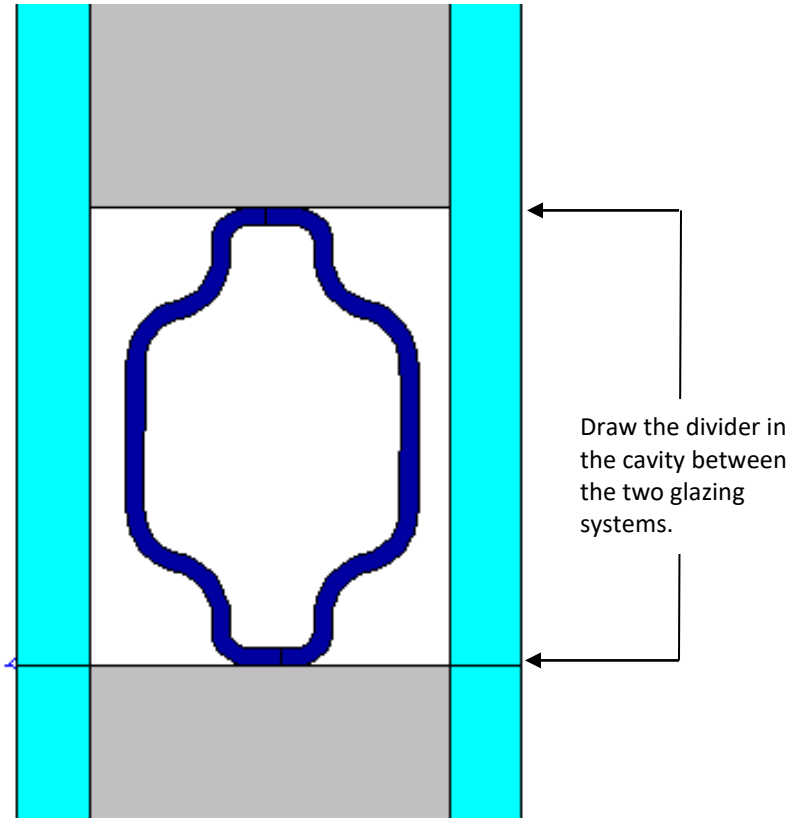


Figure 8-55. Draw the polygons to represent the divider.

5. Fill the cavities between the divider and the glass layers and inside the divider with the material "Frame Cavity". Because internal dividers are only modeled if the gap between the edge of the divider and the glass surface is less than 3 mm, the 5 mm rule for linking glazing cavities and frame cavities is not applied to internal dividers – the cavity surrounding the divider is modeled as a frame cavity, with boundaries at the upper and lower edges of the divider.

Check the emissivity values for the inside surface (by double-clicking on the surface) of the extruded metal divider – If there is a different surface finish on each side of the material, change the default material emissivity as appropriate based on the NFRC 101 values.

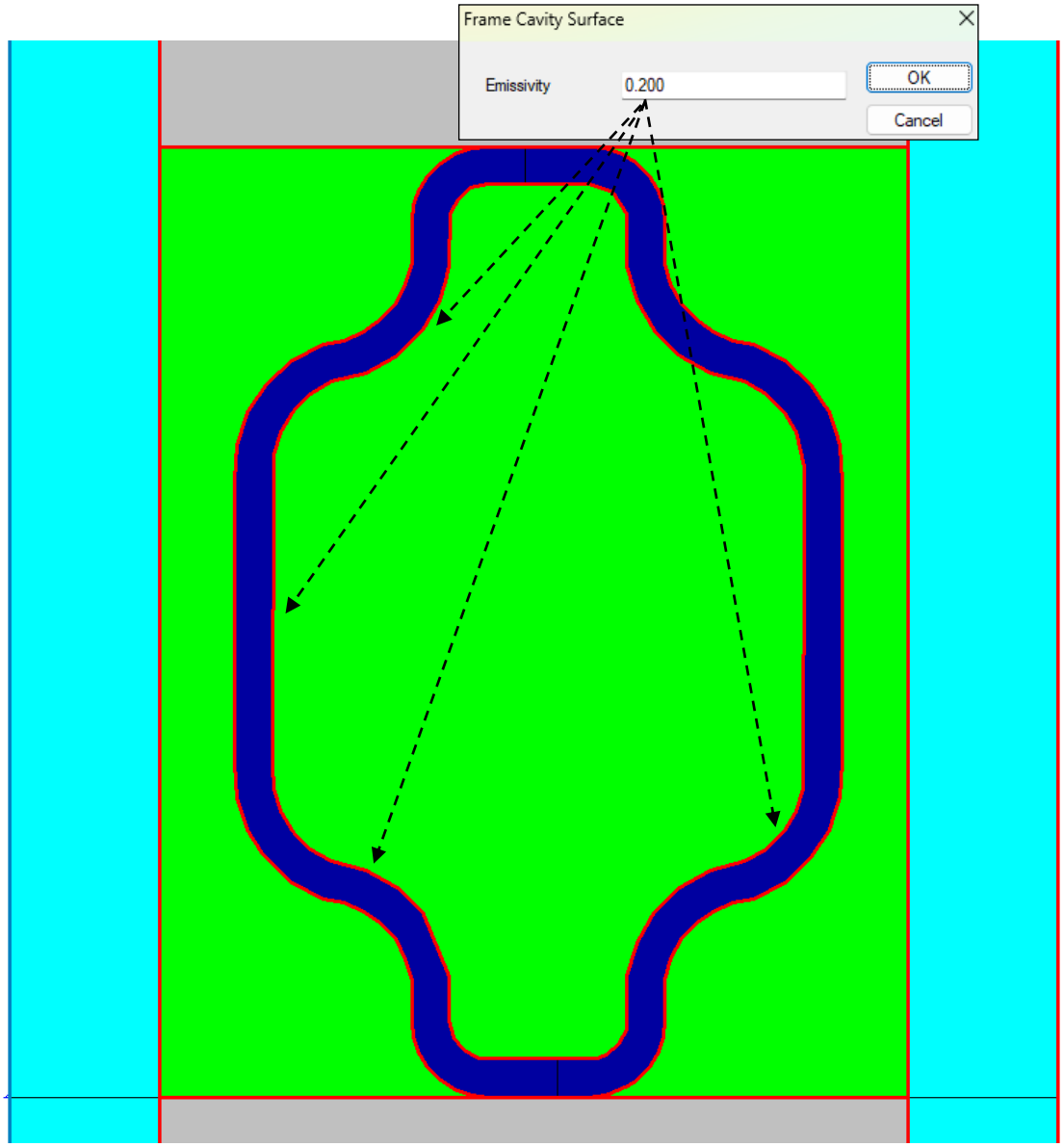


Figure 8-56. Fill the divider frame cavities with Frame Cavity.

6. Define the boundary conditions, using the "AutoEnclosure" choice for the Radiation Model.

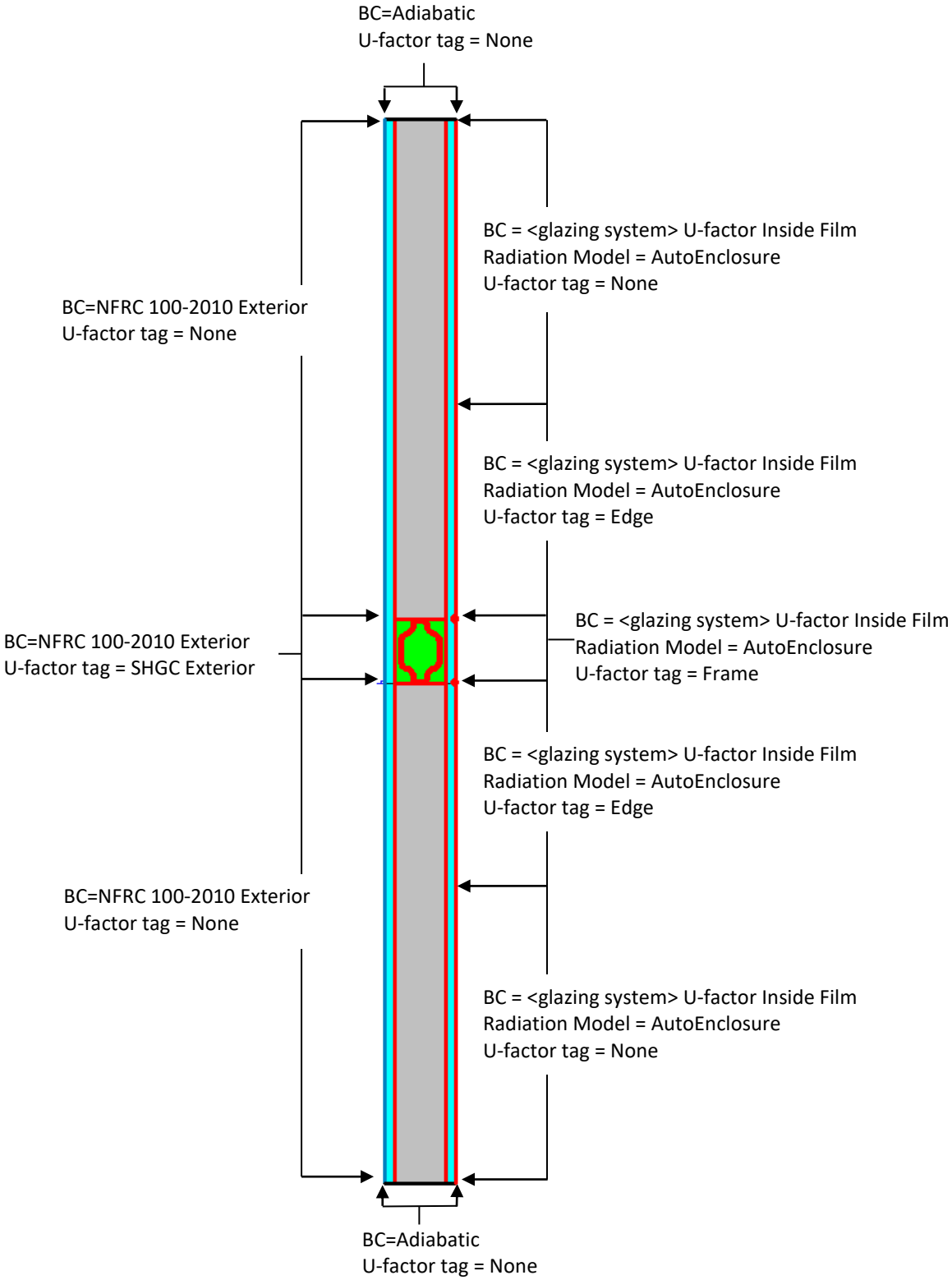


Figure 8-57. Assign the boundary conditions.

7. Calculate the results.

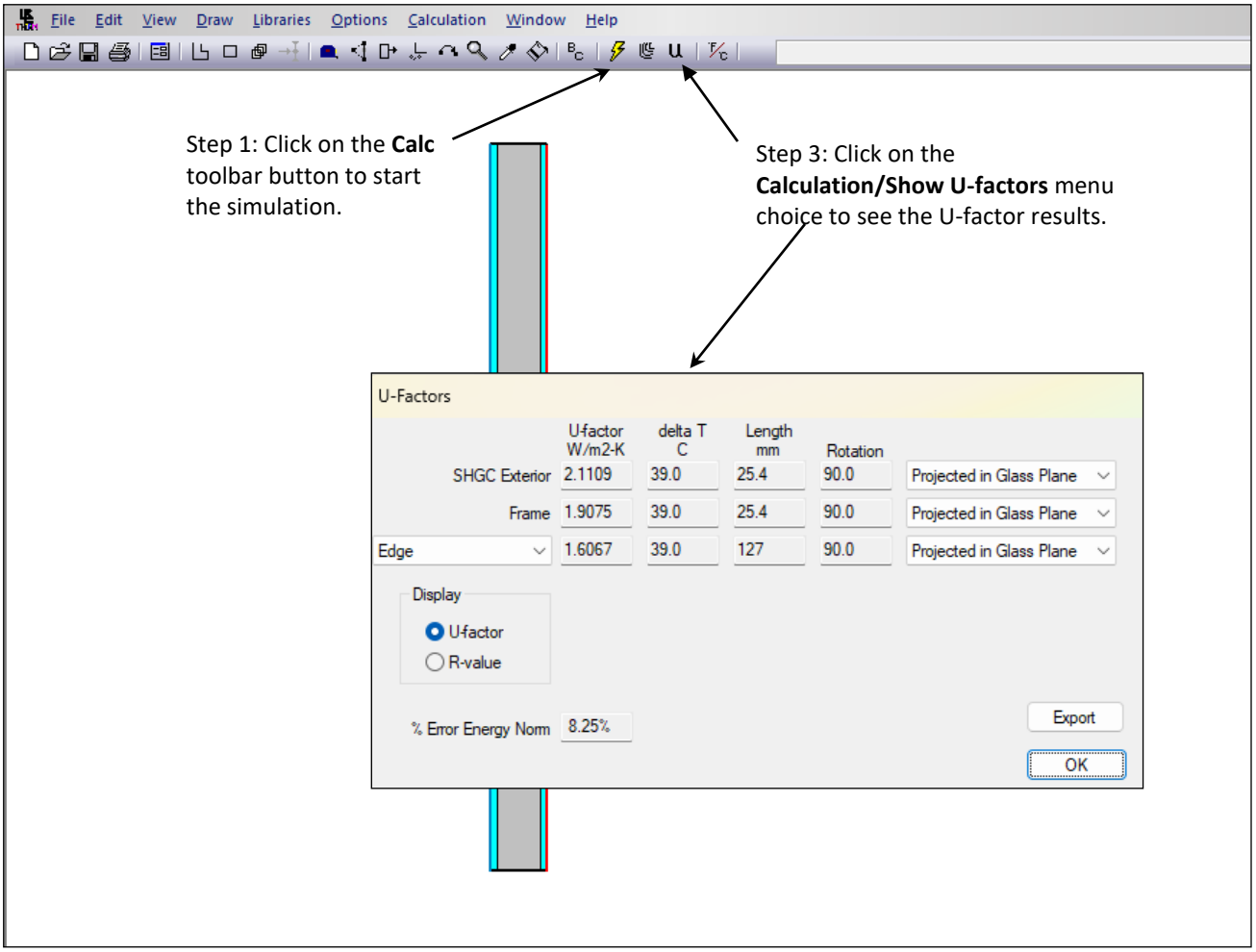


Figure 8-58. Calculate the results.

8. Save the file using the **File/Save As** menu choice.

9. Import the results to the WINDOW Divider Library, as shown below. See "Importing THERM files" in the *WINDOW User's Manual* for more information about importing THERM files.

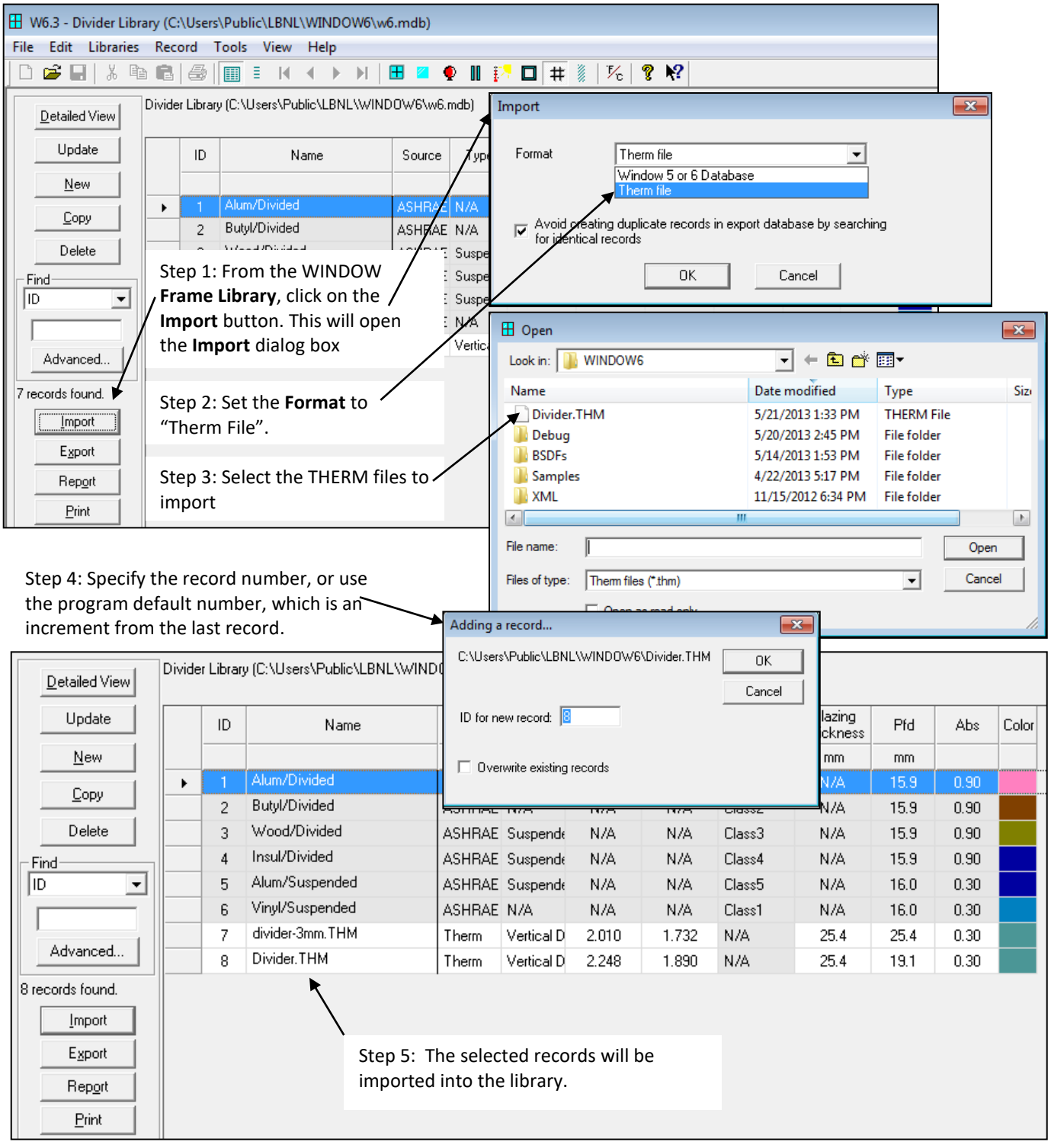


*Figure 8-59. Import the THERM file into the WINDOW **Divider Library**.*

10. Use the new divider in the calculation of the complete product values in the main screen of WINDOW.

8.8.1.3. **Gas Filled Glazing Systems**

If the glazing system being modeled with a divider is gas-filled, it is necessary to model the divider with the same gas fill as the glazing system. If the gas doesn't already exist in the THERM Gas Library, the correct gas record can be added. Select the frame cavity or cavities that are to be gas filled, press the Enter key or Double click the mouse, and the Properties for Selected Polygon dialog box will open. Set the gas to the appropriate value from the list, which is populated from the THERM Gas Library

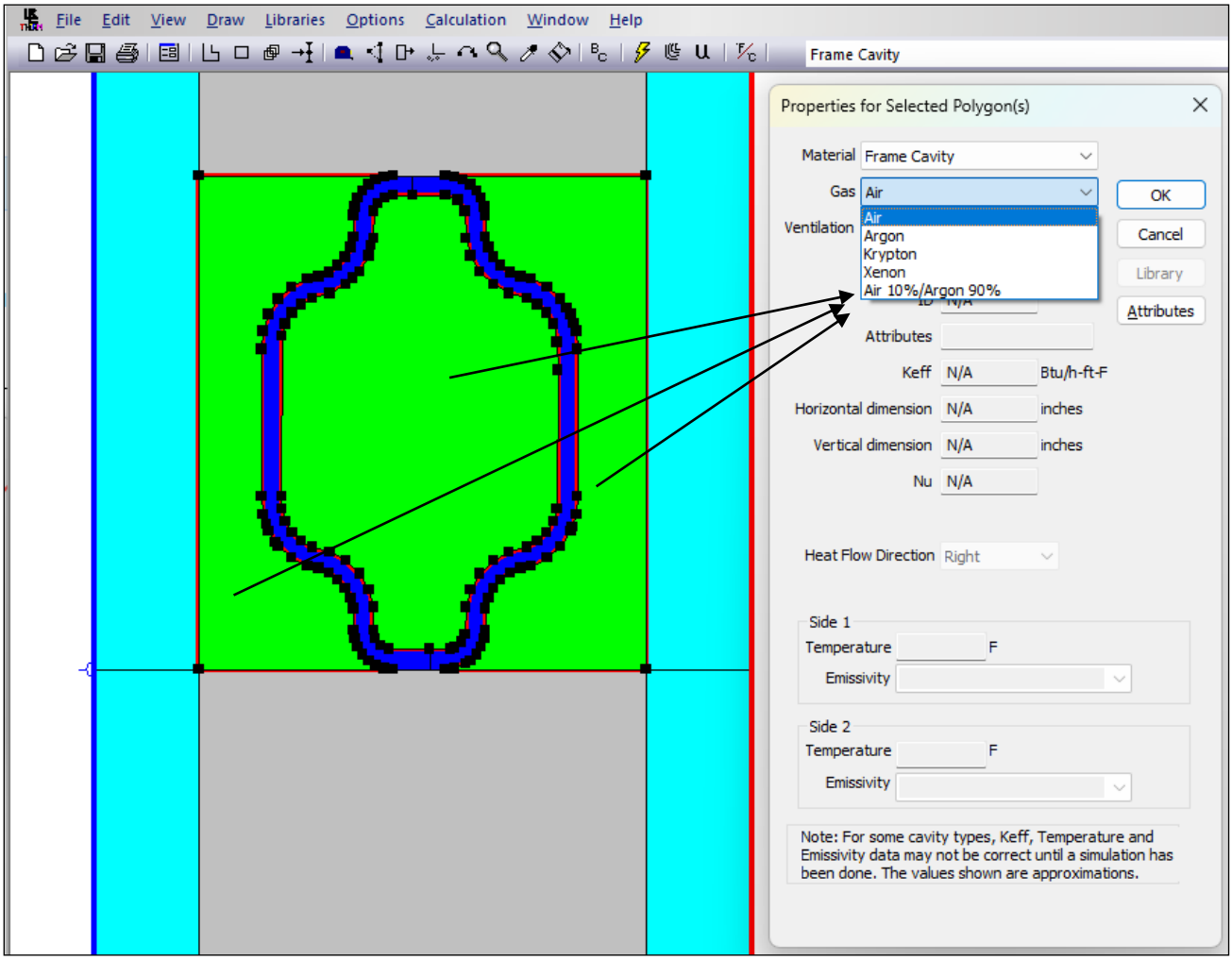


Figure 8-60. Select the Frame Cavities to open the Properties for Selected Polygons, where the appropriate Gas can be selected from the pulldown list, which is populated from the THERM Gas Library.

8.8.2. Simulated Divided Lites

A simulated divided lite is a glazing system that has elements attached to the inside and outside of a glazing system to give the appearance of true divided lites. The glazing system has a metal internal reinforcement, but is all one system. The materials applied/adhered to the outside of the insulating glazing unit can be optionally modeled.

Internal dividers, such as the dividers in the glazing system gap for simulated divided lites, are an exception to the rules for linking frame cavities to glazing cavities as discussed earlier in this section. Internal dividers are only modeled if the distance between the exterior sides of the divider and the inside surface of the glazing system are less than or equal to 3 mm. Therefore, even for contoured internal dividers, a simplifying assumption is made that the 5 mm rule is not applied. The space between the divider edge and the glass are modeled as frame cavities (with either air-filled or gas-filled as needed), and the boundary of that cavity extends to the top and bottom of the divider component. See the following examples for detailed descriptions.

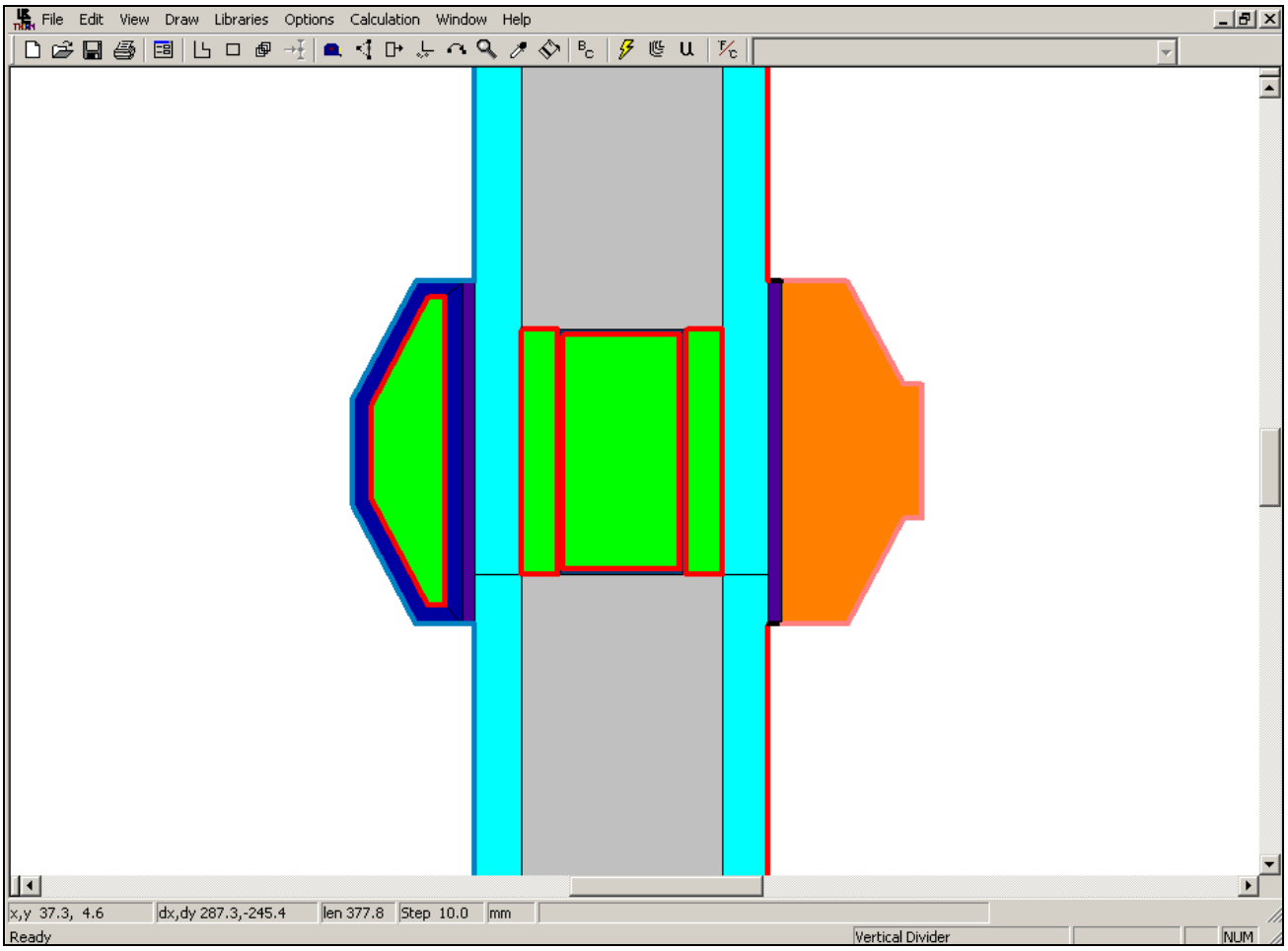


Figure 8-61. A simulated divided lite in THERM.

In THERM:

1. Set the **Cross Section Type** to "Vertical Divider".
2. Import the glazing system for the divider, which is the same glazing system as the rest of the product, with the following settings:
   - **Orientation** = Up
   - **Cavity height** = 1000 mm (39 inches)
   - **Sight line to bottom of glass** = height of the divider (in this example it is 19.05 mm [0.75 inches])
   - **Spacer height** = height of the divider (in this example it is 19.05 mm [0.75 inches)
   - **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
   - **Glazing System Height**: 150 mm (6.0 inches)
   - **Draw spacer** = Not checked

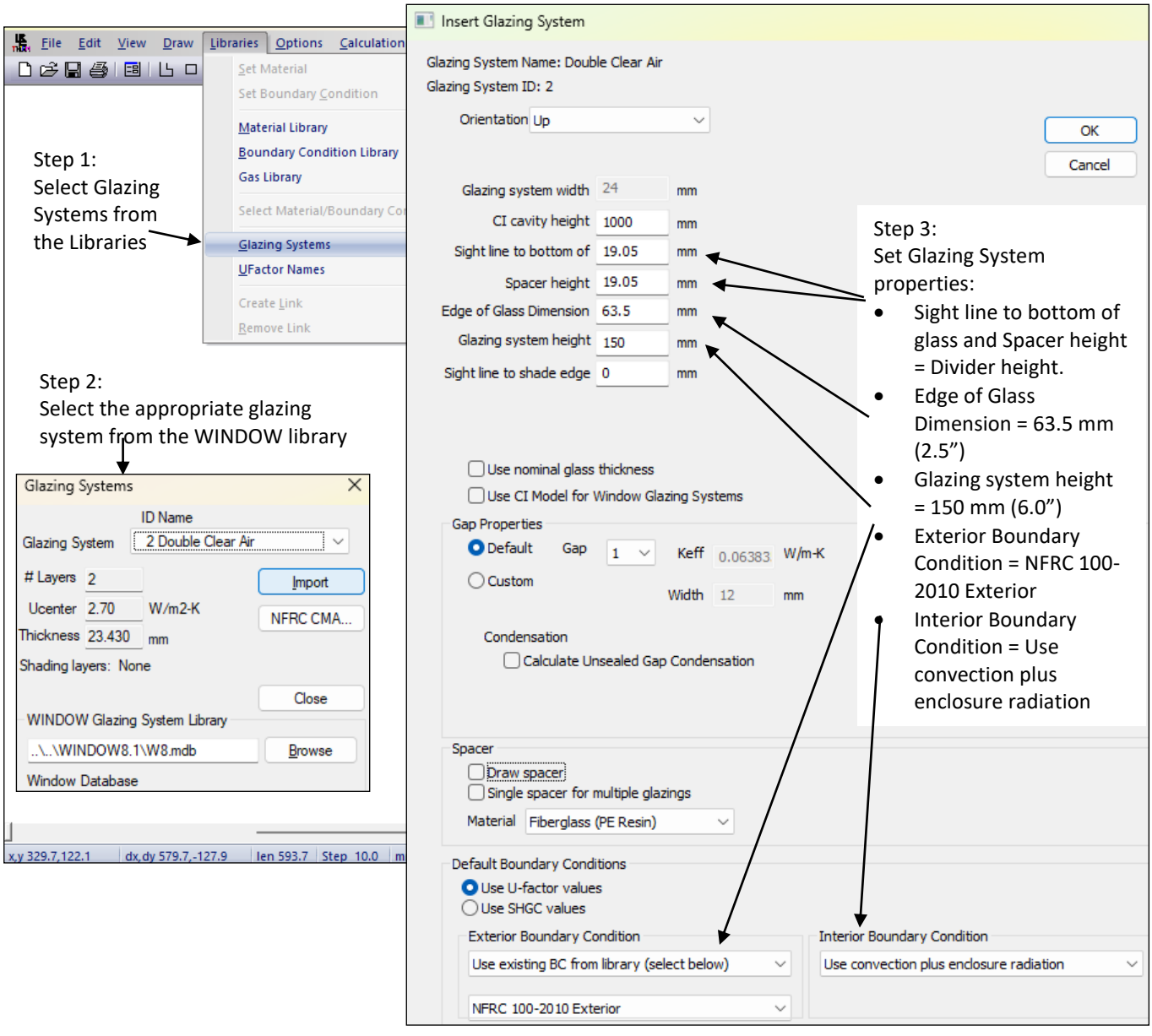


Figure 8-62. Import the first glazing system.

3. Import the glazing system again as an additional glazing system, below the first one (the locator does not have to be moved), but facing down this time. Use the following settings for this glazing system:
   - **Orientation** = Down
   - **Cavity height** = 1000 mm (39 inches)
   - **Sight line to bottom of glass** = 3.175 mm
   - **Spacer height** = 0
   - **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
   - **Glazing System Height**: 150 mm (6.0 inches)
   - **Draw spacer** = Not checked

   Insert the glazing system as an **Additional Glazing**.

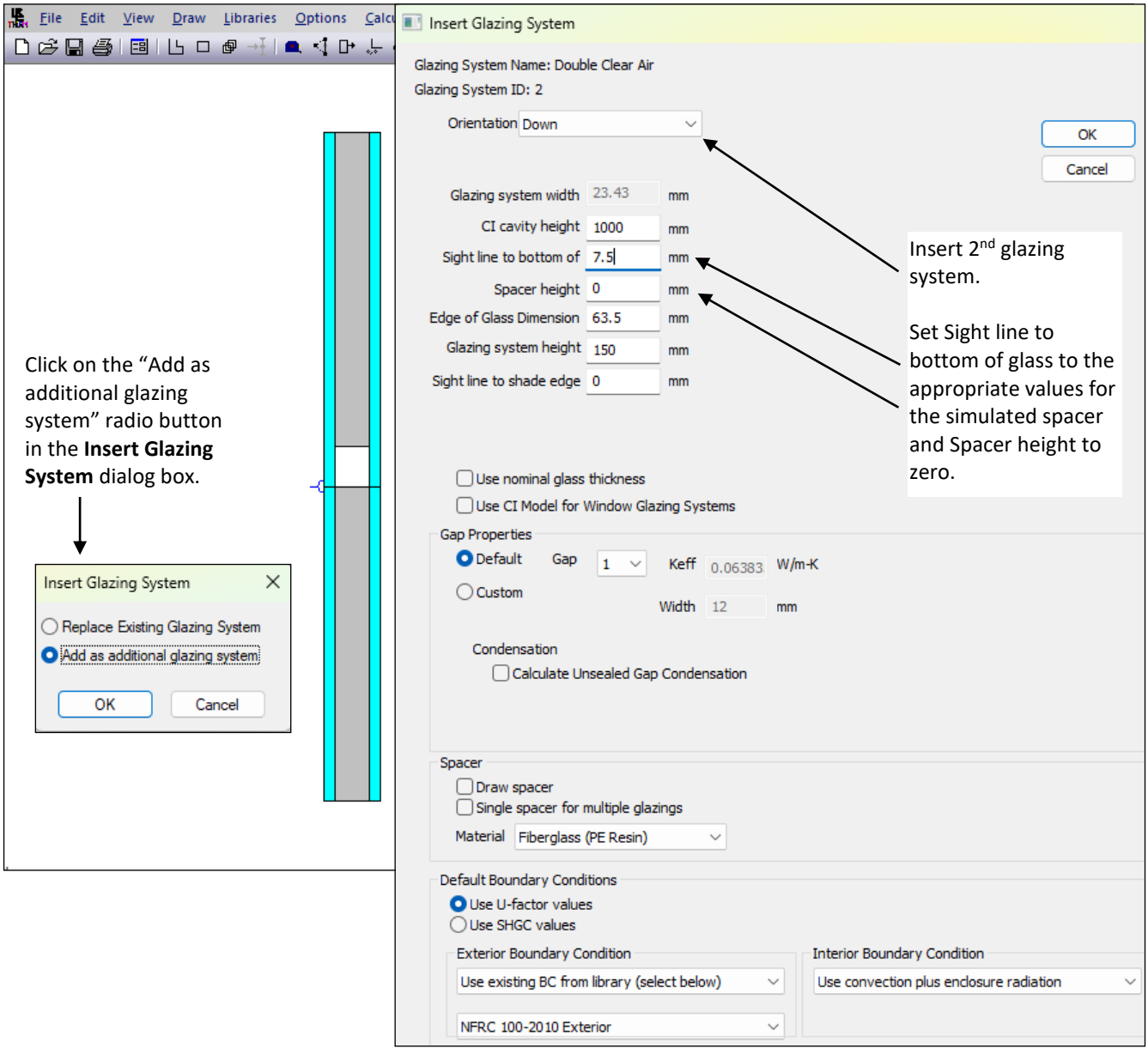


Figure 8-63. Import the second glazing system as an additional glazing system, facing down.

4. Draw (or copy and paste from another THERM file) the polygons in the cavity that represent the divider. The figure below shows the divider for this example drawn with the material set to Aluminum Alloy

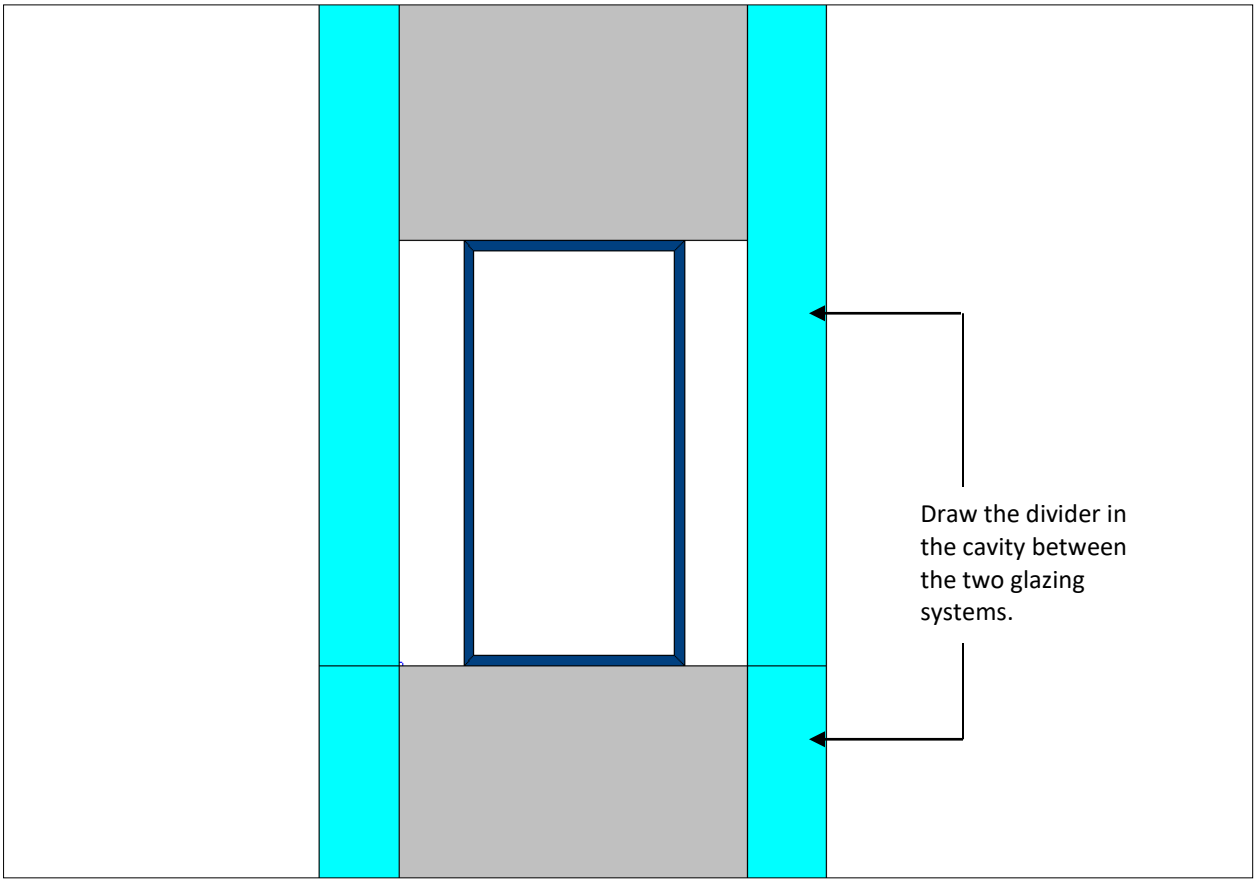


Figure 8-64. Draw the polygons to represent the divider.

5. Fill the cavities between the divider and the glass layers and inside the divider with the material "Frame Cavity". Divide the cavities up according to the 5 mm rule as necessary.

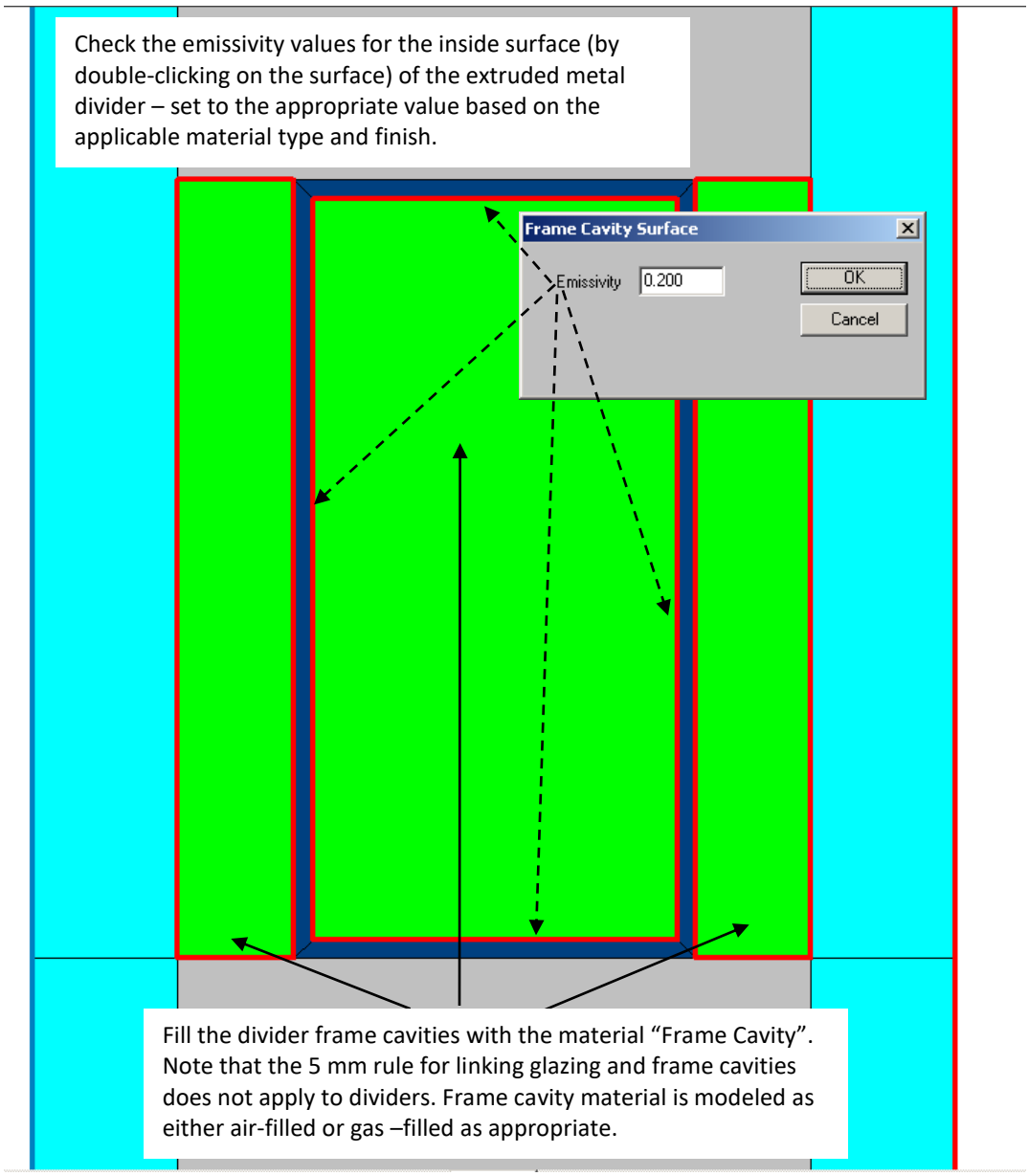


Figure 8-65. Fill the divider frame cavities with Frame Cavity.

6. Add the polygons for the simulated divided light elements on either side of the glazing system and assign the appropriate materials to them.

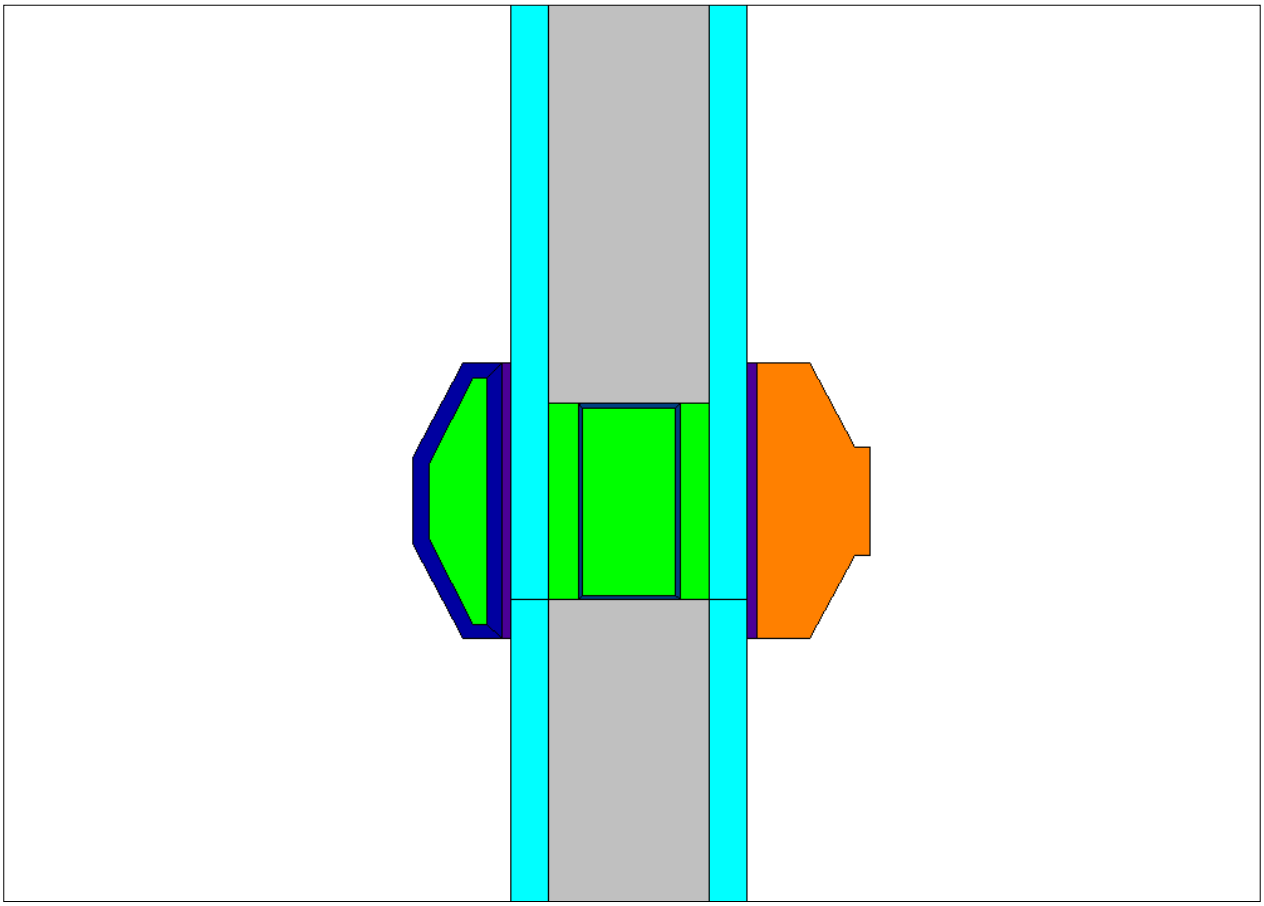


Figure 8-66. Add the polygons for the simulated divided lite elements on either side of the glazing system.

7. Define the boundary conditions, using the "AutoEnclosure" choice for the Radiation Model.

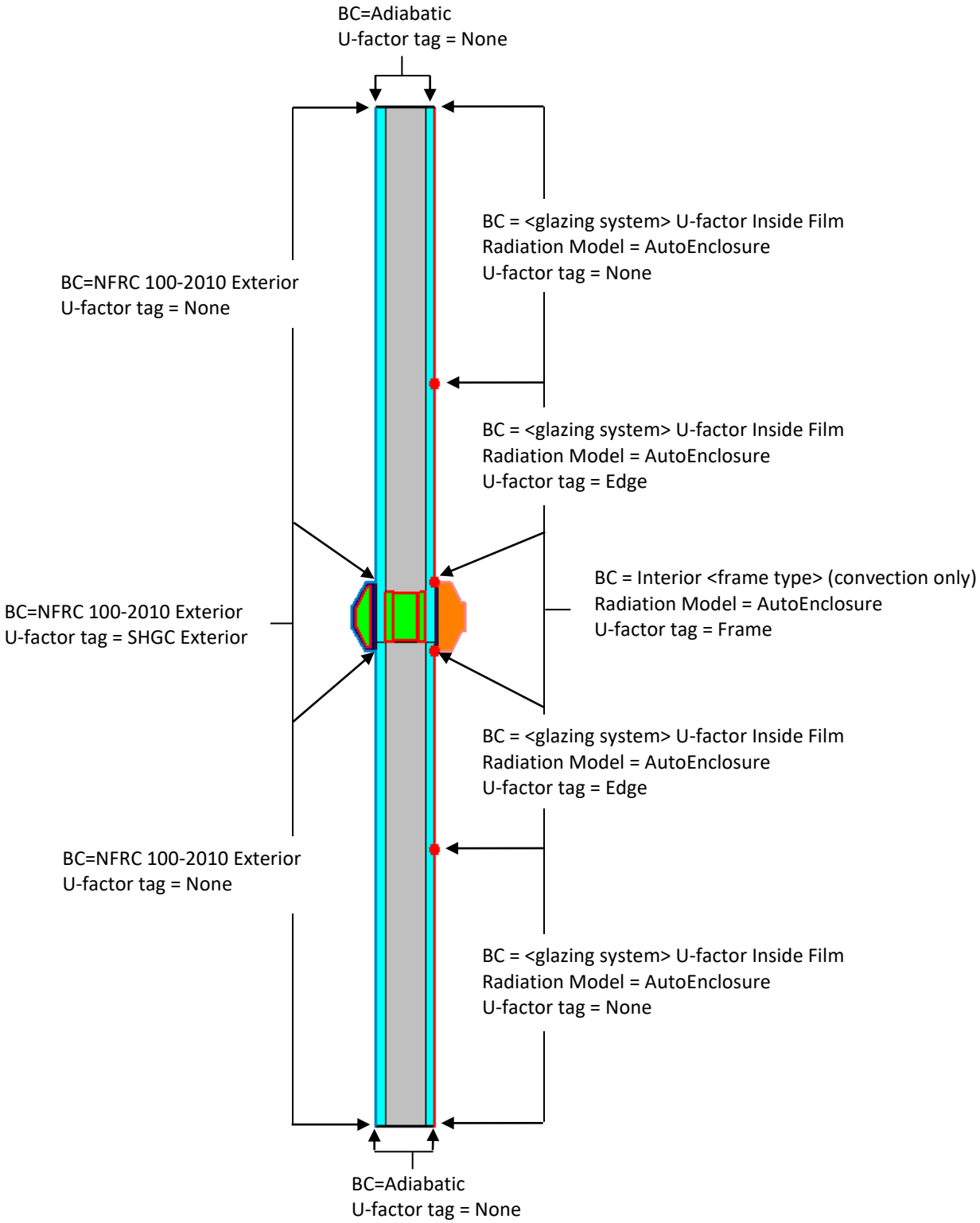


Figure 8-67. Assign the boundary conditions.

8. Calculate the results.

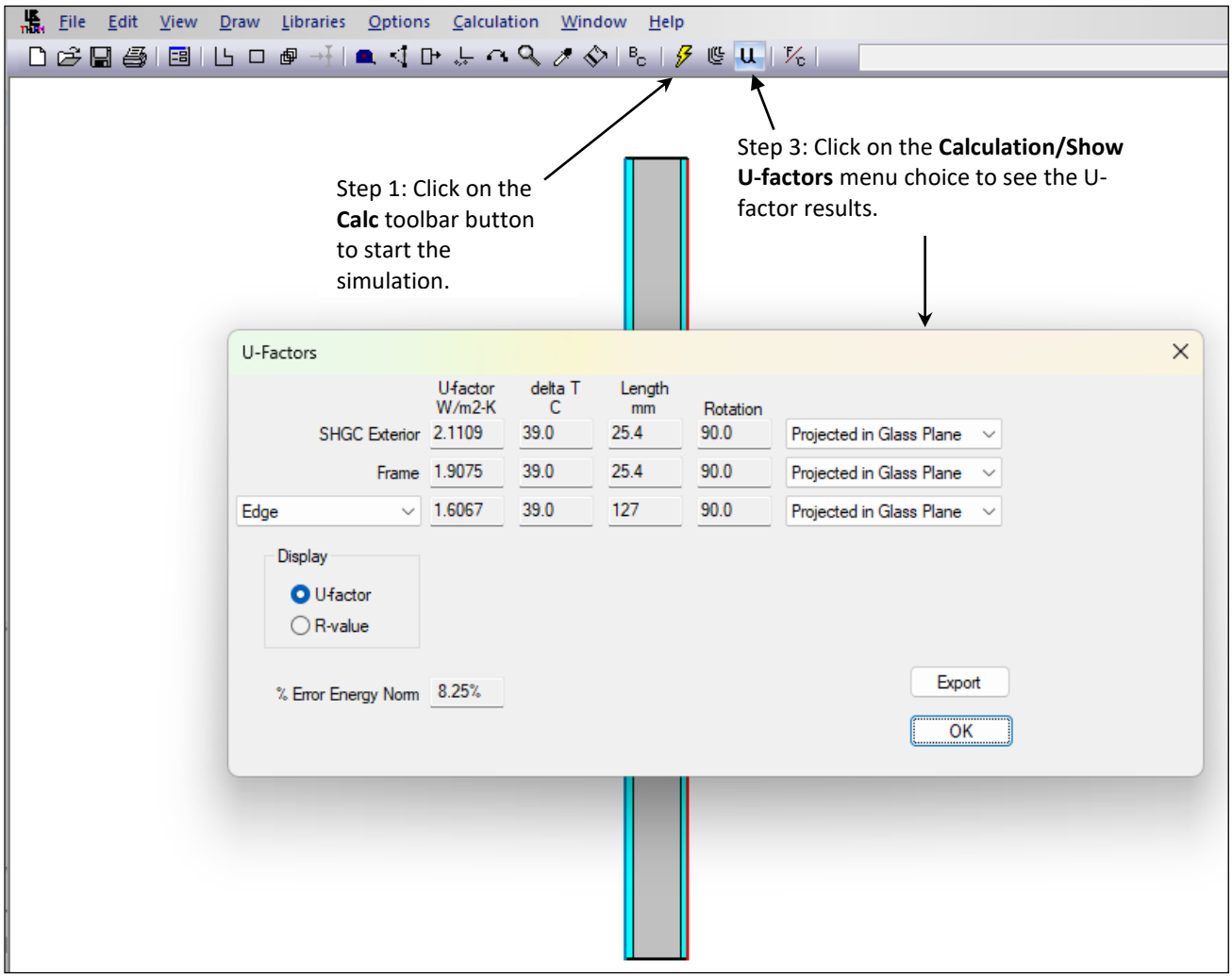


Figure 8-68. Calculate the results.

9. Save the file using the **File/Save As** menu choice.

10. Import the results to the WINDOW Divider Library, as shown below. See "Importing THERM files" in the *WINDOW User Manual* for more information about importing THERM files.

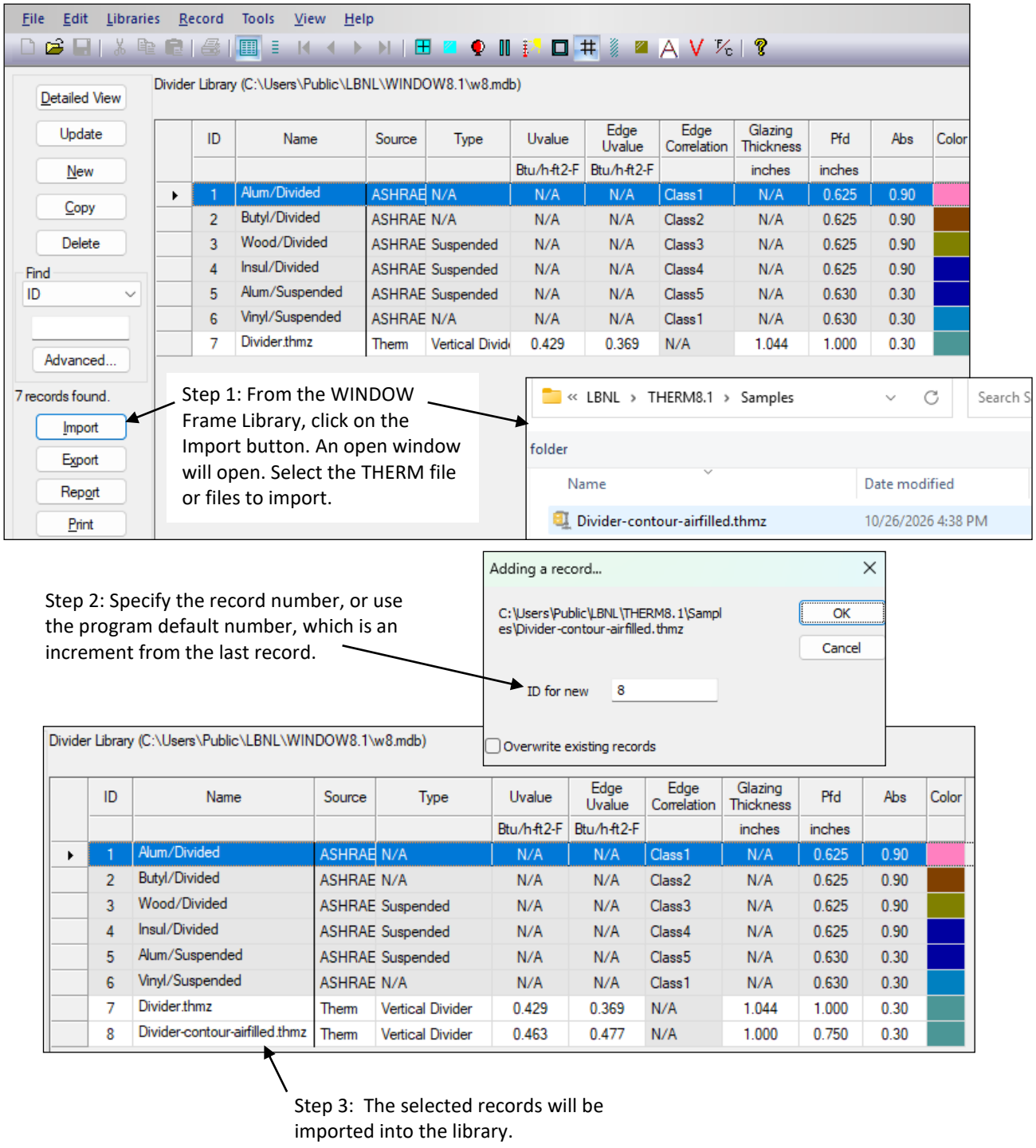


Figure 8-69. Import the THERM file into the WINDOW ***Divider Library****.*

11. Use the new divider in the calculation of the complete product values in the main screen of WINDOW.

8.8.3. True Divided Lites

The discussion below describes the methodologies in WINDOW and THERM for modeling dividers that are built as "true divided lites", which means that the dividers have separate glazing systems between the divider elements, as shown in the figure below.

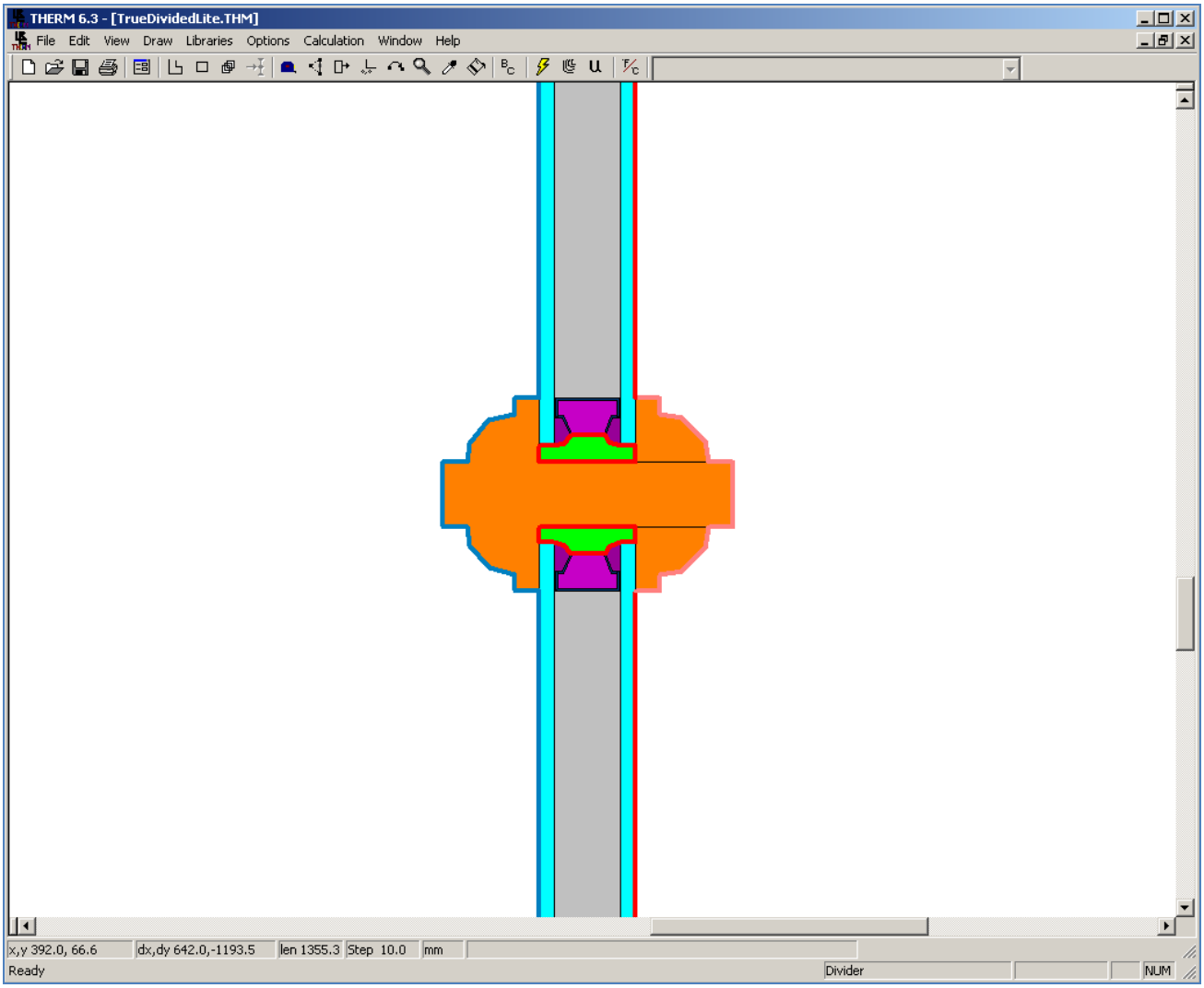


Figure 8-70. A true divided lite in THERM.

In THERM:

In this example, it is easier to start by drawing the polgyons that represent the divider, and then placing the glazing systems relative to those polygons.

1. Set the **Cross Section Type** to "Vertical Divider".
2. Draw the divider polygons as shown in the figure below.

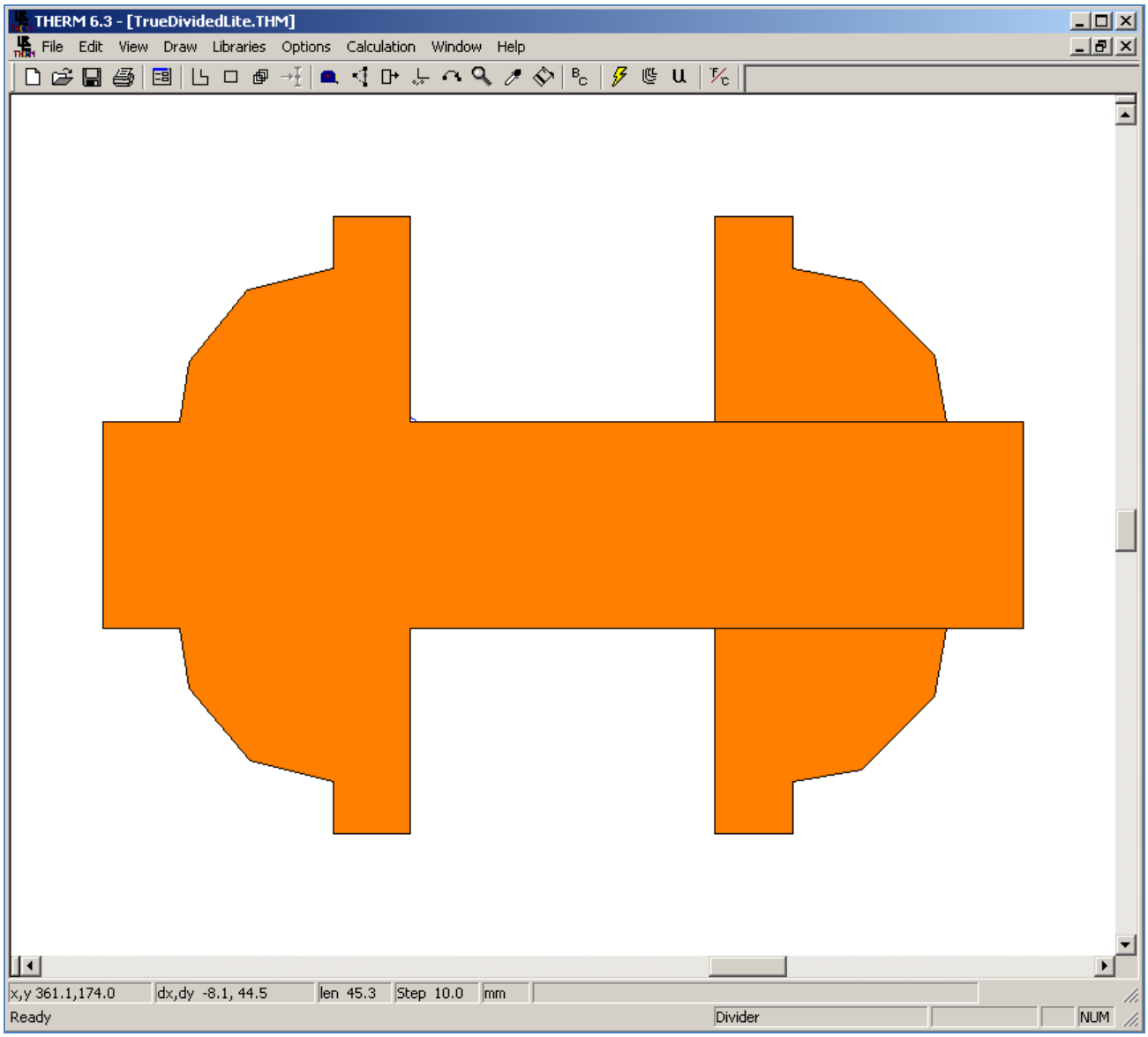


Figure 8-71. In THERM, draw the polygons for the divider elements that the glazing systems will fit into.

3. Import the first glazing system (in this case the top glazing system) for the divider, with the following settings:
   - Set the **Locator** in the appropriate location for where the glazing system will start, in this example 3.175 mm (0.125 inches) above the bottom of the divider
   - Orientation = Up
   - **Actual Cavity height** = 1000 mm (39 inches)
   - **Sight line to bottom of glass** = height of the divider (in this example it is 9.525 mm (0.375 inches)
   - **Spacer height** = height of the divider (in this example it is 9.525 mm (0.375 inches)
   - **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
   - Glazing System Height: 150 mm (6.0 inches)
   - **Draw spacer** = Not checked

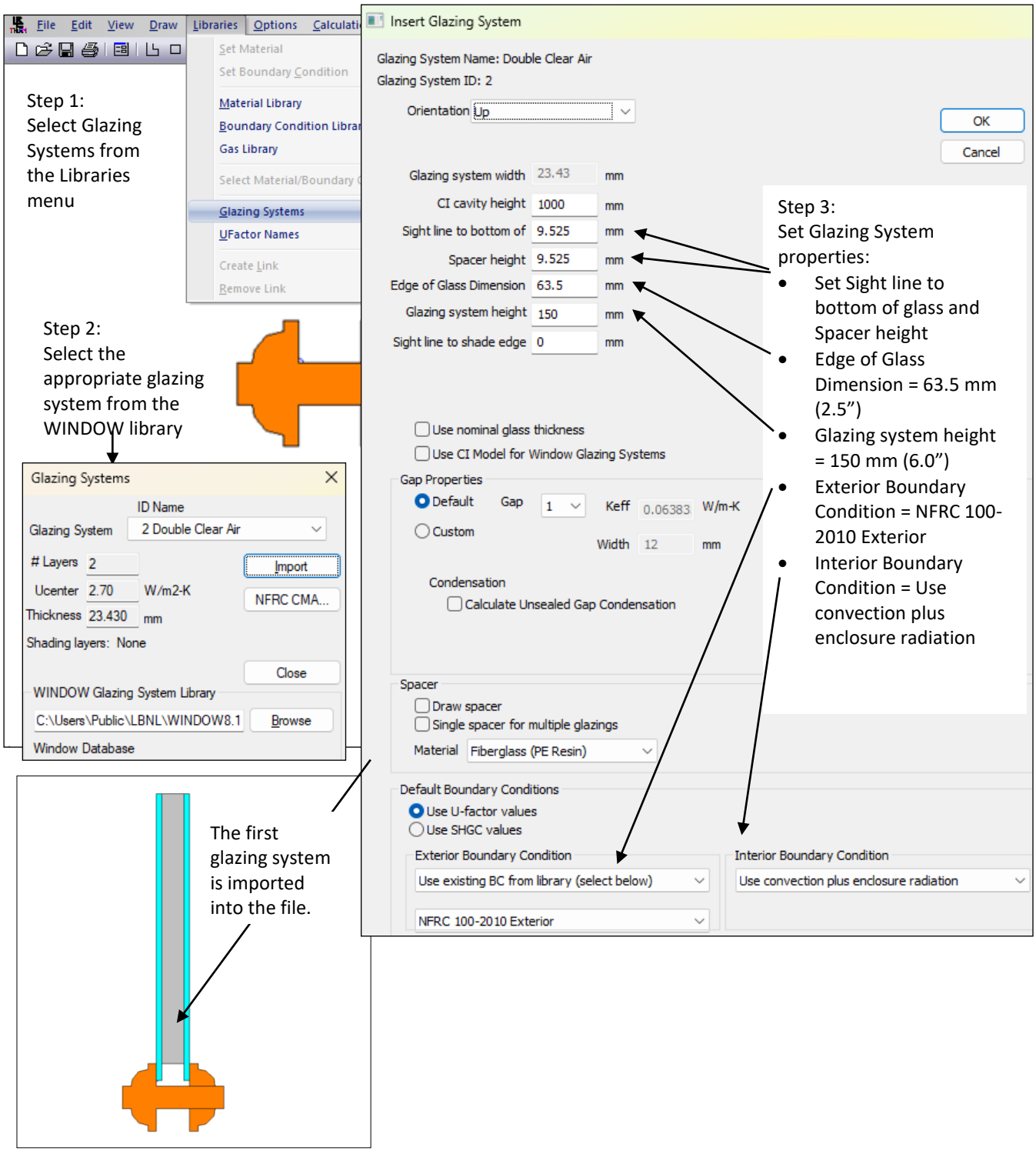


Figure 8-72. Import the first glazing system.

4. Import the glazing system again as an additional glazing system, facing down this time. Use the following settings for this glazing system:
   - Set the **Locator** in the appropriate location for where the glazing system will start, in this example 3.175 mm (0.125 inches) above the bottom of the divider
   - Orientation = Down
   - **Actual Cavity height** = 1000 mm (39 inches)

- **Sight line to bottom of glass** = height of the divider (in this example it is 9.525 mm (0.375 inches)
- **Spacer height** = height of the divider (in this example it is 9.525 mm (0.375 inches)
- **Edge of Glass Dimension** = 63.5 mm (2.5 inches)
- Glazing System Height: 150 mm (6.0 inches)
- **Draw spacer** = Not checked

Insert the glazing system as an **Additional Glazing**.

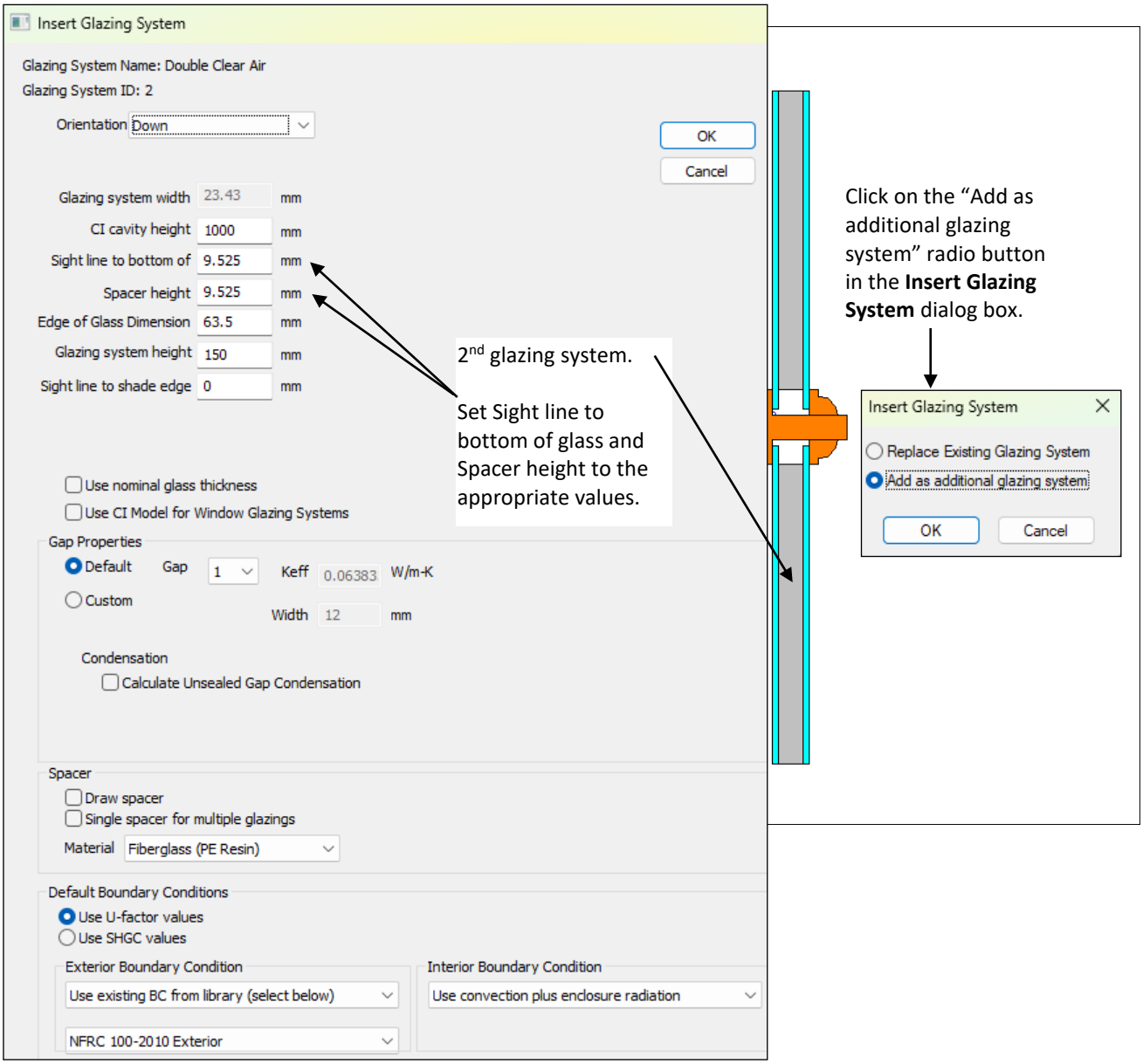


Figure 8-73. Import the second glazing system as an additional glazing system, facing down.

5. Add the spacers, sealants, desiccants, and frame cavities as appropriate.

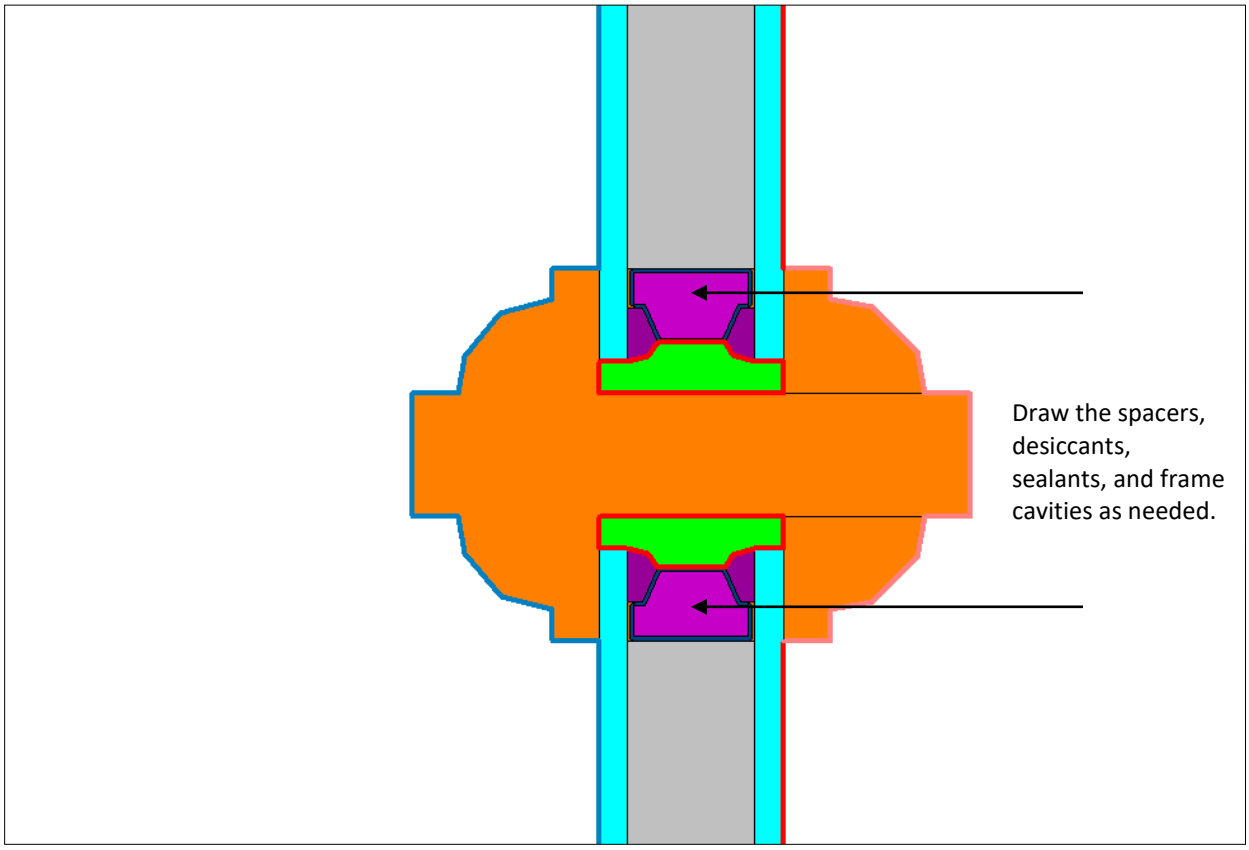


Figure 8-74. Draw the polygons to represent the divider.

6. Define the boundary conditions, using the "AutoEnclosure" choice for the Radiation Model.

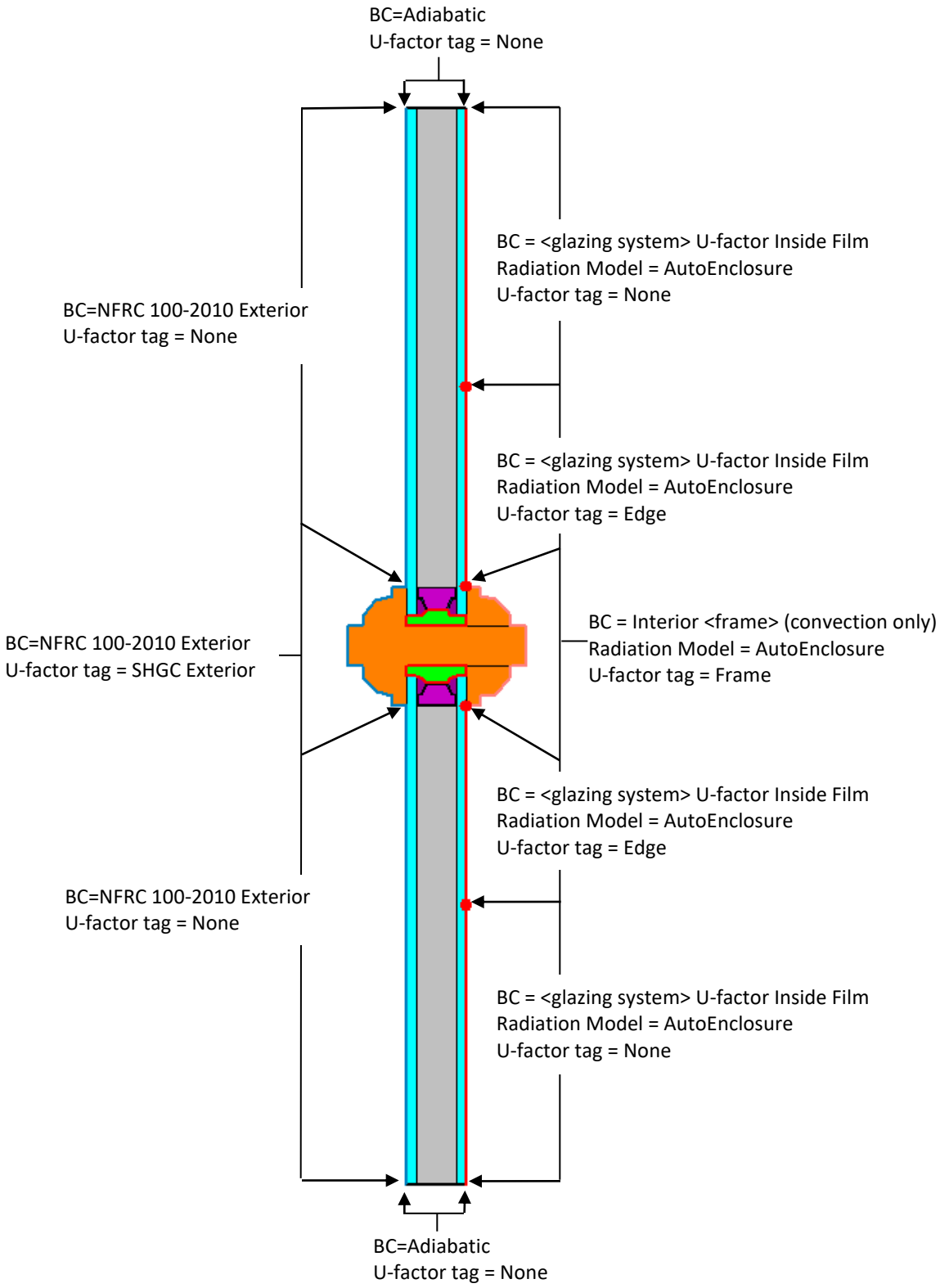


Figure 8-75. Assign the boundary conditions.

7. Calculate the results.

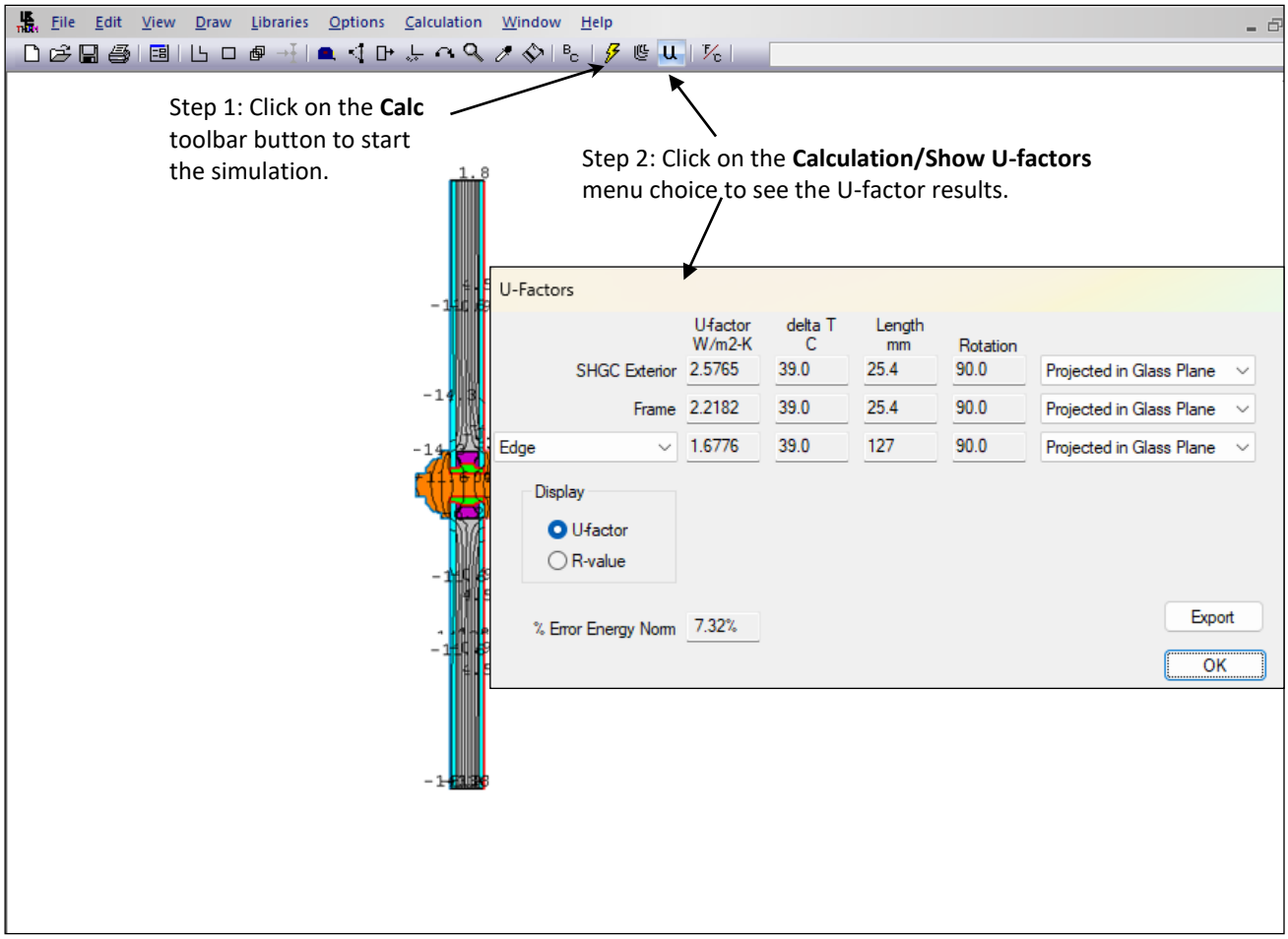


Figure 8-76. Calculate the results.

8. Save the file using the **File/Save As** menu choice.
9. Import the results to the WINDOW Divider Library, as shown below. See "Importing THERM files" in the *WINDOW User Manual* for more information about importing THERM files.

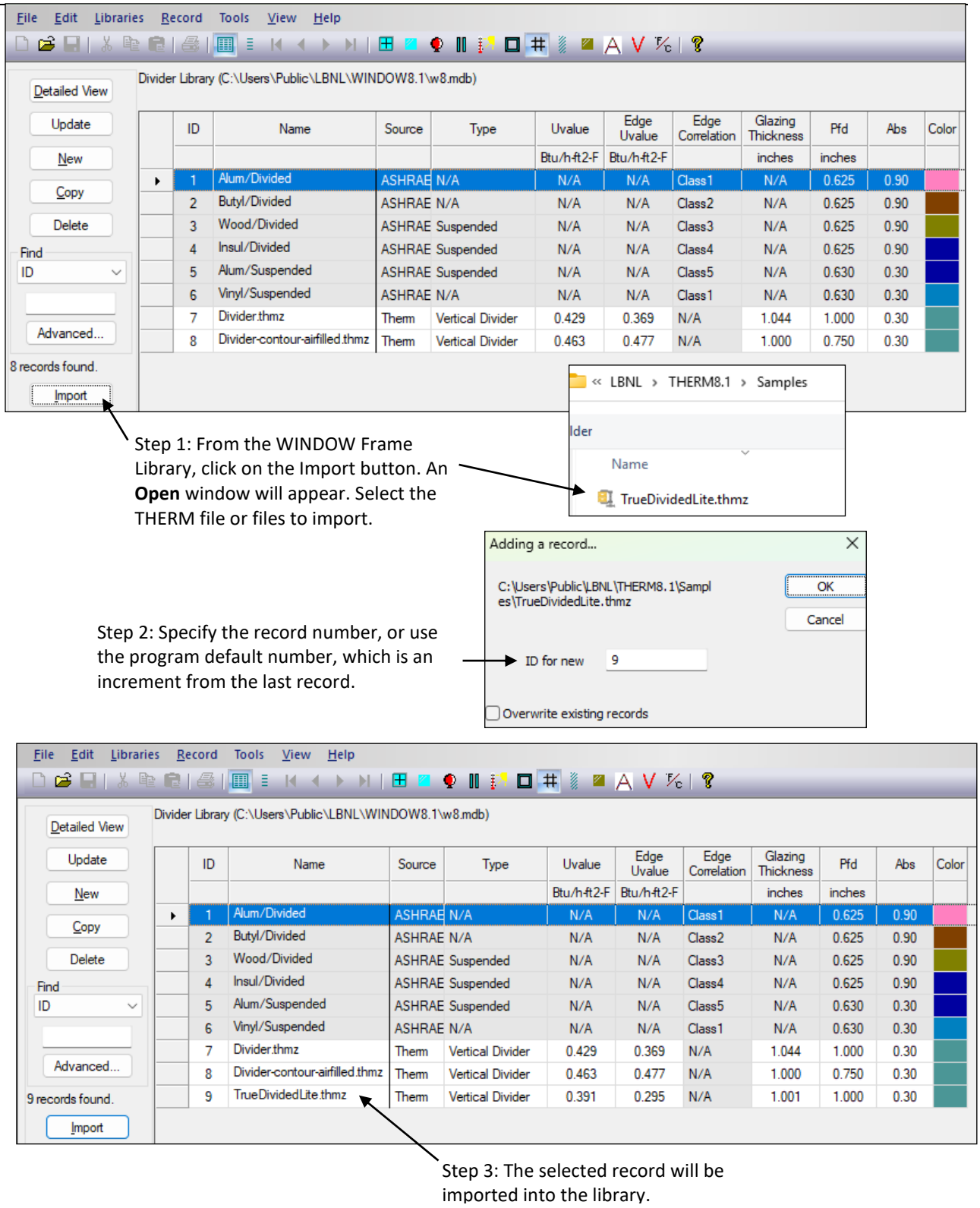


Figure 8-77. Import the THERM file into the WINDOW ***Divider Library****.*

10. Use the new divider in the calculation of the complete product values in the main screen of WINDOW.

8.9. Spacers

8.9.1. Overview

THERM has the capability to model spacers in great detail, so that modeling of spacer effective conductivity is no longer needed. Spacer models can be easily reshaped in THERM, and the program's cut and paste feature allows spacers to be copied into each cross section as needed. A library of spacer models can be produced for each spacer type. See the "Adding a Custom Spacer" in this manual. A sample spacer, spacer.thm, is included in the Samples directory when THERM is installed.

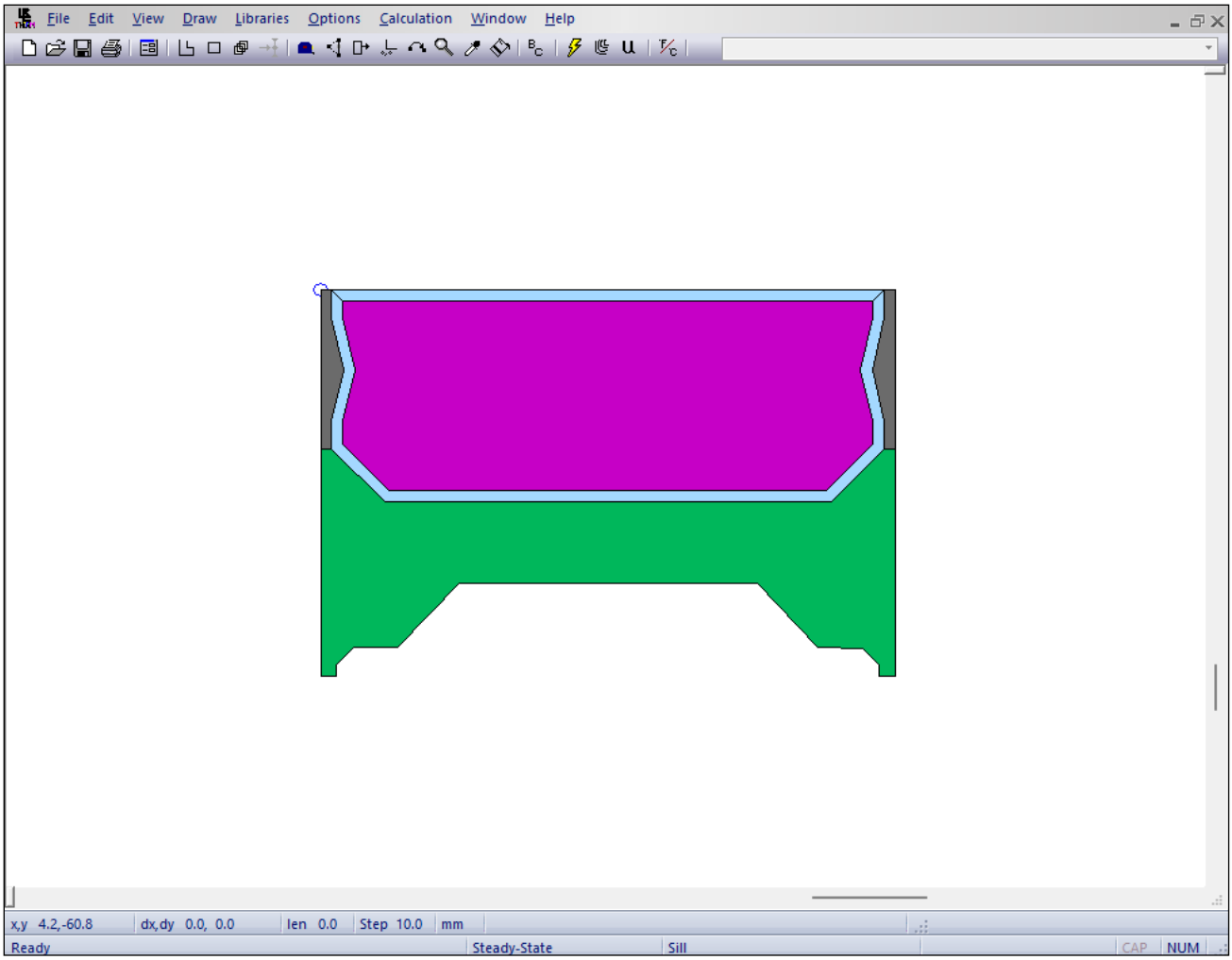


Figure 8-78. Spacer.thm sample file.

8.9.2. Linking Glazing Cavity properties (imported from WINDOW) for Open Spacers

The properties of a glazing cavity can be linked to another polygon in order to properly model spacers that are open to the glazing system cavity. Section "Linking Materials Properties of Polygons" in this manual, which explains this methodology in detail.

Follow the rules in Section 8.4.4 to determine whether the cavities in the open spacer cavity should be linked to the glazing system cavity.

To Link the properties of two materials, follow these steps:

- Select the polygon that is to linked to another polygon
- Select the **Libraries/Create Link** menu choice.
- The cursor will become an Eye Dropper. Click the Eye Dropper cursor in the polygon to be linked to. The material properties of the first polygon are not linked to the material properties of the second polygon.

When using the multiple glazing calculation option, THERM will automatically use the glazing system cavity properties for each glazing option for the linked polygon.

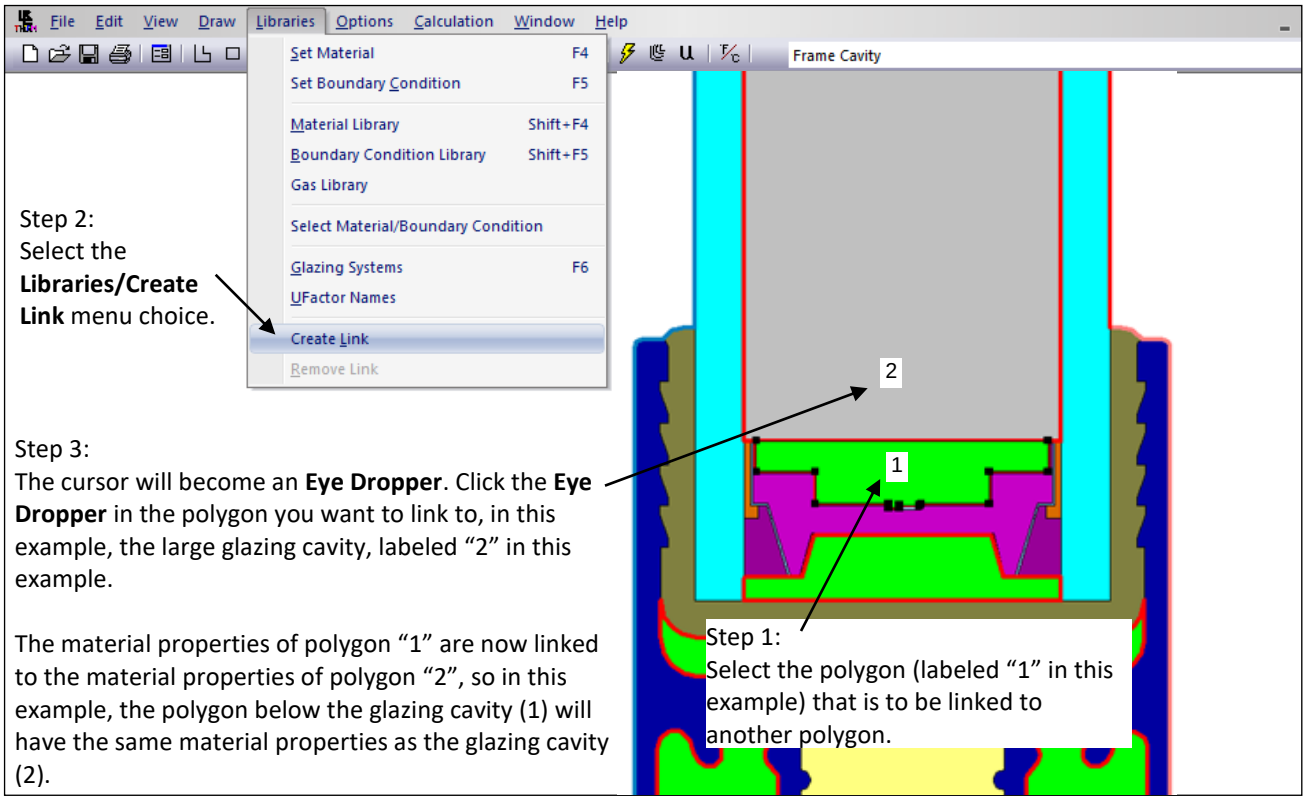


Figure 8-79. Link the open spacer cavity to the glazing system cavity using the Library/Create Link feature.

Another example of an "open" spacer is shown below. For sealed glazing systems (such as an IG) the 5 mm rule for linking frame cavities and glazing cavities does not apply. So in the example below, the frame cavity inside the spacer as well as the two small cavities on either side of it, all of which are connected to the glazing cavity, are linked to the glazing cavity even though the connection (throat) to the glazing cavity is less than 5 mm.

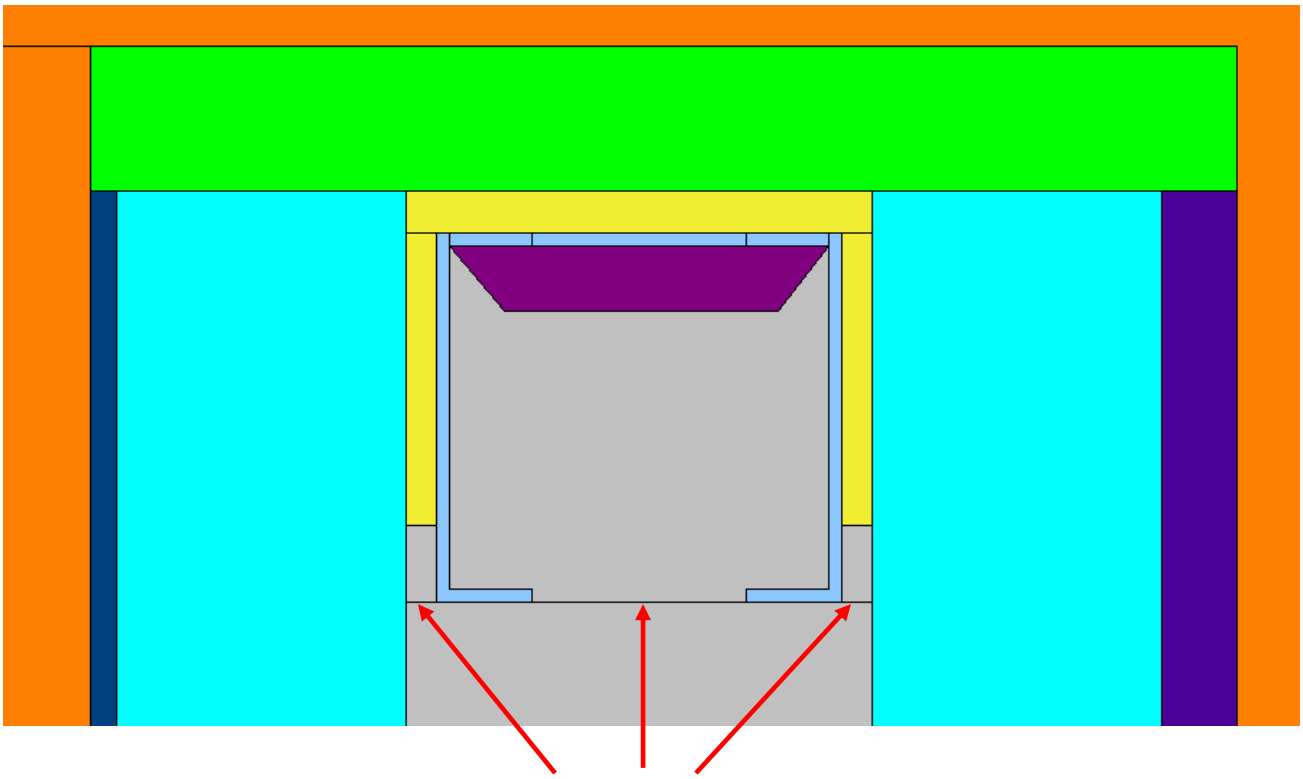


Figure 8-80. Because the "throat" between the spacer opening and the glazing system cavity is < 5 mm, the two cavities are not linked.

8.10. Non Continuous Thermal Bridge Elements

Bolts, skip and debridge thermal break, including partially de-bridged thermal break material, and thermally slotted cross section can be included in the model using the concept of *isothermal planes*. The isothermal planes methodology calculates an effective conductivity of the bridging material based on area weighting the sections of the product with and without thermal bridging material based on the bridging material spacing dimensions. This method is also valid for other regularly spaced thermal bridges such as skip-and-debridged systems.

The effect on the performance of a curtain wall system due to bolts is explained in detail in an ASHRAE paper published in 1998 entitled *"The Significance of Bolts in the Thermal Performance of Curtain-Wall Frames for Glazed Facades", by Brent Griffith, Elizabeth Finlayson, Mehrangiz Yazdanian and Dariush Arasteh*.

The THERM model to be simulated for the final result is one in which the actual materials of the thermal bridging elements are replaced with a user-defined material having an effective conductivity which represents the area-weighted value that combines the bridging and non-bridging elements.

The figure below illustrates an example of a curtain wall system which would require that the thermal bridging elements, in this case the bolts, be modeled using the isothermal planes method.

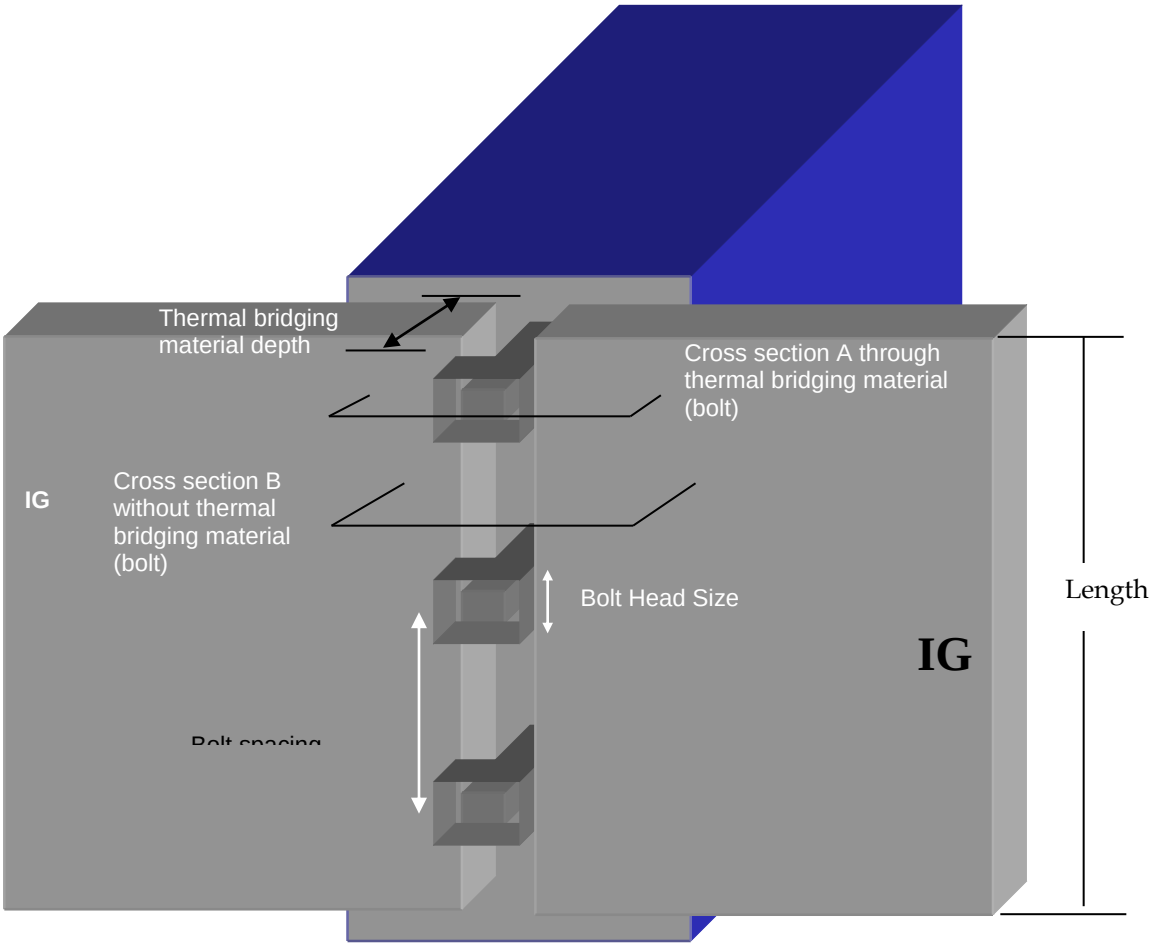


Figure 8-81. Example of a curtain wall system with regularly spaced bolts which act as thermal bridges.

8.10.3. Modeling Steps

The steps for constructing the final THERM model to be simulated are the following:

1. Draw the THERM model without the thermal bridging material.
2. Determine the conductivities of the materials that the thermal bridging material replaces.
   - Conductivities of *materials* can be obtained from the THERM Material Library
   - Conductivities of *air-filled* cavities (such as frame cavities) are assumed to be 0.024 W/m-K (or 0.014 Btu/hr-ft-^{o}F).
3. Using a cross-section that contains the non-thermal bridging material, measure the depths of each element of the non-thermal bridging material that will have a different thermal conductivity in the non-bridging cross section.
4. Use the conductivities of the non-thermal bridging materials and depths of the non-thermal bridging materials in Equation 2 below to determine the Resistance (R) for each non-thermal bridging element.
5. Sum the resistances (Rt) and divide by the total depth of the non-thermal bridging elements to obtain Kn, as shown in Equation 3, to calculate the conductivity of the non-thermal bridging elements
6. Calculate the fraction of thermal and non-thermal bridging material along the length of the façade using Equations 4 and 5.
7. Calculate the final effective conductivity value for the thermal bridging elements using Equation 1.
8. In THERM, define a new material with the Keff value derived in Step 7, and assign it to the cross section polygons that represent the thermal bridging elements.
9. Simulate the model.

8.10.4. Equations

Calculate the effective conductivity of thermal bridging elements (e.g., bolts, screws, etc.)

Keff= Fb*Kb + Fn*Kn *Equation 1*

where

Fb = Fraction of the Length which contains the thermal bridging elements (see equation 4 below)

Fn = the fraction of the Length which contains non-thermal bridging elements(see equation 5 below)

Kb = conductivity of the thermal bridging elements

Kn = conductivity of the non-thermal-bridging elements
(from the sum of the resistances, Rt, of individual elements from Equation 2 below)
Assume a default value of 0.024 W/m-K for air cavities.

This methodology should be applied with the following caveats:

- If less than 1% (to obtain percentage, multiply fraction by a 100) of the Length is made of thermal bridging elements (such as stainless steel), i.e., Fb < 0.01, do not model the thermal bridging elements.
- If between 1% and 5% of the Length is made of thermal bridging elements (0.01 <= Fb <= 0.05) and if the conductivity of the thermal bridging elements is more than 10 times the conductivity of the thermal break, model the thermal bridging elements using the keff calculated in Equation 1.
- If more than 5% of the length is made of thermal bridging elements (Fb > 0.05), model the thermal bridging elements using the keff calculated in Equation 1.
- Calculate the total resistance of the non-thermal bridging elements, Rt, by summing individual resistances (non-thermal bridging element conductivity) for each non-thermal bridging element using the formula:

Rt = Σ(D / k) *Equation 2*

Where:

Rt = Sum of the thermal resistances of the individual non-thermal bridging material. Units: $m^2 \cdot K/W$ (SI), or $hr \cdot ft^2 \cdot °F/Btu$ (IP)

D = Depth of the individual non-thermal bridging elements that will be substituted by the calculated effective conductivity. Units: m (SI), or ft (IP), or (in) (alternate IP)k = conductivity of the individual non-thermal bridging elements that will be substituted. Units: $W/m \cdot K$ (SI), or $Btu/hr \cdot in \cdot °F$ (IP), or $Btu \cdot in/hr \cdot ft^2 \cdot °F$ (alternate IP)

Therefore:

Kn=Dt/Rt *Equation 3*

Where:

Dt = Total depth, which is the sum of the depths of the individual non-thermal bridging elements

Calculate the fraction of thermal bridging material to non thermal bridging material as follows:

Fb=Wb / Sb (%Fb=Fb ·100) *Equation 4*

Fn=1-Fb *Equation 5*

Where:

Wb = Bridging material width

Sb = Bridging material spacing

8.10.5. Example 1: Bolts in Curtain Wall

Note: This example is only presented in SI units and is not translated into IP units.

The following figures show two cross sections of the curtain wall in Figure 8-81. Figure 8-82 represents the cross section of the curtain wall where the bolt occurs (screw threads should be averaged and not drawn explicitly), and Figure Figure 8-83 represents the cross section of the curtain wall where the bolt does not occur. The geometry of the cross-section in Figure 8-82 would be used for the final THERM run, and the conductivity of the materials used to define the bolt would be changed to the value derived from the methodology explained in this section. The geometry in Figure 8-83 is drawn only to obtain the conductivity values for calculating the conductivity of this "averaged" material.

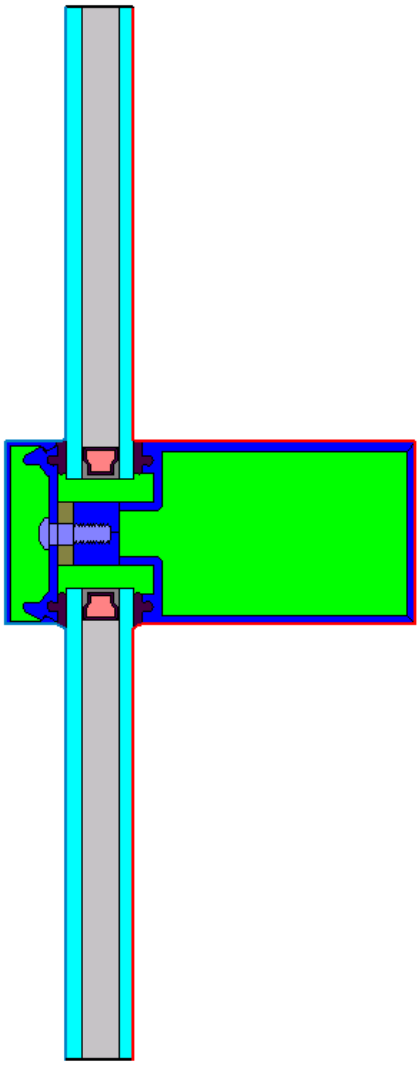


Figure 8-82. THERM cross section where the bolt occurs (curtain wall bolt.thm).

Figure 8-83 shows the conductivity values for the four materials that must be obtained for the calculation. Material 1 and 4 are air cavities, and the conductivity is assumed to be 0.024 W/m-K.

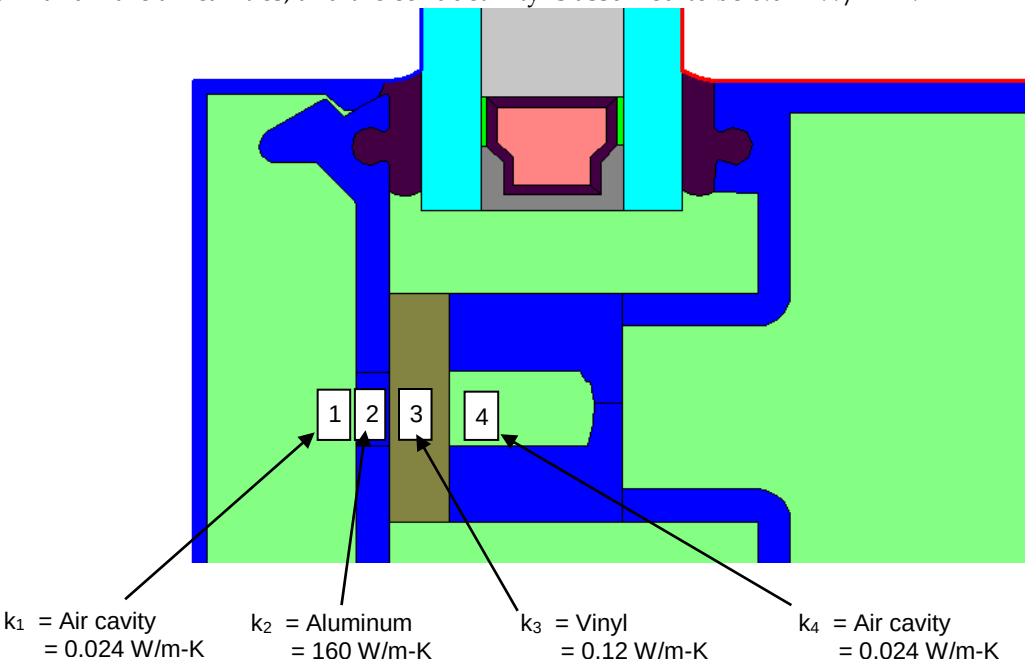


Figure 8-83. Materials in the non-bridging material cross section for which conductivities must be obtained.

Figure 8-80 shows the depths of each of the thermal bridging elements that are used in the Keff calculation.

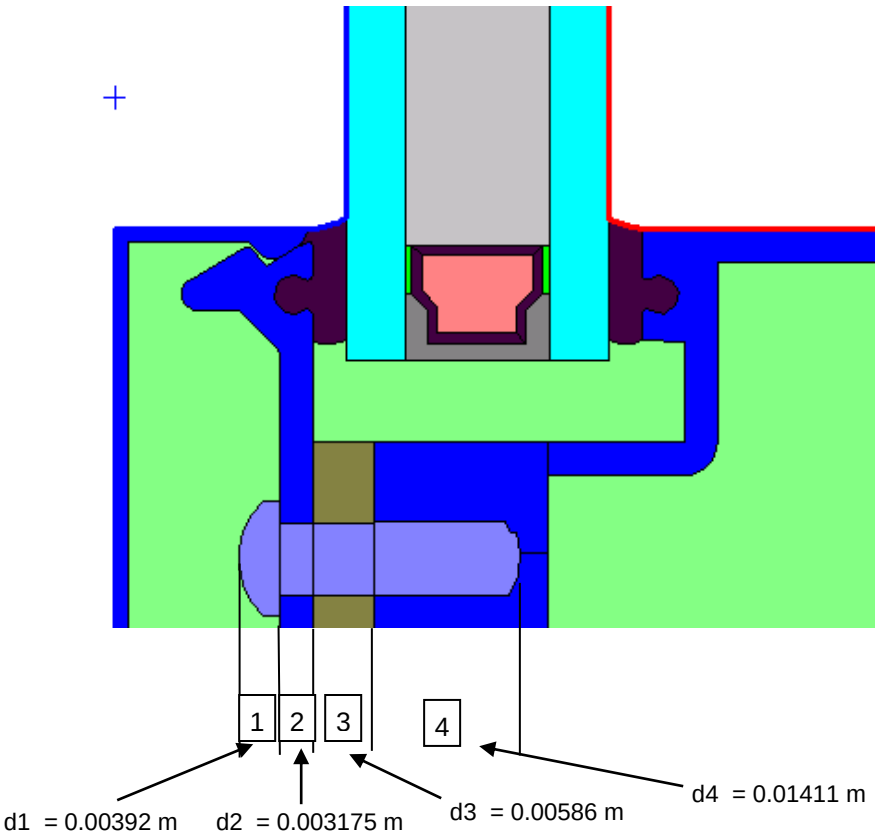


Figure 8-84. Material depths for the thermal bridging materials.

Table 8-1 shows the conductivity and depth values used to calculate the R for each non-thermal bridging element using Equation 2.

Table 8-1

Cross Section Element	Material	**Conductivity [W/mK]**	**Depth (m)**	**R [m2K/ W]**
1	Air cavity (default value)	0.024	0.00392	0.16333
2	Aluminum (conductivity from THERM Material Library)	160	0.003175	0.0000198
3	Vinyl (conductivity from THERM Material Library)	0.12	0.00586	0.049
4	Air cavity (default value)	0.024	0.01411	0.587917
	Total		**0.02706**	**0.800103**

Calculate Rt as follows:

$R_t = \Sigma(d/k)$

$= (d_1/k_1) + (d_2/k_2) + (d_3/k_3) + (d_4/k_4)$

$= (0.00392 / 0.024) + (0.003175 / 160) + (0.00586 / 0.12) + (0.01411 / 0.024)$

$= 0.800103\ m^2K/W$

$D_t = 0.00392\ m + 0.003175\ m + 0.00586\ m + 0.01411\ m$

$= 0.02706\ m$

Calculate the conductivities as follows:

$Kn = D_t/R_t$

$= 0.02706 / 0.800103$

$= 0.033821\ W/m \cdot K$

Kb = 14.3 W/m ·K (stainless steel)

Calculate the fraction of bolt to no bolt as follows:

Wb = Bolt head width

= 11.1 mm

Sb = Bolt spacing 12''

= 304.8 mm

Fb = Wb / Sb

= 11.1 mm / 304.8 mm

= 0.036 (%Fb=0.036 ·100 = 3.6%)

Fn = 1-Fb

= 1- 0.036

= 0.964

Calculate the new Keff, which will be used in THERM as follows:

Keff = Fb*Kb + Fn*Kn

Keff = (0.036 * 14.3)+(0.964 * 0.033827)

= 0.55 W/m·K

In THERM, create a new material in the Material Library with this Keff. In the THERM cross section, the bolt material should be changed from Stainless Steel to this new material. The resulting cross section is a 2-D thermal equivalent of the cross section with and without the thermal bridging material.

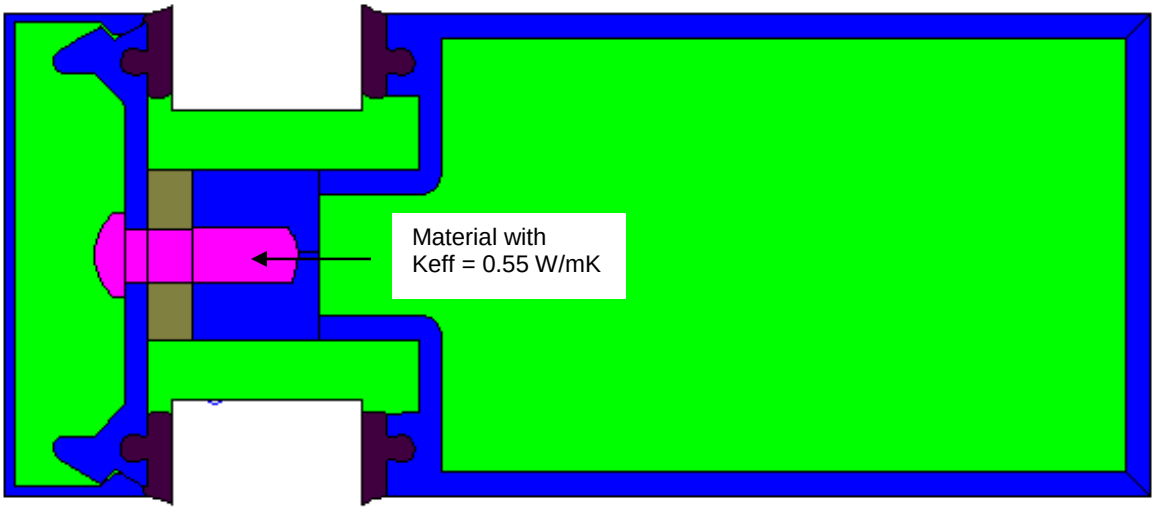


Figure 8-85. Final THERM model with boundary conditions defined.

8.10.6. Example 2: Thermally slotted cross-section

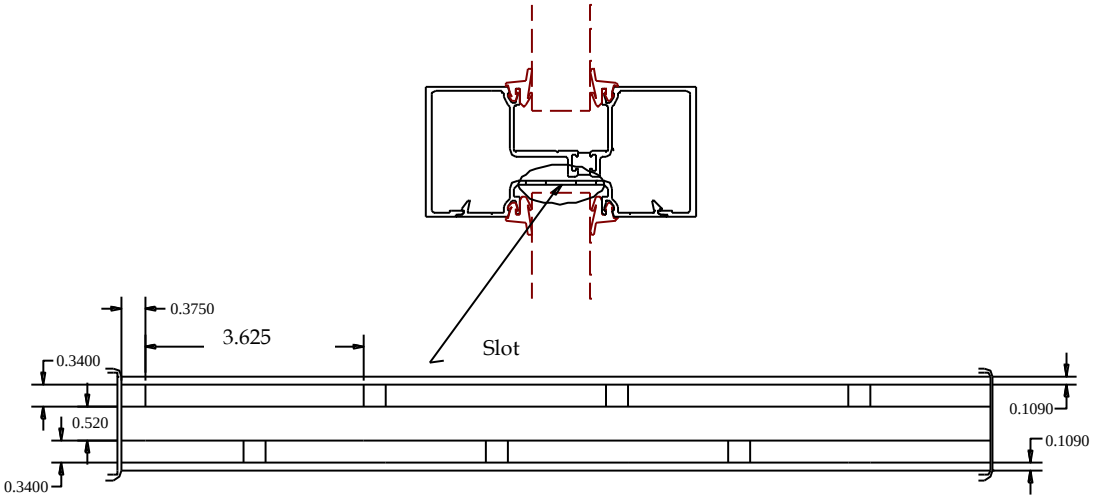


Figure 8-86. DXF for thermally slotted cross section.

Step I

Skip = 0.009525 m (0.375 in)

Slot (Air) = 0.092075 m (3.625 in)

Interval = 0.092075 m (3.625 in) + 0.009525 m (0.375 in)

= 0.1016 m (4 in)

Fb = 0.009525 m/ 0.1016 m

= 0.094

Fn = 1-Fb

= 1- 0.094

= 0.906

Percent of thermal bridge = (Fb)*100

= 0.094 * 100

= 9.4%

Because the thermal bridge is 9.4% of the length of the façade, the skip-and-debridge needs to be calculated using the isothermal plane procedure. Note: The rest of the example will be in SI units only, with no IP unit translation

Kb = 160 W/m-K (conductivity of skipped debridge, in this case Aluminum)

Rt = $\sum$ (Depth/ conductivity)

= D_d/k_d

= (0.0086 m/0.024W/m-K)

= 0.35833 m^2-K/W

where Depth is length of thermal bridge in a direction of heat flow, and the air is assumed to have the conductivity of 0.024 W/m-K

Kn = total depth/Rt

= 0.0086m/ 0.35833 W/m-K

= 0.024W/m-K

Keff = Fb*Kb +Fn*Kn

= 0.094*160 W/m-K + 0.906* 0.024W/m-K

= 15.062 W/m-K

To convert to IP:

Keff = 15.062 W/m-K * 0.57782

= 8.703 Btu/hr-ft-^{o}F

or in alternative IP units,

Keff = 15.062* 0.57782 * 12 in/ft

= 104.436 Btu-in/hr-ft^2-^{o}F

Step-2

Replace the strip of air-aluminum-air with new keff material of 15.078 W/m-K

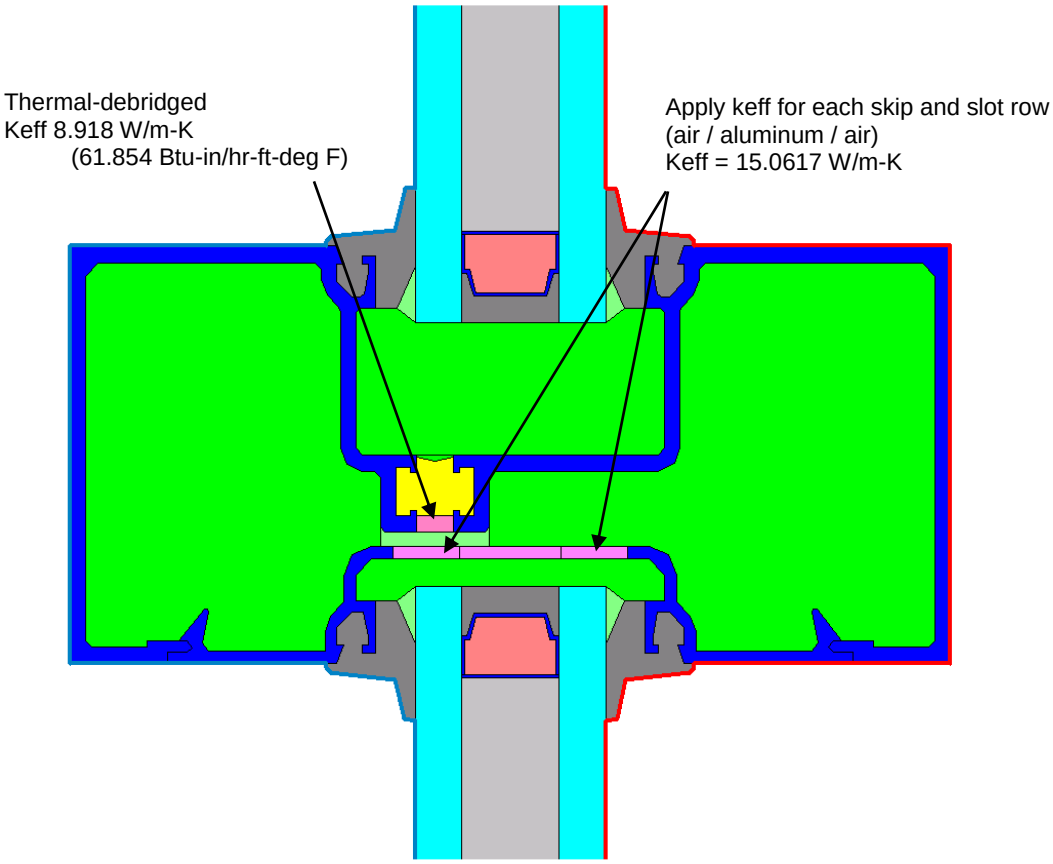
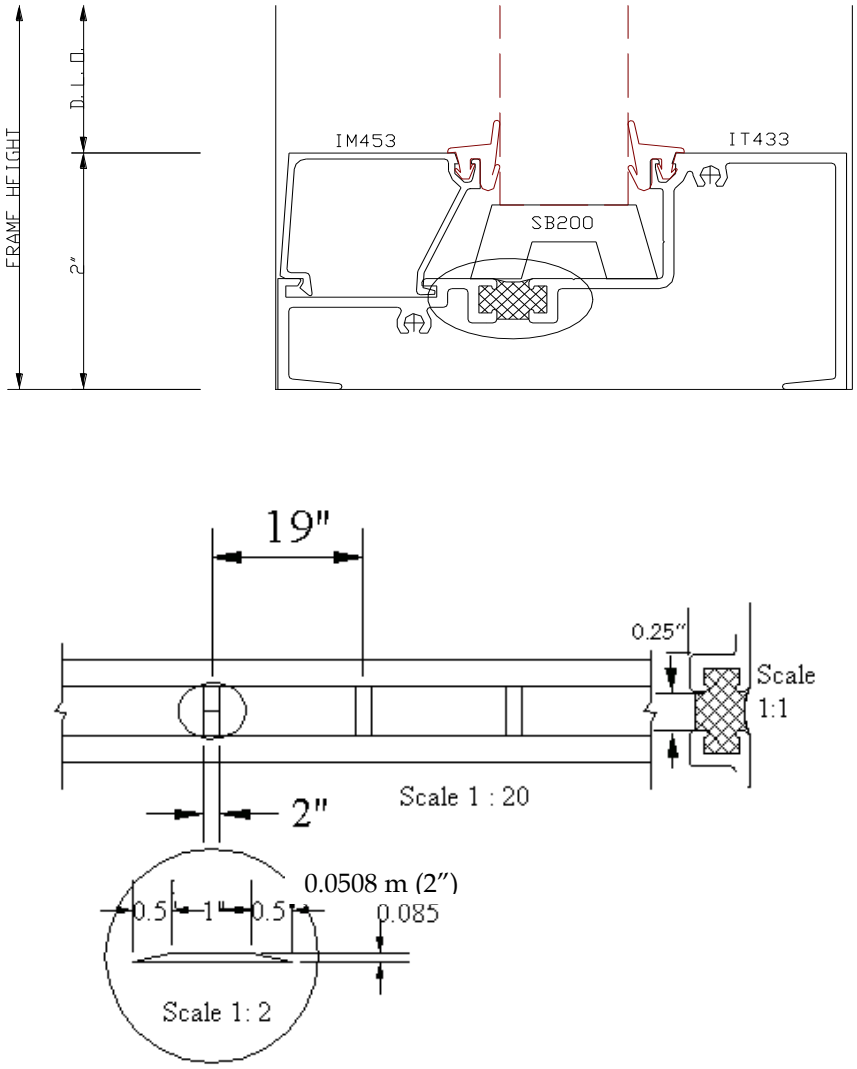


Figure 8-87. New Keff assigned to each skip and debridged row.

Step 3

Define the Boundary condition and run the model to calculate the U-factor for frame and edge-of-glass.

8.10.7. Example 3: Skip-and-debridge:



Note: the skip trapezoid is treated as a rectangle equal to the total length of the base of the trapezoid.

Figure 8-88. Drawings for Example 3 Skip and Debridge.

STEP 1

Skip = 0.0508 m (2 in)

Debridge (Air) = 0.4318 m (17 in)

Interval = 0.508 m (2 in) + 0.4318m (17 in) = 0.4826 m (19 in)

Note: The rest of the example will be in SI units only, with no IP unit translation

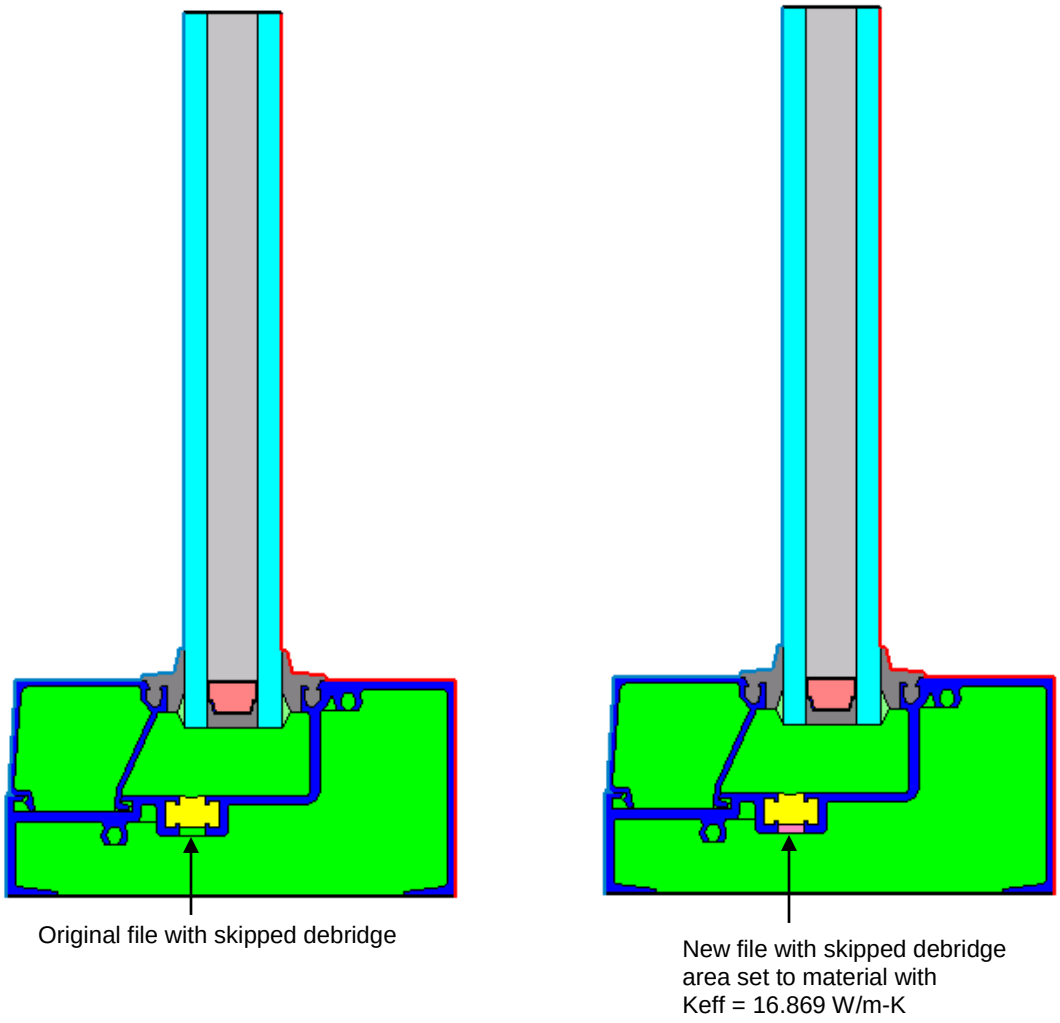


Figure 8-89. Original THERM model and new model with new Keff for skipped debridge area.

Fb = 0.0508 m/ 0.4826 m = 0.1053

Fn = 1-Fb
= 1- 0.1053 = 0.8947

%Fb = (Fb) ·100
= (0.1053) · 100
= 10.53% (Skip-and-debridge needs to be calculated using Isothermal plane procedure).

Kb = 160 W/m-K (conductivity of skipped debridge, in this case aluminum)

Rt = ∑ Length/ conductivity
= (0.00635 m/0.024 W/m-K)
= 0.2646 m^2-K/W

The length is the length of material in a direction of heat flow i.e. 0.25″ as shown in the figure above. (The air effective conductivity calculated using THERM)

Kn = length/Rt
= 0.00635m/0.2646 m^2-K/W
= 0.024 W/m-K

Keff = Fb*Kb +Fn*Kn
= 0.1053*160 W/m-K + 0.8947*0.024 W/m-K
= 16.869 W/m-K

To convert to IP:

Keff = 16.869 * 0.57782 = 9.747 Btu/hr-ft-F (or in alternative IP Units: 116.97 Btu-in/hr-ft^2-F)

8.11. Site Built fenestration products (Curtain Walls, Window Walls and Sloped Glazing)

A few things to be aware of:

- Review the drawings carefully for non-continuous elements. These systems tend to have many such elements including shear blocks, installation clips and spacers at the bolts.
- Site built modeling is partially dependent on the installation of the product. Make sure that the drawings you have show the installation.
- Site built products are typically multiple lite systems where the intermediate vertical and horizontal frame members repeat in some pattern.
- Curtain Walls should be simulated and tested with the intermediate verticals as jambs and intermediate horizontals as the head and sill members.
- Window Walls, also known as strip windows, should be simulated with intermediate verticals as jambs and standard head and sill members.

Modeling of the representative products above used to compare to the physical validation test is as follows:

1. Simulate the full vertical intermediate member twice (for the left and right jambs):
    a. Once with the left glass replaced with wood, which is the same thickness as the IG unit
    b. Once with the right glass replaced with wood, which is the same thickness as the IG unit. The wood should extend to the original sightline.
2. Simulate the horizontal members.
    a. If this is for a curtain wall, then once with the top glass replaced with wood, which is the same thickness as the IG unit and then a second time with the bottom glass replaced with wood, which is the same thickness as the IG unit. The wood should extend to the original sightline.
    b. If this is for a window wall or sloped glazing, then the standard sill and head members are modeled.

8.11.1. Curtain Walls, Window Walls, and Sloped Glazing

A curtain wall and window wall is any building wall carrying no super imposed vertical load. A *curtain/window wall system* will typically be exterior to the building framework and will typically bypass the building floors.

Curtain walls, window walls, and sloped glazed wall systems can be simulated as two lites with one vertical mullion. See the introduction to this section for the specific members of curtain walls, window walls, and sloped glazed wall systems that are to be modeled. If the intermediate vertical and horizontal members were simulated full width or height for the jambs, head and sill, then the total area of the frame as simulated would be significantly larger than actual frames, so for rating purposes, some members (depending on whether it is a curtain wall, window wall or sloped glazing) are simulated as ½ the full height (for head and sill) or width (for jambs). The vertical mullion is always simulated full width.

Modeling Procedures:

- **Curtain Walls:** See the modeling steps in the following section.

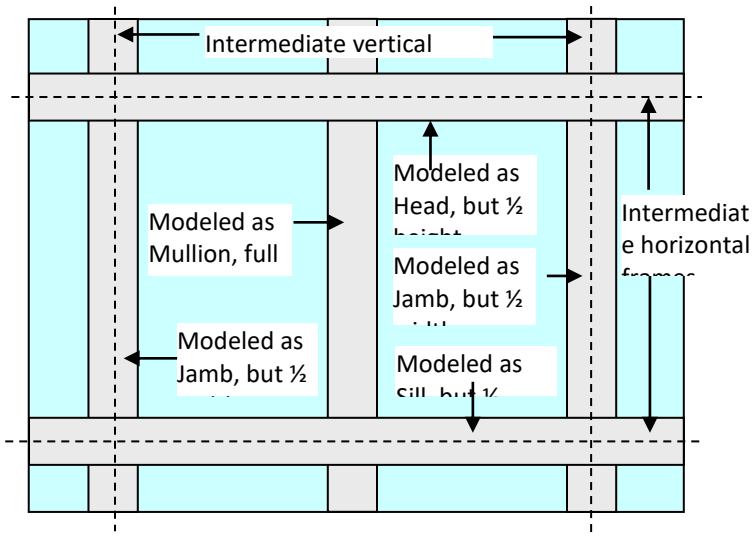


Figure 8-90. Curtain Wall Modeling Method

- **Window Walls:** For window walls, the standard head and sill members are modeled, and therefore the steps for simulation are identical to that of a normal window cross-section.

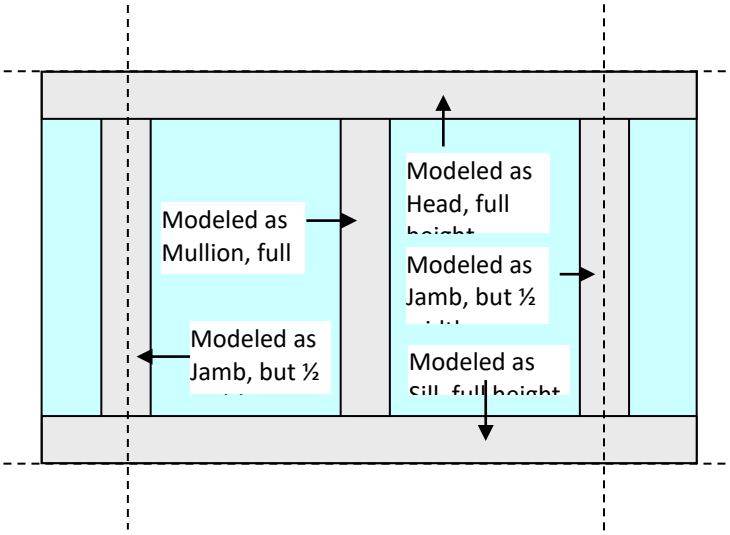


Figure 8-91. Window Wall Modeling Method

- **Sloped Glazed Wall System:** If the product to be simulated is a sloped glazed wall system, then the modeling steps are identical to a curtain wall or window wall, except that the cross-section is to be sloped 20 degrees (see skylight section for instruction on sloping products).

8.11.2. Curtain Wall Modeling Procedure

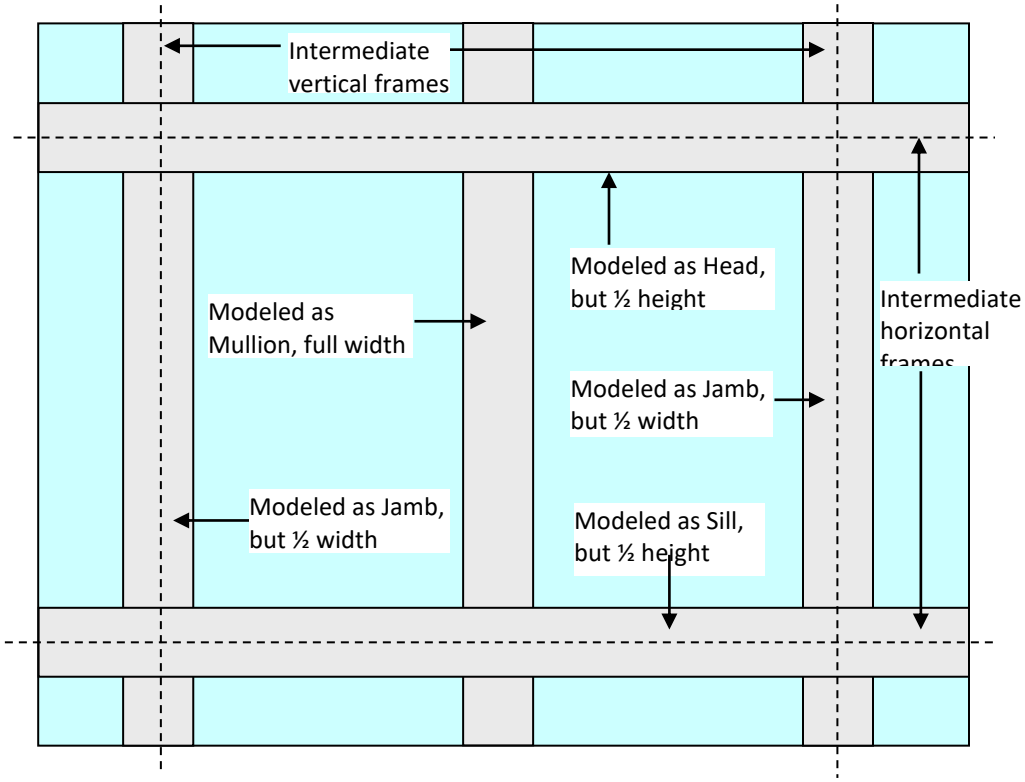


Figure 8-92. Curtain wall simulation model (represented by dotted lines) for rating, where the framing members are modeled at half their width.

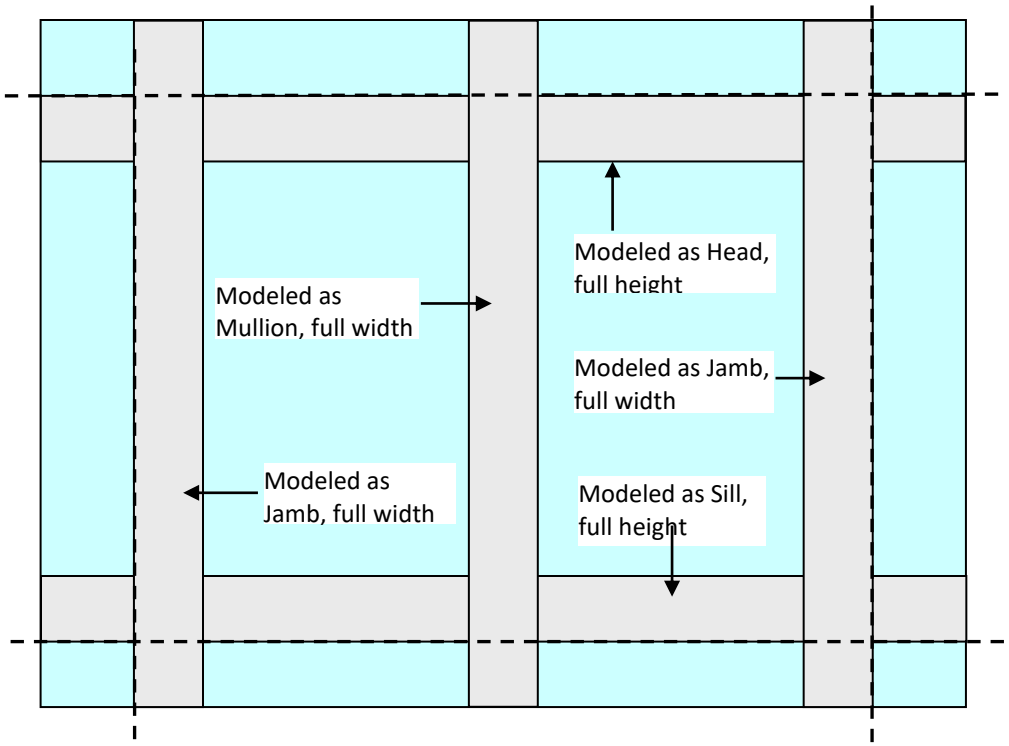


*Figure 8-93. Curtain wall simulation model (represented by dotted lines) to match **testing**.*

Steps in Modeling Curtain Walls for Rating Simulation

In WINDOW:

- Create glazing systems in WINDOW as usual, with Tilt set to “90".

In THERM:

- Draw the cross sections for curtain walls in the same manner as any other model in THERM – it will be a model similar to a meeting rail.

The following discussion lists the steps for making cross sections for intermediate horizontal and vertical frames. In the example, the horizontal frame and vertical frame are the same so only one drawing will be needed.

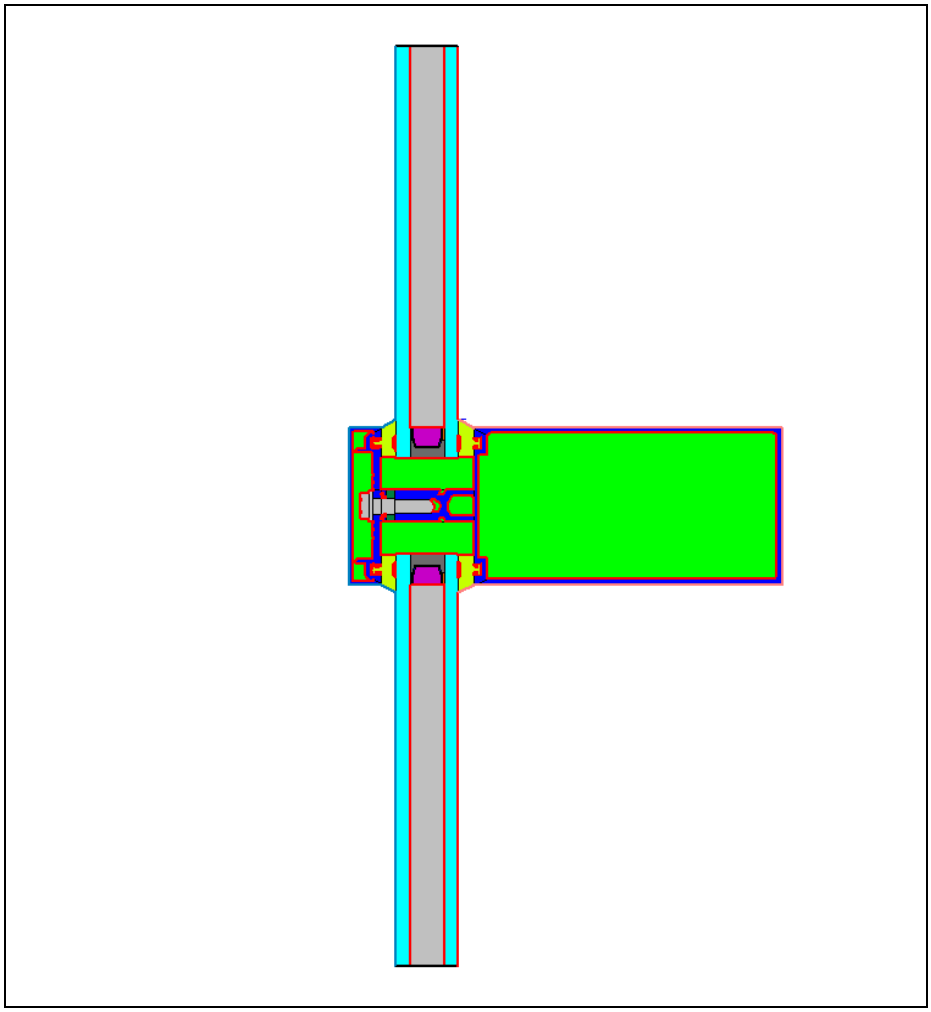
- 

Figure 8-94. Curtain Wall Mullion Cross Section (mull.thm)

Modeling the Intermediate Vertical Mullion:

Using the dimensioned drawings or a DXF file, create the cross section for the frame portion of the vertical mullion. The figure below shows a THERM cross section of the frame portion of the vertical mullion.

Make sure that the section is oriented correctly with the glazing oriented up and down.

- If bolts are present make sure to model them as non-continuous thermal bridging elements if needed. (see Section 8.10).
- Pay particular attention to the aluminum finish in order to assign the correct materials to the drawing. (In the figure below, where the arrows point to Aluminum)

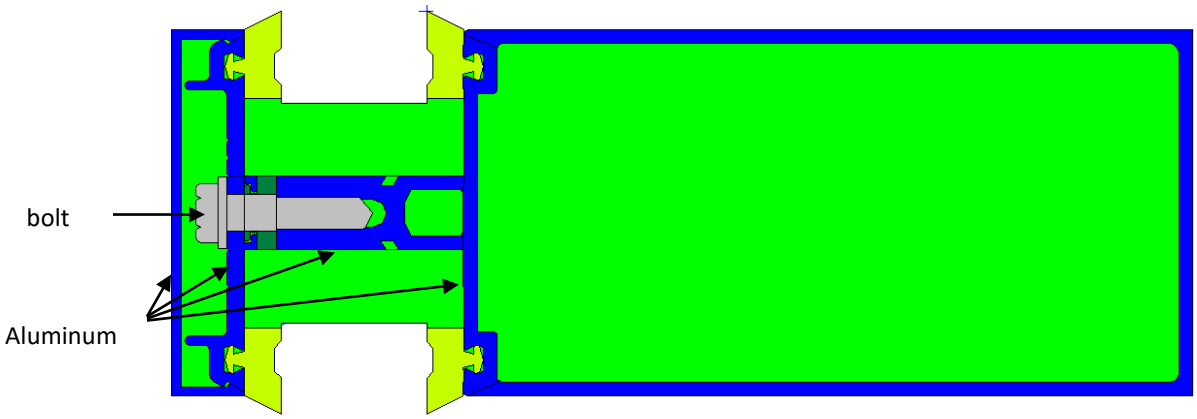


Figure 8-95. Mullion frame cross section without glass (Mull_noglass.thm)

This vertical cross section will be used for the vertical intermediate (mullion) and for the jambs. The jamb model it is necessary to determine and mark the ½ width point between the sightlines. The easiest way to do this is to make temporary reference rectangles as follows:

Determine Frame Midpoints

In order to model the "half-width" frame dimensions for the rating simulation, it is necessary to determine the mid-point of the frame, which is most easily done before the glazing systems are inserted.

- Draw a rectangle from the sightline (highest interior point) for both glazing systems, going horizontally in both directions, so that each rectangle is larger than the horizontal dimension of the frame.
- On both the right and left side of the frame draw a rectangle between the other two reference rectangles and make sure it contacts the frame.
- Measure the exterior rectangle just created in the vertical dimension. The midpoint of the frame is half of this dimension.
- For both the right and left sides of the frame cross section, select the vertical frame component and insert a point on the frame at the midpoint dimension.

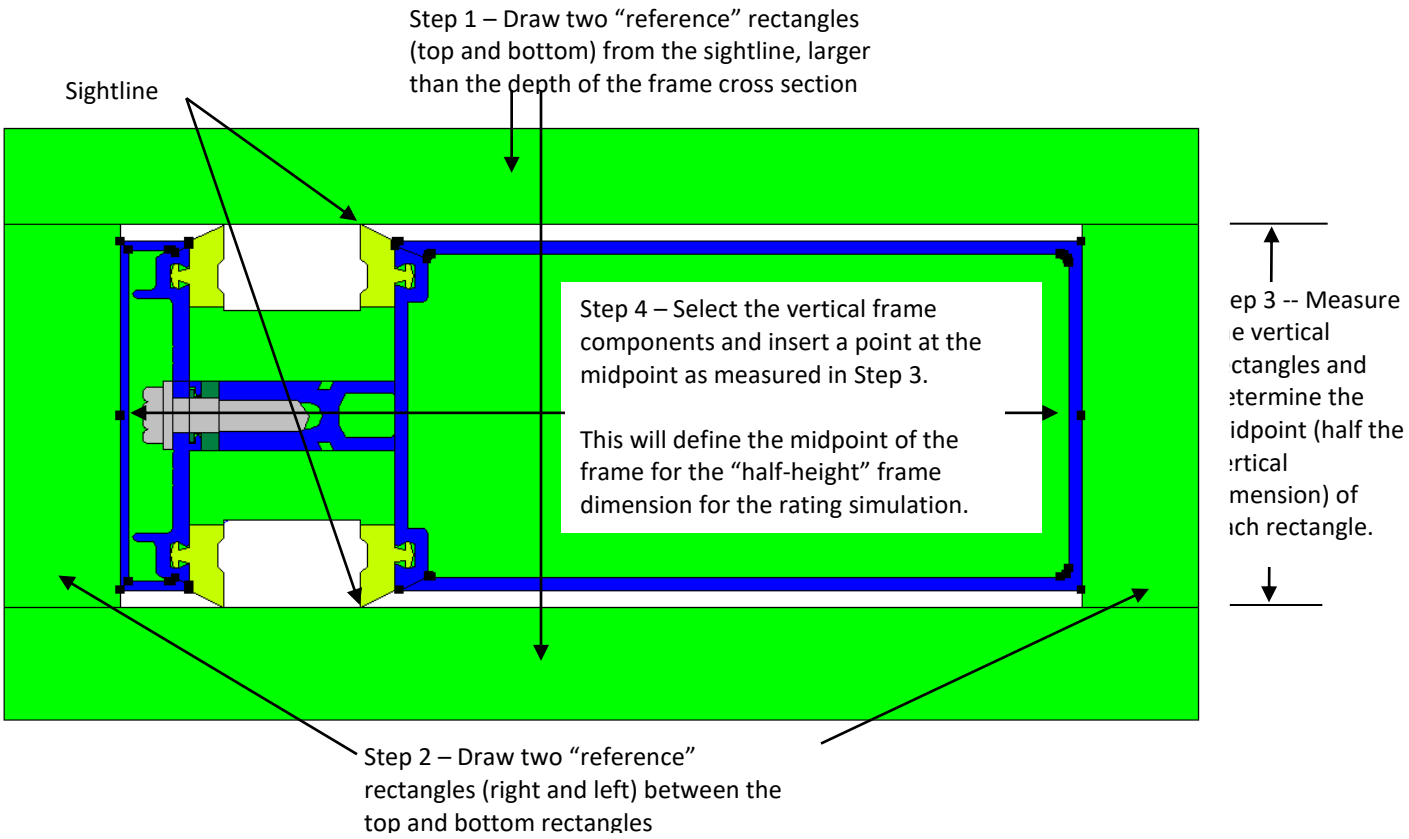


Figure 8-96. Determine the frame mid-points (mull_midpoint.thm)

Finish the cross section

- Delete the four "reference" rectangles.
- Insert the glazing and spacers as usual.
- In **Therm File Properties** (File/Properties/Model Exposure tab), set the **Cross Section Type** to "Vertical Meeting Rail".
- Assign the **Boundary Conditions** as follows:

- **Interior Frame:** For the mullion cross section, the entire frame width (not just half) is modeled, so all the interior frame boundary condition elements are assigned a **Boundary Condition** of "<frame type> Interior (Convection Only), and a **U-factor Surface** tag of "Frame".
- **Interior Glazing:** Set the **Boundary Conditions** for each glazing system to "<glazing system> U-factor Inside Film" and the **U-factor Surface** tag to "Edge" for the first 63.5 mm (2.5 inches) from the sightline, and "None" for the remainder of the glazing system.
- **Exterior Frame:** Set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "SHGC Exterior" for the exterior frame
- **Exterior Glazing:** Set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "None" for both exterior glazing systems.

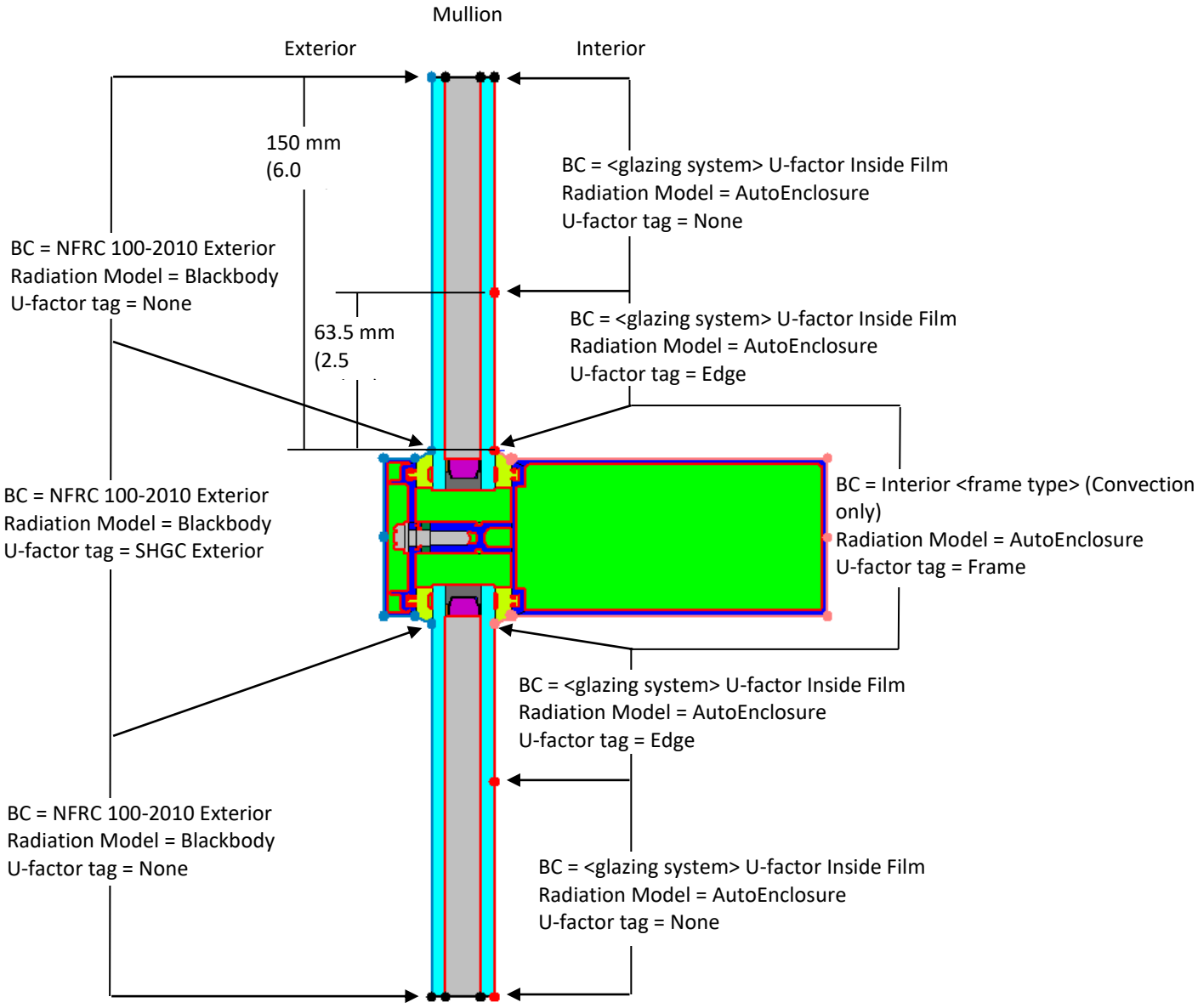


Figure 8-97. Curtain Wall Mullion Boundary Conditions (mull_boundary.thm)

Modeling Curtain Wall Jambs:

In this example, the curtain wall jambs are modeled using the same drawing as the mullion. The boundary conditions are adjusted to capture the heat flow of only half the width. Since the sections may be asymmetrical, both the left and right portions of the mullion are required to be simulated as left and right jambs.

For the Right Jamb:

- Copy the vertical mullion drawing with boundary conditions assigned .
- In **Therm File Properties** (File/Properties/Model Exposure tab), set the **Cross Section Type** to "Vertical Meeting Rail".
- Do not change the **Boundary Conditions** but assign new **U-factor Surface** tags as follows:
  - **Interior Frame:**
    - from the midpoint to the top sightline set the **U-factor Surface** tag to "Frame"
    - from midpoint to the bottom sightline and the entire bottom glass assign the **U-factor Surface** tag to "None".
    - **Interior Glazing System:** for the first 63.5 mm of the top glazing system set the **U-factor Surface** tag to "Edge", and the remainder of the glazing system to "None". For the entire bottom glass assign the **U-factor Surface** tag to "None".
  - **Exterior Frame:**
    - from the midpoint to the top sightline assign the **U-factor Surface** tag to "SHGC Exterior"
    - from midpoint to the bottom sightline assign the **U-factor Surface** tag to "None".
  - **Exterior Glazing System:** set the **U-factor Surface** to "None".

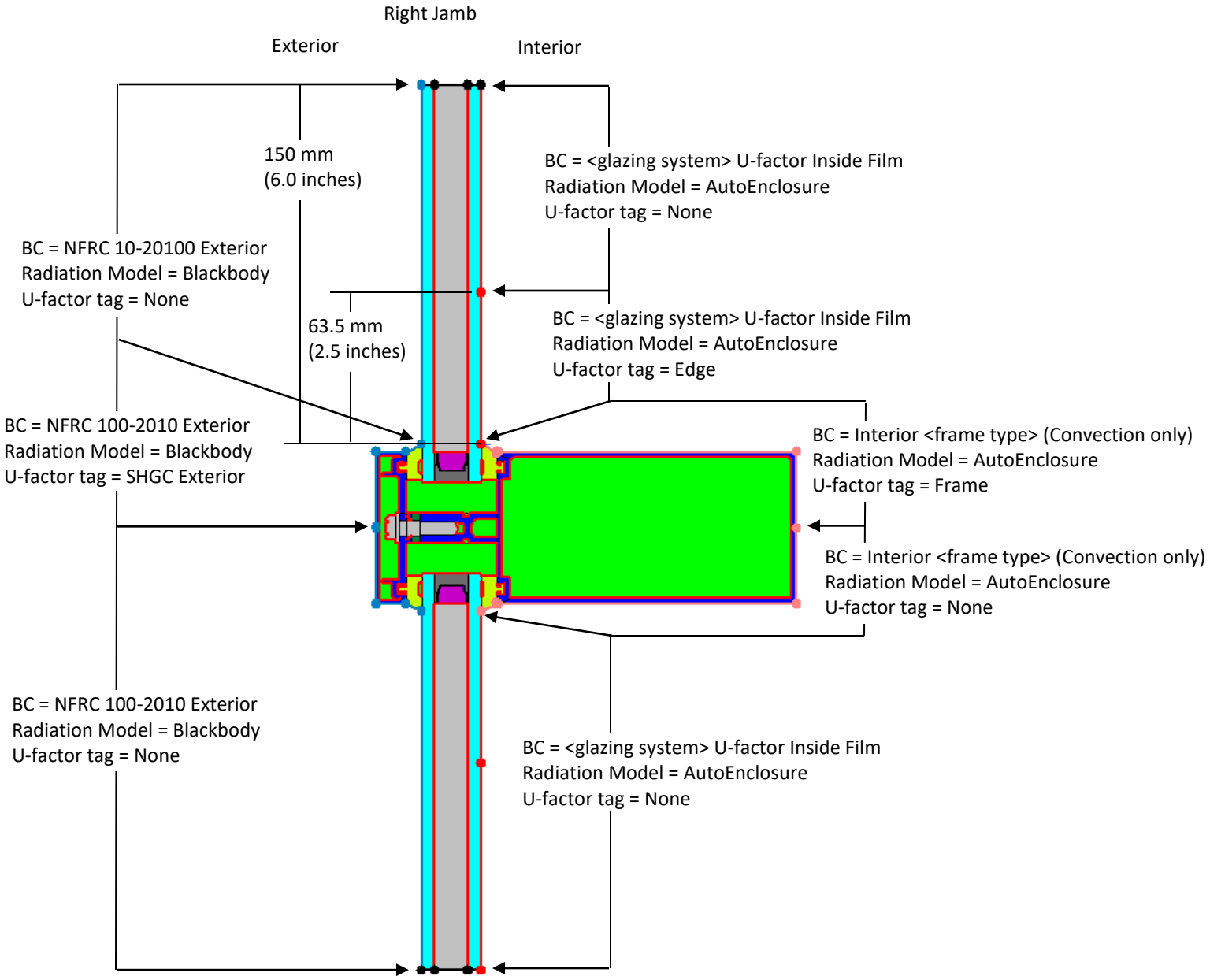


Figure 8-98. Right Jamb Boundary conditions (rightjamb_boundary.thm).

For the Left Jamb:

- The process is the same as the right jamb except that the Frame, edge and SHGC Exterior tags are assigned to the bottom half of the drawing and the top half is assigned the tag of "none"

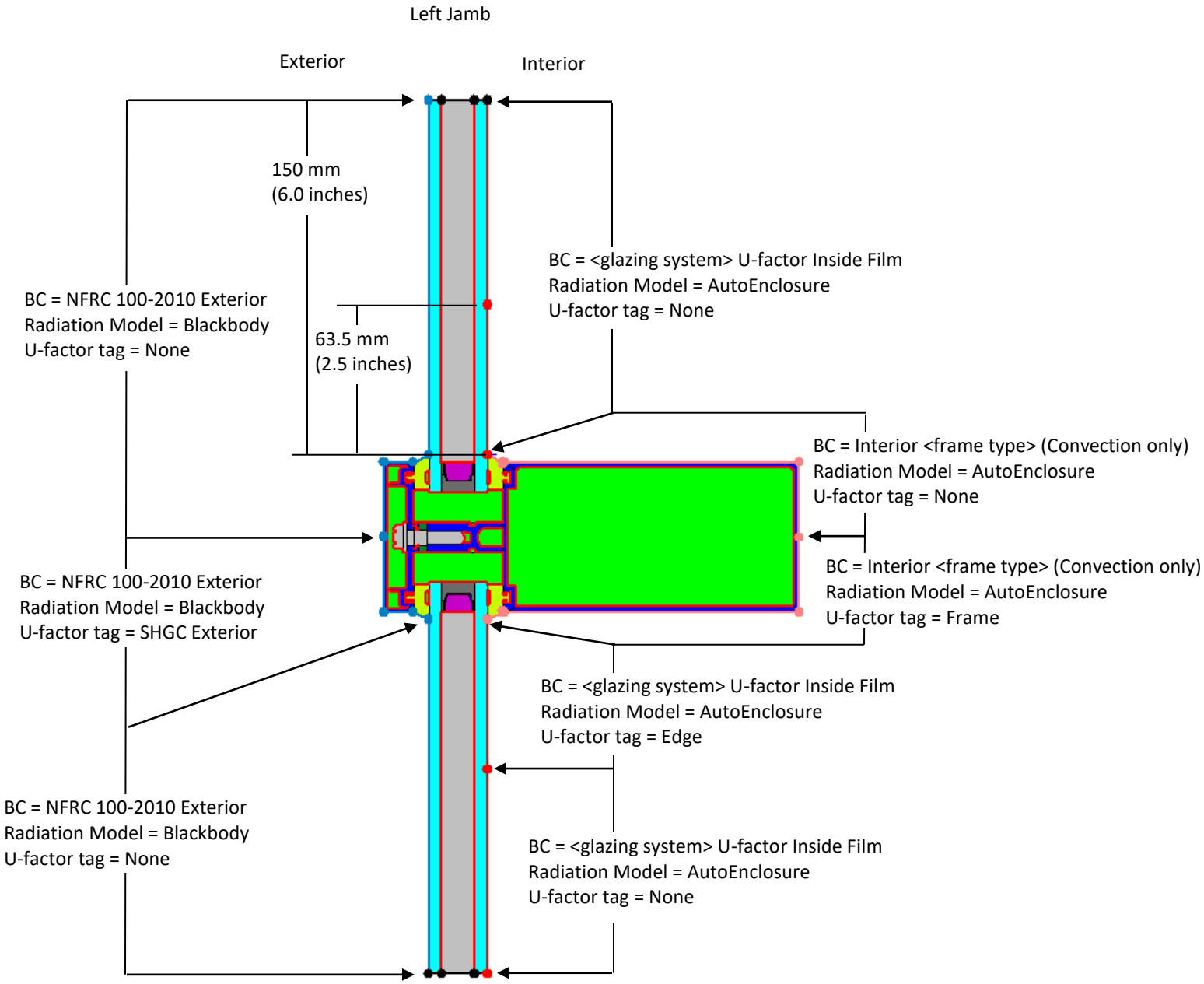


Figure 8-99. Left Jamb Boundary Conditions
(leftjamb_boundary.thm)

NOTE: The THERM File option "Use CI Model for Window Glazing System" may be left checked since THERM will not calculate Condensation resistance for a cross section Type tagged Vertical Meeting Rail or Jamb.

Modeling the Intermediate Horizontal (as Head and Sill):

The horizontal frame cross section is drawn in exactly the same way as the intermediate vertical frame cross section, including inserting the midpoint. For this example the horizontal and vertical frames are identical so the same drawing is used. Assign boundary conditions in the usual way and then assign new boundary condition U-Factor Surface tags as follows for the head and sill.

For the Head:

- Use the drawing for the intermediate horizontal.
- In **Therm File Properties** (File/Properties/Model Exposure tab), set the **Cross Section Type** to "Head".
- Assign the Boundary Conditions as follows:
  - **Interior Frame:**
    - For the interior frame from the midpoint to the *bottom* sightline, set the **Boundary Condition** to "<frame type> Interior (Convection Only), and the **U-factor Surface** tag to "Frame".
    - For the interior frame from midpoint to the *top* sightline, set the **Boundary Condition** to "<frame type> Interior (Convection Only), and the **U-factor Surface** tag to "None".
  - **Interior Glazing:** Set the **Boundary Conditions** for each glazing system of the bottom of the glass to "<glazing system> U-factor Inside Film" and the **U-factor Surface** tag to "Edge" for the first 63.5 mm (2.5 inches) from the sightline, and "None" for the remainder of the glazing system.
  - **Exterior Frame:**
    - For the exterior frame from the midpoint to the *bottom* sightline, set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "SHGC Exterior".
    - For the exterior frame from the midpoint to the *top* sightline, set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "None".
  - **Exterior Glazing:** Set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "None" for both exterior glazing systems.

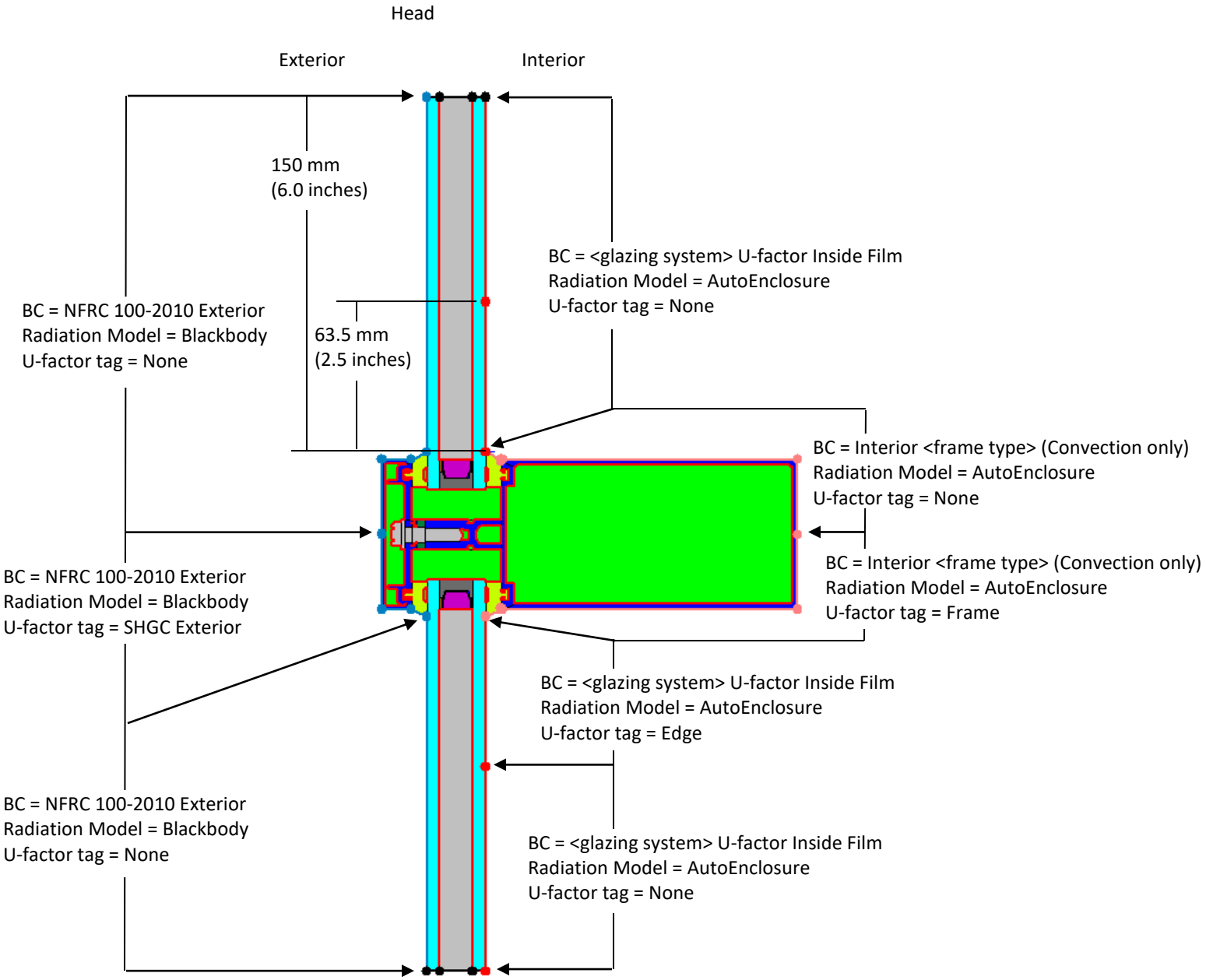


Figure 8-100. Head Boundary Conditions (head_boundary.thm)

For the Sill:

- Use the drawing for the intermediate horizontal.
- In **Therm File Properties** (File/Properties/Model Exposure tab), set the **Cross Section Type** to "Sill".
- Assign the Boundary Conditions as follows:
  - **Interior Frame:**
    - For the interior frame from the midpoint to the *top* sightline, set the **Boundary Condition** to "<frame type> Interior (Convection Only), and the **U-factor Surface** tag to "Frame".
    - For the interior frame from midpoint to the *bottom* sightline, set the **Boundary Condition** to "<frame type> Interior (Convection Only), and the **U-factor Surface** tag to "None".
  - **Interior Glazing:** Set the **Boundary Conditions** for each glazing system of the *top* of the glass to "<glazing system> U-factor Inside Film" and the **U-factor Surface** tag to "Edge" for the first 63.5 mm (2.5 inches) from the sightline, and "None" for the remainder of the glazing system.
  - **Exterior Frame:**
    - For the exterior frame from the midpoint to the *top* sightline, set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "SHGC Exterior".
    - For the exterior frame from the midpoint to the *bottom* sightline, set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "None".
  - **Exterior Glazing:** Set the **BoundaryCondition** to "NFRC 100-2010 Exterior" and the **U-factor Surface** tag to "None" for both exterior glazing systems.

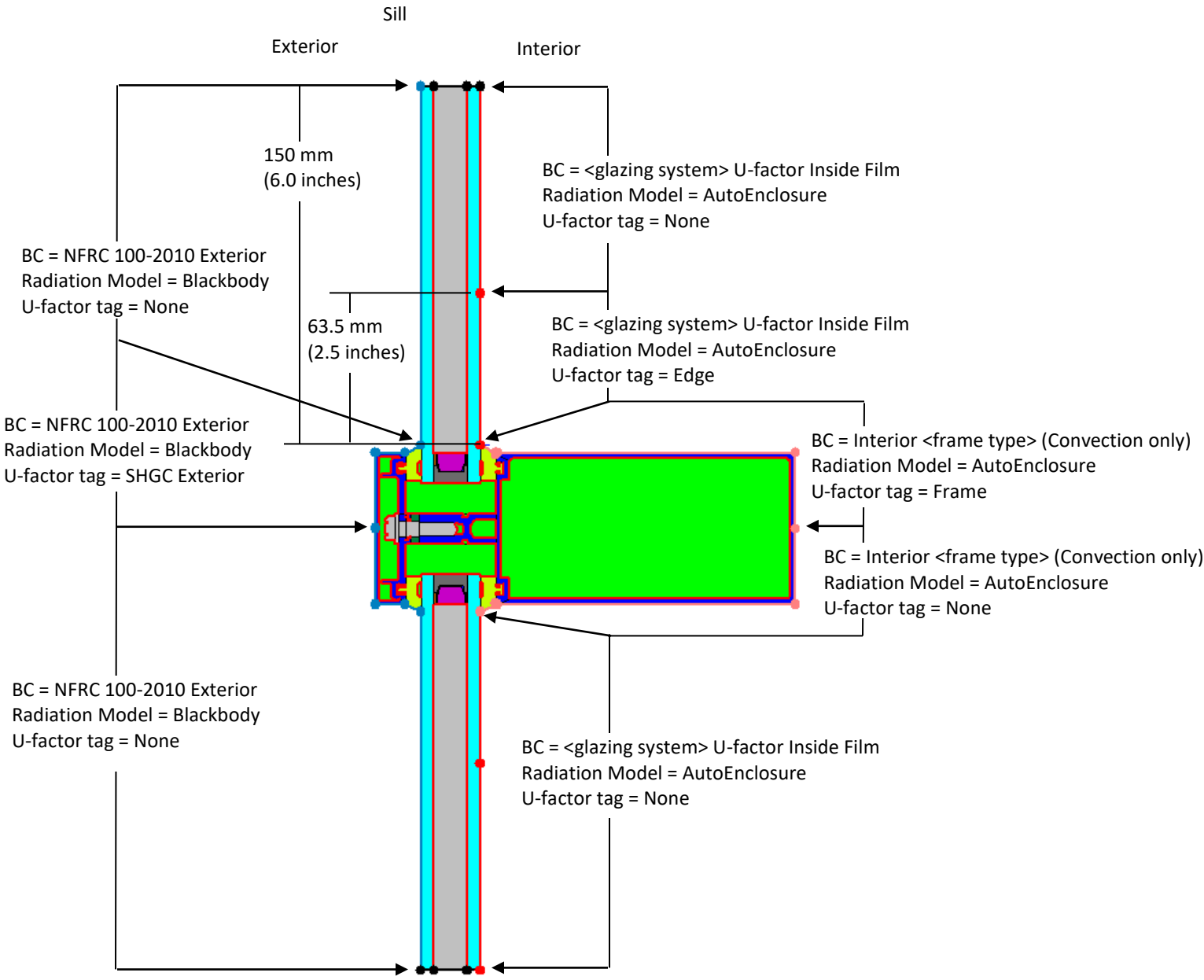


Figure 8-101. Sill Boundary conditions (sill_boundary.thm)

It is a good idea at this point to verify the orientation and the direction of the gravity arrows of these sections.

In WINDOW:

Calculate these files and import into WINDOW. Use the files to create the whole product in the Window Library as applicable.

8.12. Framed Intermediate Pane

8.12.1. Overview

For a product that has an intermediate pane with it's own frame, the modeling steps are as follows:

In WINDOW:

- Create the glazing system, including the intermediate pane.

In THERM:

- Model the head, sill and jamb sections including the intermediate pane and frame, using the glazing system created in WINDOW. Model caming, if applicable, as shown in the Dividers section of this manual

In WINDOW:

- Import the THERM cross sections into the Frame (and Divider if modeling caming) Libraries.
- Model the whole product using the Glazing System, Frame and Divider components.

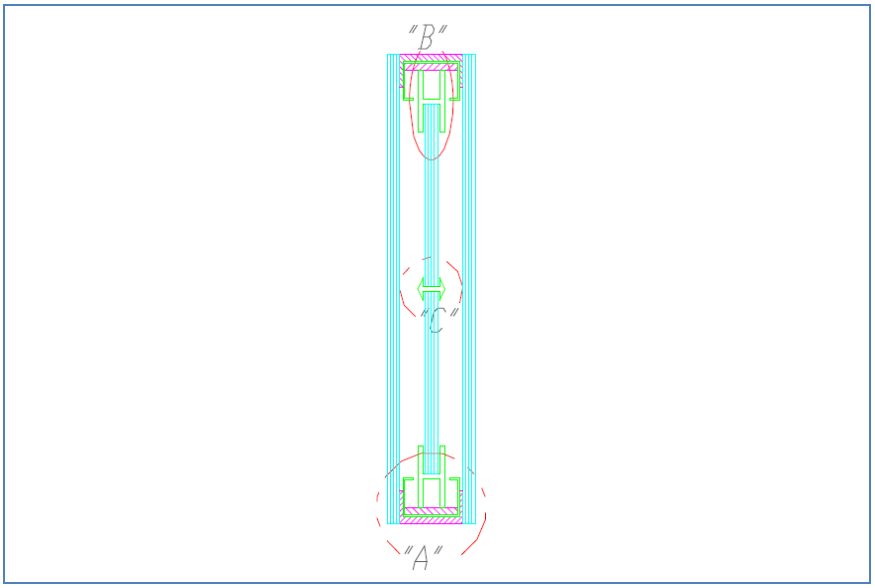


Figure 8-102. An example of a framed intermediate pane. Section A is modeled for the Sill/Jamb, Section B is modeled for the Head, and Section C is modeled as a Divider.

8.12.2. THERM Modeling Details

The THERM modeling is relatively straightforward, with a few details to be aware of shown below.

The head and sill are modeled normally, except that the extra frame for the intermediate pane is also modeled. The sightline will remain defined as the highest interior point on the window/door frame.

If the glazing system is gas filled, any frame cavity opening into the glazing cavity should also be gas filled. See "Dividers" in this manual for information about creating gas filled frame cavity materials.

Example Sill

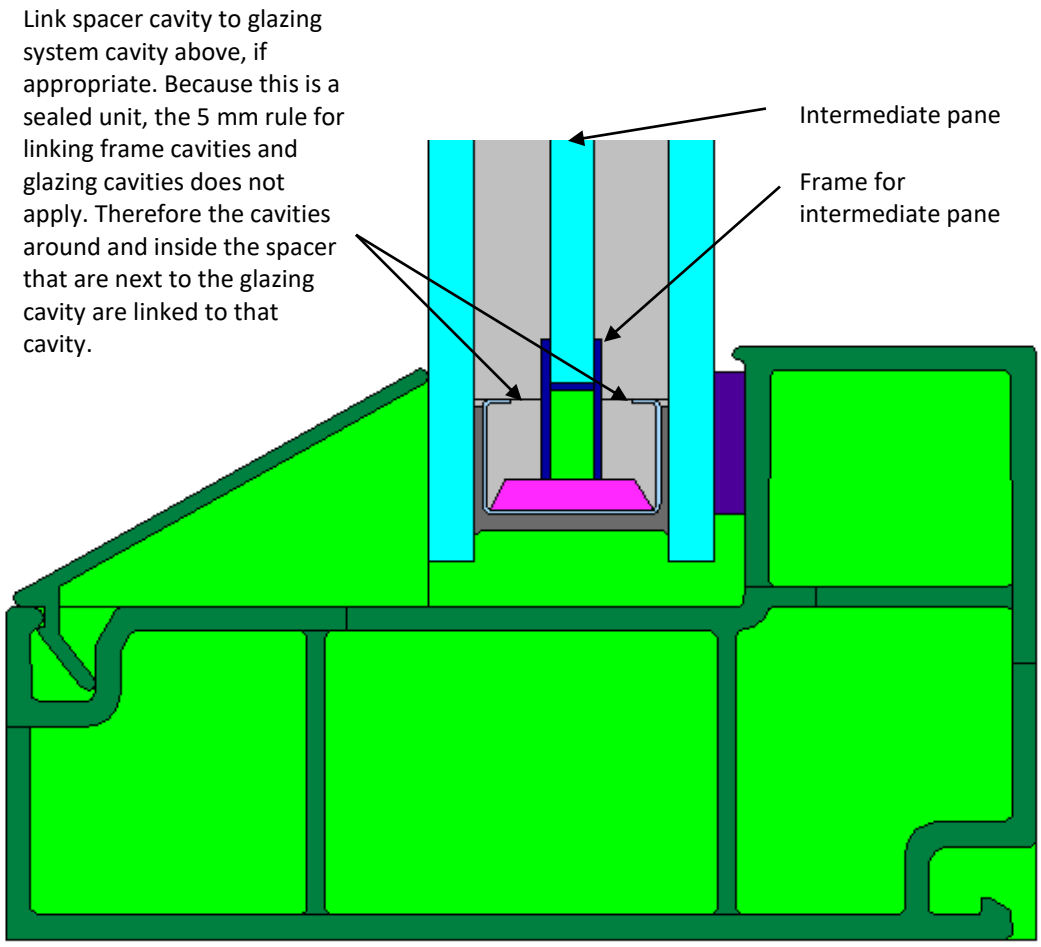


Figure 8-103. Example sill with framed intermediate pane

Example Head

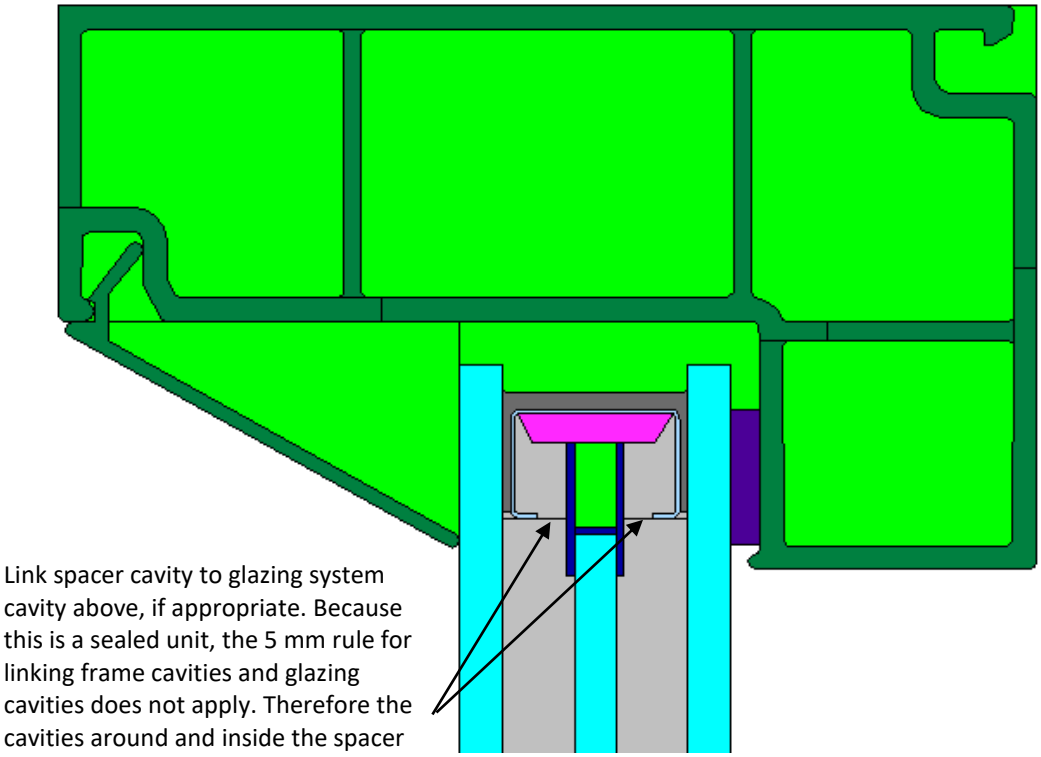


Figure 8-104. *Example head with framed intermediate pane*

Example Caming / Divider

Caming, if it exists, is modeled as a Divider. (See Section 8-8, Dividers for modeling rules.)

9. Transient Moisture and Thermal Simulations

9.1. Overview

THERM allows modeling both transient moisture and thermal simulations, as well as stead-state thermal simulations.

The sections below describe the settings for transient moisture and thermal simulations.

9.2. File / Properties

The **File / Properties / Calculation Options tab** controls what model is used for a given file. All the settings in the THERM File Properties dialog box are saved with individual files, not as program-level settings.

Calculation Options tab

Below is a description of the fields in the **File / Properties / Calculation Options tab**.

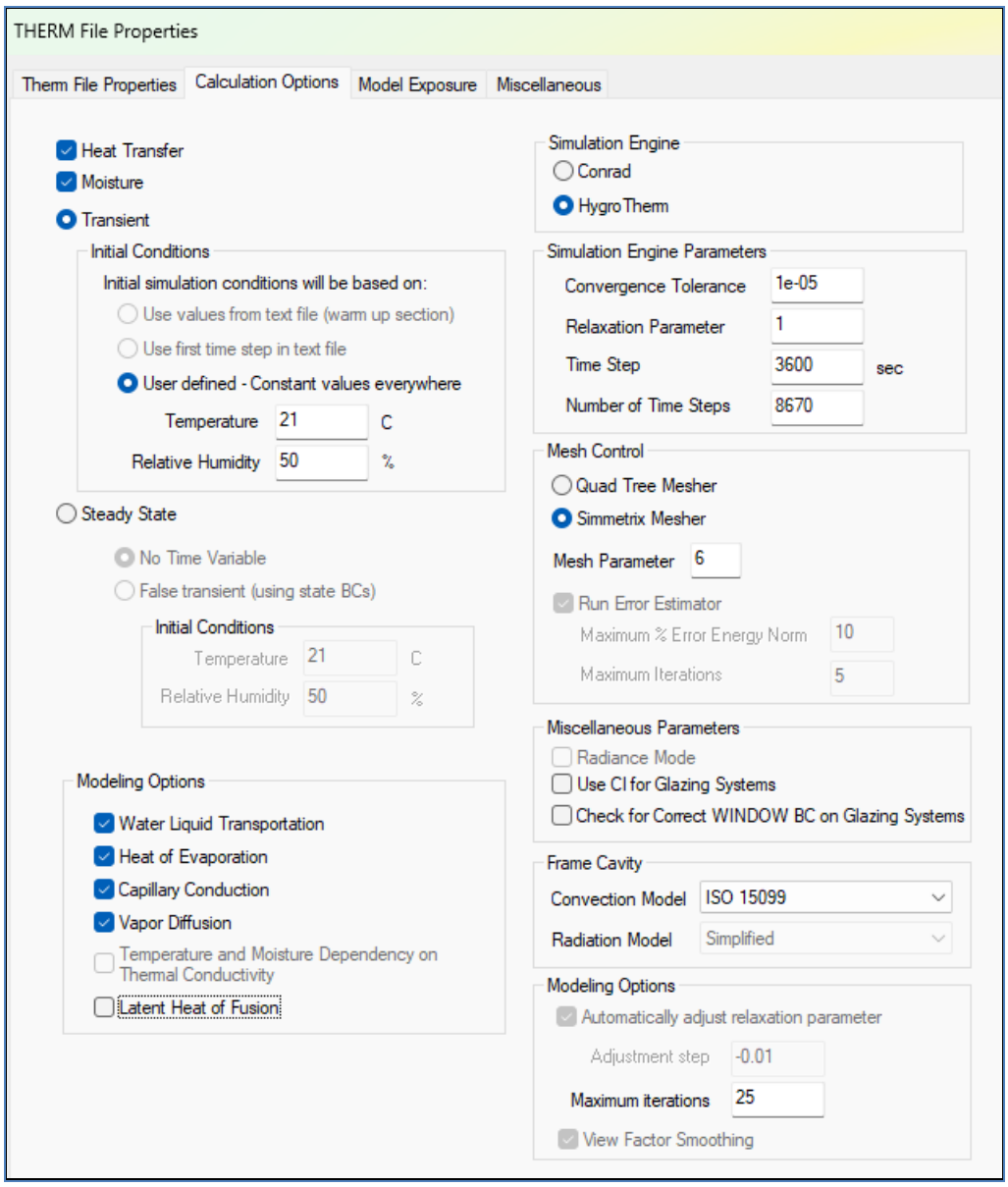


Figure 9-1. File/Properties/Calculation Options tab contains settings for both the steady-state and transient models.

Heat Transfer	*This only applies to the Transiet Thermal + Moisture simulation engine.* Check this box to model heat transfer. Checking this box is appropriate for ▪ Simulation engine = Steady-State Thermal (ConRad) ▪ Simulation engine = Transient Therm + Moisture (HygroTherm)
Moisture	*This only applies to the Transiet Thermal + Moisture simulation engine.* Check this box to model moisture. Checking this box is appropriate for ▪ Simulation engine = Transient Therm + Moisture (HygroTherm)
Transient *This only applies to the Transiet Thermal + Moisture simulation engine.*	
Initial Conditions	Initial simulation conditions will be based on either: ▪ User values from text file (warm up section): *currently not implemented* ▪ Use first time step in text file: *currently not implemented* ▪ User defined – Constant values everywhere ▪ Temperature: Default: 21 C. ▪ Relative Humidity: Default: 10. Units: percentage (%)
Steady State	
No Time Variable	This uses the same steady-state equation to calculation heat transfer as ConRad, and there is no time variable. See technical documentation
False transient (using state BCs)	This runs a transient simulation until it reaches a steady-state solution. Initial Conditions: this sets the temperature and relative humidity of the entire model to start the simulation. It is also necessary to have transient-style Boundary Conditions (ie, XML file) that have contant temperature and relative humidity values over the timesteps.
Initial Conditions	If False transient is selected, the temperature in Initial Conditions can be set by the user.

Modeling Options *This only applies to the HygroTherm simulation engine when Heat Transfer and Moisture are selected.*	
	These are modeling options to set when modeling moisture using the HygroTherm engine: ▪ **Water Liquid Transportation:** Models transport of water in a liquid state. If this is not checked, then only water vapor is modeled. ▪ **Heat of Evaporation:** Models phase change of water. ▪ **Capilary Conduction:** Models capillary transport through ▪ **Vapor Diffusion Conduction:** Models diffusion of vapor thorugh material. ▪ **Temperature and Moisture Dependency on Thermal Conductivity:** *Currently not available* ▪ **Latent Heat of Fusion:**
Simulation Engine	
ConRad	Steady-state thermal simulation method, used in previous versions of THERM
HygroTherm	Transient thermal and moisture simulation engine, which can now do both Steady State, Moisture and Transient modeling.
Simulation Engine Parameters	
Convergence Tolerance	This controls the simulation error norm tolerance. If the error norm calculation is greater than this value, the program will display a "non-convergence" error message. If it doesn't converge, this value can be reduced. *Used only in the HygroTherm simulation engine.* **Default:** 1e-05.
Relaxation Parameter	This is the initial value for relaxation parameter used in iterations. *Used only in the HygroTherm simulation engine.* **Default:** 1.
Time Step	*This only applies when Transient is selected.* This parameter is used by the HygroTherm Transient model to set the length of the time step. The default value of 3600 seconds equals 1 hour. **Default:** 3600. **Units:** seconds.
Number of Time Steps	*This only applies when Transient is selected.* This parameter determines how many times steps are calculated when the HygroTherm Transient model is selected. It works in conjunction with the Time Step value. The default Time Step value of 3600 seconds (1 hour) used with the default value of the Number of Time Steps (8670) would result in a simulation at 1 hour intervals for a year. **Default:** 8670. **Units:** unitless

Mesh Control	
Quad Tree Mesher	The mesher used in previous versions of THERM (7 and lower).
Simmetrix Mesher	A new mesher that will mesh much more complex model geometries than the Quad Tree Mesher.
Mesh parameter	This value determines how fine the mesh is generated. If the program determines that the mesh cannot be generated, increasing this value may help. The relative size of the finite element mesh created for the model. The larger the number the finer the mesh. Increasing this value may help mesh files that return "mesh errors". **Quad Tree Mesher** ▪ Default: 6; upper limit usually 9 **Simmetrix Mesher** ▪ Default: 8; it can be set much higher
Run Error Estimator	If this option is checked, the program will perform the calculation of the error energy norm. If the simulation does not meet the value in Maximum % Error Energy Norm, the program will increment the Quadtree Mesh Parameter (if Automatically increment mesh parameter is checked on the Simulation tab), until the Maximum Iteration value is reached. **Default:** checked.
Maximum % Error Energy Norm	THERM incorporates a local error estimator and subsequent local mesh refinement. The error estimator is based on a published and well regarded methodology based on the estimate of Error Energy Norm (EEN). In this technique, only regions with an EEN larger than the threshold value are refined, resulting in optimum mesh size. An EEN value of 10% is chosen for the default because it results in overall computational accuracy of well below 1%, which is the ISO 10211 requirement. **Default:** 10%
Maximum Iterations	The number of iterations for the program to run in an attempt to meet the Maximum % Error Energy Norm specified. **Default:** 5

Miscellaneous Parameters	
Radiance Mode	*Currently only available for the Conrad simulation engine* If this option is checked, THERM will use Radiance to model the optical properties of a THERM model. This is used for modeling shading systems whose geometry is modeled in THERM. The THERM background will turn gray when this option is checked, to indicate that this option is selected, because this is not an option that would normally be used. Default: unchecked.
Use CI for Glazing Systems	If this option is checked, THERM will perform a Condensation simulation as well as the standard U-factor calculation. Default: unchecked.
Check for Correct WINDOW BC on Glazing Systems	If this option is checked, THERM will check to see that the Boundary Conditions applied to an imported WINDOW glazing system match those that were imported with that glazing system. Default: unchecked.
Frame Cavity	
Convection Model	The convection model used for Frame Cavities, from these options ▪ ISO 15099 ▪ EN 10077
Radiation Model	Always set to Simplified
Modeling Options	
Automatically adjust relaxation parameter	When equations are not converging, the relaxation parameter modifies previous iteration result in order to better converge. The values are from 0 to 1. The solver starts from a value of 1 and reduces it from there. *Only available for the Conrad simulation engine.* Default: checked
Adjustment step	Incremental reduction of the relaxation parameter from 1 to 0. *Only available for the Conrad simulation engine.* Default: -0.01

Maximum iterations	Number of iterations to apply the automatic relaxation parameter adjustment to. *For both the HygroTherm and Conrad simulation engines* Default: 25
View Factor Smoothing	This is used for gray body radiation calculation. If the view factors (surfaces being viewed) are very different, such as blocking surfaces, the view factor smoothing helps accuracy of results. See TARCOG technical documentation on the THERM documentation webpage. *Only available for the Conrad simulation engine.*

Typical settings for a transient thermal and moisture simulation (HygroTherm)

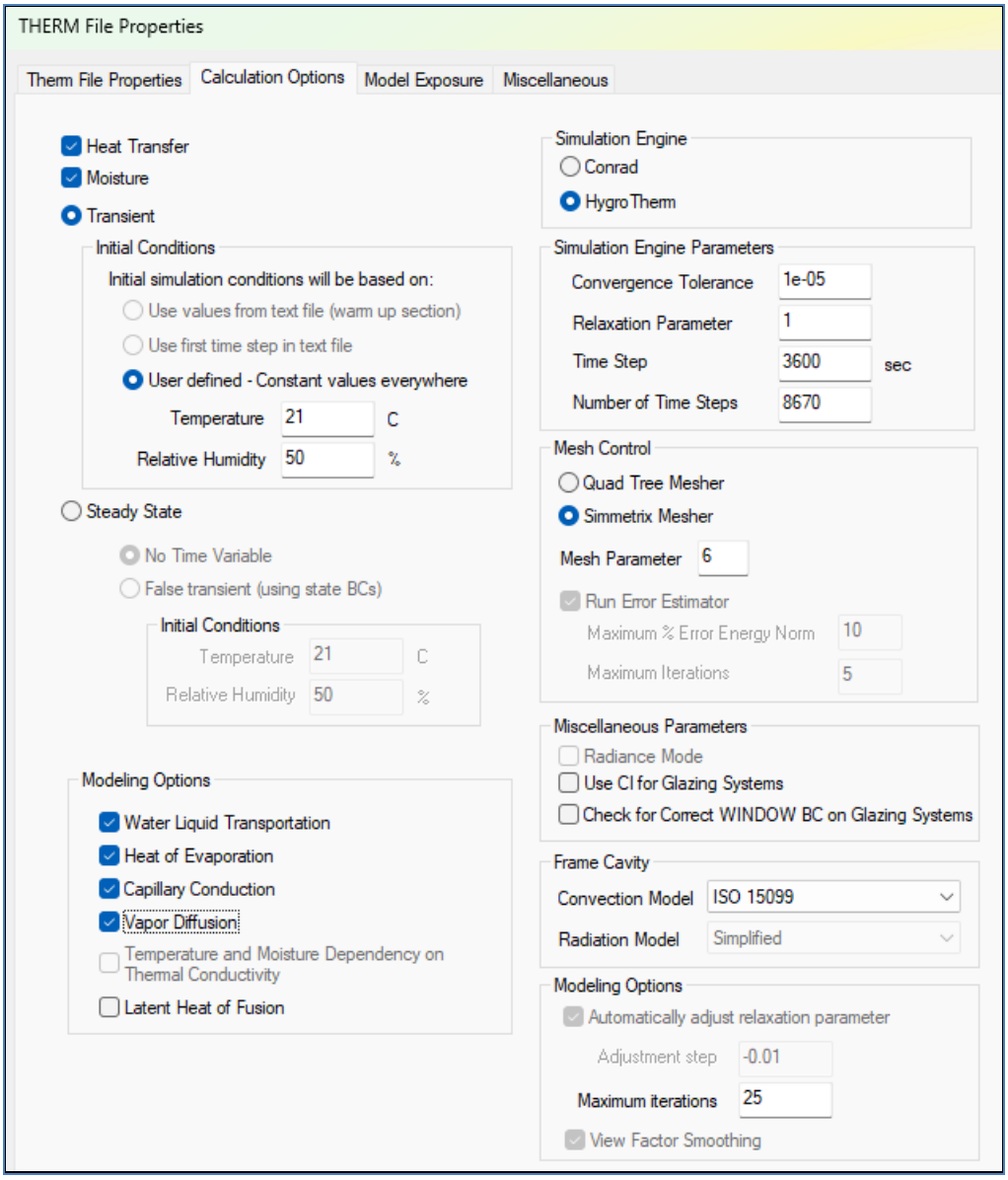


Figure 9-2. Typical File/Properties/Calculation Options settings for a transient thermal and moisture model.

Model Exposure tab

Below is a description of the fields in the File / Properties / Model Exposure tab.

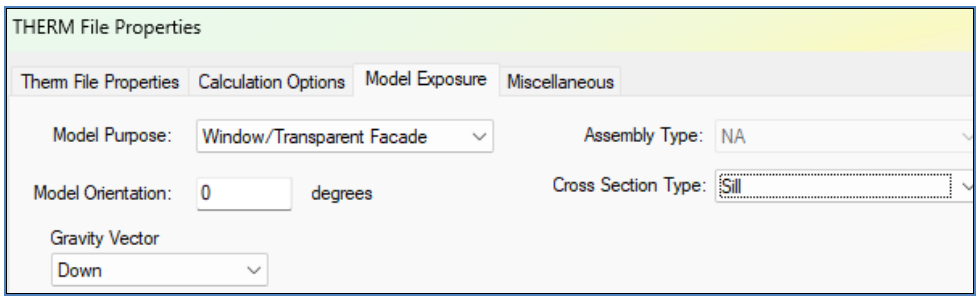
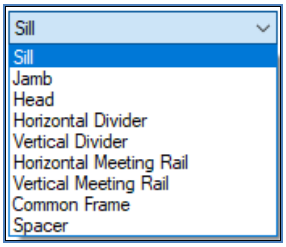
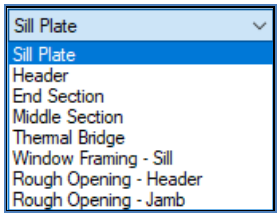
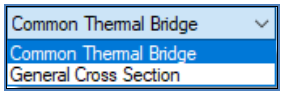
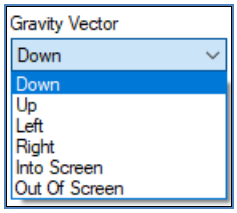


Figure 9-3. File Properties / Model Exposure tab.

Model Purpose	This pulldown list consists of the following options ▪ Window / Transparent Façade ▪ Opaque Façade ▪ Other
Assembly Type	If **Model Purpose** is set to **Opaque Façade**, this pulldown list becomes active with the following choices: ▪ Wall ▪ Roof If **Model Purpose** is set to either **Window/Transparent Façade** or **Other**, this value is set to "N/A".
Model Orientation	The direction the exterior surface of the model faces. This orientation of the exterior surface is used to determine the wind direction for the simulation. Below are the standard Cardinal orientations, but any value from 0 – 360 may be entered. ▪ North = 0 ▪ East = 90 ▪ South = 180 ▪ West = 270 **Legal values:** 0 – 360. **Units:** degrees

Cross Section Type	The values in this pulldown depend on the Model Purpose selection. ▪ If Model Purpose = Window / Transparent Façade, the choices are  *Figure 9-4. Cross-Section Type if Model Purpose = Window/Transparent Facade.* *Note: this setting was located under the Therm File Properties tab in previous versions of the program.* ▪ If Model Purpose = Opaque Façade, the choices are  *Figure 9-5. Cross-Section Type if Model Purpose = Opaque Facade.* ▪ If Model Purpose = Other, the choices are  *Figure 9-6. Cross-Section Type if Model Purpose = Other.*
Gravity Vector	The choices for the gravity vector are  *Figure 9-7. Gravity Vector options.* ***Note:*** *Setting the gravity vector was done from the View menu in previous versions of THERM* ***Note:*** *This setting will change the View/Gravity Arrow to match. Don't change gravity vector from the View menu in this version of THERM.*

Miscellaneous tab

Below is a description of the fields in the File / Properties / Miscellaneous tab.

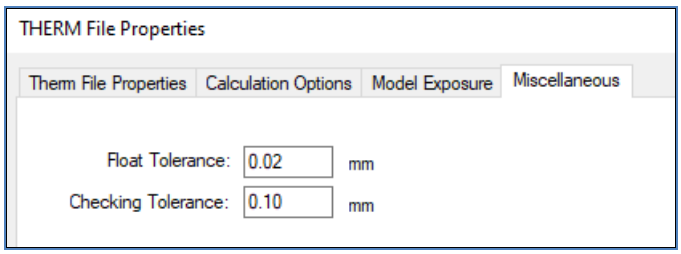


Figure 9-8. File Properties / Miscellaneous tab.

Float Tolerance	Default: 0.02 mm
Checking Tolerance	Default: 0.10 mm

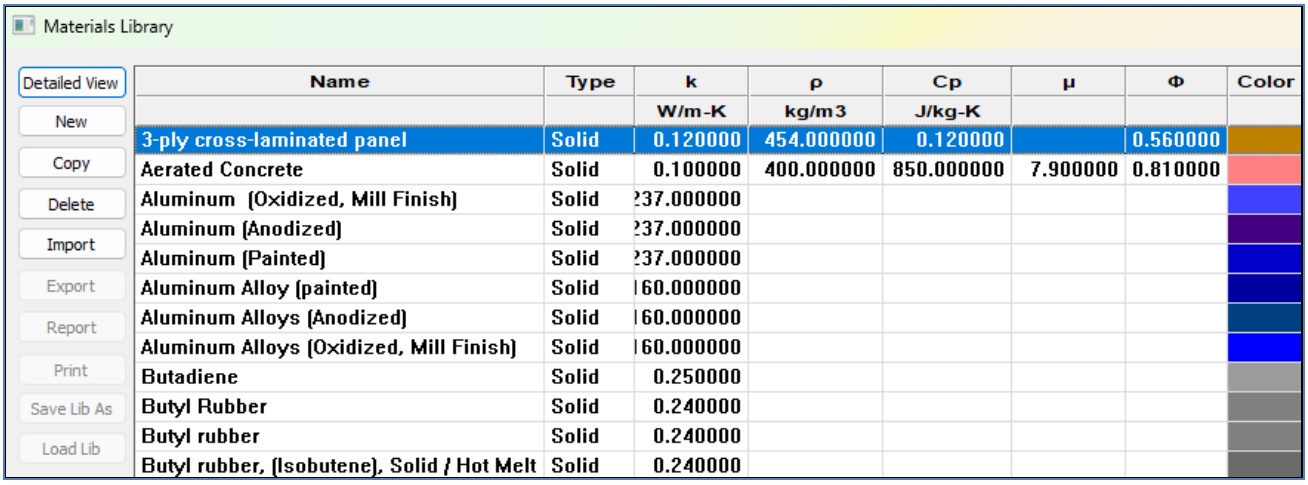
9.3. Material Library

The Material Library is different depending on the Simulation Engine selected.

- ConRad: If the ConRad simulation engine is selected, the Material library is the same as in previous versions of THERM (such as THERM 7), ie, stored in binary files with the extension *.BIN.
- HygroTherm: If the HygroTherm simulation engine is selected, the Material library is much different. The values are stored in an XML file format.

9.3.1. Material Library List View

Selecting the Material choice from the Libraries menu when the HygroTherm simulation model is selected displays the Materials in "List View".



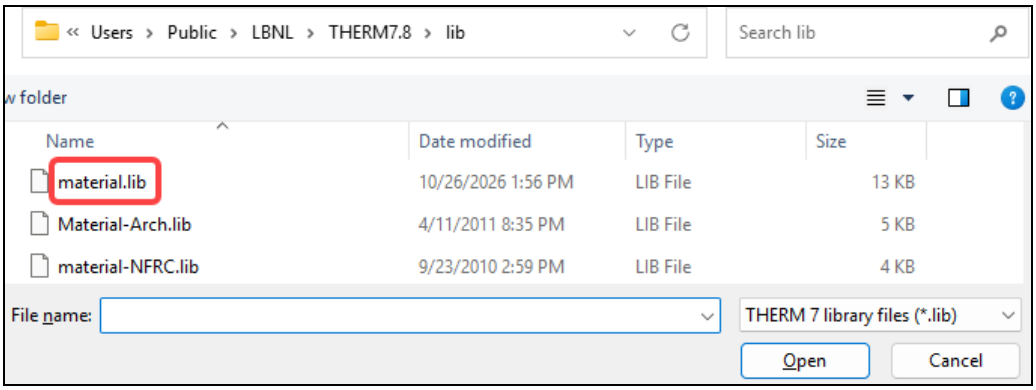
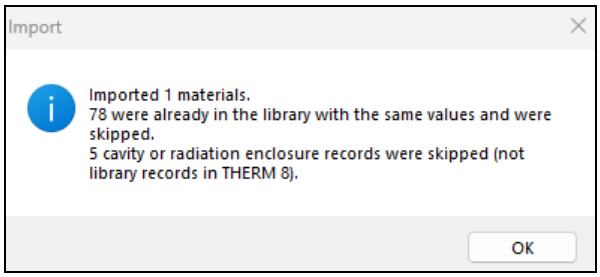
Name	Type	k	ρ	Cp	μ	Φ	Color
		W/m-K	kg/m3	J/kg-K			
3-ply cross-laminated panel	Solid	0.120000	454.000000	0.120000		0.560000	
Aerated Concrete	Solid	0.100000	400.000000	850.000000	7.900000	0.810000	
Aluminum (Oxidized, Mill Finish)	Solid	237.000000					
Aluminum (Anodized)	Solid	237.000000					
Aluminum (Painted)	Solid	237.000000					
Aluminum Alloy (painted)	Solid	160.000000					
Aluminum Alloys (Anodized)	Solid	160.000000					
Aluminum Alloys (Oxidized, Mill Finish)	Solid	160.000000					
Butadiene	Solid	0.250000					
Butyl Rubber	Solid	0.240000					
Butyl rubber	Solid	0.240000					
Butyl rubber, (Isobutene), Solid / Hot Melt	Solid	0.240000					

Figure 9-9. Material List View.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.

Import	Used to import records from previous versions of THERM (7) that were stored in a "lib" file. Clicking this button opens a dialog box to browse to a *.lib file to import from, such as the "material.lib" file in the THERM7.8/lib folder  *Figure 9-10. Importing materials from older THERM *.lib files.* THERM will import all the records in the *.lib file, without user intervention. It will not import records that have the same values, to prevent duplicates, and a dialog box explaining how many records were imported will be displayed after the import.  *Figure 9-11. THERM will report the number of records actually imported, as well as the number of records skipped*
Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Type	The material type, from the options of ▪ Solid ▪ Frame Cavity ▪ Glazing Cavity ▪ External Radiation Enclosure ▪ Shading Material
k	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI)
ρ	Density of the material. **Units:** lb/ft3 (IP); kg/m3 (SI)
Cp	Specific heat of the material. **Units:** Btu/lb-F (IP);J/kg-K (SI)
μ	Water Vapor Diffusion Resistance Factor. **Units**: unitless
Φ	Porosity of the material. **Units:** unitless
Color	The color assigned to the material.

9.3.2. Material Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each material.

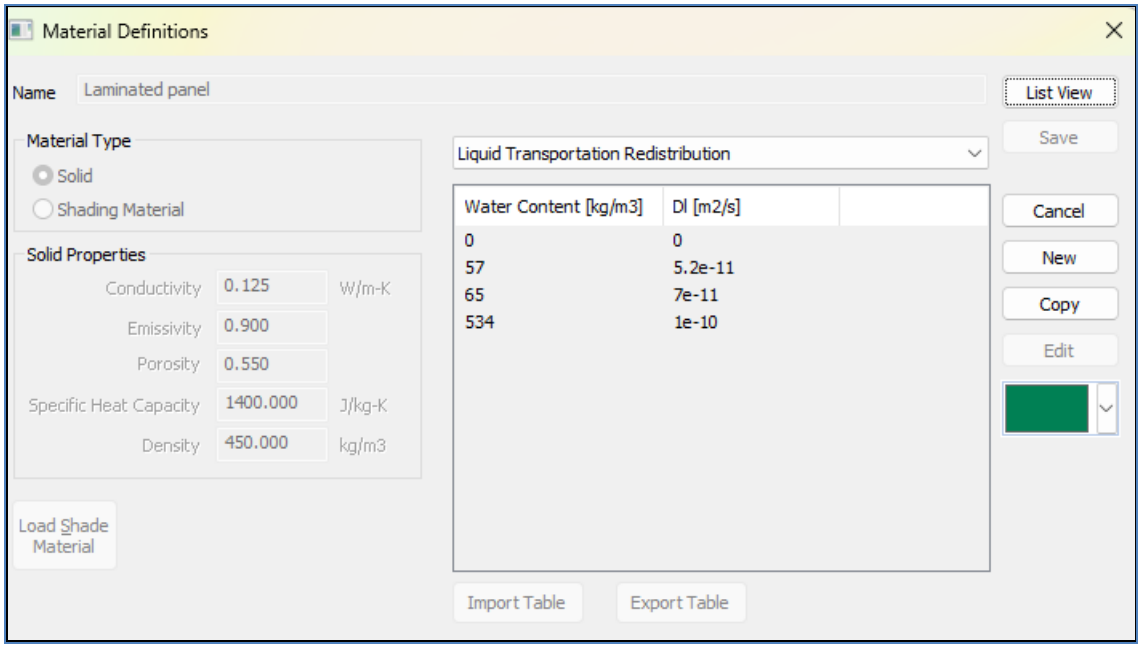


Figure 9-12. Material Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Material Library.
Save	Saves any changes made to the Materials XML file. The default materials in the library are "protected" from editing, but if you make a new record, or copy an existing record (with a different name), the values can then be changed.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. The current record will be copied and all the parameters will be editable. Change the name, set the parameters accordingly and Save the copied record.
Edit	*Currently not implemented.*

Color	The color assigned to the material. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.

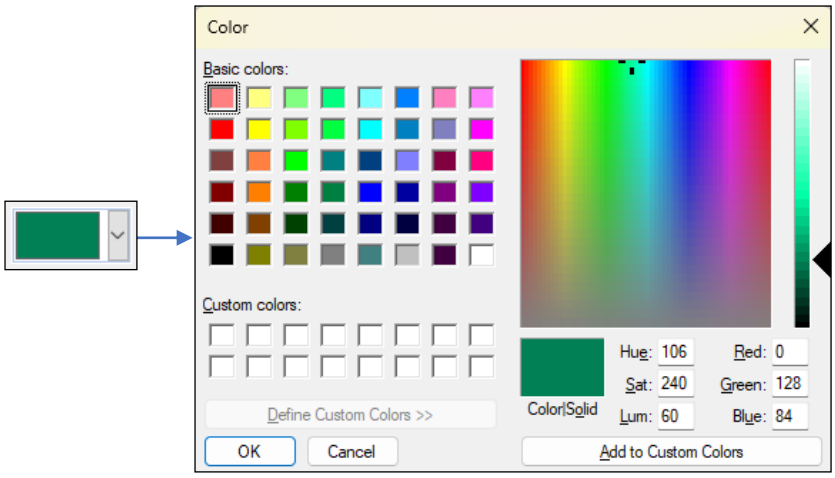


Figure 9-13. Color Settings in the Material Library Detail View.

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Material Type	
	The material type, from the options of ▪ Solid ▪ Shading Material
Solid Properties *These input values are for Material Type = Solid*	
Conductivity	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Emissivity	Emissivity of the material. **Units:** unitless
Porosity	Porosity of the material. **Units:** unitless
Specific Heat Capacity	Specific Heat Capacity of the material. **Units:** Btu/lb-F (IP); J/kg-K (SI).
Density	Density of the material. **Units:** lb/ft3 (IP); kg/m3 (SI).

Liquid Transportation Redistribution

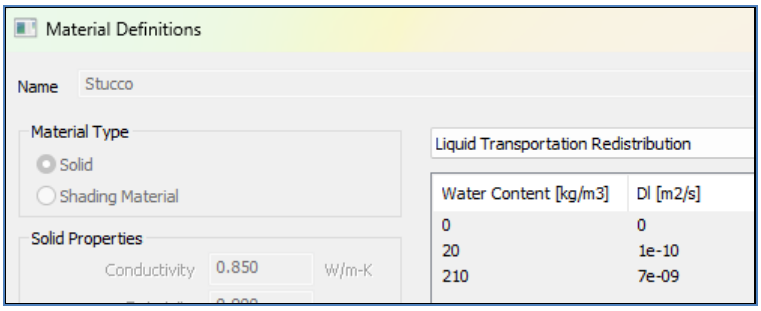


Figure 9-14. Liquid Transportation Redistribution variables for Stucco material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Dl	Liquid transportation coefficient. Units: ft2/s (IP); m2/s (SI).
Import Table	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content ▪ Dl (liquid transportation coefficient) THERM8-stucco-liquidTransportationRedistribution.csv 1 0,0 2 20,1e-10 3 210,7e-09 *Figure 9-15. Import CSV file format for Liquid Transportation Redistribution variables, always in SI units.*
Export Table	The Liquid Transportation Redistribution properties for the material will be exported as a CSV in the same format as shown for importing the file.

Liquid Transportation Suction

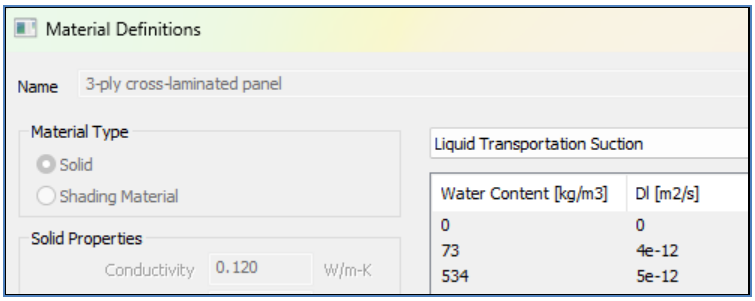


Figure 9-16. Liquid Transportation Suction variables for 3-ply cross-laminated panel material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Dl	Liquid transportation coefficient. Units: ft2/s (IP); m2/s (SI).
Import Table	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content (kg/m3) ▪ Dl (liquid transportation coefficient, m2/s) THERM8-3plypanel-liquidTransportationSuction.csv 1 0,0 2 73,4e-12 3 534,5e-12 *Figure 9-17. Import CSV file format for Liquid Transportation Suction variables for 3-ply cross-laminated panel material, always in SI units.*
Export Table	The Liquid Transportation Suction properties for the material will be exported as a CSV in the same format as shown for importing the file.

Moisture Dependent Thermal Conductivity

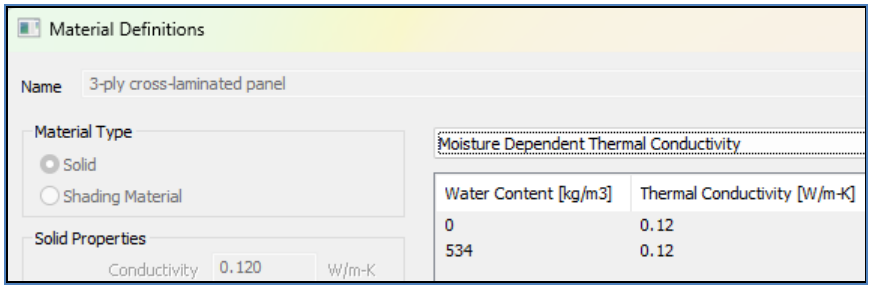


Figure 9-18. Moisture Dependent Thermal Conductivity variables for Laminated Panel material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Thermal Conductivity	Thermal conductivity. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Import Table	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Water Content (kg/m3) ▪ Thermal Conductivity (W/m-K) THERM8-laminatedPanel-MoistureDependentThermalConductivity.csv 1 0,0.12 2 534,0.12 *Figure 9-19. Import CSV file format for Moisture Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.*
Export Table	The Mosture Dependent Thermal Conductivity properties for the material will be exported as a CSV in the same format as shown for importing the file.

Moisture Storage Function

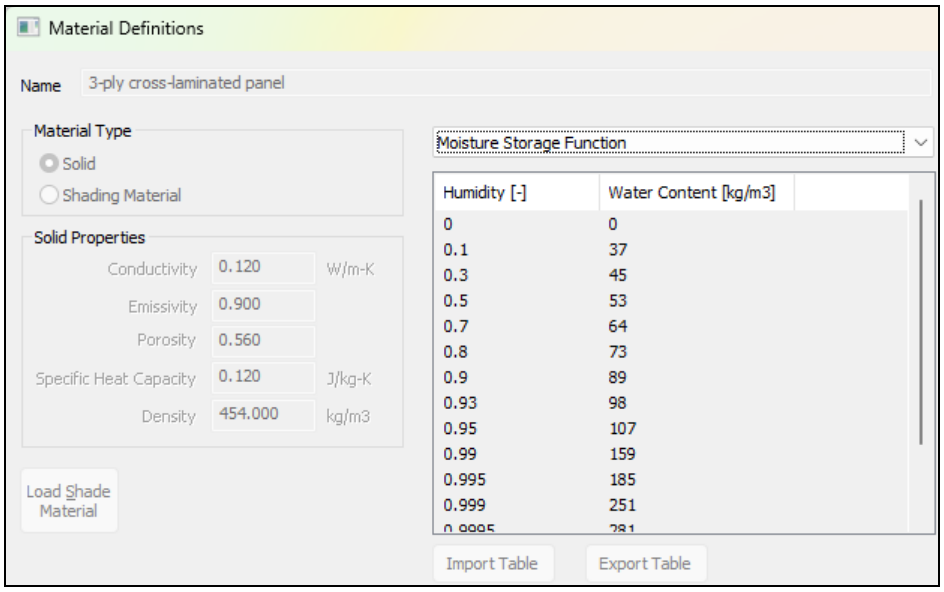


Figure 9-20. Moisture Storage Function variables for Laminated Panel material.

Humidity	Humidity fraction. Legal values: 0 – 1. Units: unitless.
Water content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Import Table *Export Table*	To import data into this function, create a CSV file with the two values separated by a comma. The data can also be exported into a CSV file of the same format as the import file. The values (always in SI units) are: ▪ Humidity (fraction, 0-1) ▪ Water Content (kg/m3) THERM8-LaminatedPanel-MoistureStorageFunction.csv 1 0,0 2 0.1,37 3 0.3,45 4 0.5,53 5 0.7,64 6 0.8,73 7 0.9,89 8 0.93,98 9 0.95,107 10 0.99,159 11 0.995,185

Figure 9-21. Import CSV file format for Moisture Storage Function variables for Lamianted Panel material, always in SI units.

Temperature Dependent Thermal Conductivity

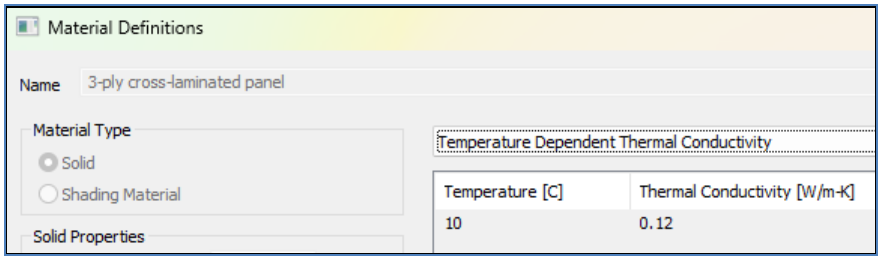
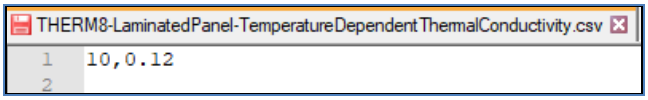


Figure 9-22. Temperature Dependent Thermal Conductivity variables for Laminated Panel material.

Temperature	Temperature. Units: F (IP); C (IP).
Thermal Conductivity	Thermal conductivity. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Import File *Export File*	To import data into this function, create a CSV file with the two values separated by a comma. The data can also be exported into a CSV file of the same format as the import file. T The values (always in SI units) are: ▪ Temperature (C) ▪ Thermal Conductivity (W/m-K)  *Figure 9-23. Import CSV file format for Temperature Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.*

File Format and Location

The Material Library is stored in an XML file called "Materials.xml" which is located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.0\lib

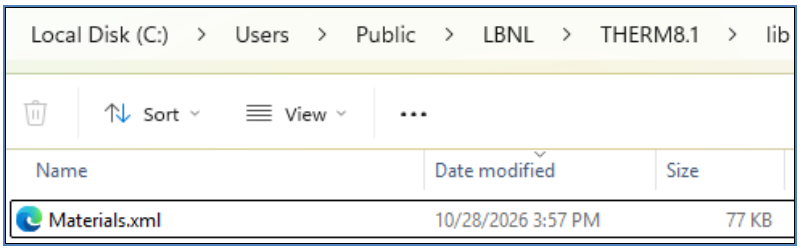


Figure 9-24. The Materials.xml file is located in the "lib" subfolder of the THERM working directory.

The definitions of all the materials displayed in the THERM Material Library are stored in this XML file.

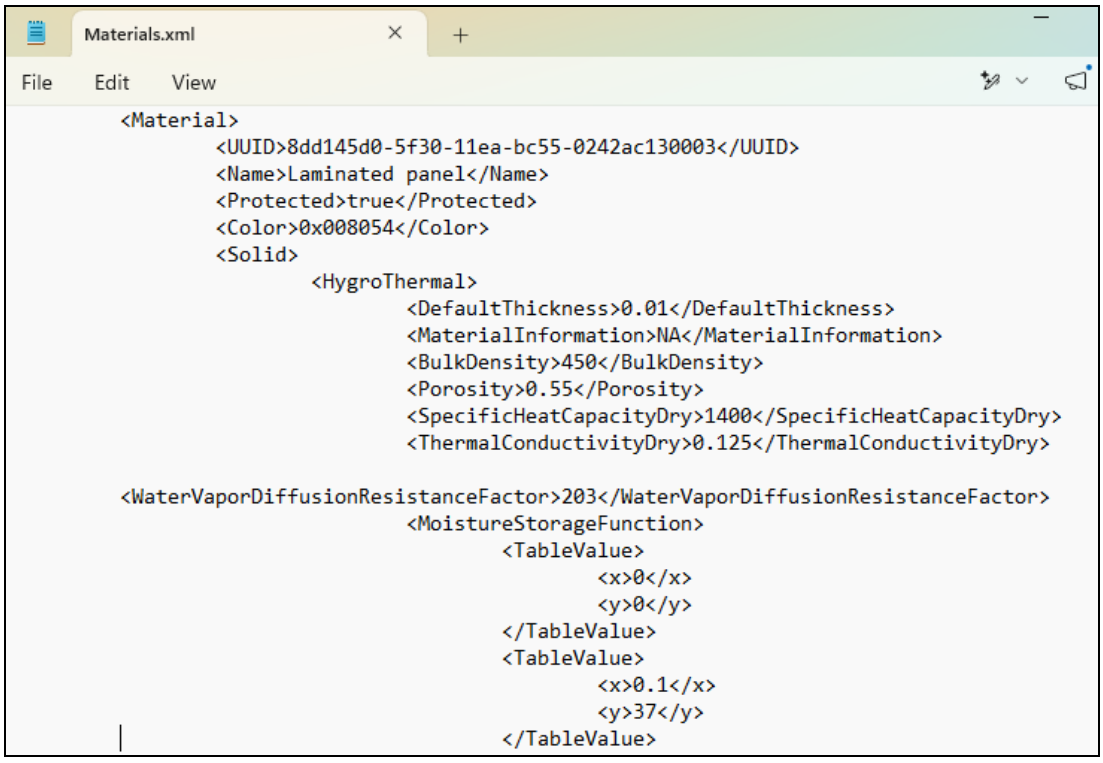


Figure 9-25. The Materials.xml file contains all the values for all the material records in the Material Library.

Making New Material Library Records

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the Material.xml file.

For example, if a new "solid" material record is created, and values added for the Solid Properties section as well as the different functions on the right side, all those changes are then saved in the Materials.xml file.

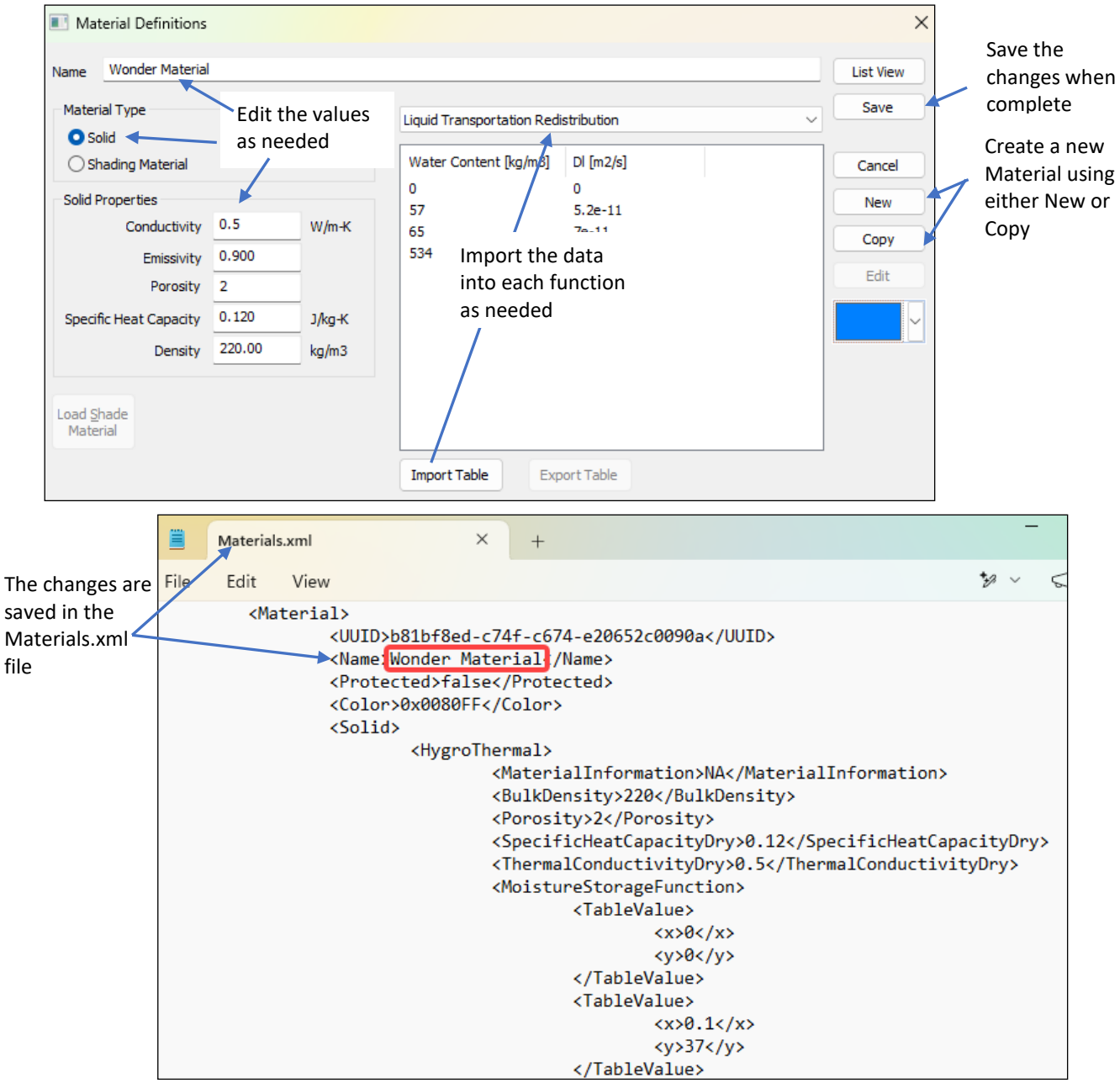


Figure 9-26. Make a new material using either New or Copy and change the input values as needed.

9.4. Boundary Condition Library

9.4.1. Boundary Condition Library List View

Selecting the Boundary Condition choice from the Libraries menu displays the Boundary Condition records in "List View".

There are two sets of boundary conditions, one for steady-state simulations and one for transient calculations. In future versions, these will be combined into one library.

Simulation Engine: Conrad

XML File: BoundaryConditionsSteadyState.xml

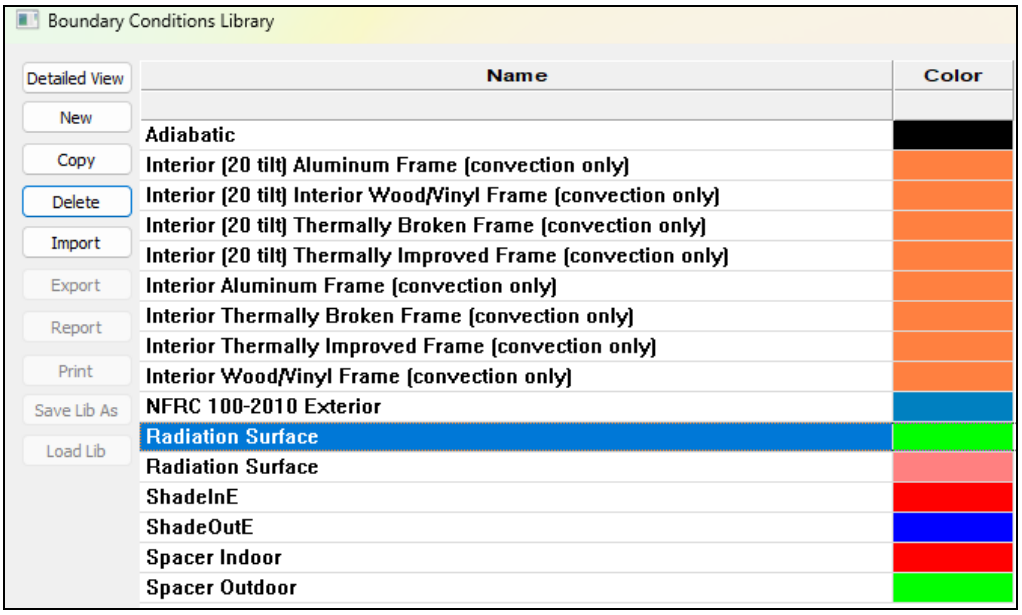


Figure 9-27. Boundary Condition Library List View of the default steady-state records.

Simulation Engine: HygroTherm, Transient model

XML File: BoundaryConditionsTransient.xml

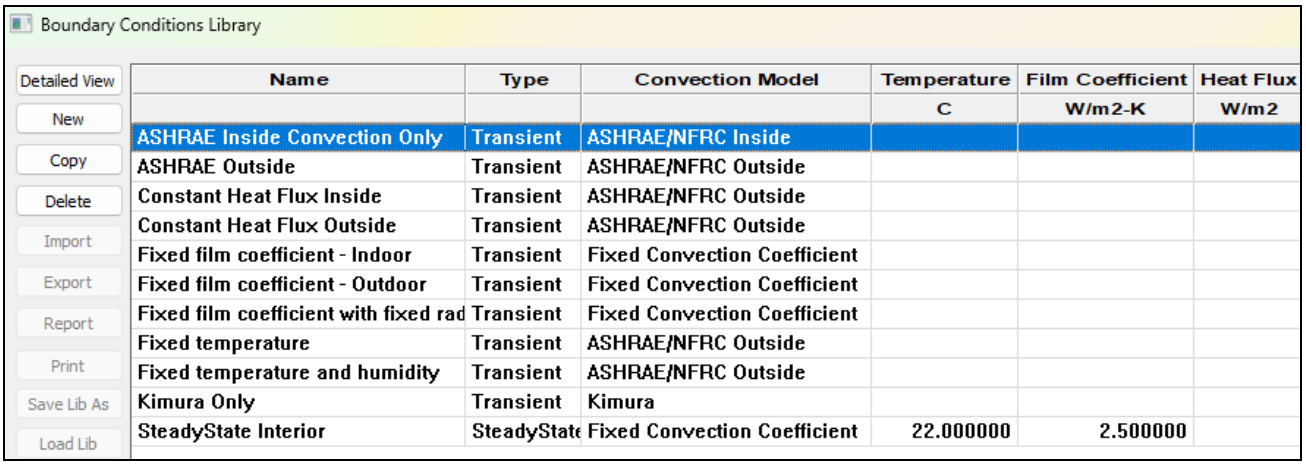
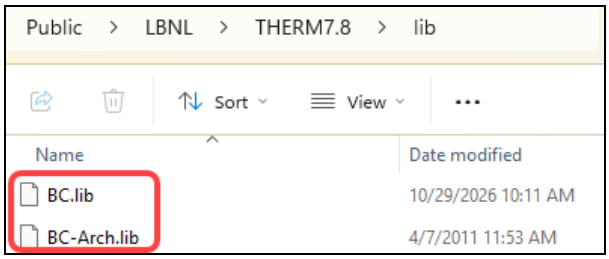
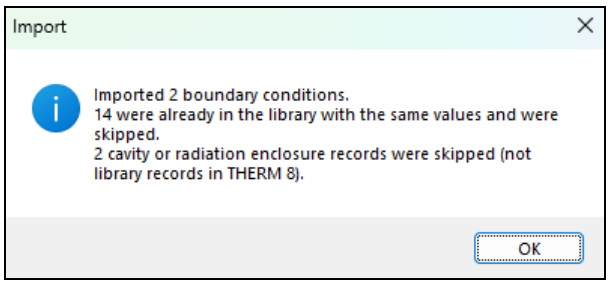


Figure 9-28. Boundary Condition Library List View of the default steady-state records.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.
Import	*Only available for steady-state modeling.* Used to import records from previous versions of THERM (7) that were stored in a "lib" file. Clicking this button opens a dialog box to browse to a *.lib file to import from, such as the "material.lib" file in the THERM7.8/lib folder  *Figure 9-29. Importing materials from older THERM *.lib files.* THERM will import all the records in the *.lib file, without user intervention. It will not import records that have the same values, to prevent duplicates, and a dialog box explaining how many records were imported will be displayed after the import.  *Figure 9-30. THERM will report the number of records actually imported, as well as the number of records skipped*

Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The columns in the list view will depend on the XML library being displayed, which is controlled by the simulation engine and model type (see above), ie, **steady-state** or transient. The List view contains columns with selected results (from the Detail view) for each record, as described below.

Steady State Boundary Conditions, HygroTherm XML File: BoundaryConditionsSteadyState.xml	
Name	The name of the boundary condition.
Color	The color assigned to the boundary condition.
Transient Boundary Conditions, HygroTherm XML File: BoundaryConditionsTransient.xml	
Name	The name of the boundary condition.
Type	The boundary condition type, from the options of ▪ Transient ▪ Steady State (this is for the case where the transient moisture model is used with steady-state algorithms)
Convection Model	Options are: ▪ ASHRAE/NFRC Outside ▪ ASHRAE/NFRC Inside ▪ Fixed Convection Coefficient ▪ TARP ▪ Yazdanian Klems ▪ Kimura ▪ Montazeri
Temperature	Convection Model Air Temperature when modeling steady-state conditions. **Units:** F (IP); C (SI)

Film Coefficient	Convection Model Film Coefficient when modeling steady-state conditions. **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Heat Flux	Convection Model Heat Flux when modeling steady-state conditions. **Units:** Btu/h-ft2 (IP); W/m2 (SI)
Solar Radiation	Convection Model Solar Radiation when modeling steady-state conditions. **Units:** Btu/h-ft2 (IP); W/m2 (SI)
Radiation Model	For the Neumann Boundary Condition Type, if the Radiation Checkbox is checked, the Radiation model will be one of the following: ▪ Automatic Enclosure Model ▪ Black Body Radiation ▪ Fixed Radiation Coefficient
Film Coefficient	Radiation Model Film Coefficient when modeling steady-state conditions. **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Temperature	Radiation Model Radiation Surface Temperature when modeling steady-state conditions. **Units:** F (IP); C (SI)
Humidity	Radiation Model Humidity fraction when modeling steady-state conditions. **Legal Values: 0-1** **Units:** unitless

9.4.2. Boundary Condition Library Detail View: HygroTherm, Transient

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each boundary condition.

When the HygroTherm simulation engine is selected, the detailed input will depend on the simulation engine and whether the boundary condition is for steady-state or transient modeling

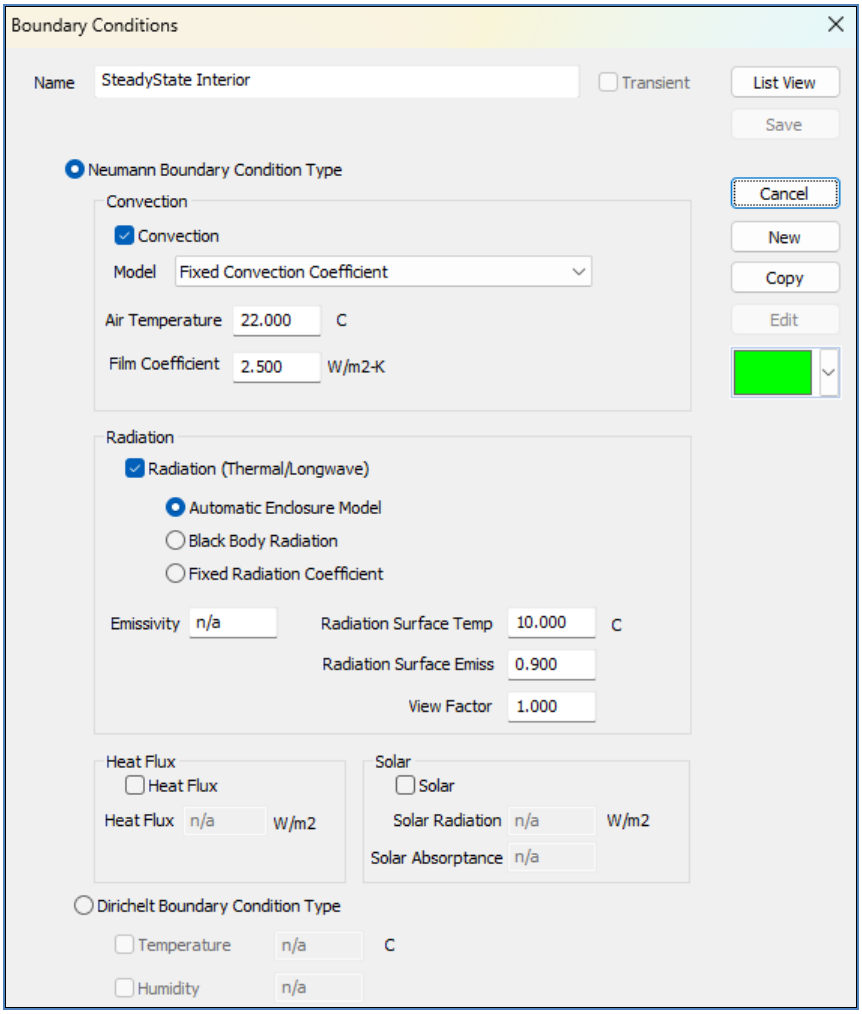


Figure 9-31. Boundary Condition Detail View for simulation engine HygroTherm, steady-state boundary conditions.

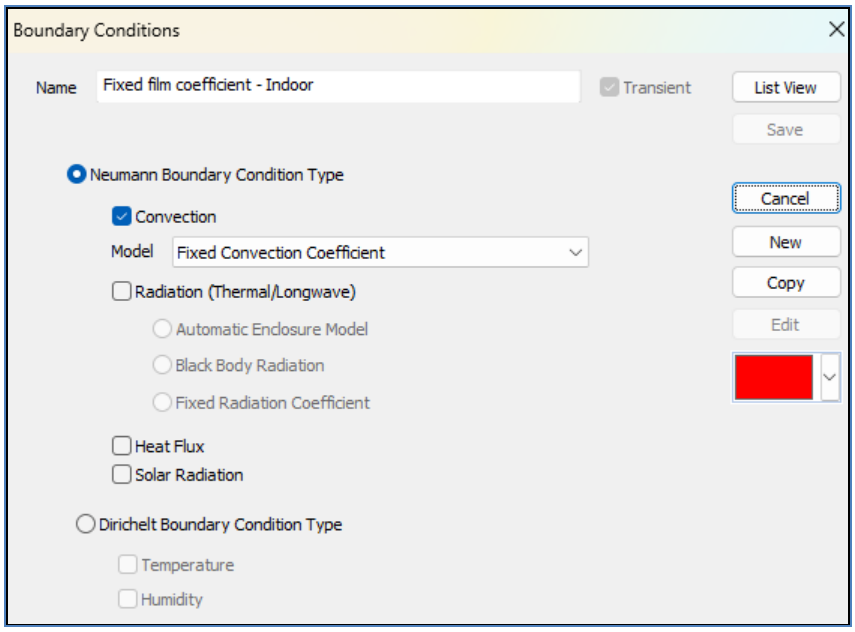
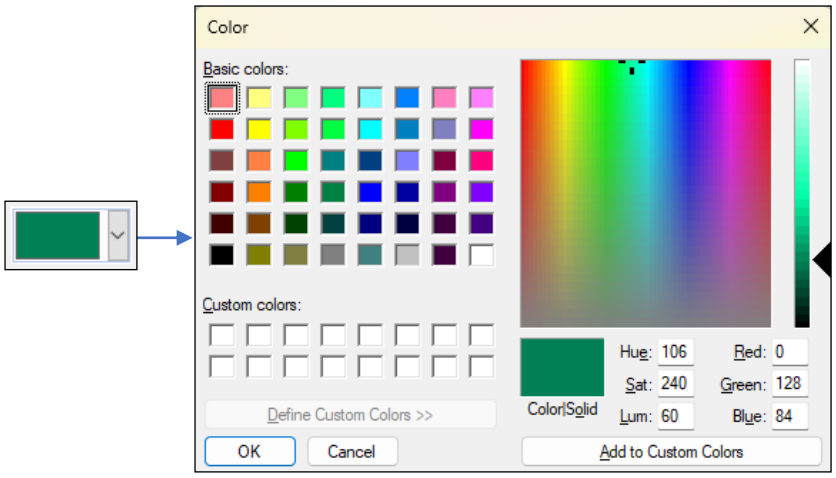


Figure 9-32. Boundary Condition Detail View for simulation engine HygroTherm, transient boundary conditions.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Boundary Condition Library.
Save	Saves any changes made to the BoundaryConditions XML file. If changes are made to a field, move off the field in order for the Save button to become activated.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record and edit the name and parameters accordingly. Then click the Save button to save the changes.
Edit	*Currently not implemented.*

Color	The color assigned to the boundary condition. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.  *Figure 9-33. Color Settings in the Boundary Conditions Library Detail View.* One convention is to make the interior (warm) boundary conditions in the "red" colorway, and the exterior (cold) boundary conditions in the "blue" colorway.

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Steady State Boundary Conditions	
Name	The name of the boundary condition.
Transient	Unchecked for the SteadyState boundary conditions for the HygroTherm simulation engine.
Neumann Boundary Condition Type	
Convection	This option is checked if the boundary condition will include convection modeling.
Model	The Model pulldown list pertains to the type of convection model to be used, from the following choices: ▪ ASHRAE/NFRC Outside ▪ ASHRAE/NFRC Inside ▪ Fixed Convection Coefficient ▪ TARP ▪ Yazdanian Klems ▪ Kimura ▪ Montazeri

Air Temperature	The Air Temperature set in the Convection section of the Boundary Condition Detail View. **Units:** F (IP); C (SI)
Film Coefficient	Convection Model Film Coefficient when modeling steady-state conditions. **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Radiation	This option is checked if the boundary condition will include Radiation modeling The Radiation model options are: ▪ Automatic Enclosure Model ▪ Black Body Radiation ▪ Fixed Radiation Coefficient
Heat Flux	This option is checked if the boundary condition will include Heat Flux modeling
Solar Radiation	This option is checked if the boundary condition will include Solar Radiation modeling
Dirichelt Boundary Condition Type	
Temperature	Check this option to include Temperature in this Boundary Condition Type
Humidity	Check this option to include Humidity in this Boundary Condition Type

File Format and Location

The Boundary Condition Library is stored in two XML files which are located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

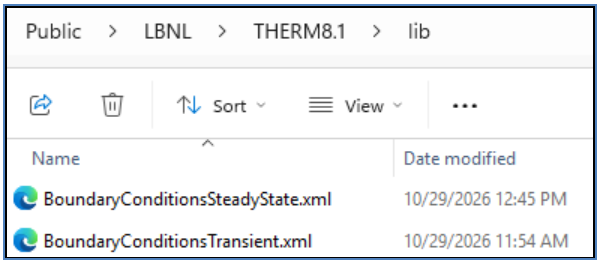


Figure 9-34. The Boundary Conditions XML files are located in the "lib" subfolder of the THERM working directory.

The definitions of all the boundary conditions displayed in the THERM Boundary Condition Library are stored in this XML file.

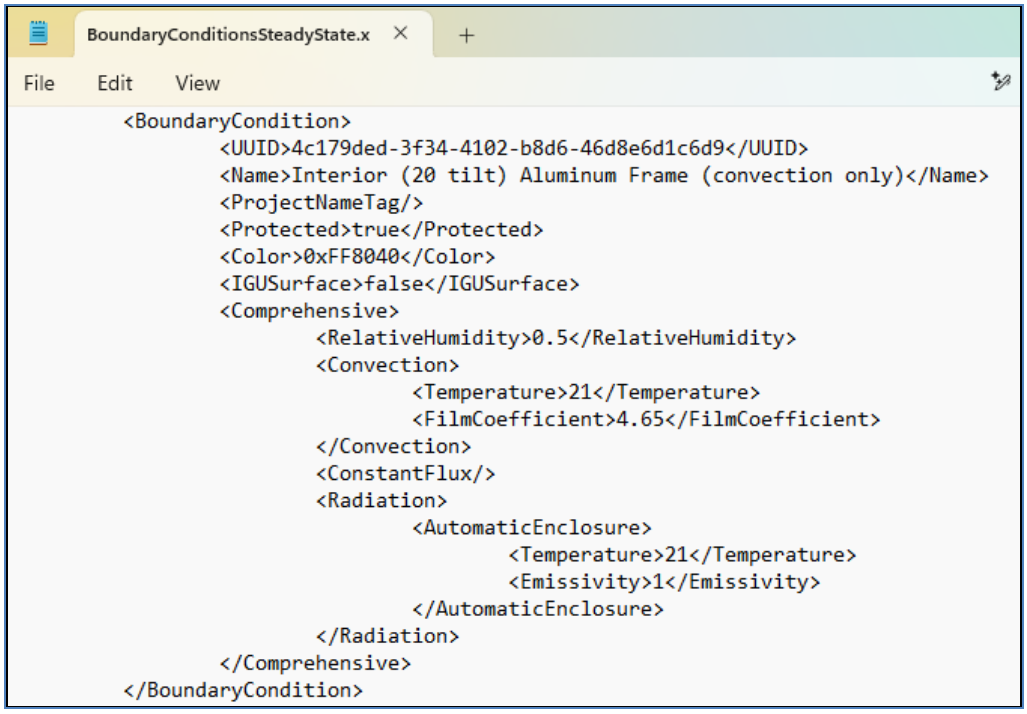
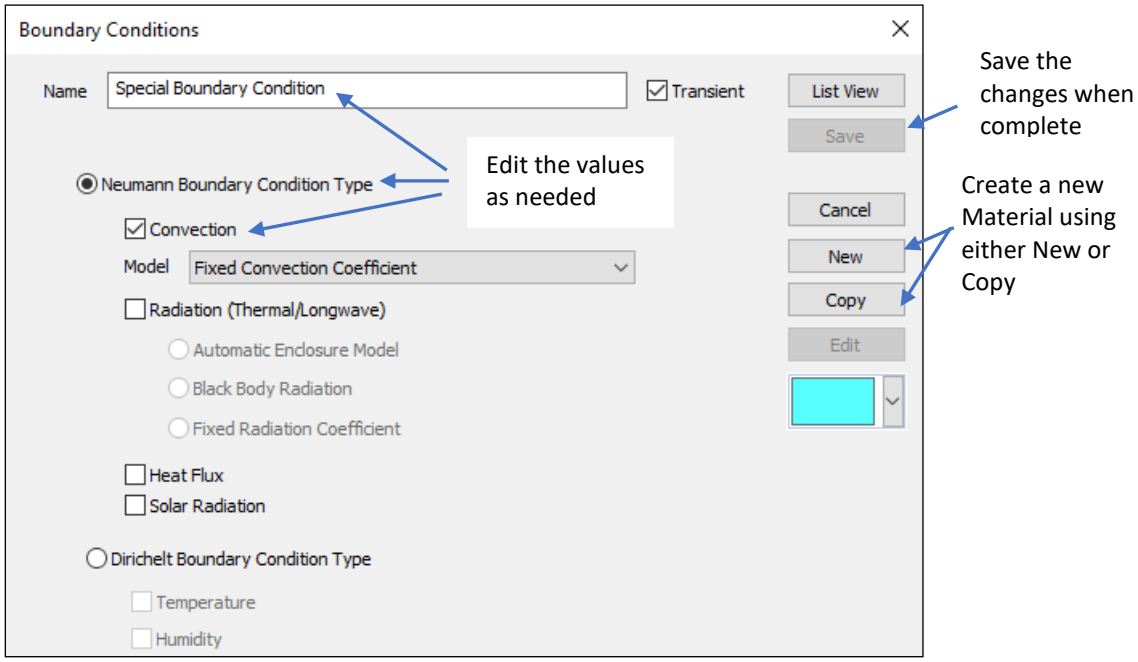


Figure 9-35. The BoundaryConditionsSteadyState.xml file contains all the values for all the material records in the Material Library.

Making New Boundary Condition Library Records

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the BoundaryCondition.xml file.

For example, if a new boundary condition record is created, all the settings for that boundary condition are saved in the BoundaryConditions.xml file.



The changes are saved in the BoundaryConditions.xml file

```
147      <Name>Special Boundary Condition</Name>
148      <Protected>false</Protected>
149      <BCType>Transient</BCType>
150      <BCModel>Neumann</BCModel>
151      <Convection>
152        <Model>Fixed Convection Coefficient</Model>
153      </Convection>
154      <UseHeatFlux>false</UseHeatFlux>
155      <UseTemperature>false</UseTemperature>
156      <UseHumidity>false</UseHumidity>
157      <Color>0x57FFFF</Color>
158    </BoundaryConditionType>
159  </BoundaryConditionsType>
```

Figure 9-36. Make a new boundary condition using either New or Copy and change the input values as needed. The changes will be saved into the BoundaryConditions.xml file.

Time Step Boundary Condition Files

In addition to specifying a standard Boundary Condition for a boundary condition segment when using the transient simulation engine (HygroTherm), it is also necessary to specify a Time Step Boundary Condition, which are stored in separate XML files.

There are several examples of Time Step files that are installed with THERM 8, and they are found in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.1\lib

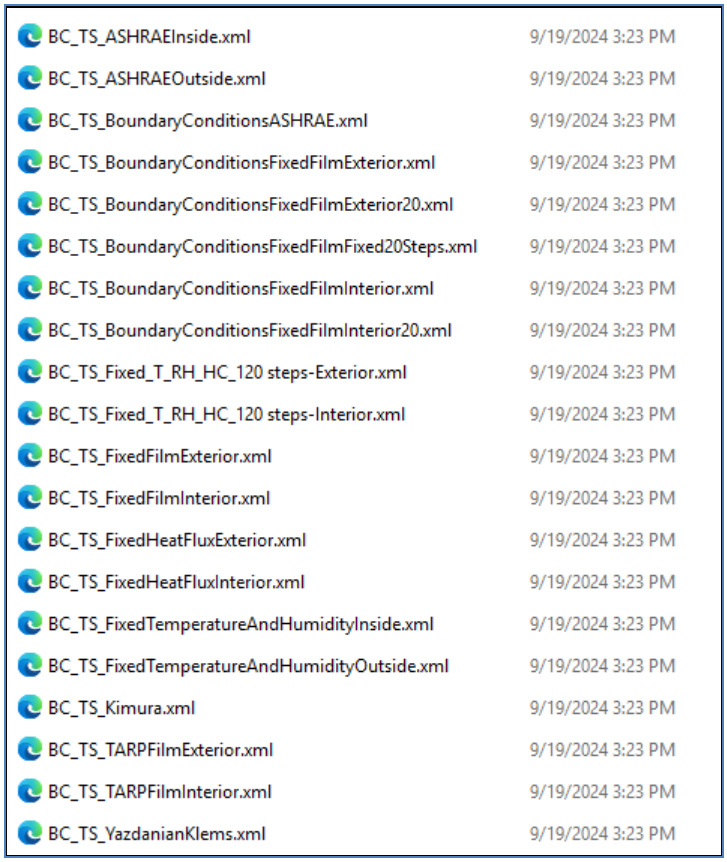
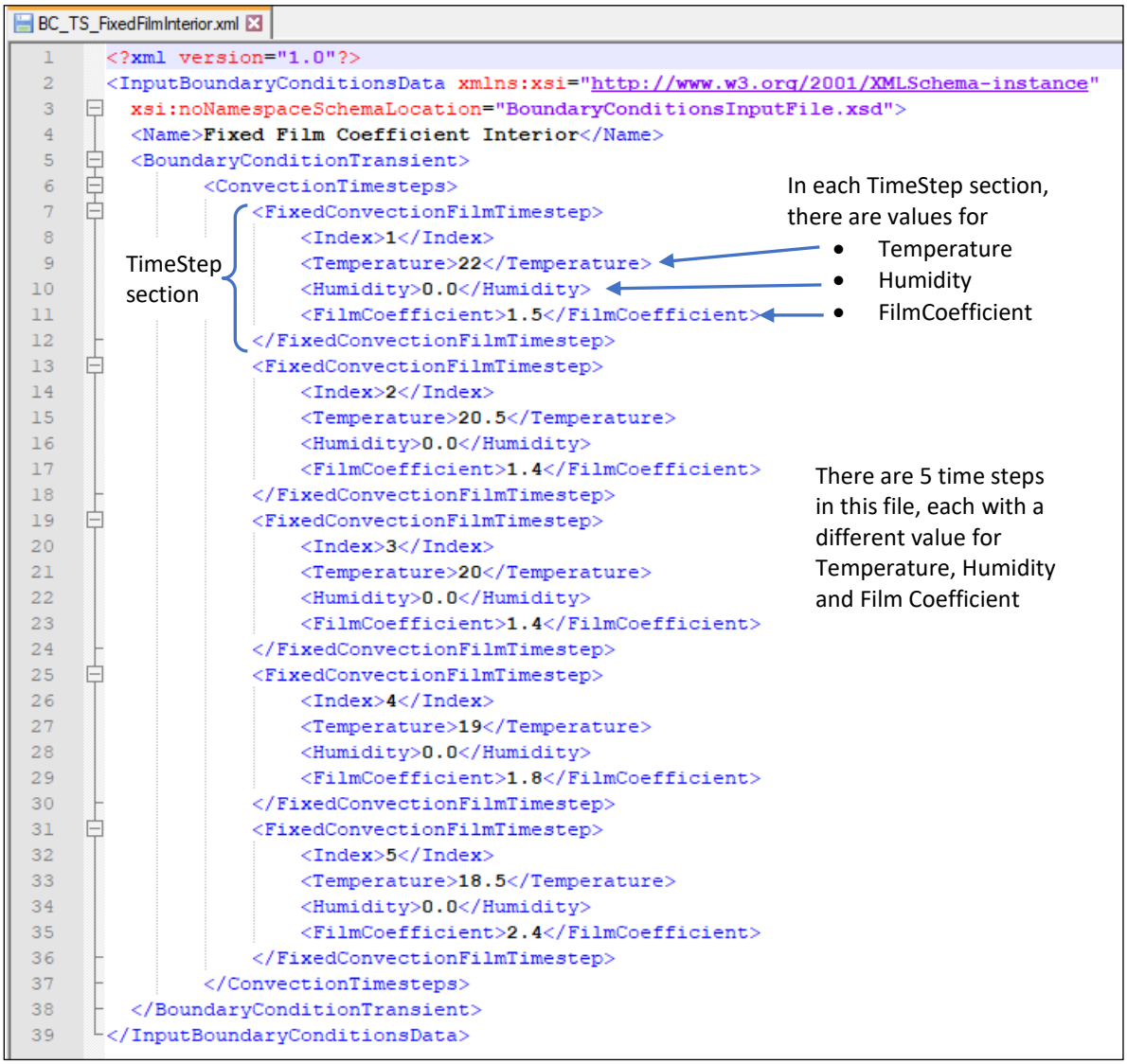


Figure 9-37. The Boundary Condition Time Step files are located in the "lib" subfolder of the THERM working directory.

These example files have a common naming convention of "BC_TS_" to indicate that they are **B**oundary **C**ondition **T**ime **S**tep files, but the files can have any name desired.

The Time Step files contain values for as many time steps as are being defined in each.

For example, the example file "BS_TS_FixedFilmInterior.xml" file has 5 timestep sections, with definitions for Temperature, Humidity and Film Coefficient for each time step.




```
<?xml version="1.0"?>
<InputBoundaryConditionsData xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
 xsi:noNamespaceSchemaLocation="BoundaryConditionsInputFile.xsd">
 <Name>Fixed Film Coefficient Interior</Name>
 <BoundaryConditionTransient>
      <ConvectionTimesteps>
          <FixedConvectionFilmTimestep>
              <Index>1</Index>
              <Temperature>22</Temperature>
              <Humidity>0.0</Humidity>
              <FilmCoefficient>1.5</FilmCoefficient>
          </FixedConvectionFilmTimestep>
          <FixedConvectionFilmTimestep>
              <Index>2</Index>
              <Temperature>20.5</Temperature>
              <Humidity>0.0</Humidity>
              <FilmCoefficient>1.4</FilmCoefficient>
          </FixedConvectionFilmTimestep>
          <FixedConvectionFilmTimestep>
              <Index>3</Index>
              <Temperature>20</Temperature>
              <Humidity>0.0</Humidity>
              <FilmCoefficient>1.4</FilmCoefficient>
          </FixedConvectionFilmTimestep>
          <FixedConvectionFilmTimestep>
              <Index>4</Index>
              <Temperature>19</Temperature>
              <Humidity>0.0</Humidity>
              <FilmCoefficient>1.8</FilmCoefficient>
          </FixedConvectionFilmTimestep>
          <FixedConvectionFilmTimestep>
              <Index>5</Index>
              <Temperature>18.5</Temperature>
              <Humidity>0.0</Humidity>
              <FilmCoefficient>2.4</FilmCoefficient>
          </FixedConvectionFilmTimestep>
      </ConvectionTimesteps>
 </BoundaryConditionTransient>
</InputBoundaryConditionsData>
```

Figure 9-38. The Boundary Condition Time Step file format.

Applying Boundary Conditions to a Model

Applying boundary conditions to a model is similar to how they are applied in previous versions of THERM, with a few exceptions.

- After the model is completed, click the BC toolbar button to assign the boundary conditions. Intially, all the boundary conditions for the model will be set to "Adiabatic".

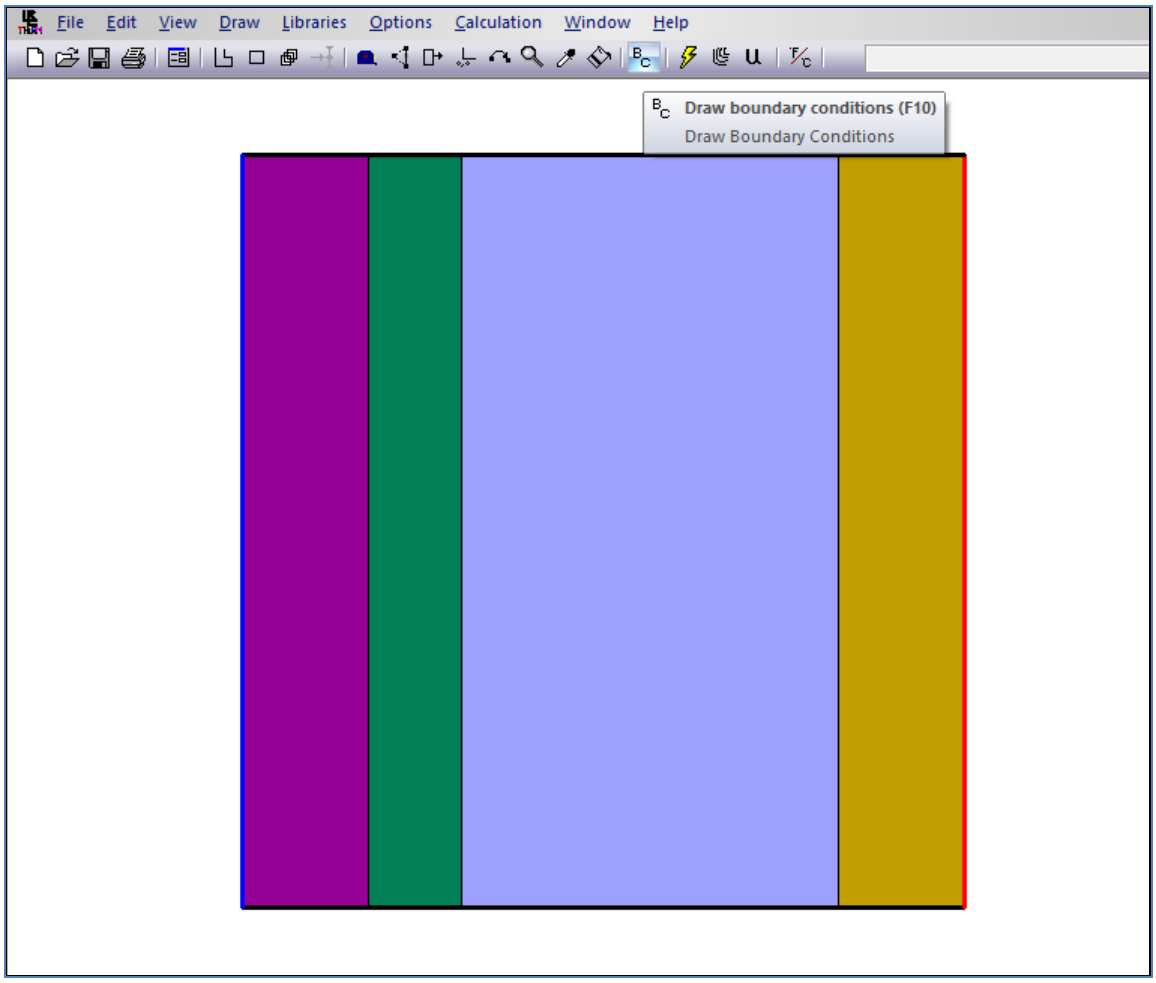
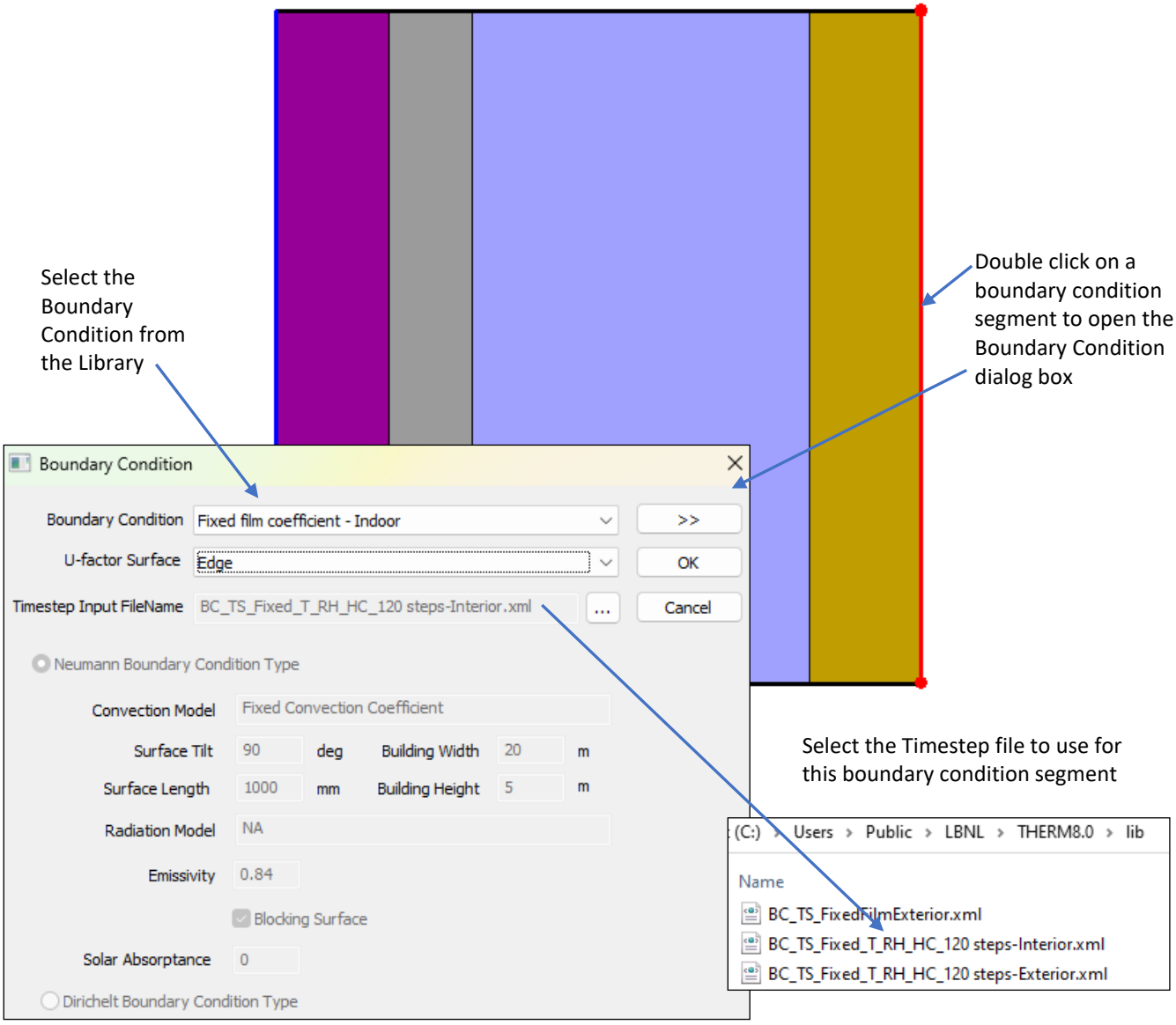


Figure 9-39. Click the BC toolbar icon to generate the intial boundary conditions for the model, which will initially all be set to Adiabatic (with a black color0.

- Select the desired boundary condition segments to assign boundary conditions to and either double click the left mouse button or press the Enter key, in order to open the Boundary Condition selection dialog box.
- Select the Boundary Condition from the pulldown list which displays the records from the Boundary Condition Library
- Select the U-factor Surface tag as applicable
- Select the Timestep Input Filename as applicable. The OK button will not be activated until a Timestep Input Filename has been specified

Figure 9-40. Specify both a Boundary Condition and a Timestep File for each boundary segment in a model.



9.5. Simulate the Model

When the model has been drawn, and materials and boundary conditions assigned, the model can be simulated by clicking the Calc toolbar icon, or selecting the Calculation/Calculation (F9) menu option.

In the example file below (from the sample file called "Stucco Wall – Moisture.thm"), the **File/Properties Time Step** is set to **3600 seconds** (1 hour), the **Number of Time Steps** is set to **100**, and the **QuadTree Mesh Parameter** is set to **4**.

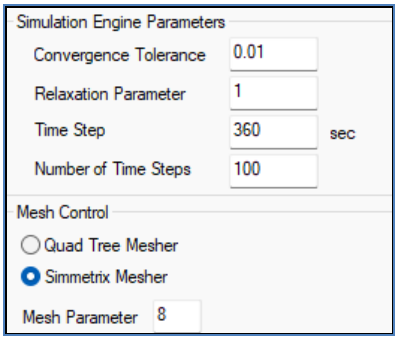


Figure 9-41. File/Properties setting for the sample file called "Stucco Wall – Moisture.thm".

The sample file Stucco Wall – Moisture.thm consists of five polygons.

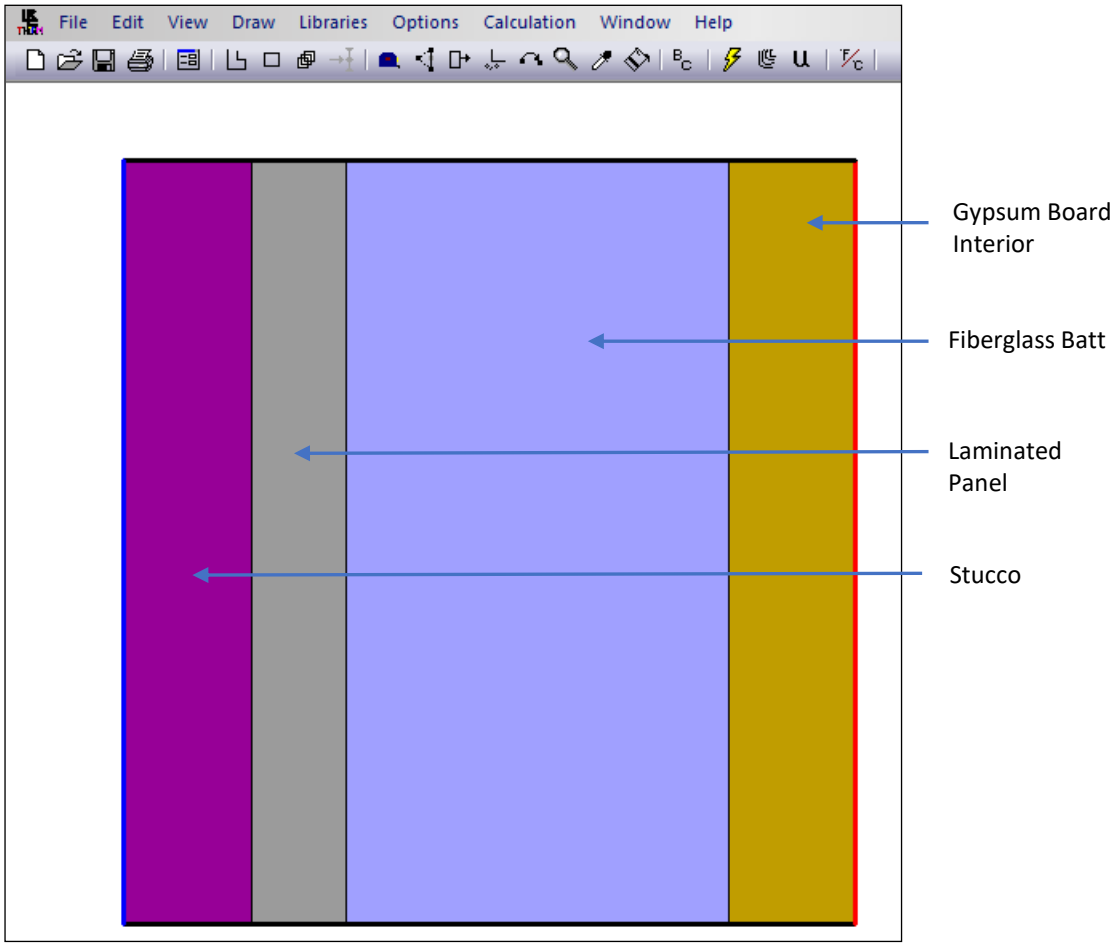


Figure 9-42. Stucco Wall – Moisture.thm.

The boundary condition definitions are shown below. The Timestep Input files have a timestep of 120, and the File/Properties Time Step value is set to 100, so the lower of the two timestep settings will determine the time step for the simulation, in this case 100 timesteps.

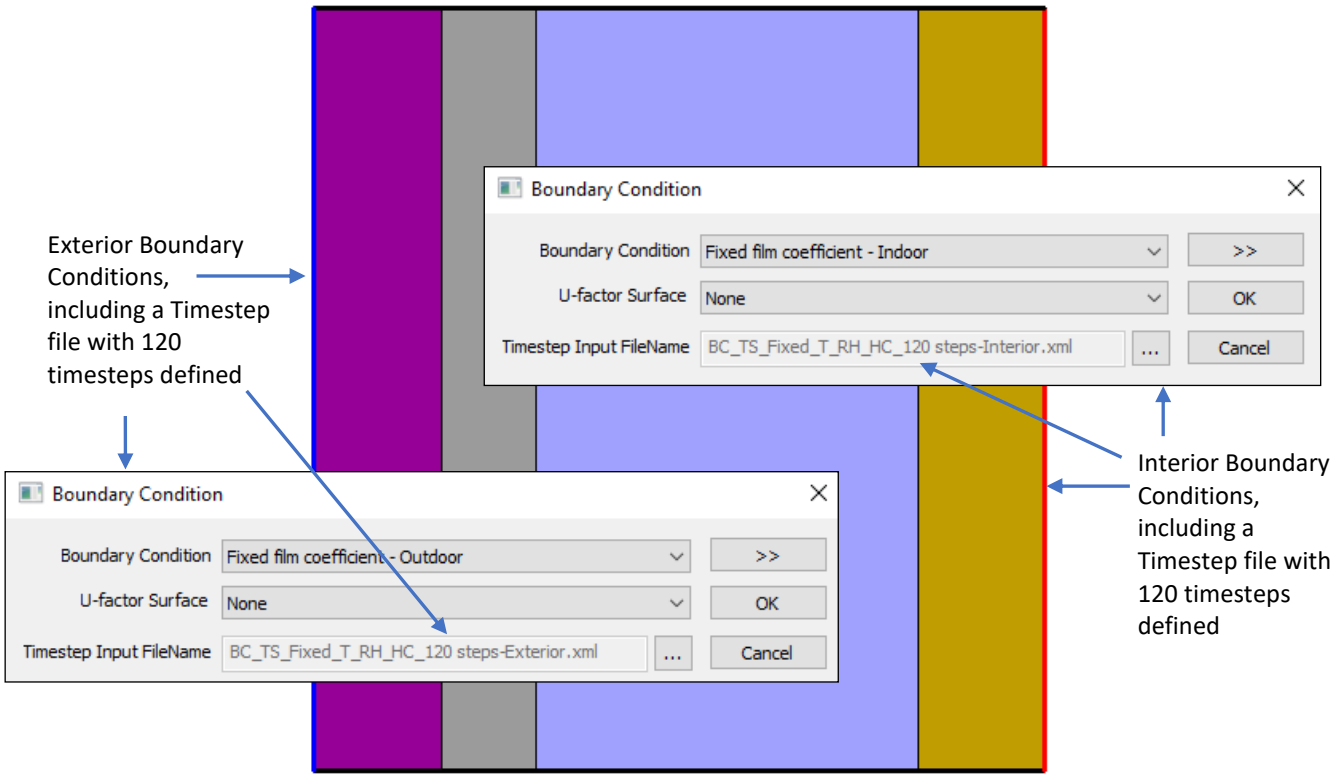


Figure 9-43. Boundary Condition definitions.

During the transient simulation, the program will display a small progress window showing the current timestep.

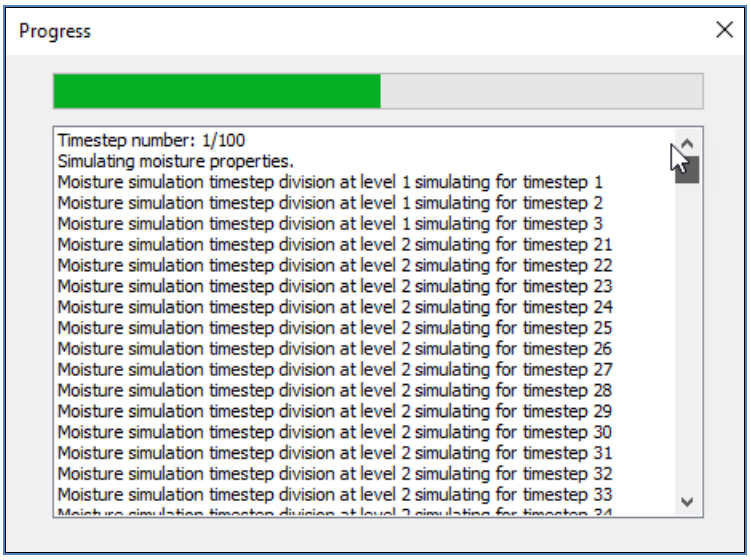


Figure 9-44. A progress window shows the status of the transient simulation.

9.6. View the Results

When the simulation has finished, the progress window will close, and the results will be automatically displayed.

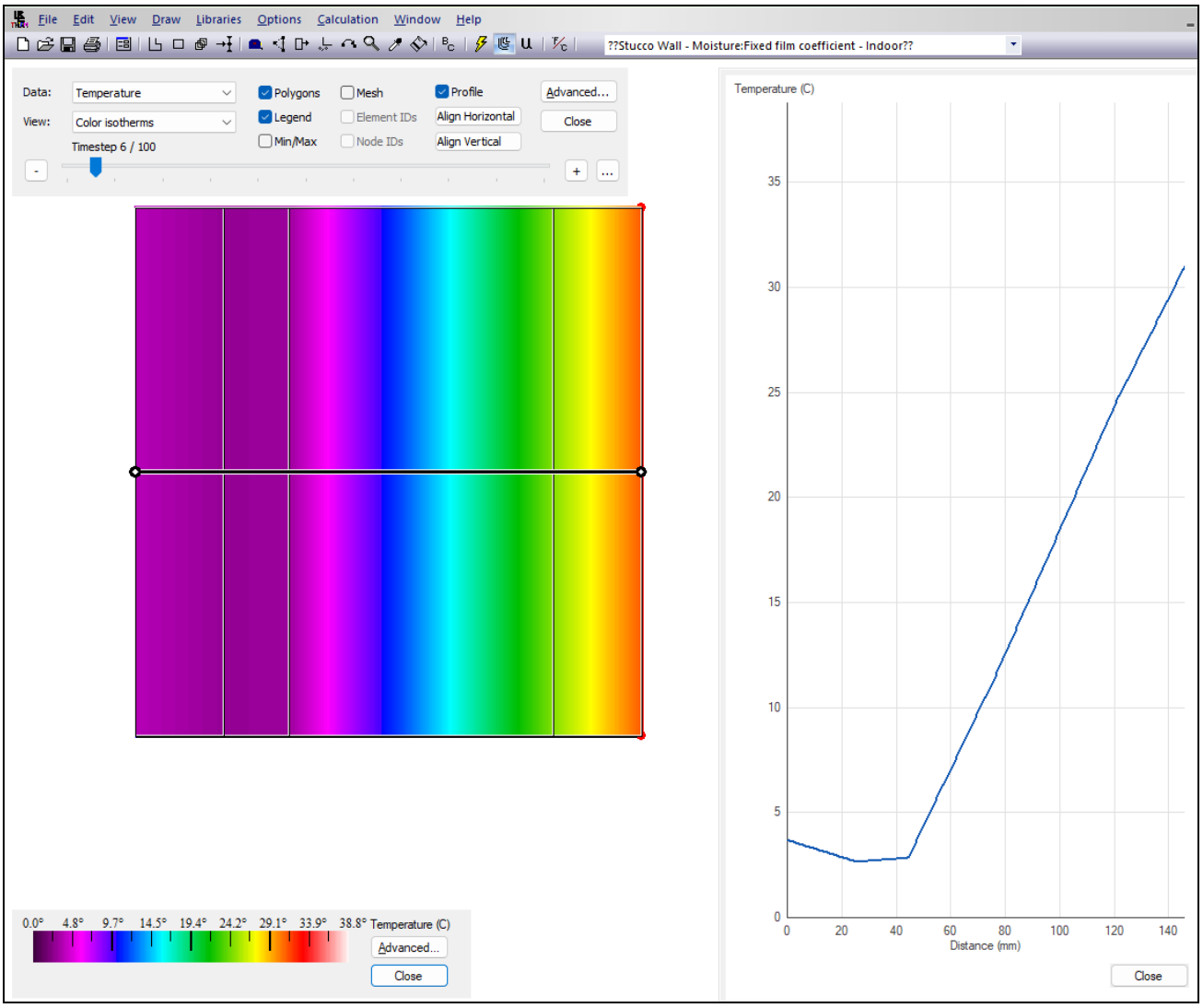


Figure 9-45. When a simulation has completed, the results are displayed.

The slider at the top of the visualization tool allows results to be viewed at each timestep. There are also different values that can be displayed, chosen from the Data pulldown list:

- Temperatures
- Humidities
- Water Content
- Ice Content

The slider on the left side allows placement of the horiztonal location line, which determines the values displayed in the graph on the right hand side. This is useful when viewing results for a non-symmetrical model.

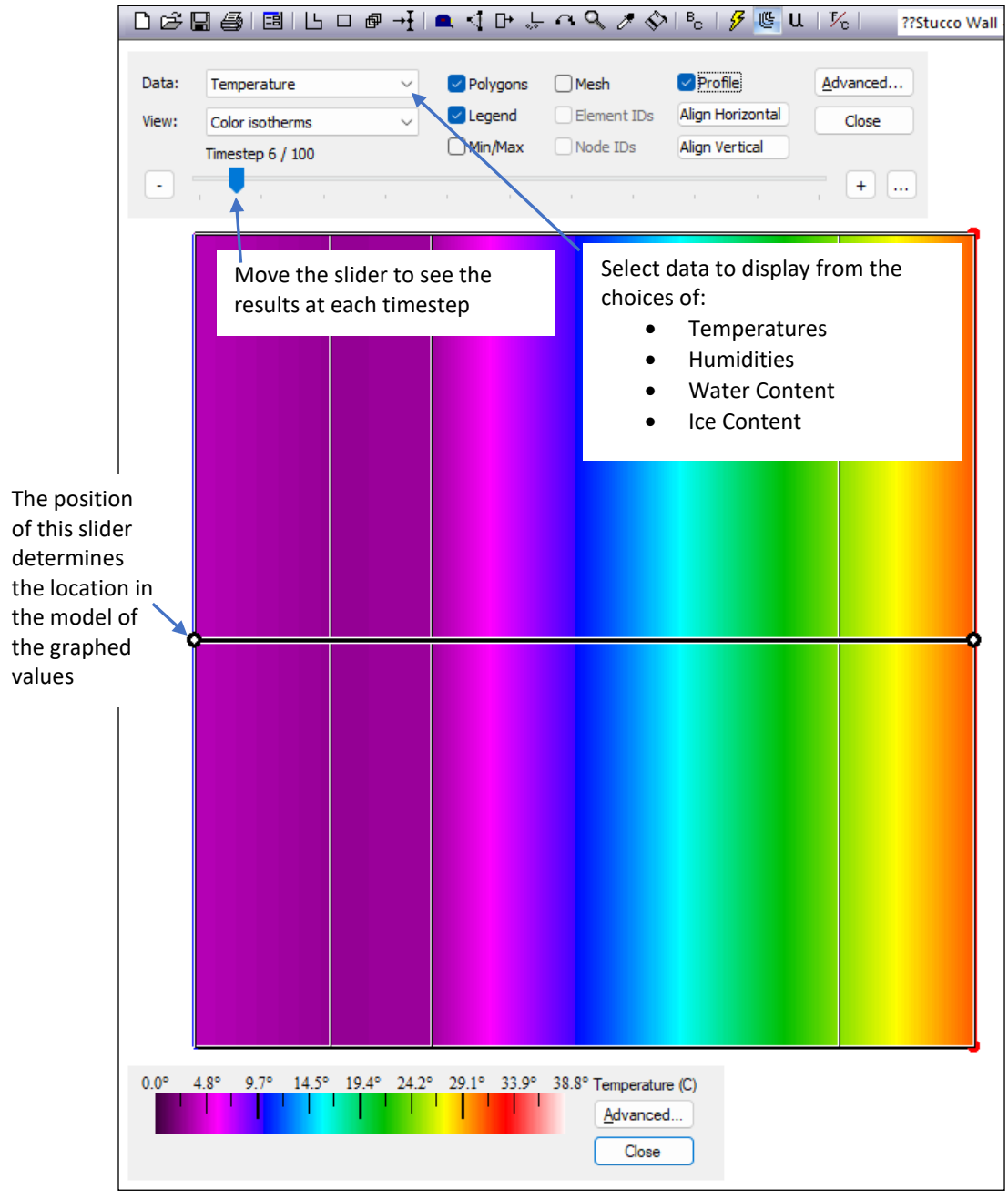


Figure 9-46. The THERM Visualization tool control options

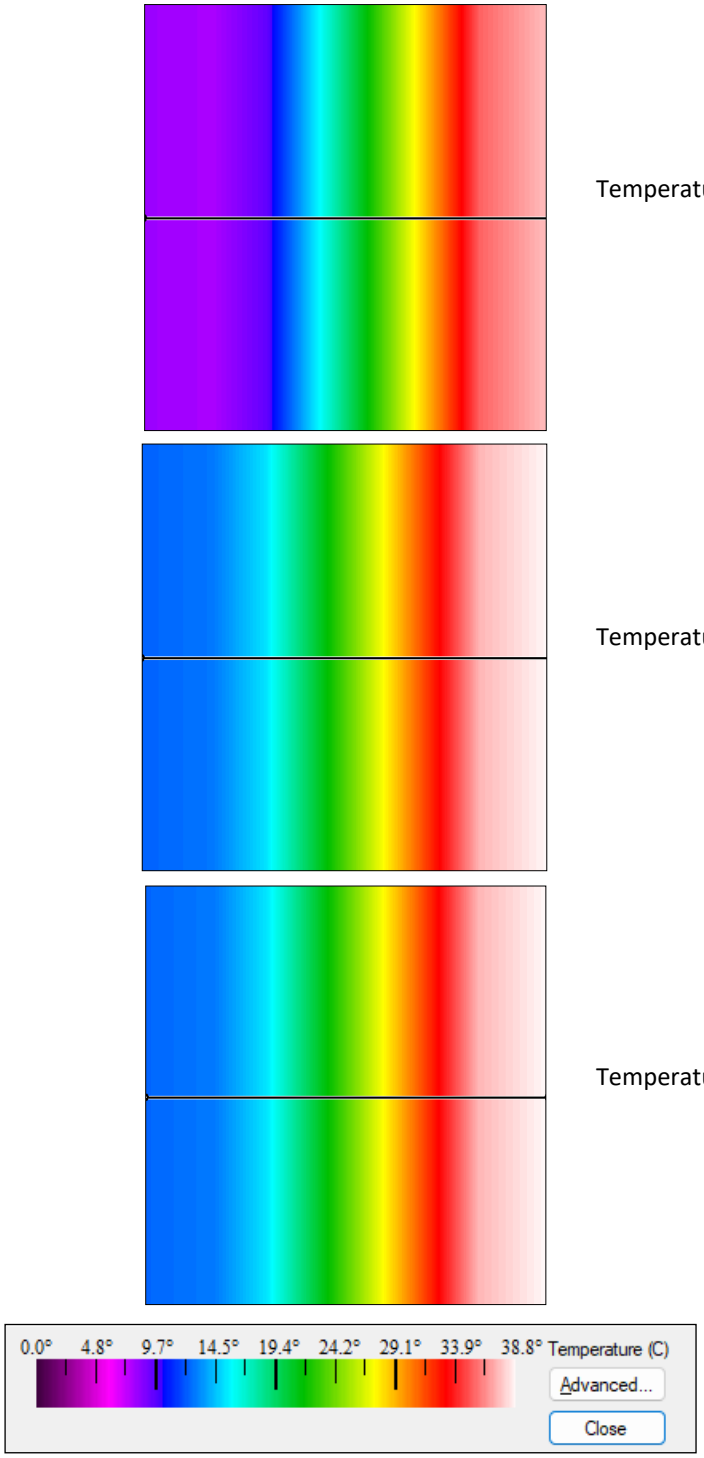


Figure 9-47. Temperatures over the timesteps

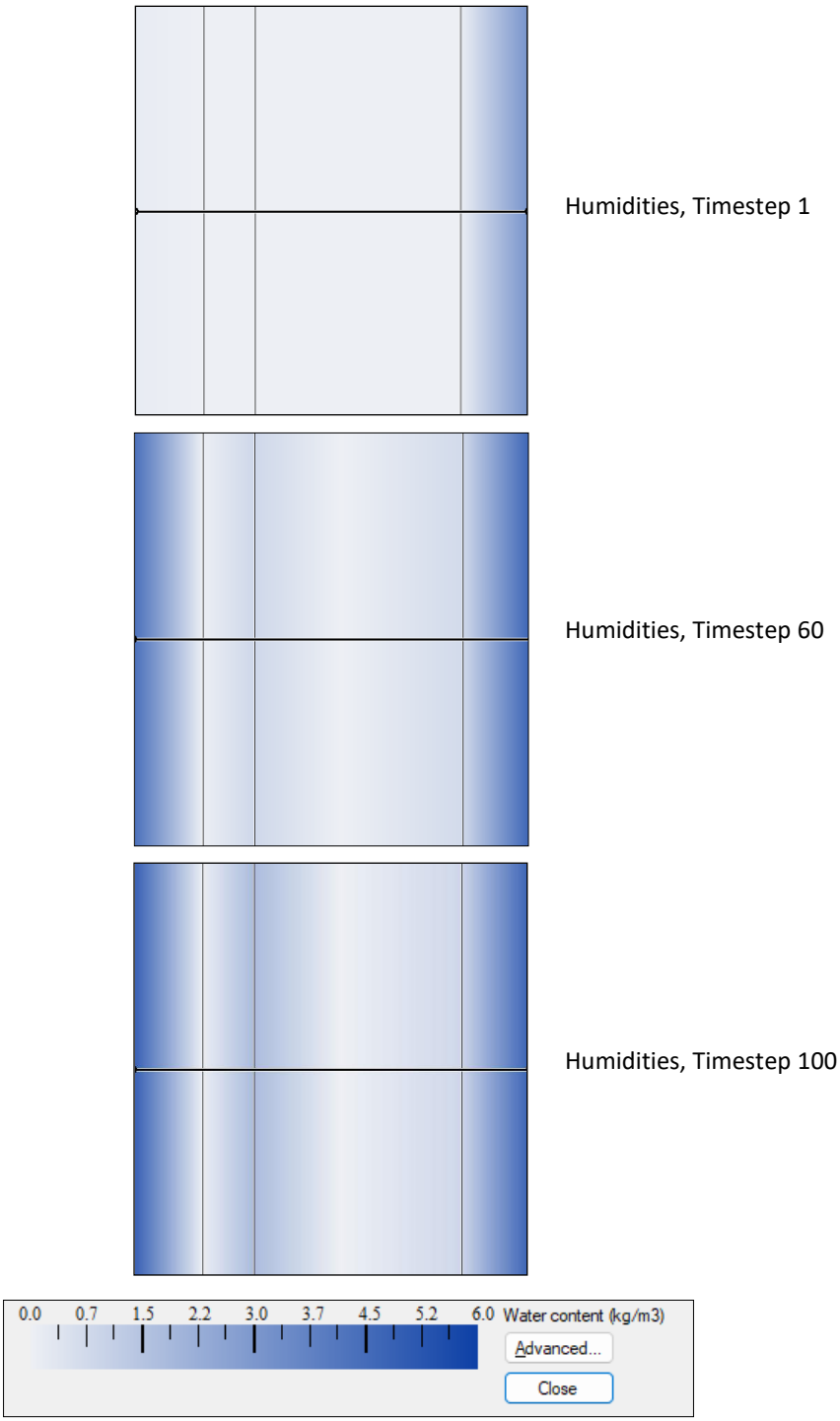


Figure 9-48. Humidities over the timesteps

10. TUTORIAL

10.1. Tutorial Organization

The tutorial section of this manual is organized as follows:

10.2. Start THERM.

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

10.3.1. Using THERM's autoconvert DXF file feature,

10.3.2. Iimporting and tracing an underlay file,

10.3.3. Using a dimensioned drawing.

10.4. Import a glazing system from WINDOW (a version compatible with the THERM version)

10.5. Define the Boundary Conditions.

10.6. Run and View the Results.

10.2. Start THERM

From the Microsoft Windows™ **Start** button, **Programs** choice, click on THERM to activate the program.

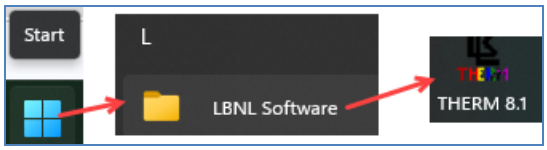


*Figure 10-1. Start THERM from the Windows™ **Start** button.*

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.

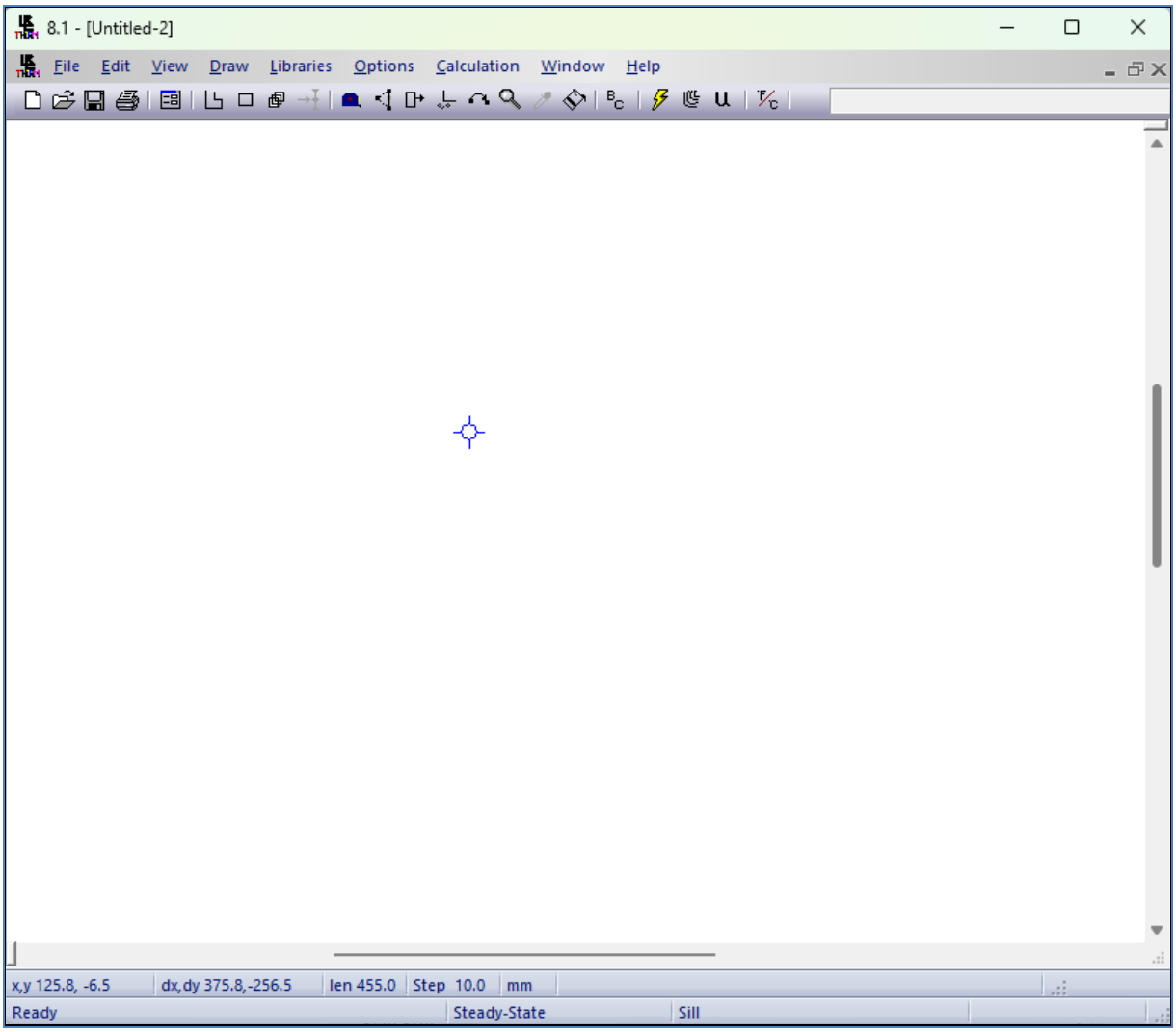


Figure 10-2. THERM starts with a blank drawing area and no files loaded.

10.3. Draw the Frame Cross-Section Geometry

You can use three methods to input a cross-section geometry. Section 10.3.3 shows how to input a cross section using the THERM underlay autoconvert feature, which automatically converts polygons from a DXF file. Section 10.3.4 shows how the same DXF underlay file can be used to trace the polygons. Section 10.3.5 shows how the same profile can be input from a dimensioned drawing.

10.3.3. Import Underlay with Autoconvert On

- Select the dominant material from the Material Library: If there is a dominant material in the DX file, if you select it in the **Mateiral Library**, the auto-converted polygons will be set to that material.
- From the **File** menu, select the **Underlay** menu choice.

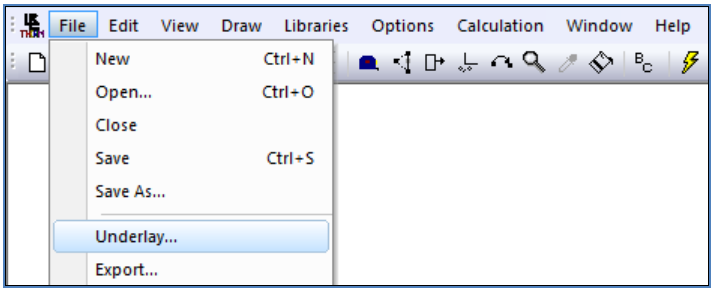


*Figure 10-3. To import an underlay, use the **File** menu, **Underlay** choice.*

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

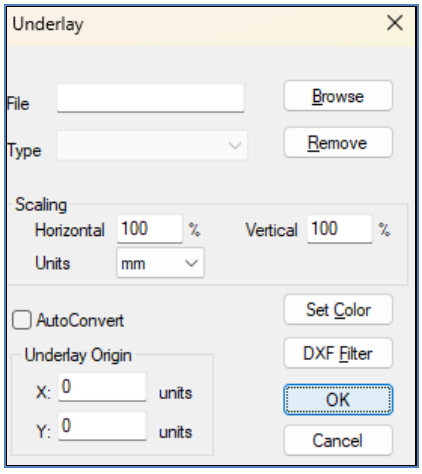


*Figure 10-4. The **Underlay** dialog box is used to import a file to use as an underlay for a THERM cross section.*

- 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.

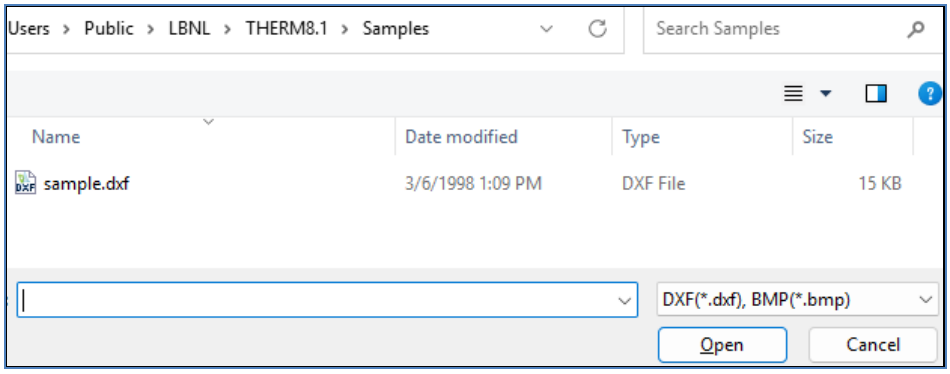


Figure 10-5. Select "sample.dxf" as the underlay file.

- 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

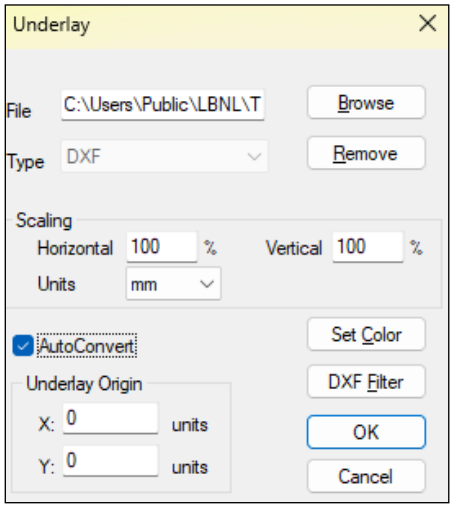


*Figure 10-6. Click the **Autoconvert** check box when importing the underlay file.*

- 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.

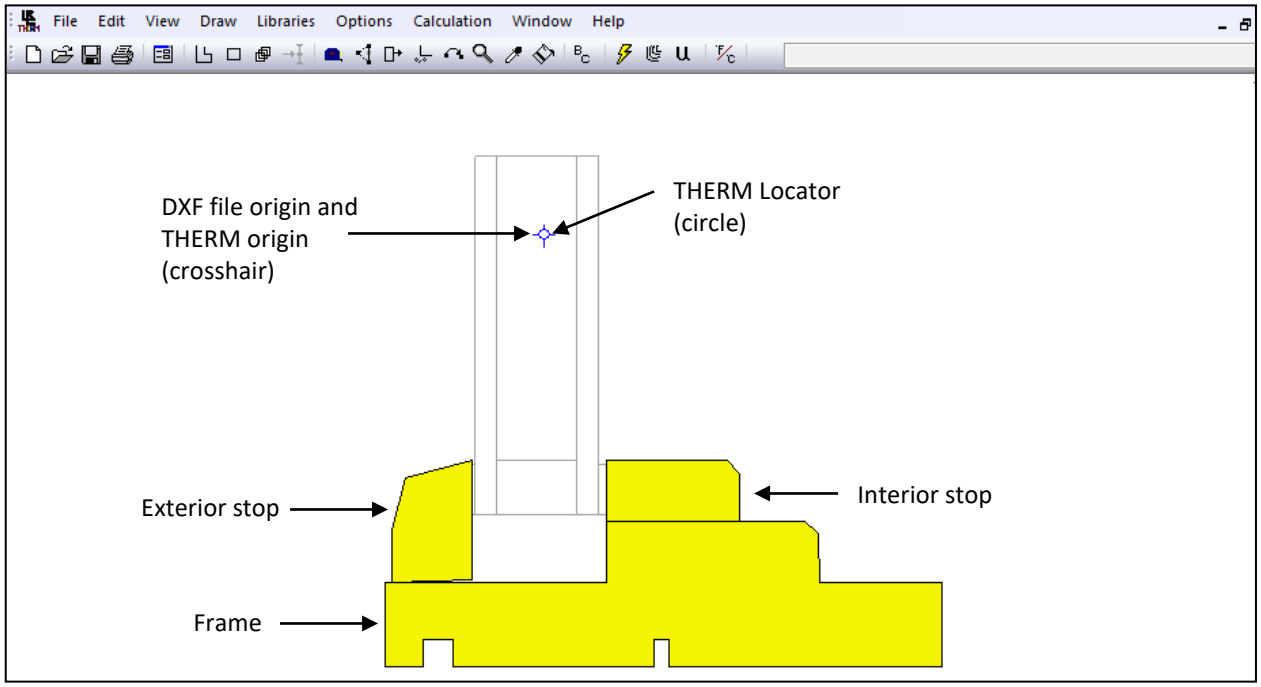


Figure 10-7. The components of the cross section

10.3.3.1. 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".

10.3.3.2. 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.

10.3.3.3. 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 **Hardwood (Oak, Maple)** 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 5.11, "Assigning Materials After Drawing Polygons" for more information about selecting materials).

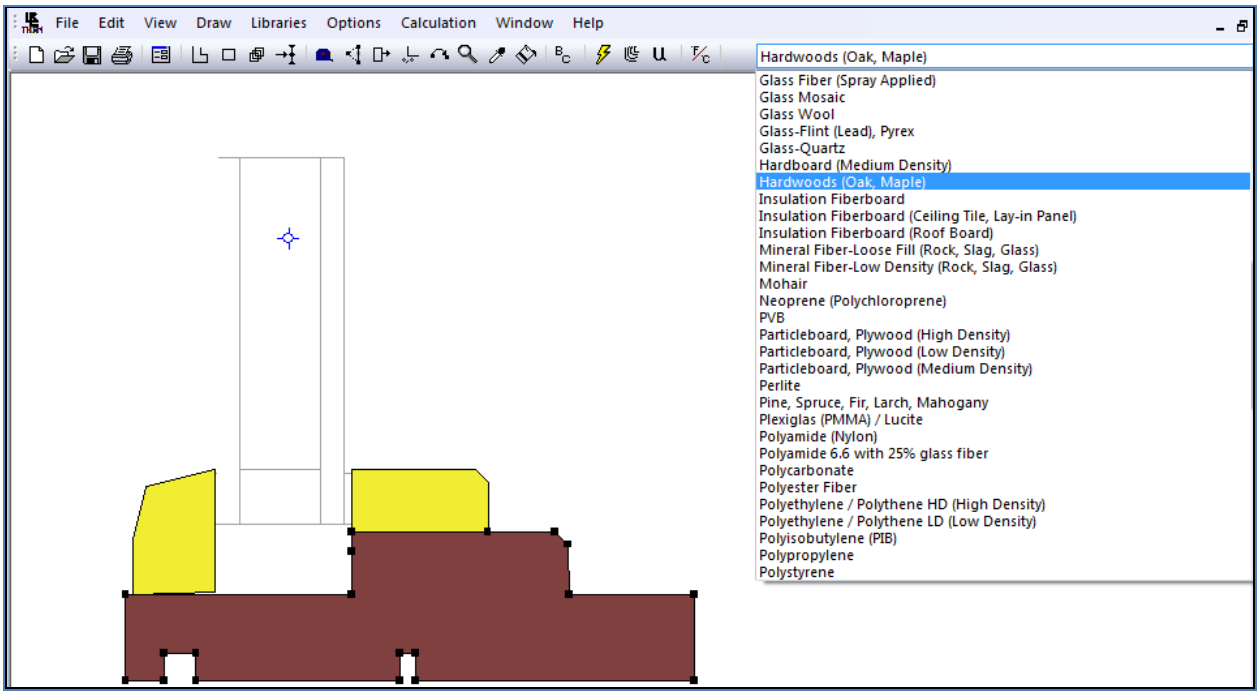


*Figure 10-8. Select the polygon representing the frame and assign it the **Wood (Pine, Spruce, Fir)** material.*

- 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 **Redwood, Cedar, Hemlock, Douglas Fir, White Fir, Cypress** from the **Materials** list, as shown in Figure 10-9.

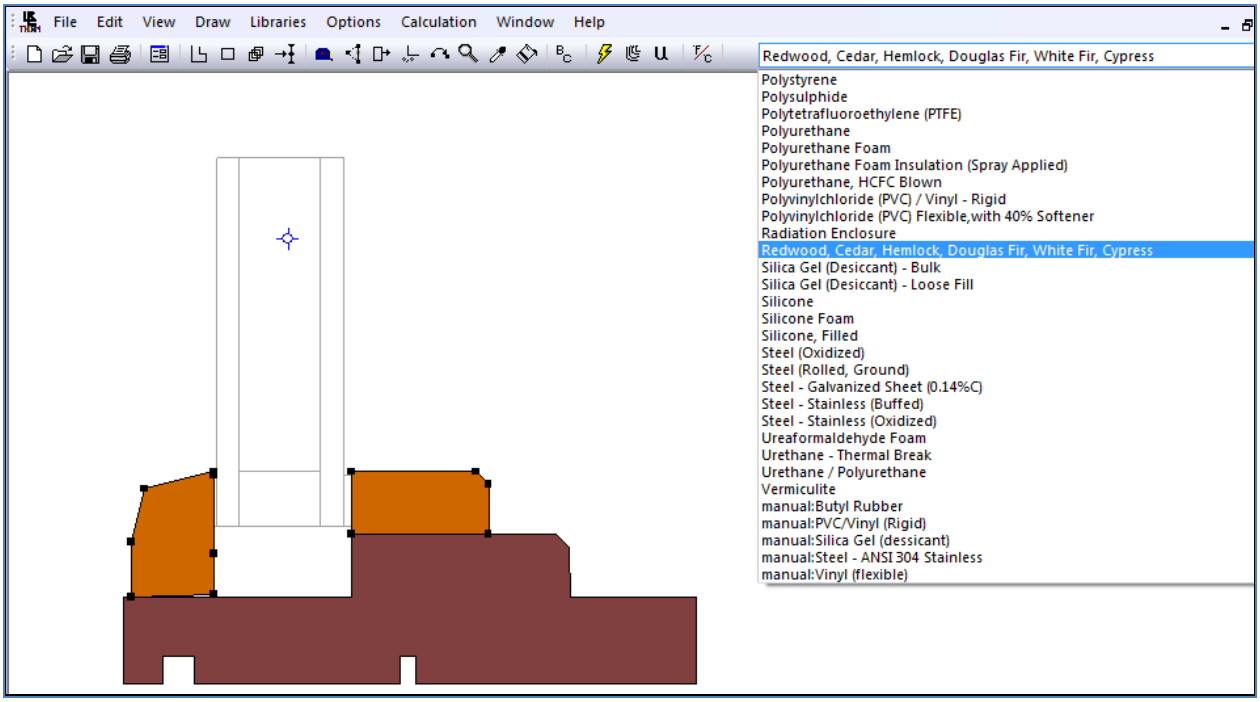


*Figure 10-9. Select multiple polygons by holding the **Shift** key down while clicking on multiple polygons.*

10.3.3.4. 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.

10.3.4. 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.

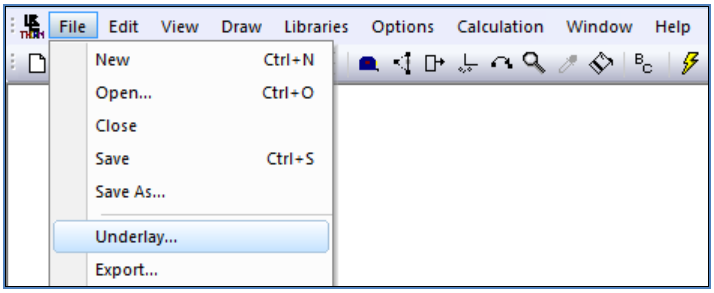


*Figure 10-10. To trace the DXF file, select the **Underlay** choice from the **File** menu.*

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

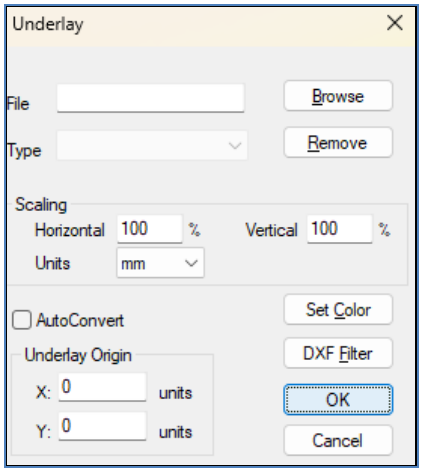


*Figure 10-11. The **Underlay** dialog box is used to select the DXF file to trace.*

- 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

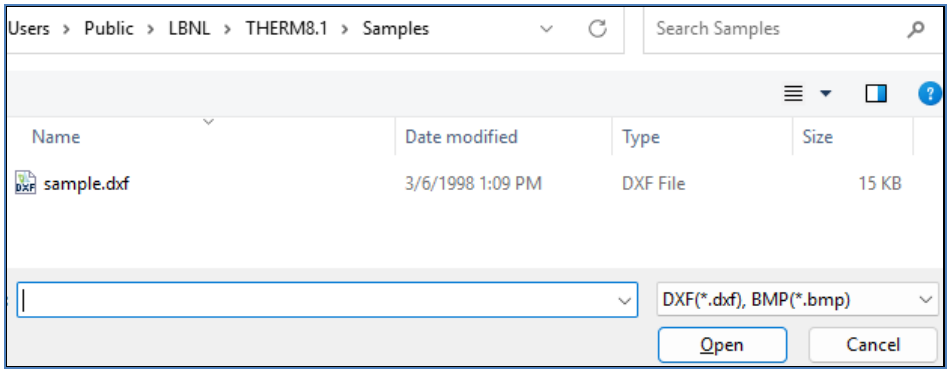


Figure 10-12. The Open dialog box shows all files with a DXF or BMP extension.

- 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.
- 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.

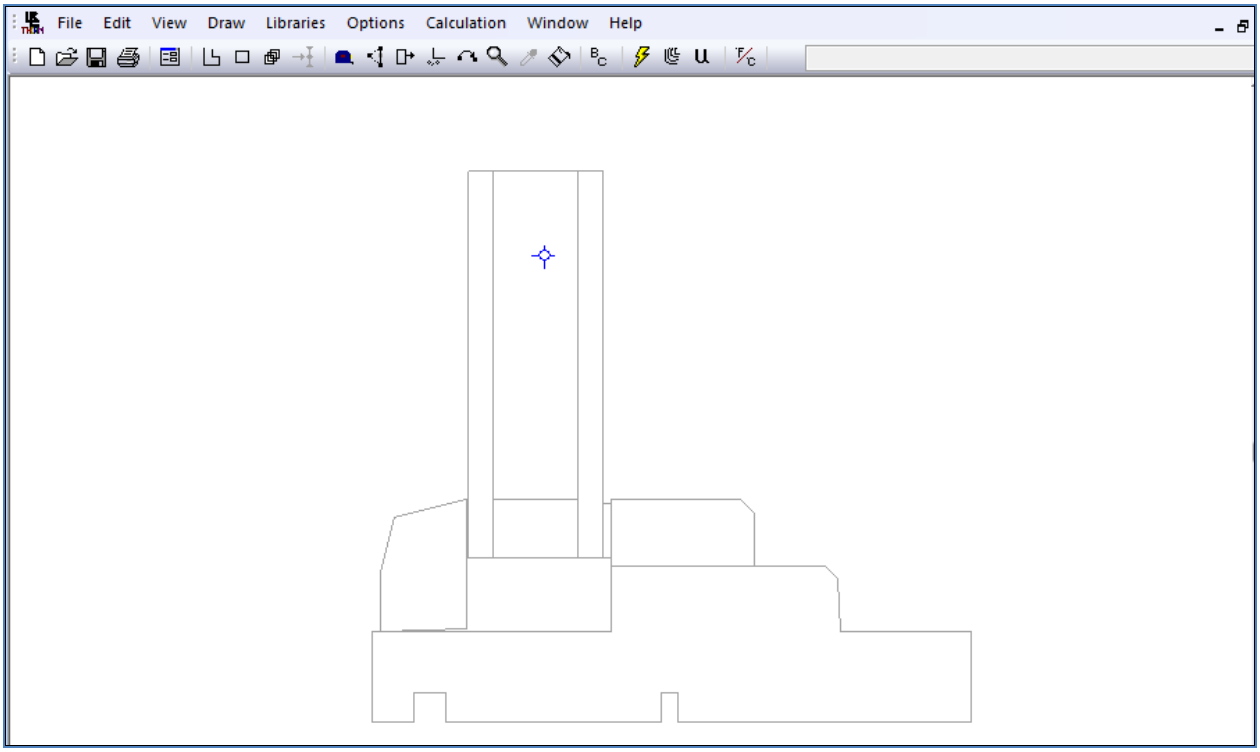


Figure 10-13. The DXF file has been imported as an underlay, which can now be traced to make the individual polygons.

- THERM attempts to fit the DXF cross section in the drawing area while still making the scale as big as possible. In Figure 10-13, 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.

10.3.4.1. 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.

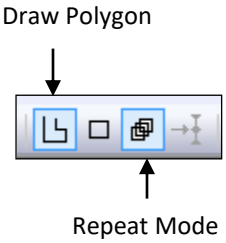


*Figure 10-14. Push the **Draw Polygon** and **Repeat Mode** toolbar buttons.*

- 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 **Redwood, Cedar, Hemlock, Douglas Fir, White Fir, Cypress** 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.

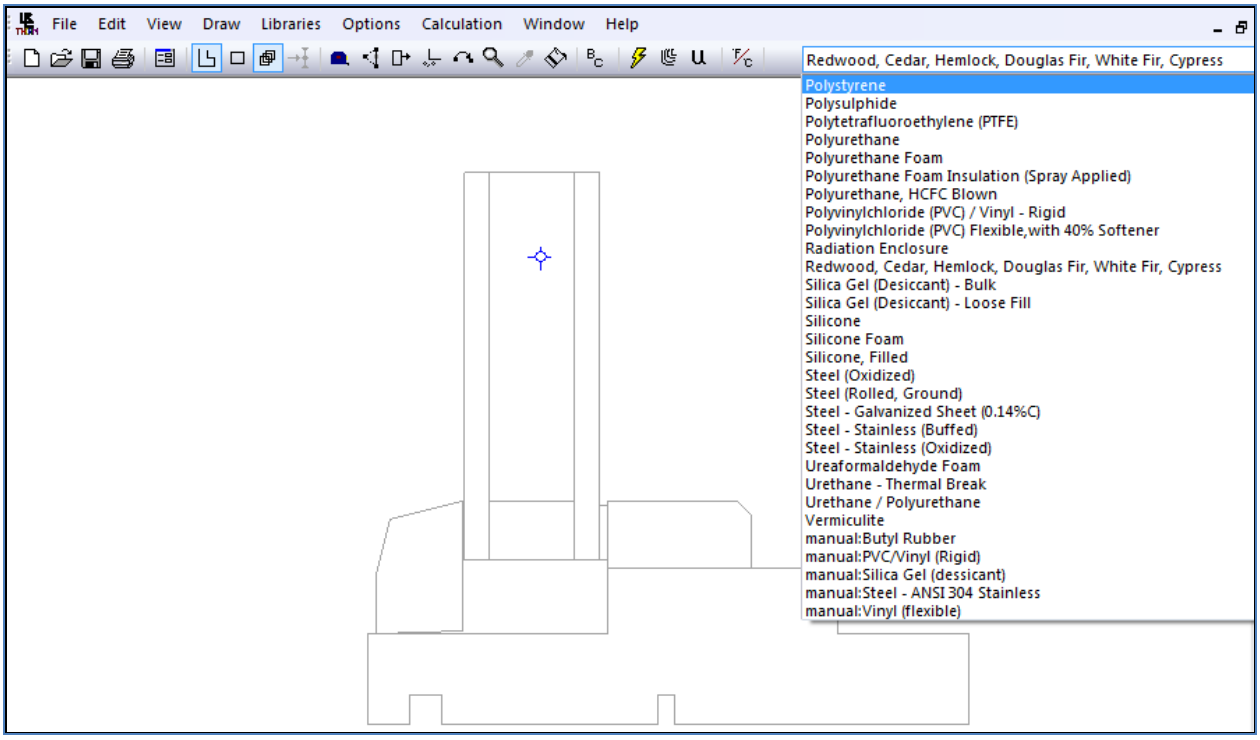


Figure 10-15. Select the appropriate material from the Materials pull-down list (on the right side of the toolbar) for the first polygon to be drawn.

- 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 10-16.
- 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 10-16. 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.

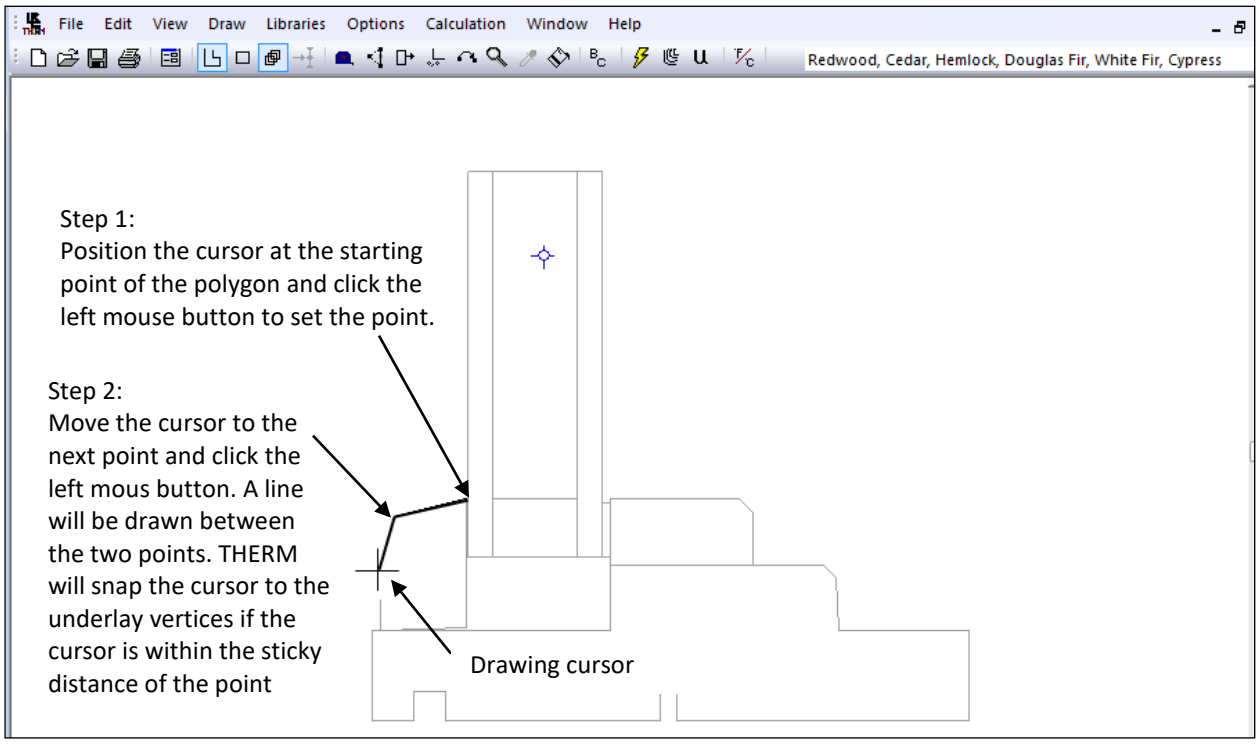


Figure 10-16. Start drawing the first polygon.

- 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 10-17, and fill it in with the color corresponding to the material definition (or to a shade of gray on monochrome monitors).

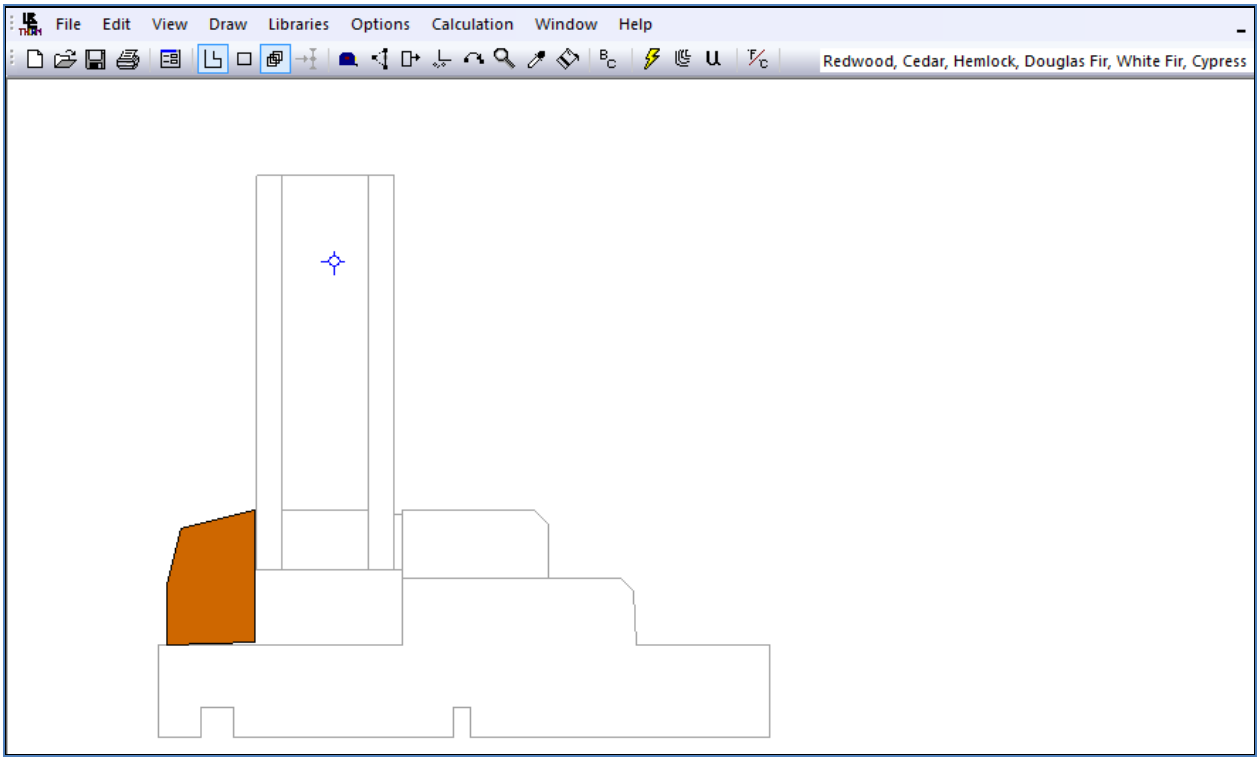


Figure 10-17. The first polygon, representing the exterior stop, has been drawn.

- 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 10-18.
- 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.

10.3.4.2. 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, **Hardwoods (Oak, Maple)** from the **Materials** pull-down list on the right-hand side of the toolbar.

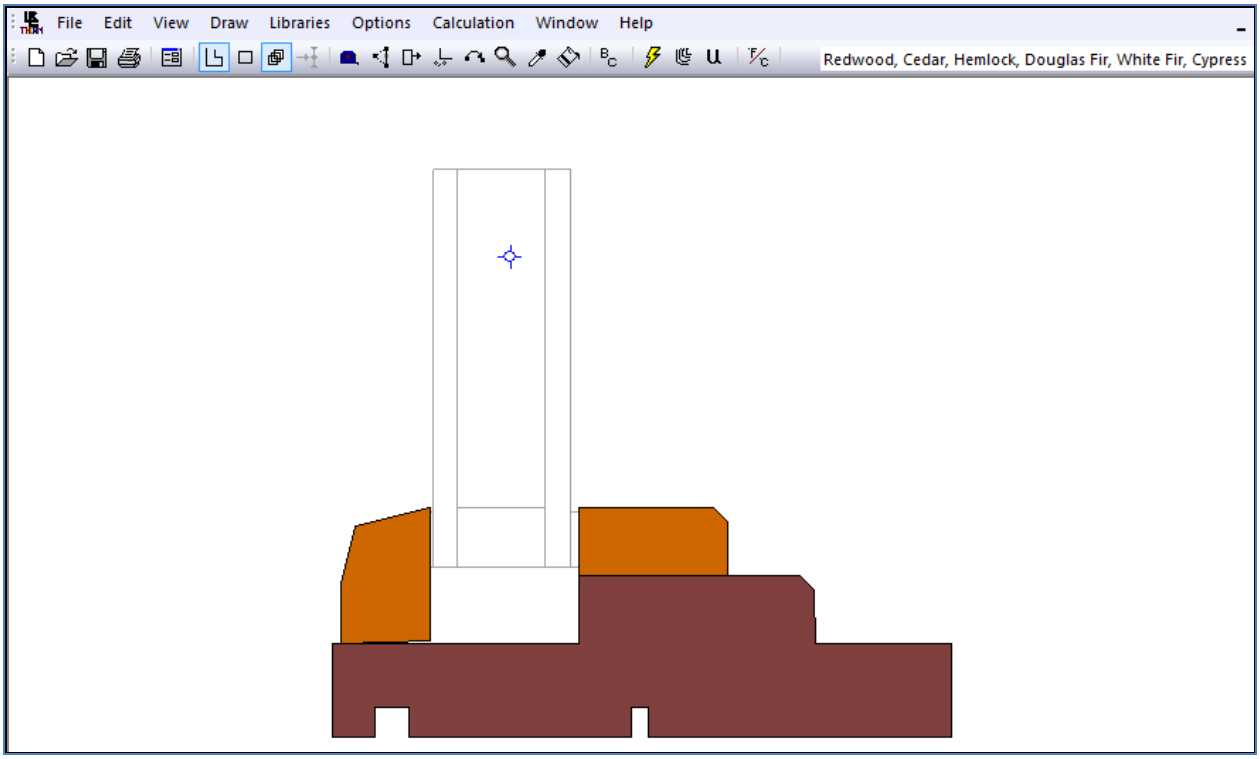


Figure 10-18. The wood components have been defined.

- 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).

10.3.4.3. 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".

10.3.5. 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.

10.3.5.1. 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.*

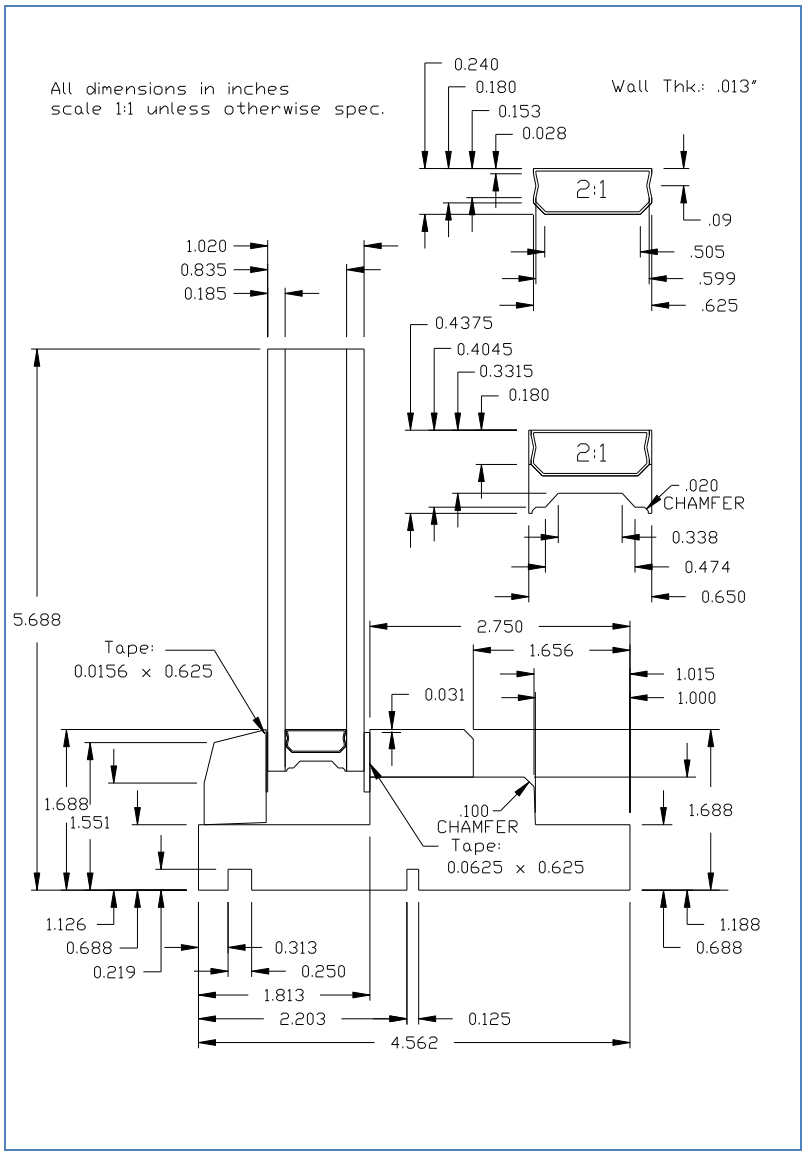


Figure 10-19. Dimensioned drawing of cross section

10.3.5.2. 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

10.3.5.3. 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.)

10.3.5.4. 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.

10.3.5.5. Drawing the Dimensioned Polygons

- 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.*

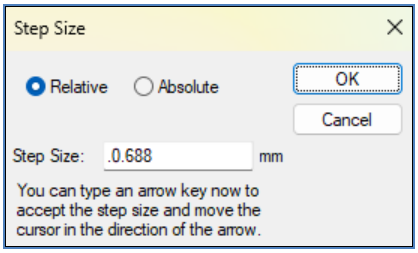


Figure 10-20. The Step Size dialog box appears any time you type a numeric value. You can click on the OK button to set the step size, or just use the arrow keys while the dialog box is open to move the step size distance.

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

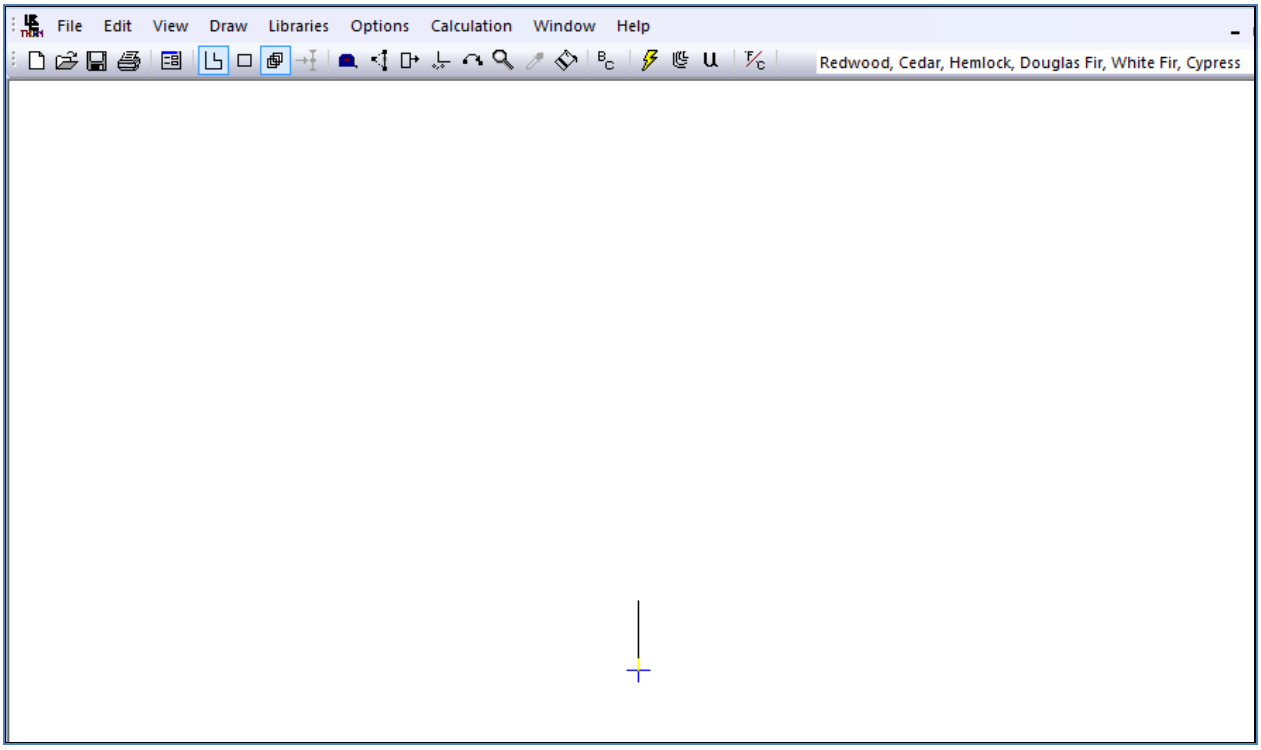


*Figure 10-21. To draw a line using the keyboard: from the origin, type in the step size, move the **up arrow key**, and press **Enter**.*

- 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 10-19. 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 10.3.5.8.
- 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.1 1.813

- 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:

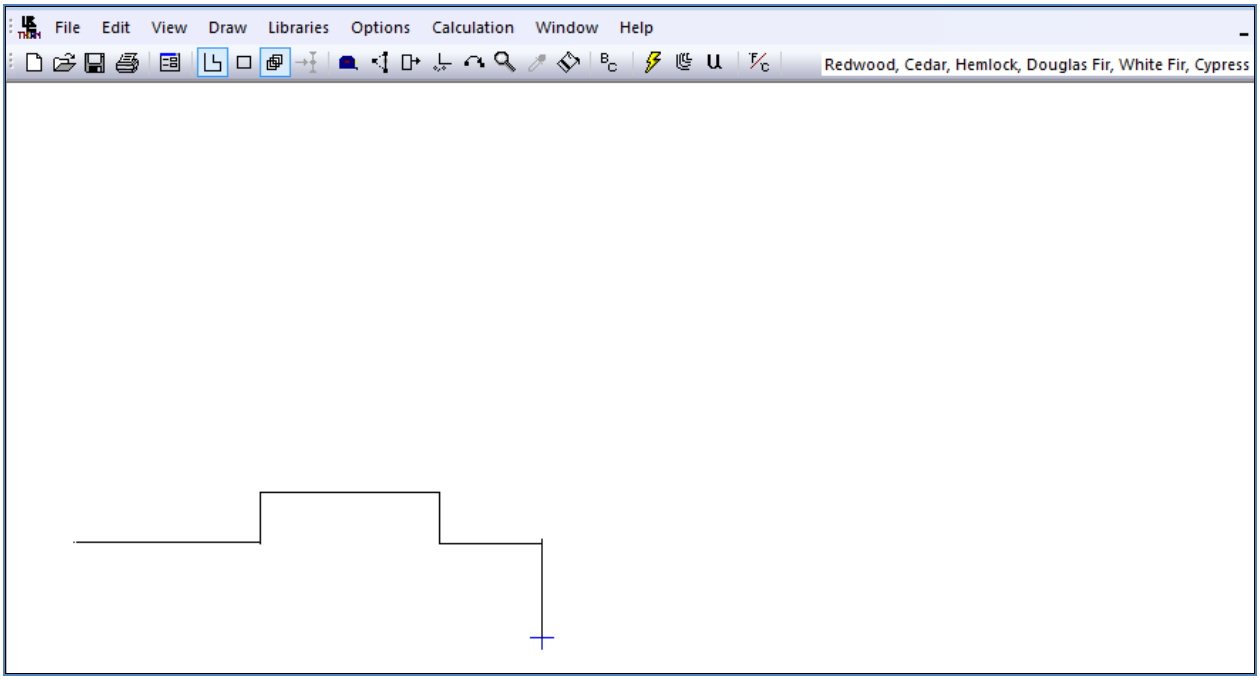


Figure 10-22. Using the keyboard, you can type the polygon dimensions and use the arrow keys to draw the lines.

10.3.5.6. 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 10-23.

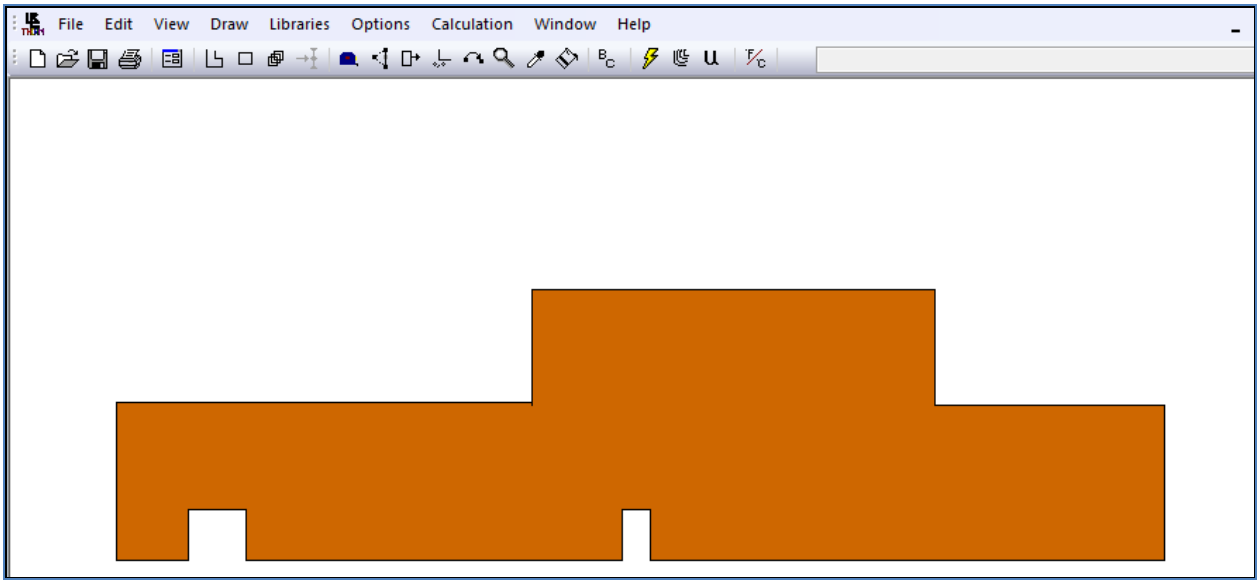


Figure 10-23. The completed polygon that represents the frame

10.3.5.7. 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 10-24 for deleting additional points or the entire polygon.

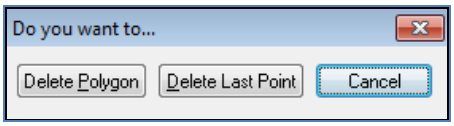


Figure 10-24. This dialog box allows you to delete the last point in a polygon. You can use your mouse or the arrow keys to move to each button in the box.

10.3.5.8. 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 10-25 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.

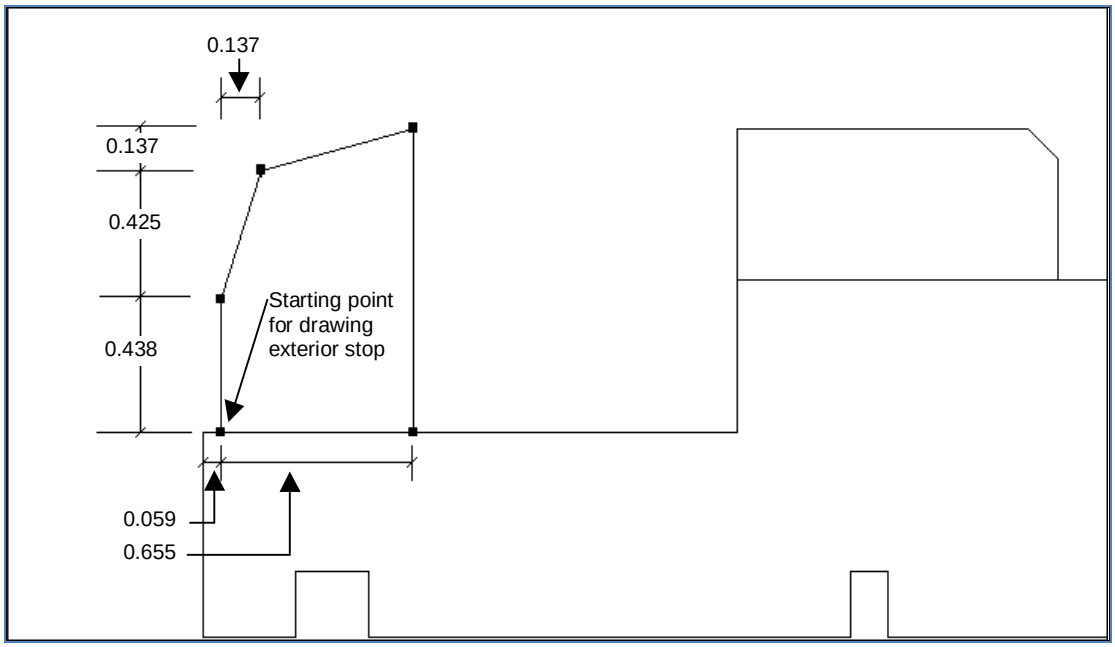


Figure 10-25. Dimensions for the exterior stop.

- 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.

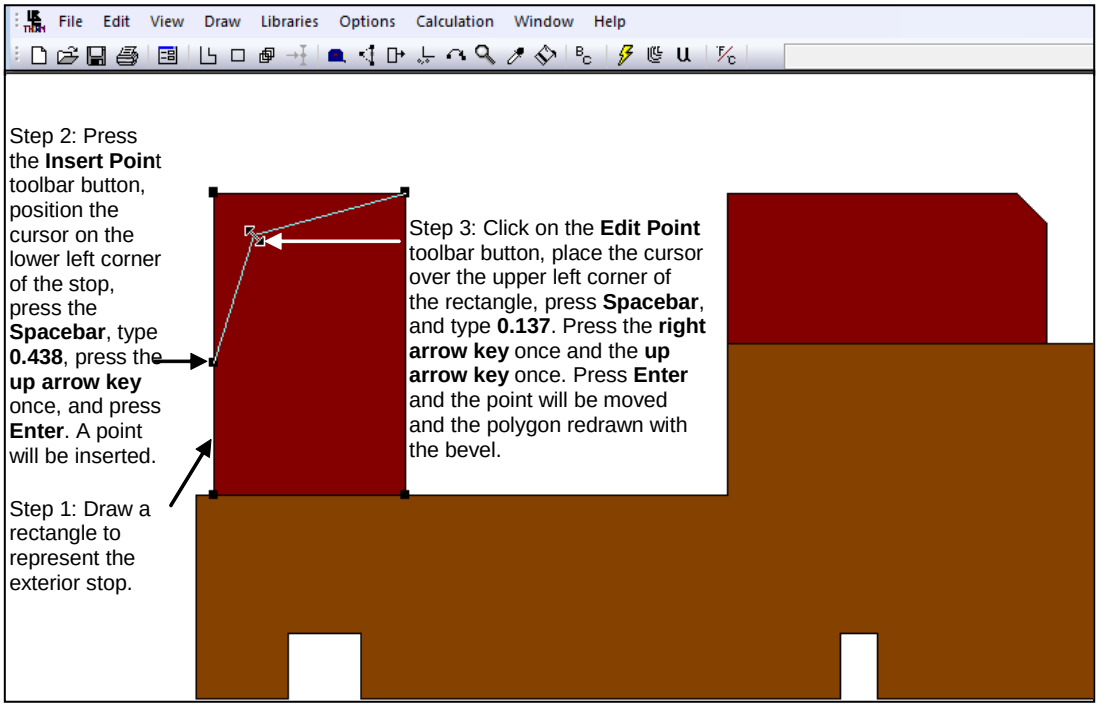


Figure 10-26. Adding the beveled edge to the exterior stop

10.3.5.9. 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".

10.4. Import a Glazing System from WINDOW

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.

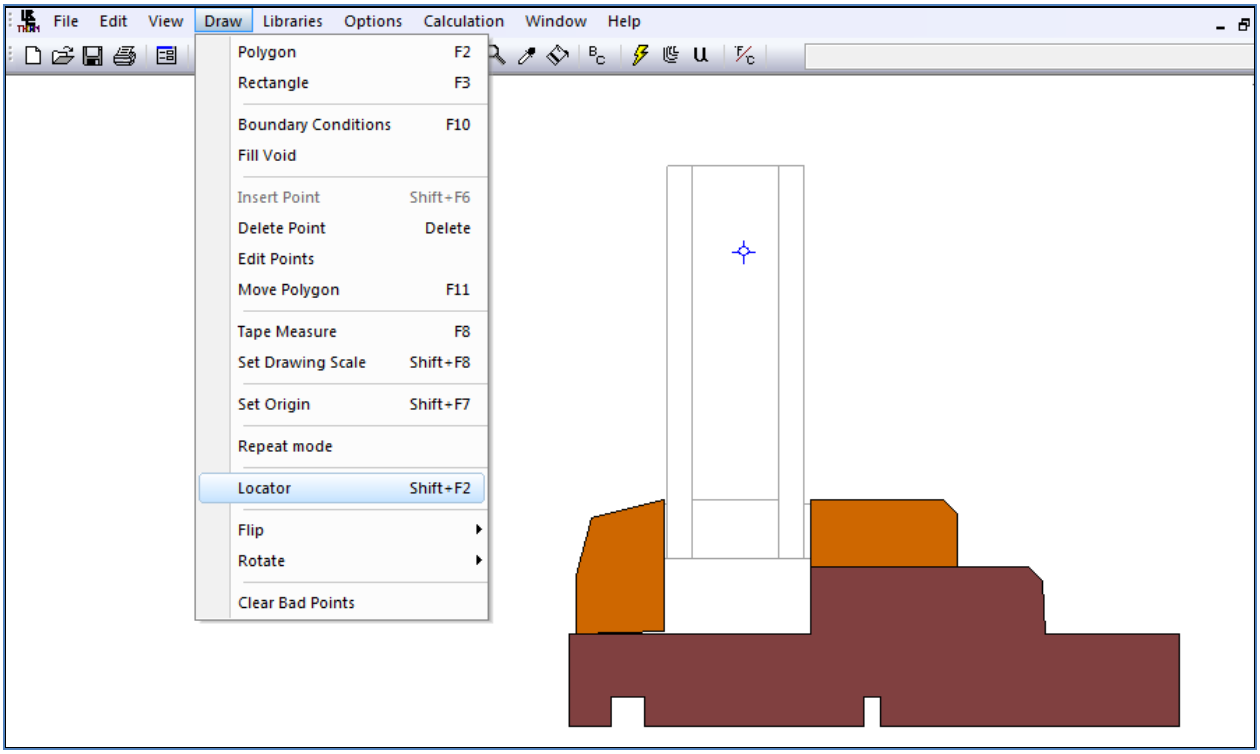


Figure 10-27. Select the Locator choice from the Draw menu.

- 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 10-28. (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**.)

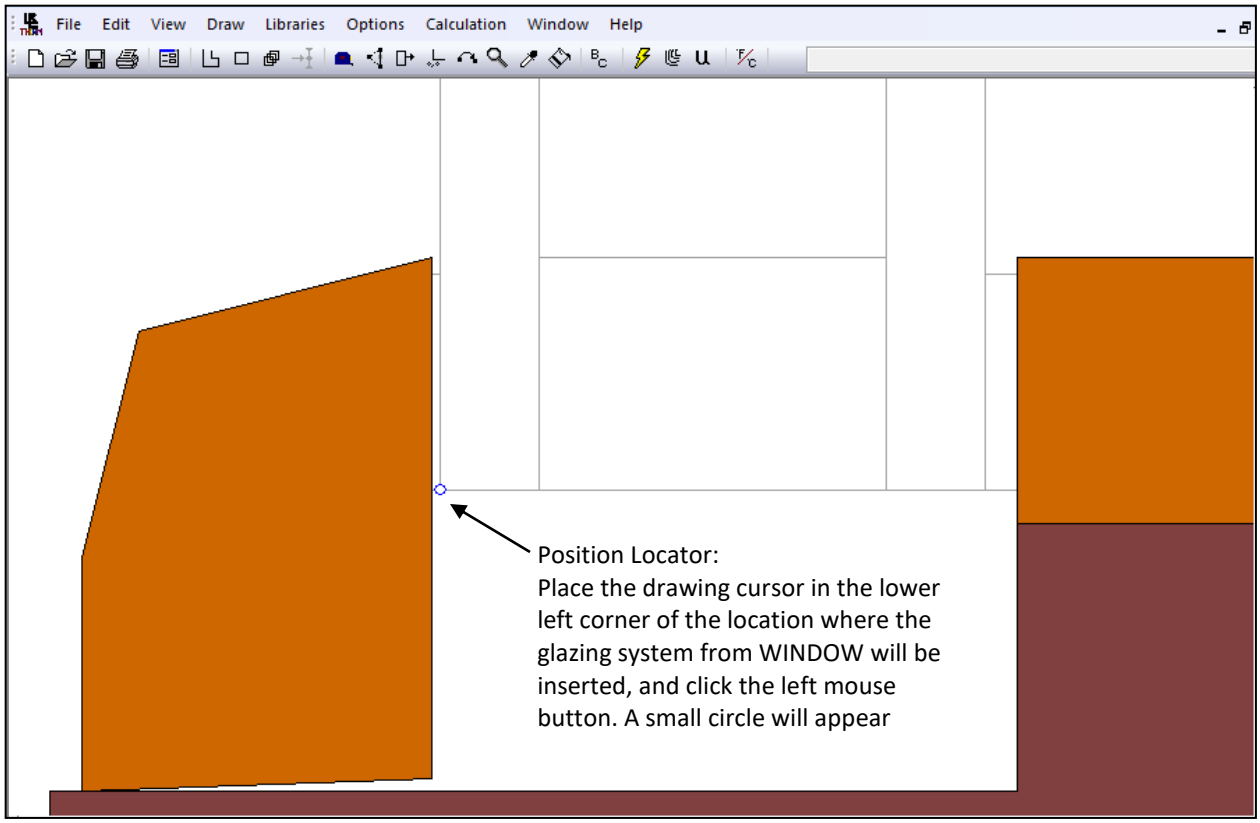


Figure 10-28. Position the locator where you want the lower left corner of the glazing system to be placed.

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

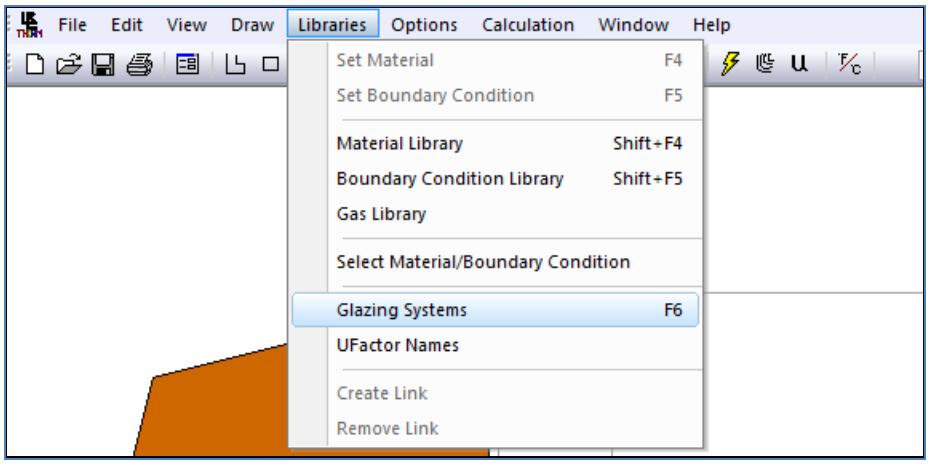


Figure 10-29. Select the Glazing Systems choice from the Libraries menu.

- The **Glazing Systems** dialog box will appear with a pull-down list showing all the choices from the WINDOW database shown next to the **Browse** button. You will need to have downloaded a compatible version of WINDOW in order to import a glazing system from a WINDOW database.

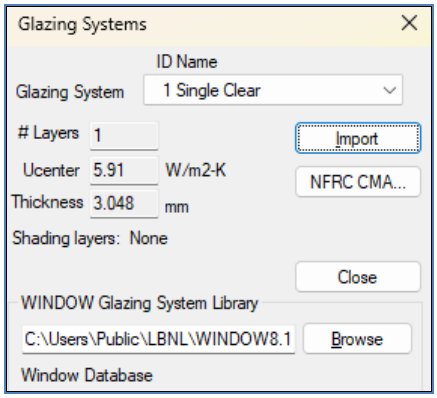


Figure 10-30. The Glazing Systems dialog box displays the information about each glazing system choice.

- 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 "Sample GlzSys" choice.

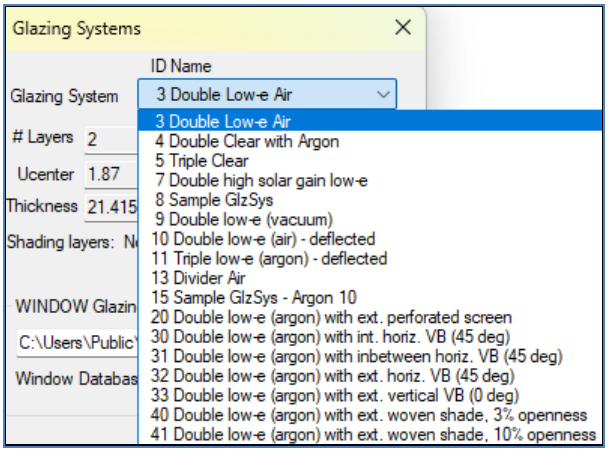


Figure 10-31. Select the THERM SAMPLE glazing system to be inserted in the tutorial example.

- 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).*

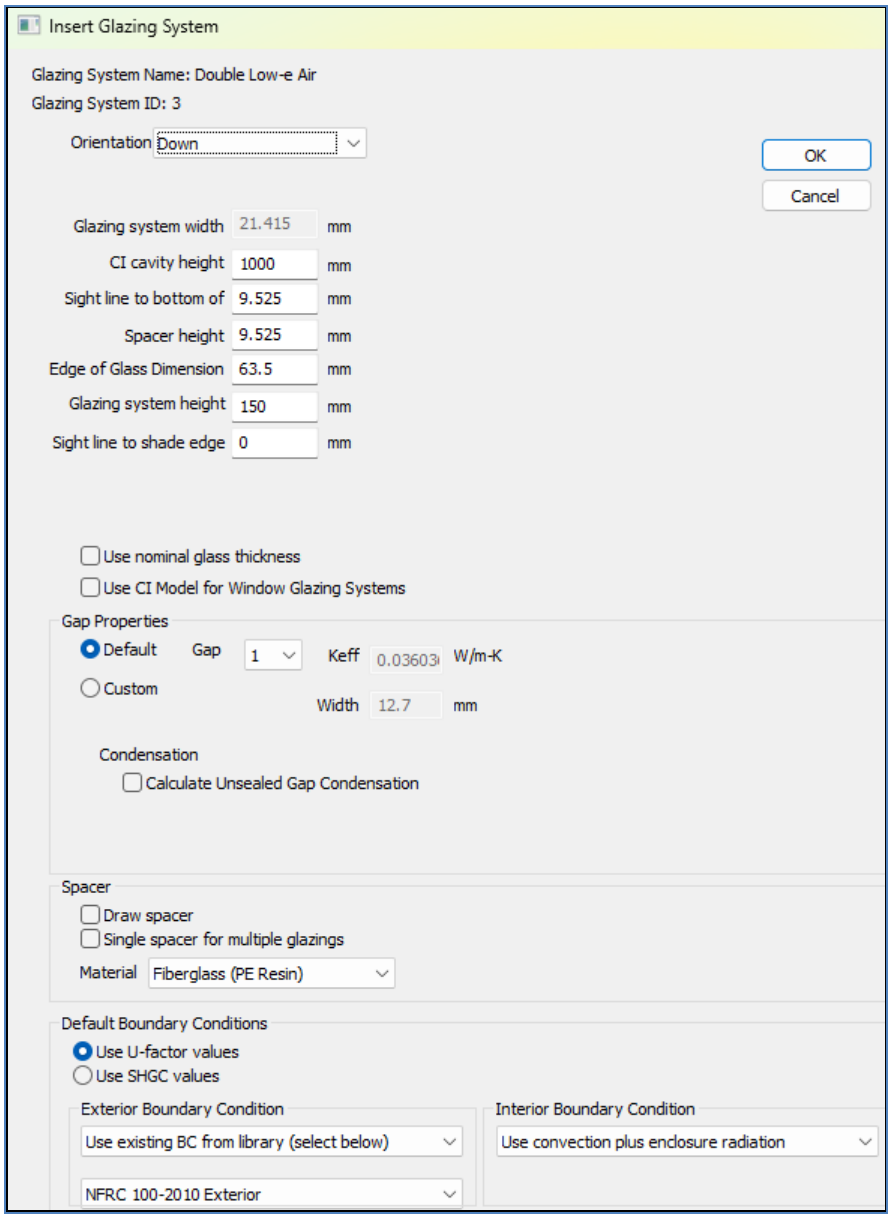


*Figure 10-32. The **Insert Glazing System** dialog box appears when you are importing a glazing system from WINDOW.*

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

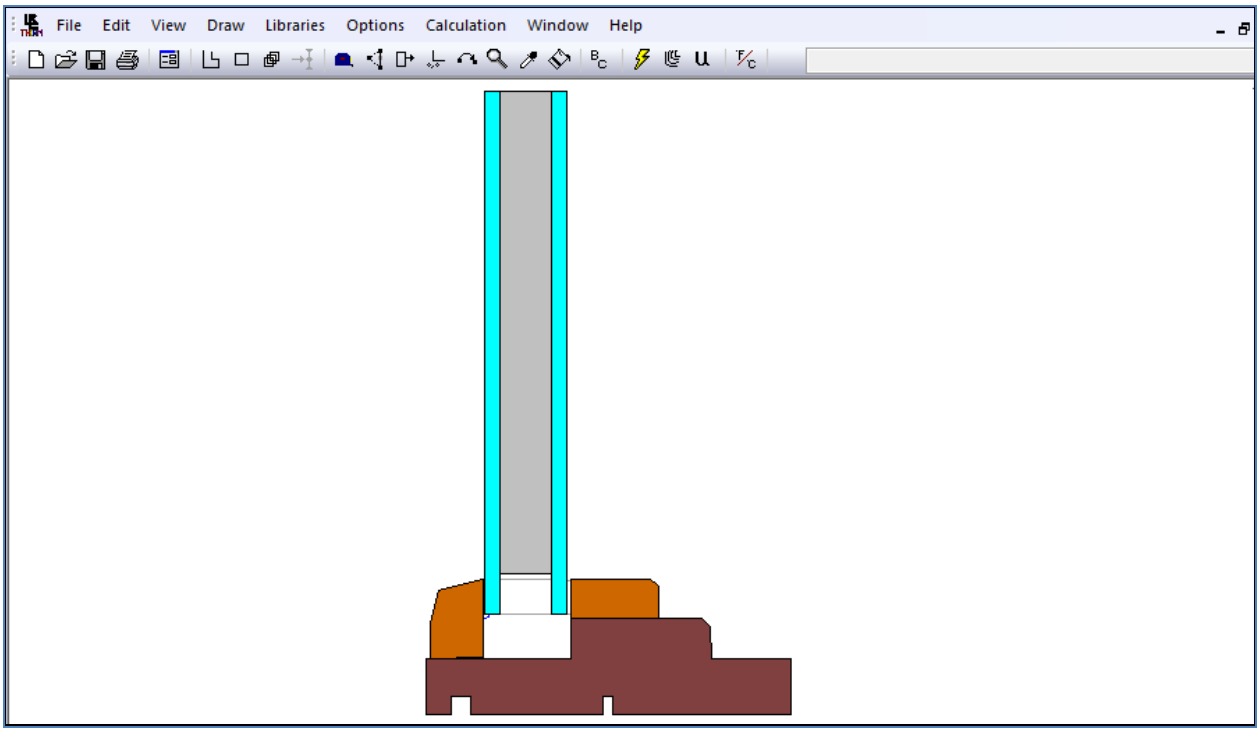


Figure 10-33. The glazing system from WINDOW has been inserted into the frame components.

- 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 (PIB)** 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.

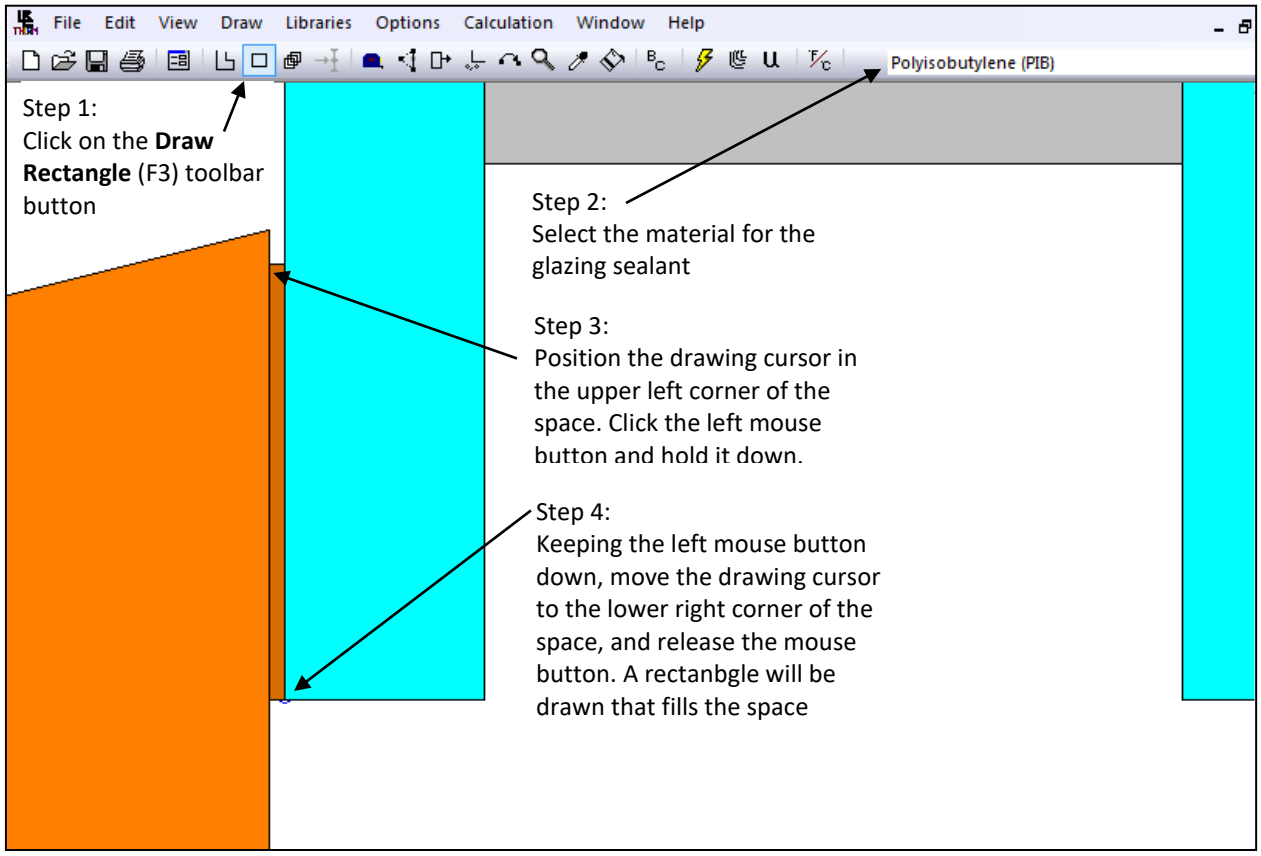


Figure 10-34. Draw a rectangle that fills the space between the exterior stop and the glazing system.

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

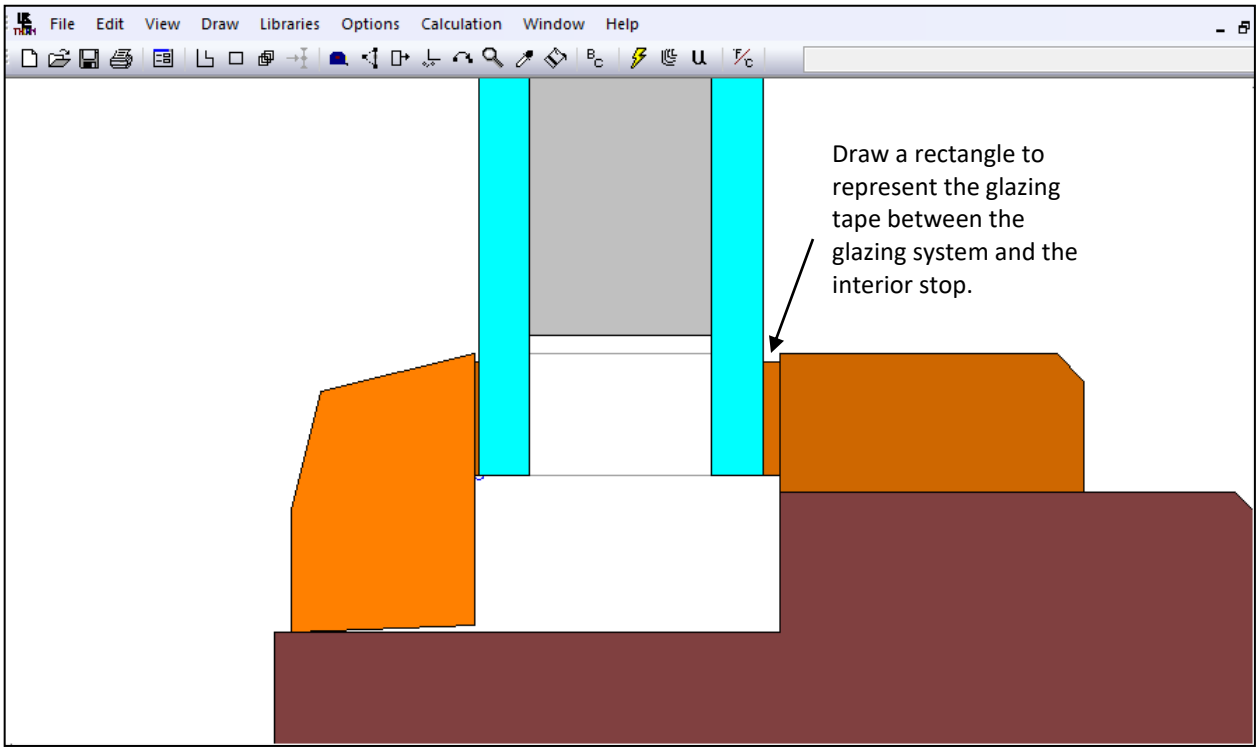


Figure 10-35. Draw a rectangle that fills the space between the interior stop and the glazing system.

- 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.

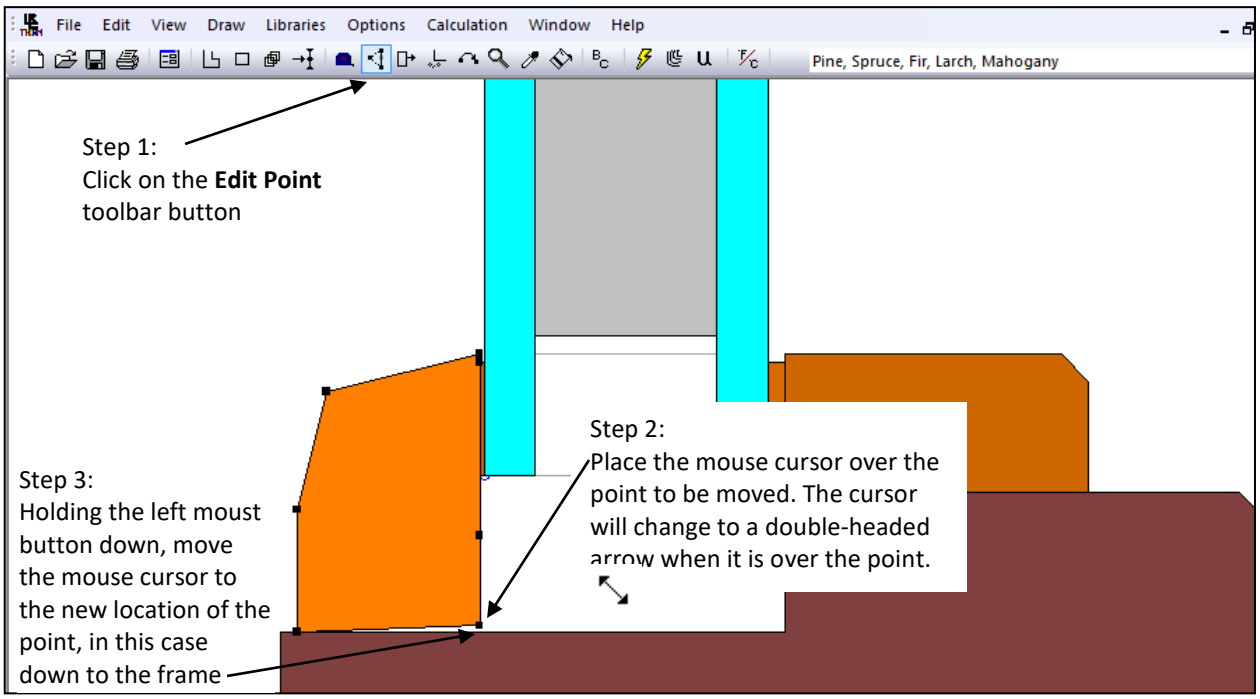


*Figure 10-36. With the **Edit Point** toolbar button activated, place the mouse cursor over the point to be moved, hold the left mouse button down, and move the mouse to the new point location.*

- The point and attached lines are moved, and the polygon is redrawn, as shown in Figure 10-37.

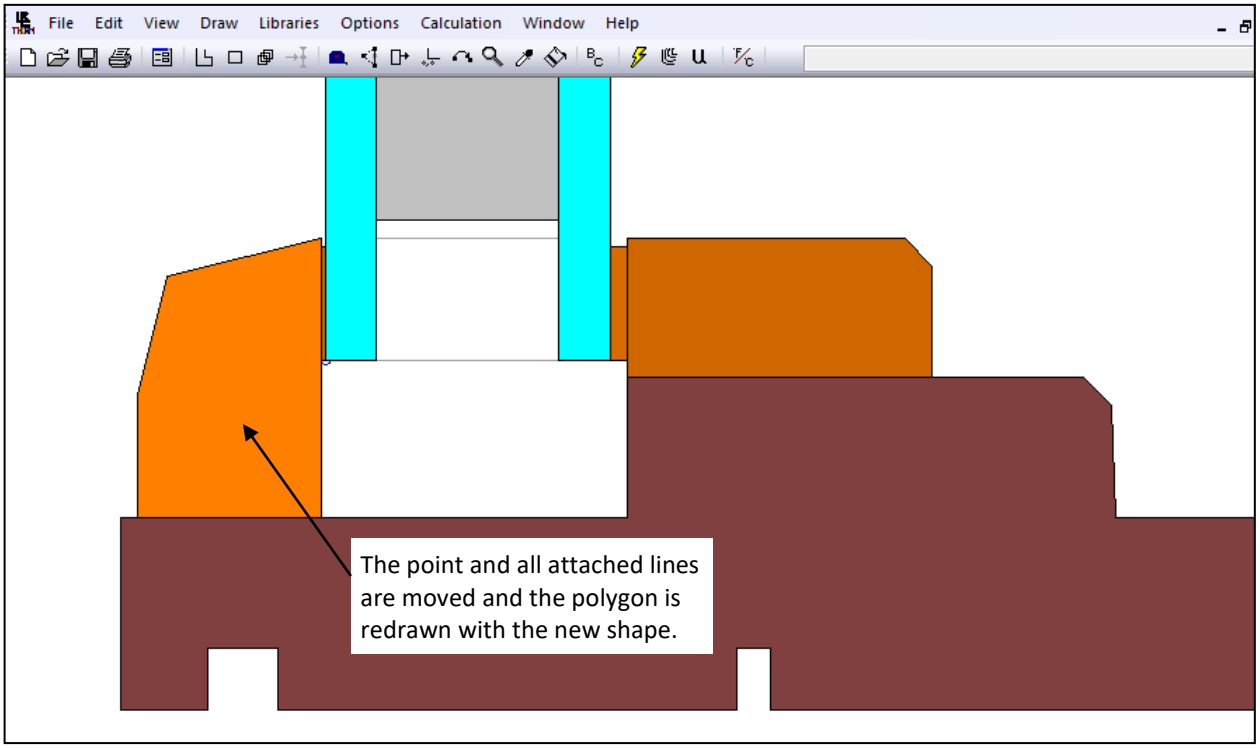


Figure 10-37. THERM will move the point and the lines attached to that point and redraw the cross section with the new geometry.

- 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.

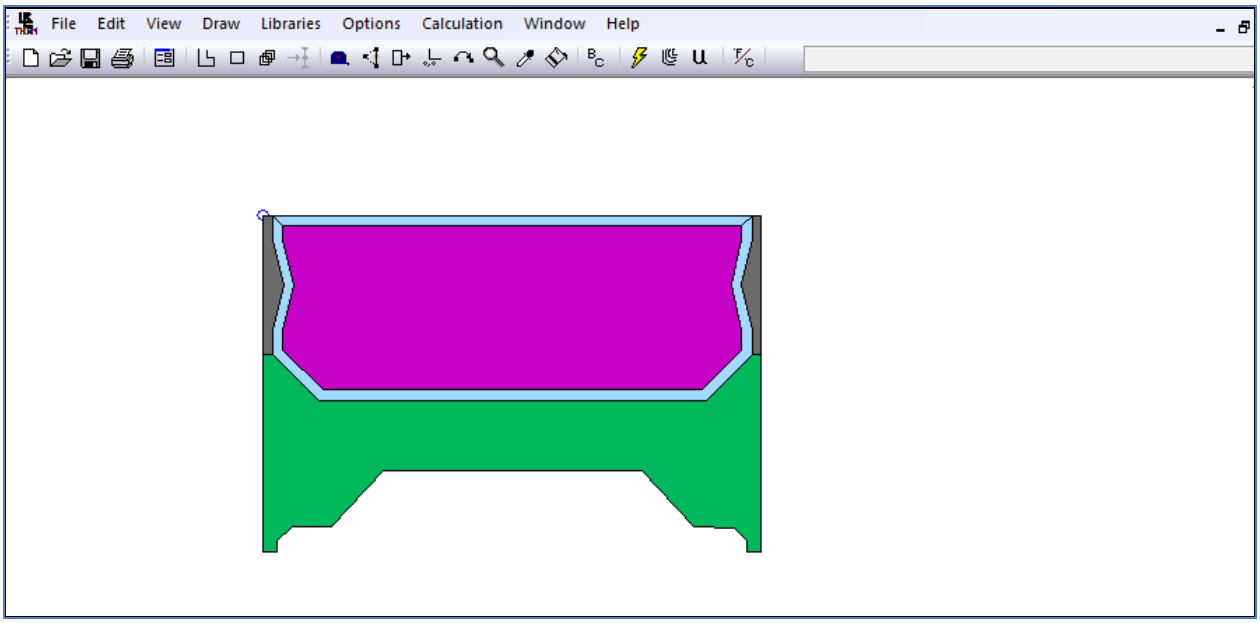


*Figure 10-38. Use the **Draw/Locator** menu choice to position the **Locator** in the upper left corner of the spacer.*

- 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 10-39.

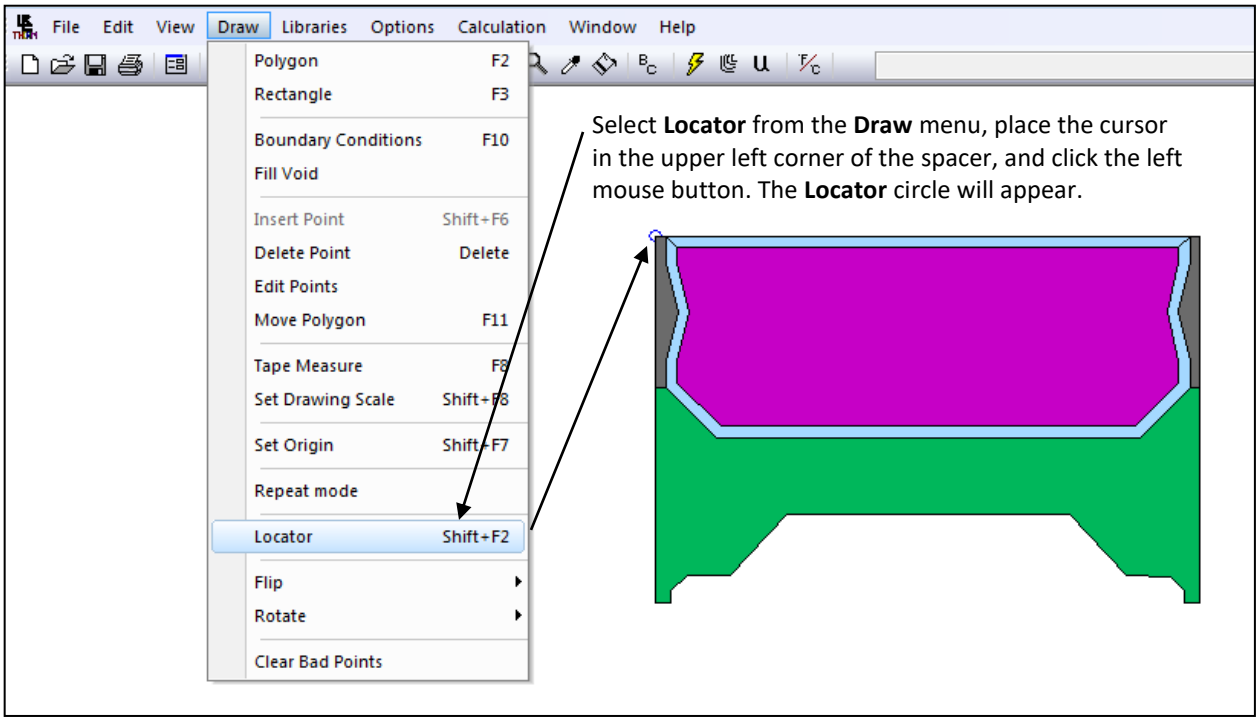


*Figure 10-39. Position the **Locator** in the upper left corner of the spacer.*

- 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.

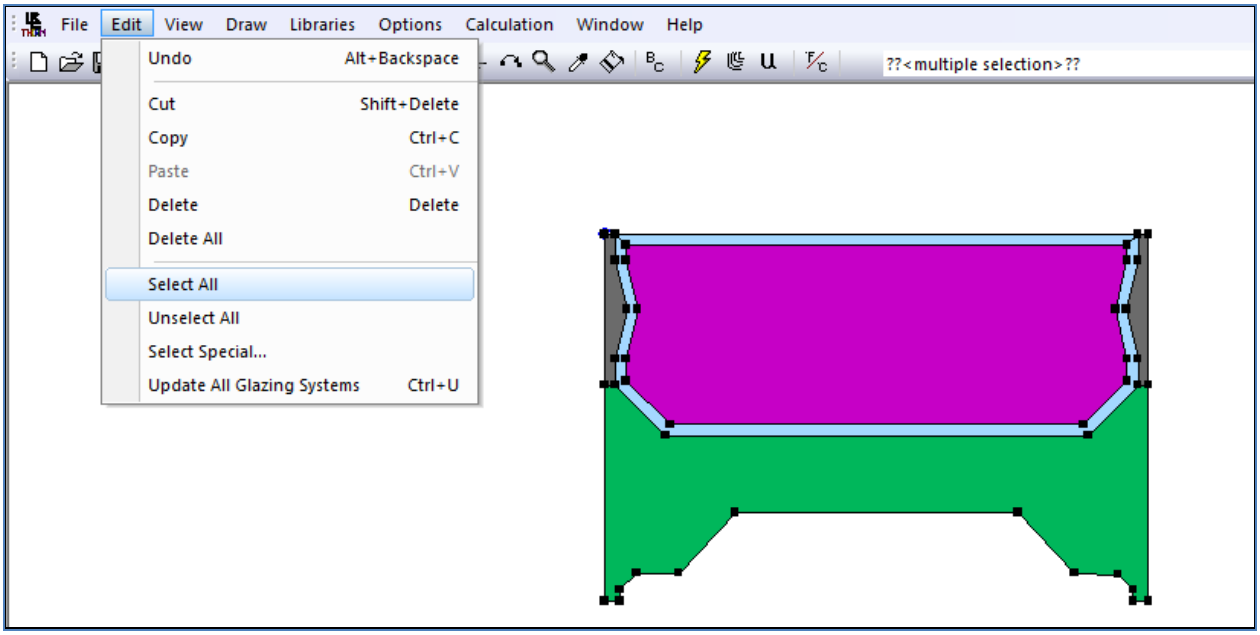


*Figure 10-40. Use the **Edit/Select All** menu choice to select all the polygons in the spacer; then, use **Edit/Copy** (or **Ctrl-C**) to copy these polygons into memory.*

- 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.

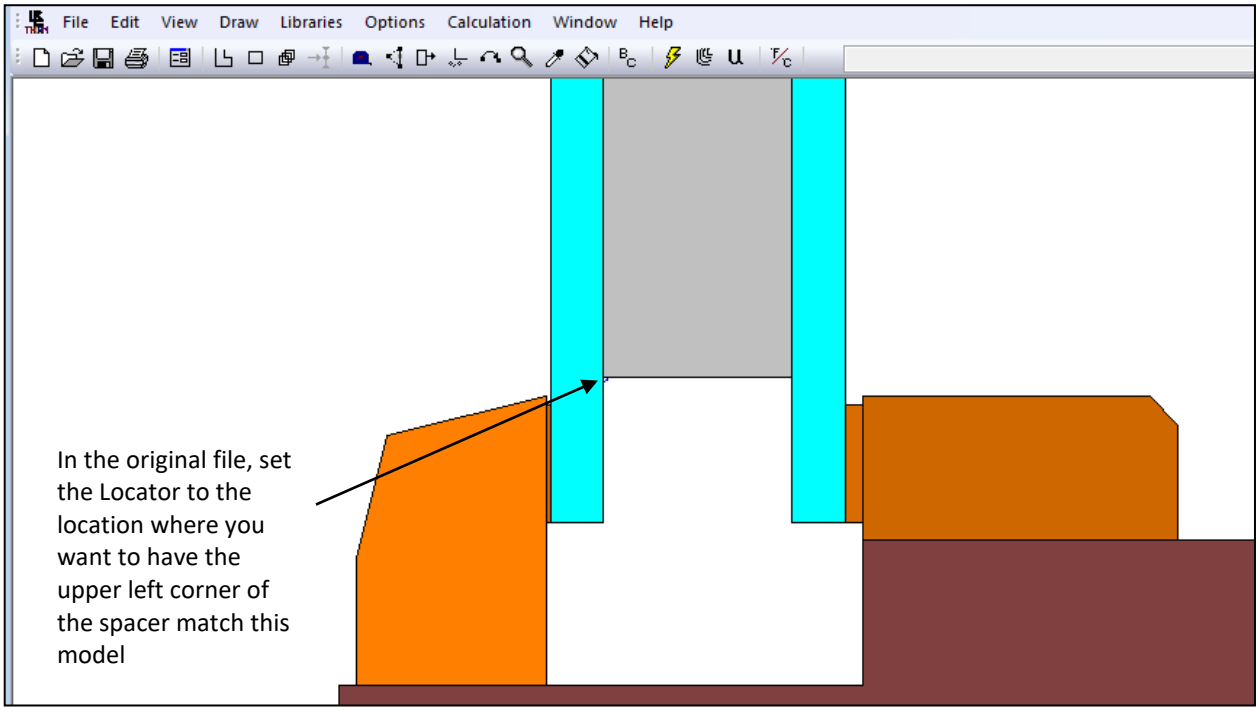


Figure 10-41. Set the locator to the place where you want the upper left corner of the spacer to be matched.

- Use the **Draw/Paste menu**, or **Ctrl-V**, to paste the spacer into the cross section. The spacer will be placed in the cross section by matching up the positions of the two Locators.

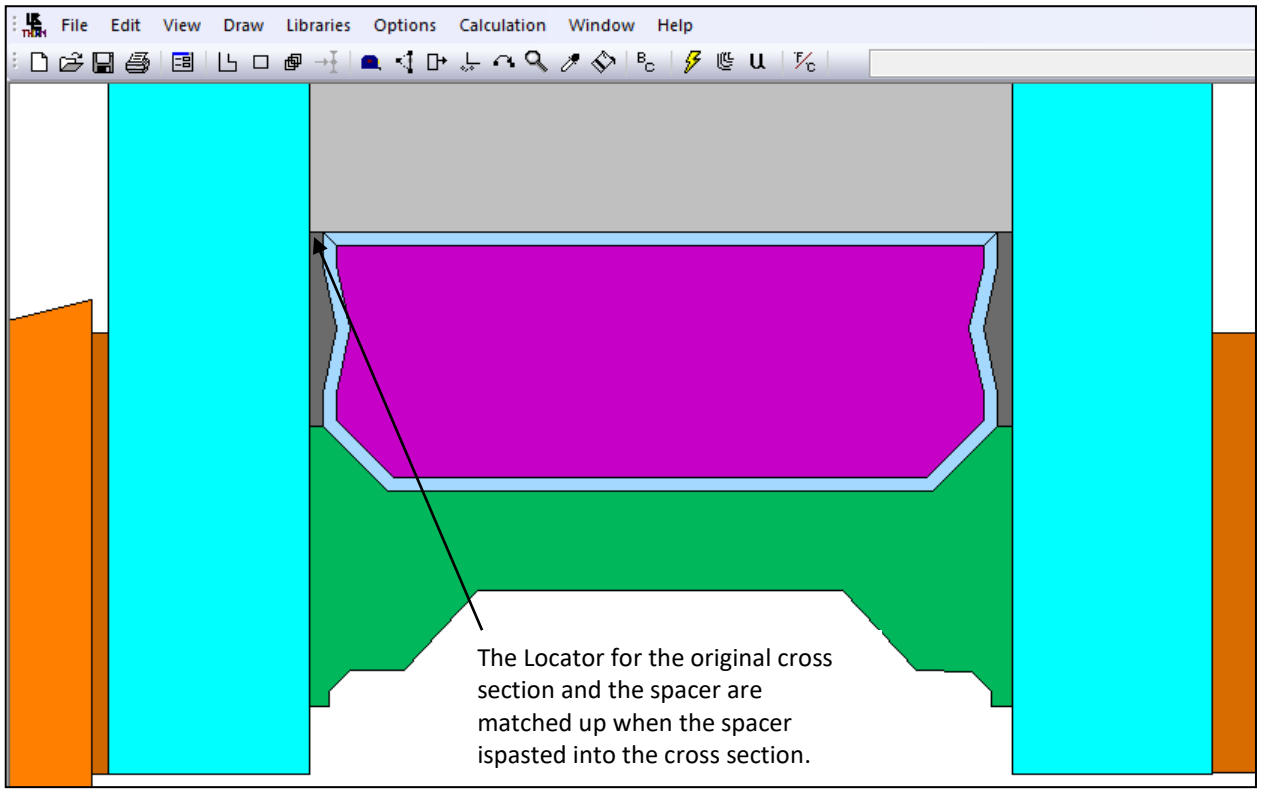


*Figure 10-42. Use the **Edit/Paste** menu choice to paste the spacer into the cross section.*

- 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**.

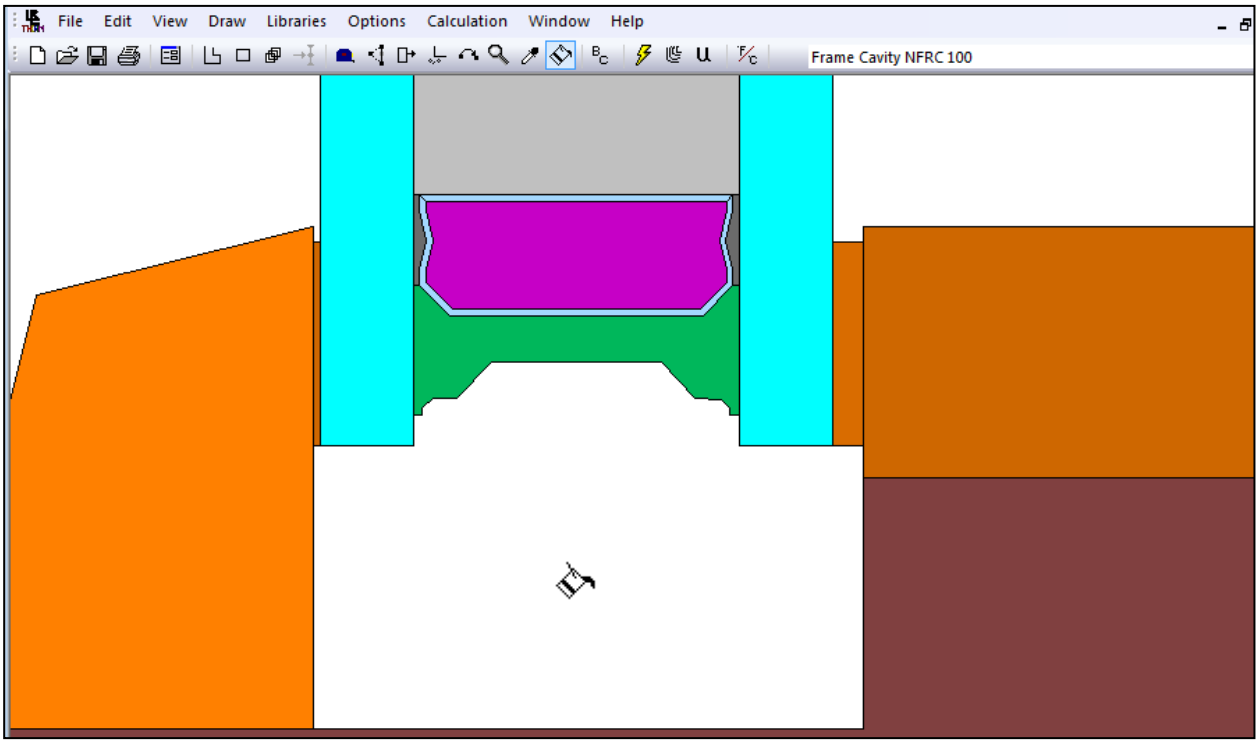


*Figure 10-43. Use the **Fill** tool to fill in the cavities in the cross section.*

- 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.

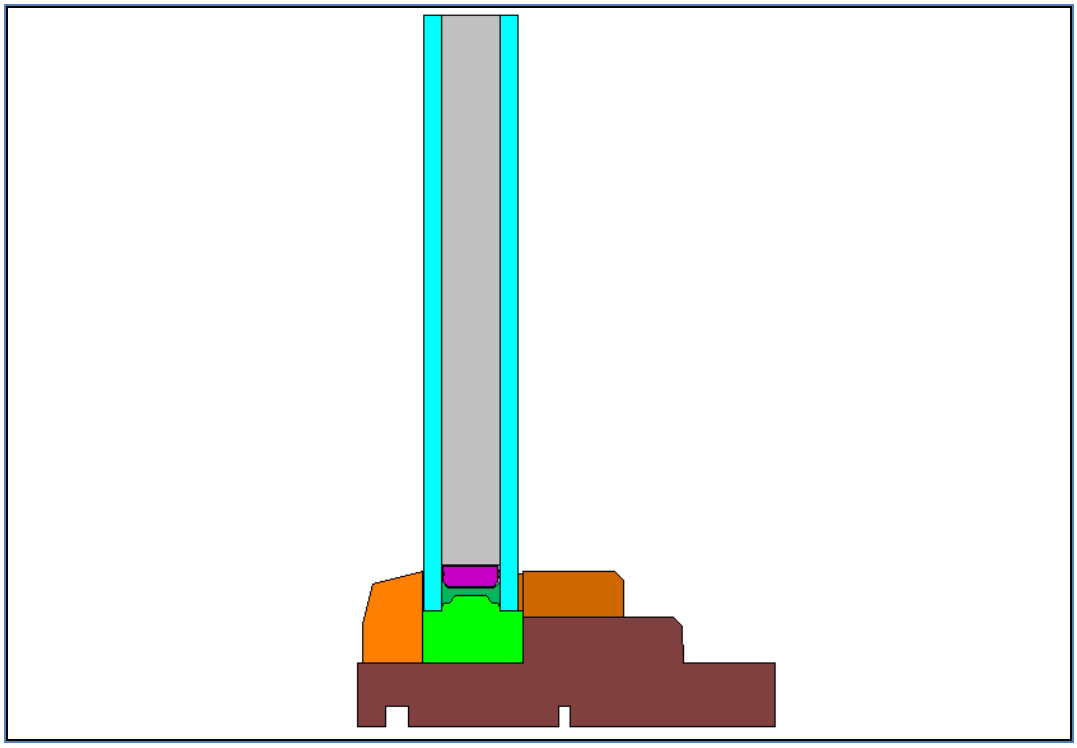


Figure 10-44. The finished cross section.

10.5. 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.

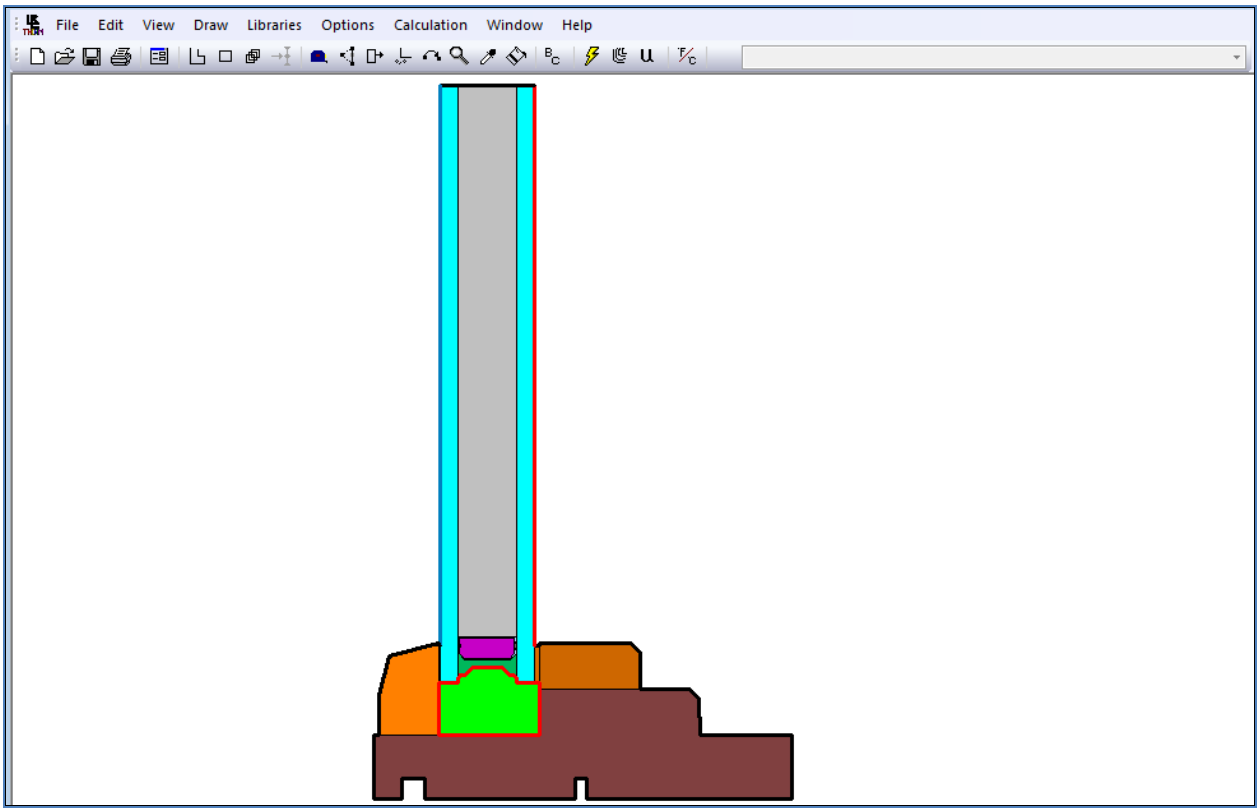


Figure 10-45. When you press the Boundary Conditions toolbar button, THERM draws an exterior boundary around all the cross-section components.

- 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 10-46.
- 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 10-46. Press the OK button.

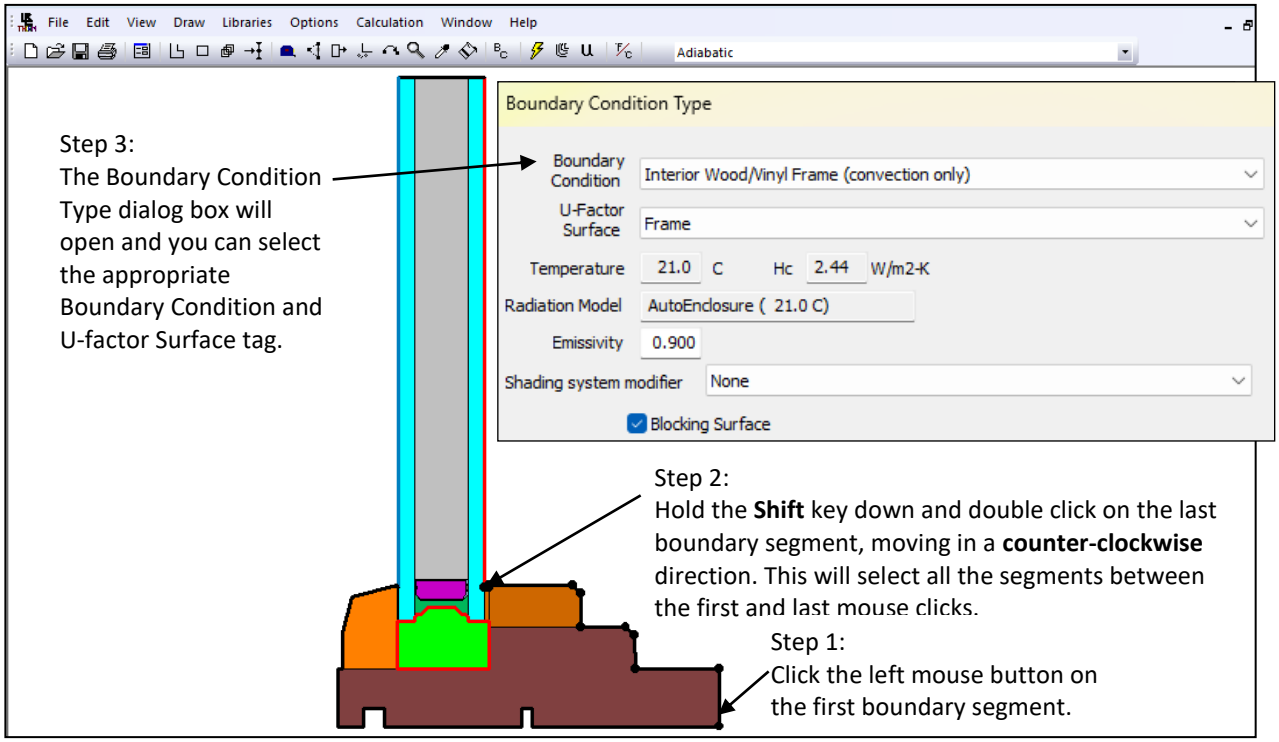


Figure 10-46. Select multiple boundary segments for the interior side of the cross section, and define their boundary conditions.

- Use the same technique to define the boundary conditions for the exterior (left side) of the cross section, as shown in Figure 10-47. Select the exterior boundary segments for the exterior stop and the frame, and assign them the **Boundary Condition** of **Exterior Surface**.

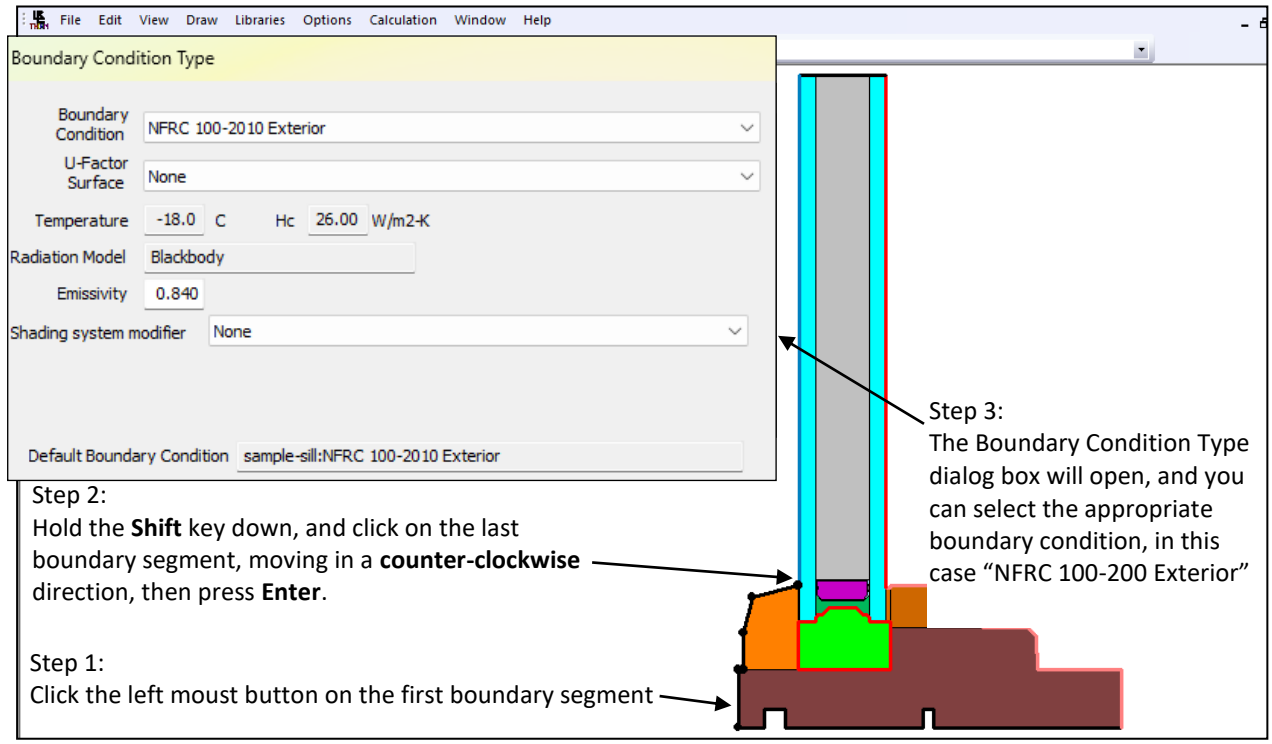


Figure 10-47. Select multiple boundary segments for the exterior side of the cross section, and define their boundary conditions.

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 10-48. Press the **OK** button.

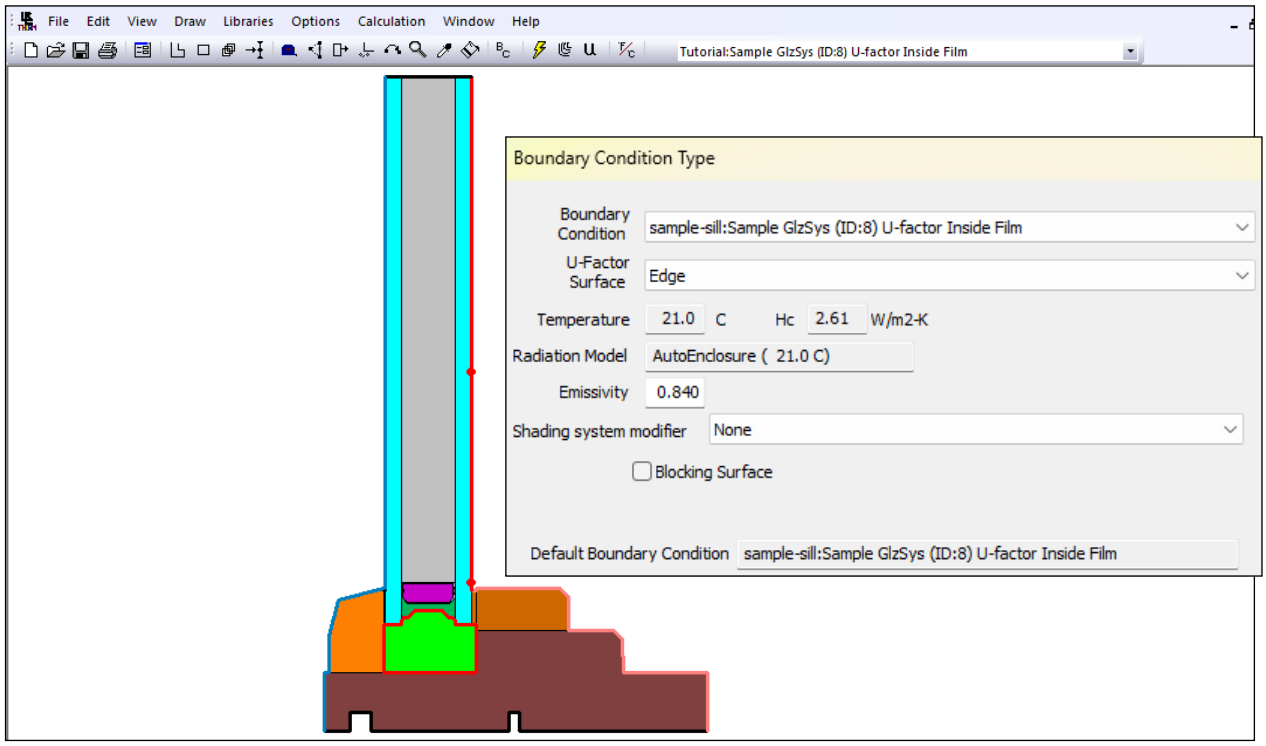


Figure 10-48. You can view the boundary condition definitions for the glazing system by double-clicking the left mouse button on the glazing system boundary.

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

10.6. 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 10-49.

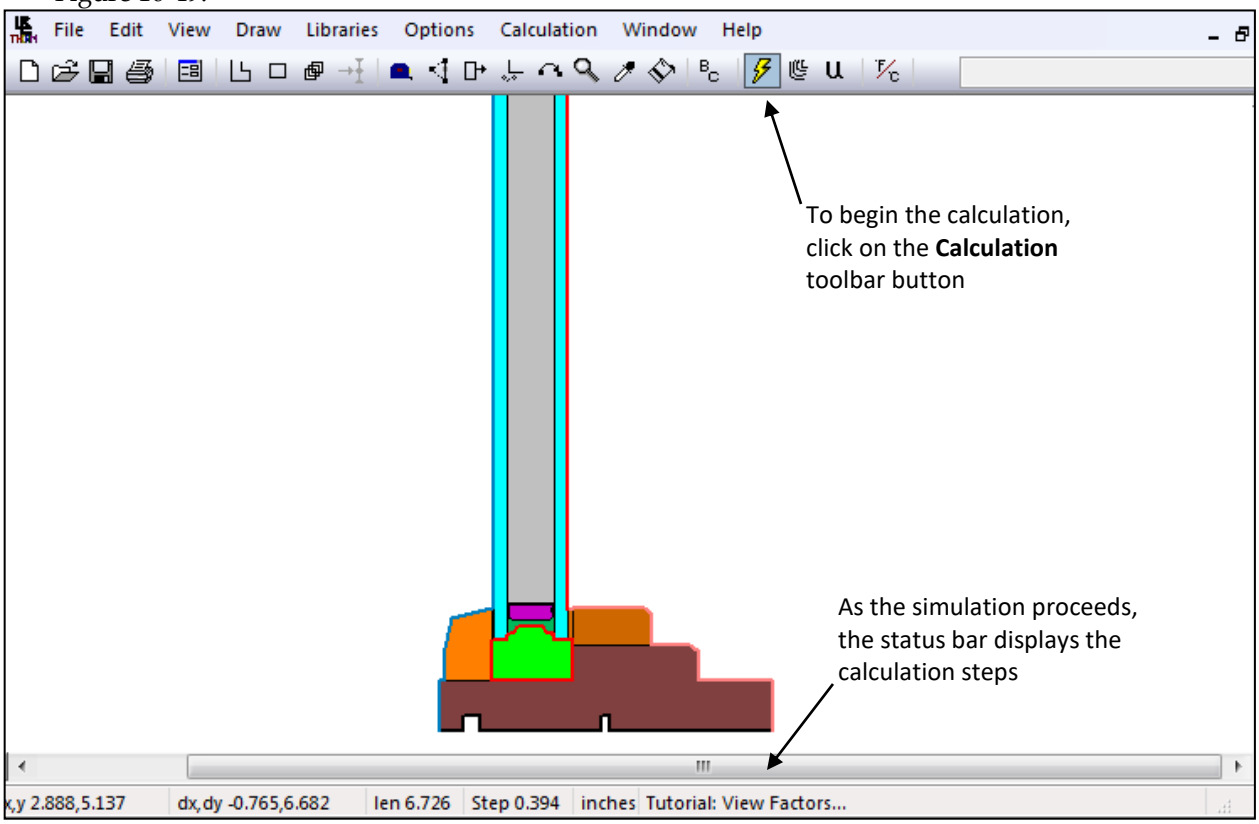


Figure 10-49. Click on the Calculation toolbar button; as the calculation proceeds, the steps will be displayed on the status bar.

- When the simulation is finished, the default isotherm results will be drawn on the cross section, as shown in Figure 10-50.

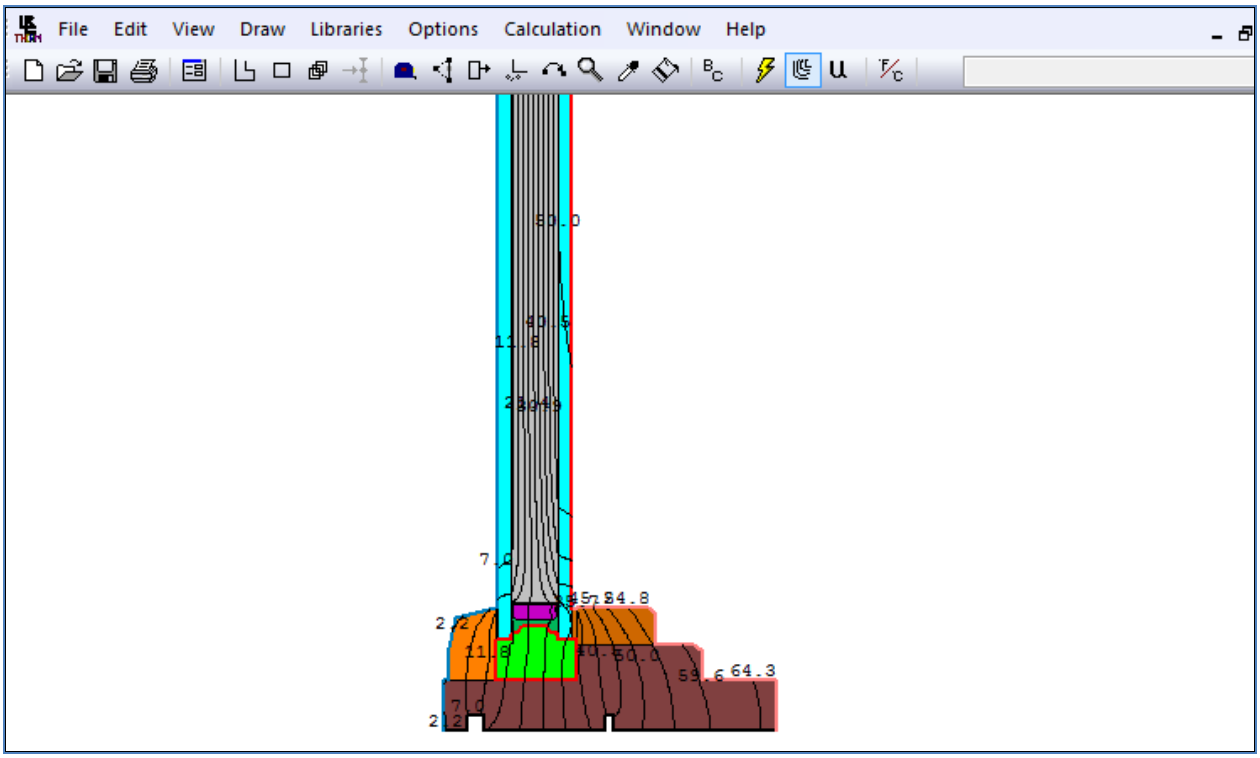


Figure 10-50. When the calculation has finished, the program will display the results as isotherms (constant temperature lines).

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

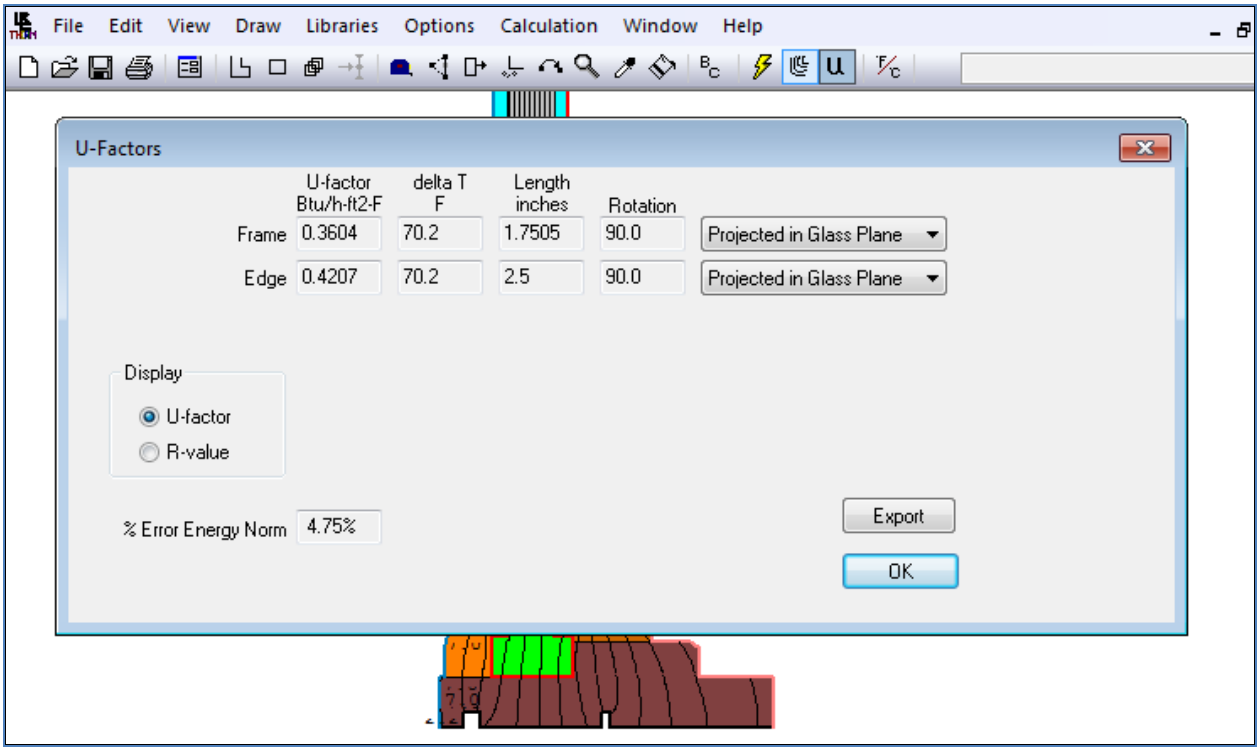


*Figure 10-51. U-Factor results, accessed from **Calculation/Show U-Factors** menu choice.*

- When the simulation finishes, the results will automatically display.

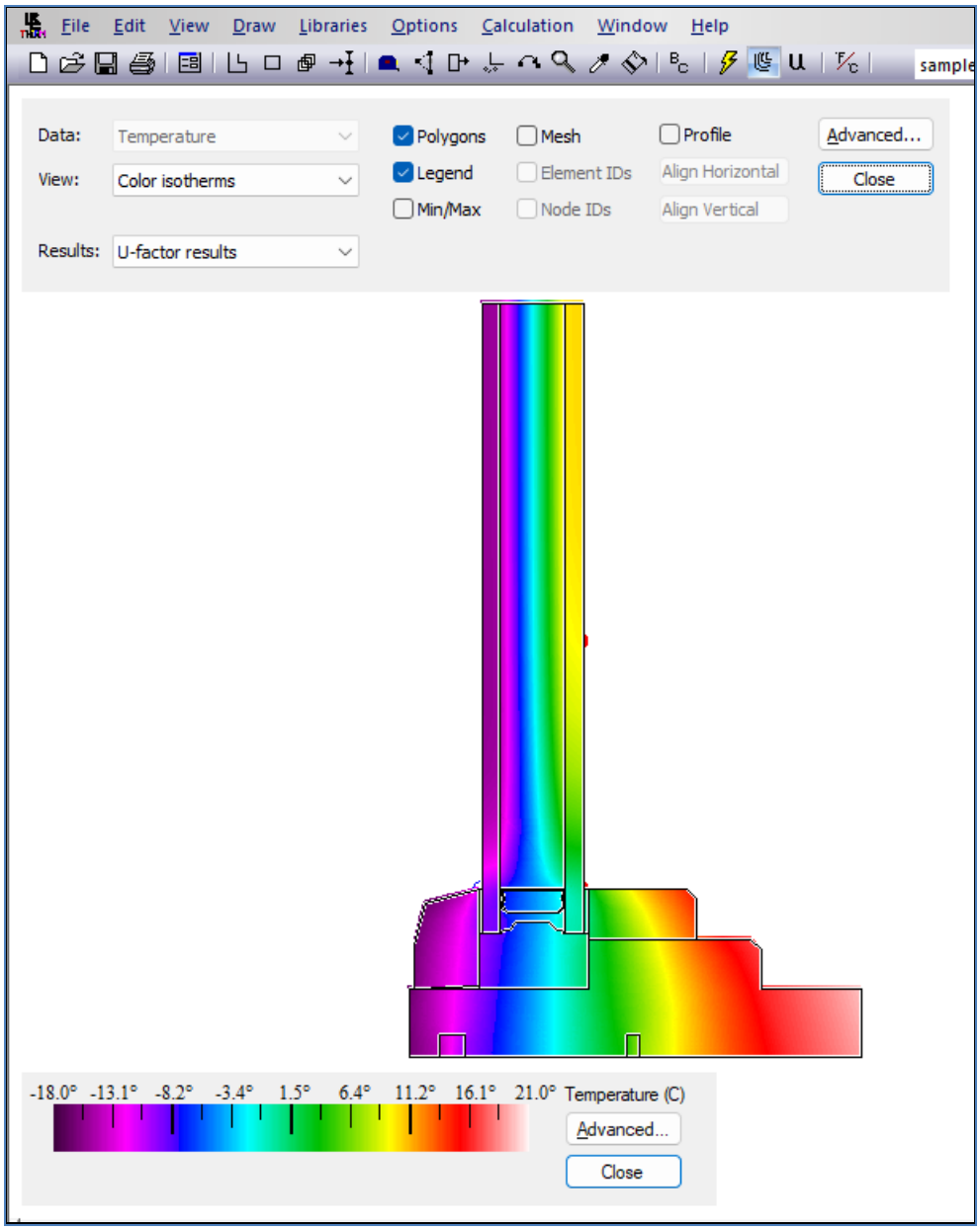


Figure 10-52. After the simulation is completed, the results will display.