

6. MODELING FRAME AND EDGE HEAT TRANSFER WITH THERM

6.1. Overview

In order to calculate the total product U-factor, the U-factor must be determined for the following components, and then these values must be area-weighted for the whole product:

- Center-of-glazing values from WINDOW
- Frame and edge-of-glazing values from THERM

The *NFRC 100: Procedures for Determining Fenestration Product U-factors* is the definitive source for the methodology for calculating U-factors, and should be used for situations not covered in this manual. In addition, NFRC Technical Interpretations supplement the NFRC 100 document, and these should also be consulted when questions arise.

6.2 Cross-Sections to be Modeled

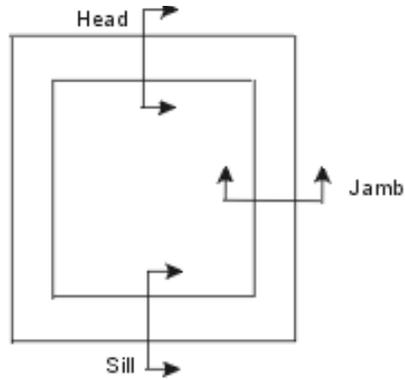
The fenestration product being modeled must be divided into the cross-sections which completely define the heat transfer through the product. THERM is used to calculate the U-factor values for the edge-of-glazing as well as the frame components (sill, head, jamb, meeting rail, and divider).

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 must be modeled. Table 6-1 lists the minimum and maximum cross sections that must be modeled for each operator type, and Figure 6-1 illustrates where the cross sections must be taken. The maximum assumes that each frame cross section is unique and therefore must be modeled. (Note that significant operating hardware may require more cross sections to be simulated. See Section 6.3.3, "What is Not 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 Resistance, 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.

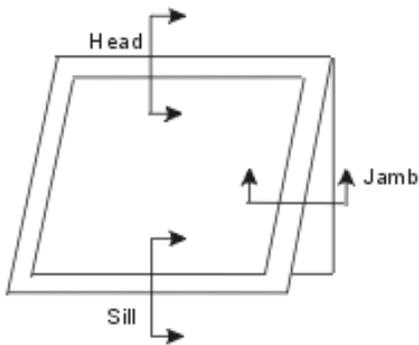
Although all the Product types listed in Table 1 of NFRC 100 are not listed here in Table 6-1 or Figure 6-1, these four basic categories cover all the geometries in NFRC 100.

Table 6-1. Cross sections to be modeled

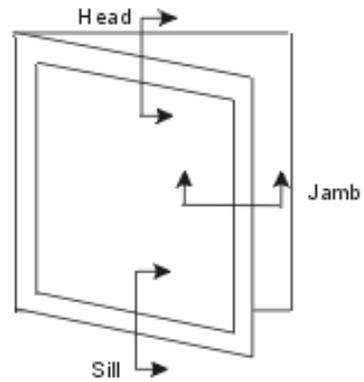
Operator Type	Number of Cross Sections to Model		Maximum	Cross Sections
	Minimum			
Single Lite (Fixed, Picture, Transom, Casement, Awning, etc)	CR & U Calc	U only Calc	4	<ul style="list-style-type: none"> ▪ Head ▪ Jamb (1 or 2) ▪ Sill
	3 If Jambs are the same	2 If Head and Sill are the same		
Two Lite 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	<ul style="list-style-type: none"> ▪ 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	<ul style="list-style-type: none"> ▪ 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	<ul style="list-style-type: none"> ▪ Head Fixed & Vent ▪ Jamb Fixed & Vent ▪ Sill Fixed & Vent ▪ Meeting Rail



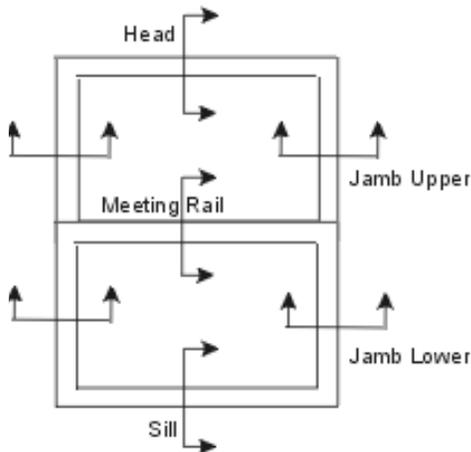
Fixed
(Max. 4, Min 3 cross sections)



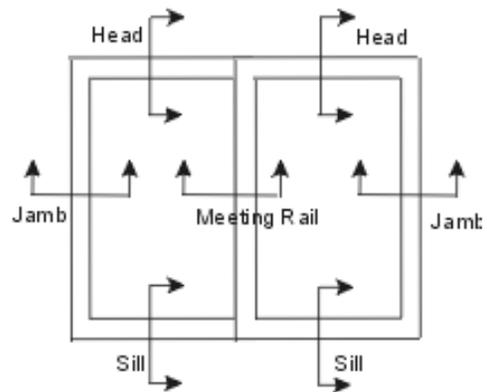
Projecting/Awning
(Max. 4, Min 3 cross sections)



Casement
(Max. 4, Min 3 cross sections)



Vertical Slider
(Max. 7, Min 5 cross sections)



Horizontal Slider
(Max. 7, Min 5 cross sections)

Figure 6-1. Minimum and maximum cross sections to be modeled based on operator type.

6.3 Draw the Cross Sections

After the number of cross sections to be modeled has been determined, draw them in THERM. The Section 5, "Drawing Cross-Section Geometry" in the *THERM User's Manual* contains detailed instructions about how to draw cross sections. This section of the THERM NFRC Simulation Manual contains information pertaining specifically to the NFRC procedures.

6.3.1. 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. Section 5.2, "Importing a DXF or Bitmap File as an Underlay" in the *THERM User's 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. Section 5.5, "Drawing Using the Keyboard (Numeric Cursor Positioning)" of the *THERM User's Manual*.

6.3.2. File Preferences -- Cross-Section Type

It is extremely important to specify the **Cross Section Type** in the **THERM File Properties** dialog box (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.

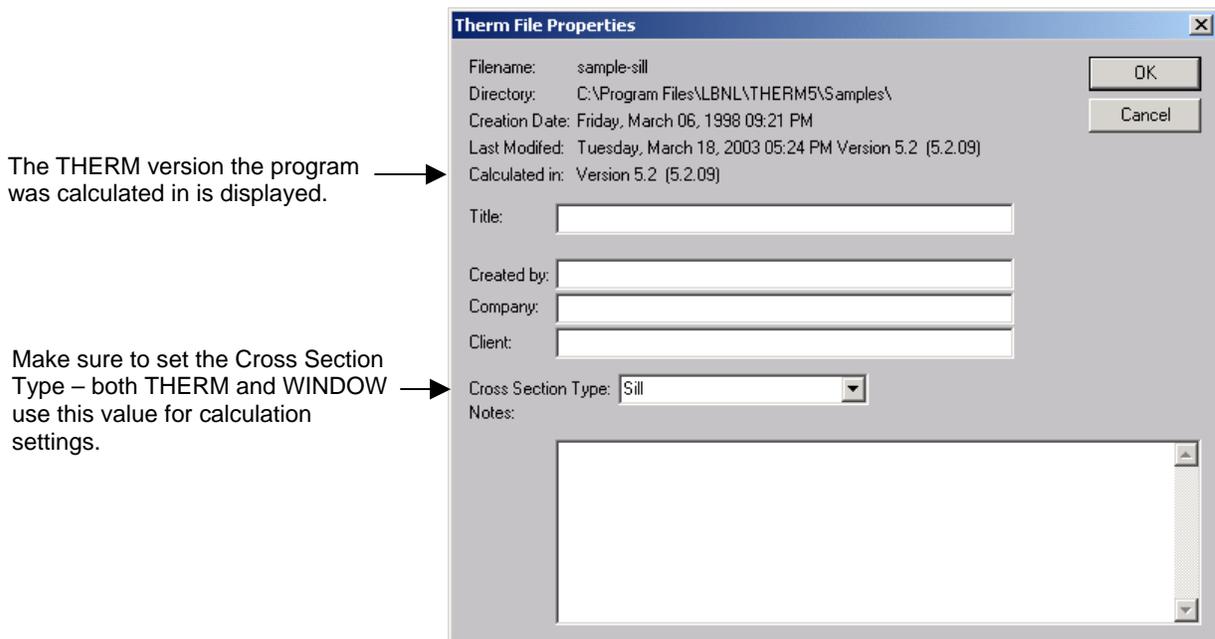


Figure 6-2 The Cross Section Type value from the THERM File Properties is used by the ISO 15099 models to determine the gravity vector and also in WINDOW to determine the method for calculating the Condensation Resistance values.

6.3.3. Cross-Section Orientation

For NFRC simulations using THERM, all cross sections shall be oriented in a vertical direction, i.e., all the glazing systems must be pointing either up or down, not horizontally. (The exception is for skylights, which are modeled at a 20 degree tilt; see Section 8.5 for a discussion of modeling skylights). 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 must be rotated into a vertical position for modeling. Figure 6-3 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 must 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.

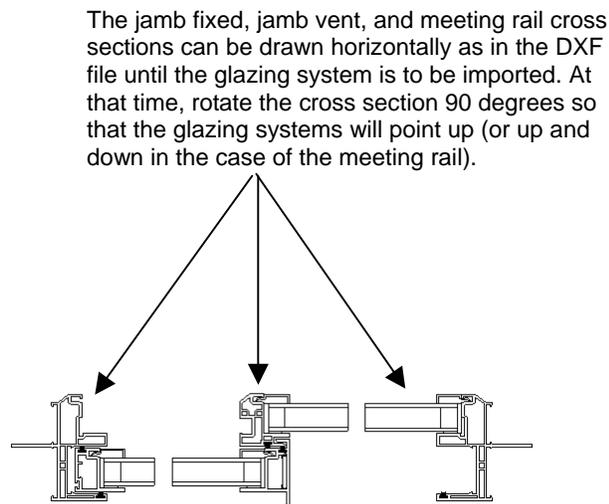


Figure 6-3. DXF file with horizontal cross sections that must be rotated 90 degrees before importing glazing systems and simulating.

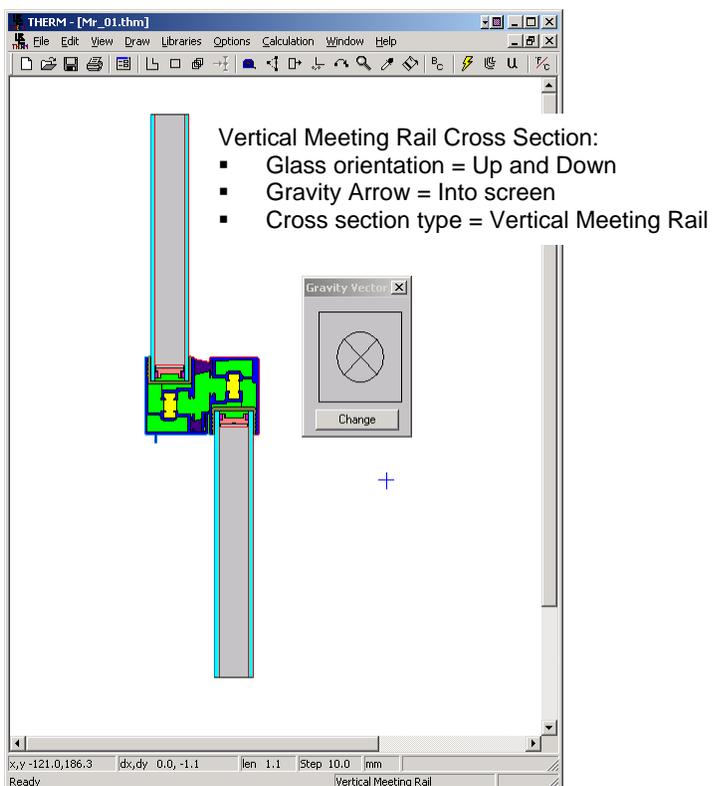
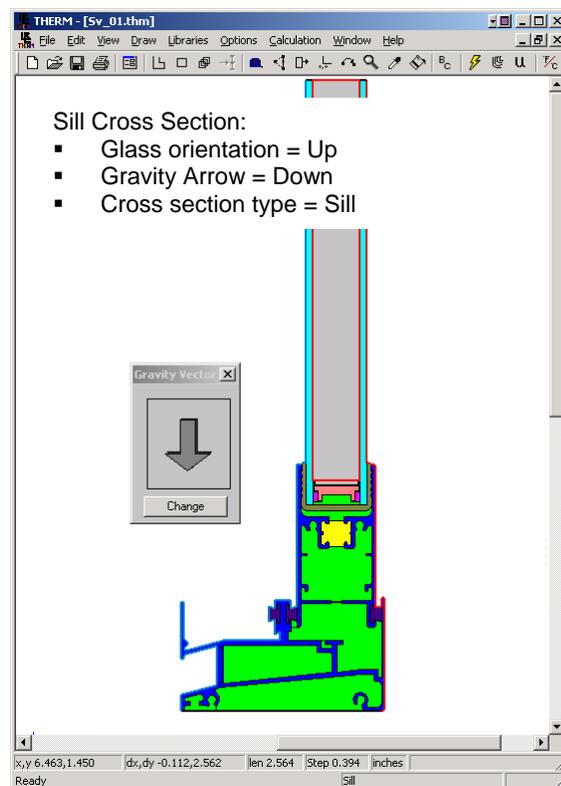
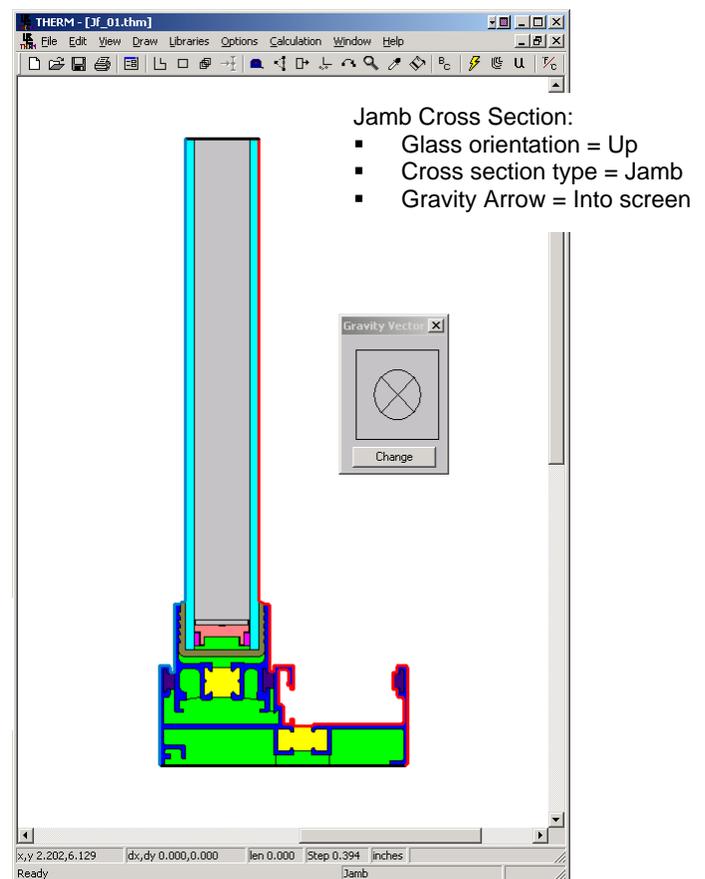
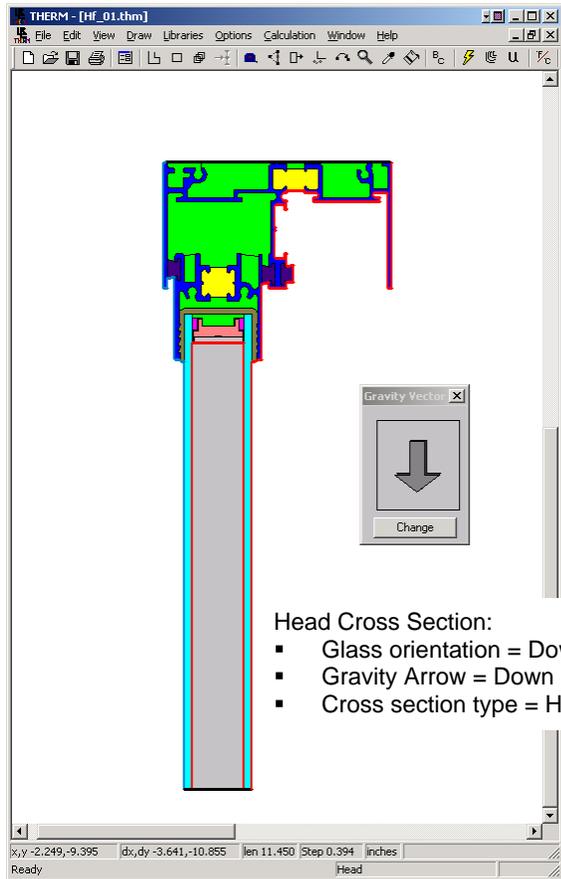


Figure 6-4 Sample cross sections from a horizontal slider showing the cross section orientation, cross section type label in the lower right corner of the screen, and gravity arrow orientation..

6.3.4. What is Not Modeled

NFRC 100 specifically excludes certain options on fenestration products which are not modeled in THERM. These options include:

- Screens
- Optional Interior trim
- Removable grilles applied to the interior or exterior surface of the glass (i.e., snap-on grilles). Grilles or dividers between glass layers must be modeled under some circumstances, as discussed in Section 8.3, "Modeling Internal Dividers."
- Optional jamb, head, and sill extensions
- Interior or exterior shading devices
- 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 shall be modeled.

In addition, as stated in *NFRC 100*, "including but not limited to screws and bolts in curtainwalls and pour-and-debridge thermal breaks which are not full debridged, shall be simulated". For the time being, components that *do not* have to be modeled include:

- Hinges
- Locks
- Balances
- Non-continuous Operator Hardware
- Weep Holes
- Setting Blocks
- Shear Blocks
- Corner Keys

6.3.5. 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.

6.3.6. Frame Cavities

For NFRC Simulations, frame cavities are modeled using the ISO 15099 Cavity Model. There are default items in the Material Library that can be used to model frame cavities:

- **Frame Cavity NFRC 100-2001:** to be used for all interior frame cavities
- **Frame Cavity Slightly Ventilated NFRC 100-2001:** to be used for frame cavities open to the exterior. See Section 6.3.7, Vented Exterior Cavities, for a complete description of when to apply these cavities.

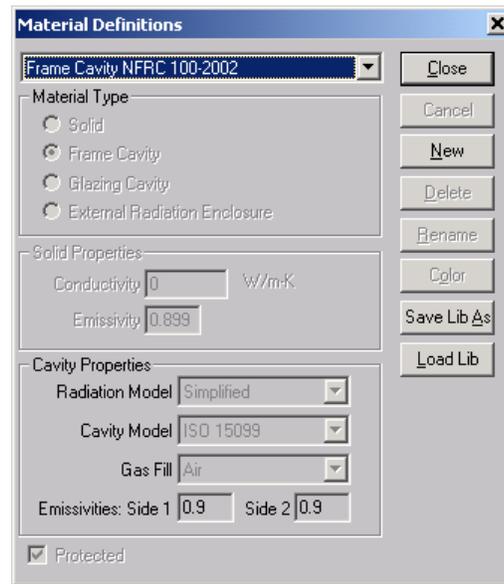


Figure 6-5. Default frame cavity material for NFRC Simulations.

As shown in the figure above, the Frame Cavity NFRC 100-2001 material has the following characteristics:

- **Radiation Model:** Simplified
- **Cavity Model:** ISO 15099
This cavity model is an implementation of the ISO 15099 standard, and will cause THERM to automatically calculate the cavity wall temperatures, emissivities and heat flow direction.
- **Gas Fill:** Air
- **Emissivities:**
The default emissivities that are included in this frame cavity material are essentially irrelevant because THERM will recalculate and override them during the simulation of the cross section. However, when either the “Frame Cavity NFRC 100-2001” or the “Frame Cavity Slightly Ventilated NFRC 100-2001”, the frame cavity surfaces 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.

THERM also makes the following assumptions about the frame cavity:

- **Default frame cavity height:** 1 meter
- **Gravity vector:** based on Cross Section Type (see Section 6.3.2, “Cross Section Type”, and Section 6.3.3, “Cross Section Orientation”)

If the frame cavity model is set to “ISO 15099”, which it is for **Frame Cavity NFRC 100-2001** the emissivities, temperatures and heat flow direction will be automatically calculated by the program during simulation. 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 menu. 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. (See the *THERM User’s Manual* for a discussion of the gravity vector). 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 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.

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 **NFRC 100-2001**" choice from the **Material Library** pulldown list. Because the **Cavity Model** for that material is set to "ISO 15099", the heat flow direction, emissivities, and temperatures will be automatically calculated.

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 relevant.

The dialog box displays the following values:

- Material: Frame Cavity NFRC 100-2002
- ID: 30
- Keff: 0.0395 W/m-K
- Horizontal dimension: 16.857 mm
- Vertical dimension: 3.411 mm
- Nu: 1.00
- Heat Flow Direction: Left
- Side 1 Temperature: -4.04 C
- Side 1 Emissivity: 0.43
- Side 2 Temperature: -9.683 C
- Side 2 Emissivity: 0.43

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 simulation.

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.

Figure 6-6. 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 rectangularization of the cavity according to ISO 15099 specifications. The current NFRC procedure is to break up cavities that are separated by a connection, or "throat", less than or equal to 5 mm (0.20 inches). 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.

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 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 cavity does not need to be broken up.

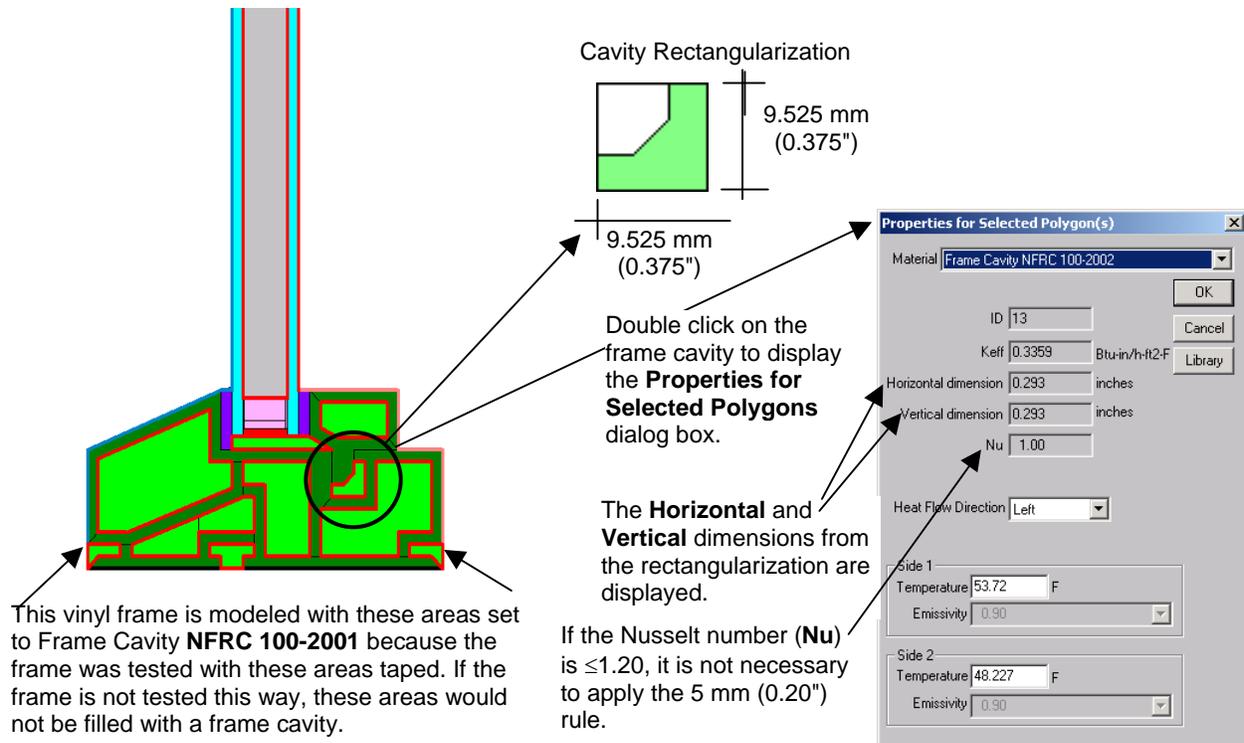


Figure 6-7. THERM uses the ISO 15099 rectangularization specification to model the thermal characteristics of cavities.

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 cavity. 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.
2. Check the Nusselt Number of the undivided cavity. If the Nusselt Number is less than or equal to 1.20, the simulator shall not break the cavity into smaller polygons.
3. If the Nusselt Number is greater than 1.20, the simulator shall identify all cavities that are less than or equal to 5 mm (0.20 inches) in both opening and depth and define them as separate cavities.
4. Re-check the Nusselt number of the remaining large cavity. If it is less than or equal to 1.20, no further action is necessary. If the number is greater than 1.20, the simulator shall:

- a. Identify throats (constriction) in the cavity that are less than 5 mm (0.20 inches) and working from the largest part of the cavity to the smallest, break-up the larger cavity into smaller cavities, splitting first horizontally, then vertically, and finally (as a last resort) diagonally.
- b. Each time the cavity is broken, the simulator shall re-check the Nusselt number of the new cavities. If the Nusselt number in the remaining cavity/cavities is less than or equal to 1.20, no further action is necessary. If the Nusselt number for the remaining cavity/cavities is greater than 1.20, proceed, again, with step 4 until no further constrictions can be defined.

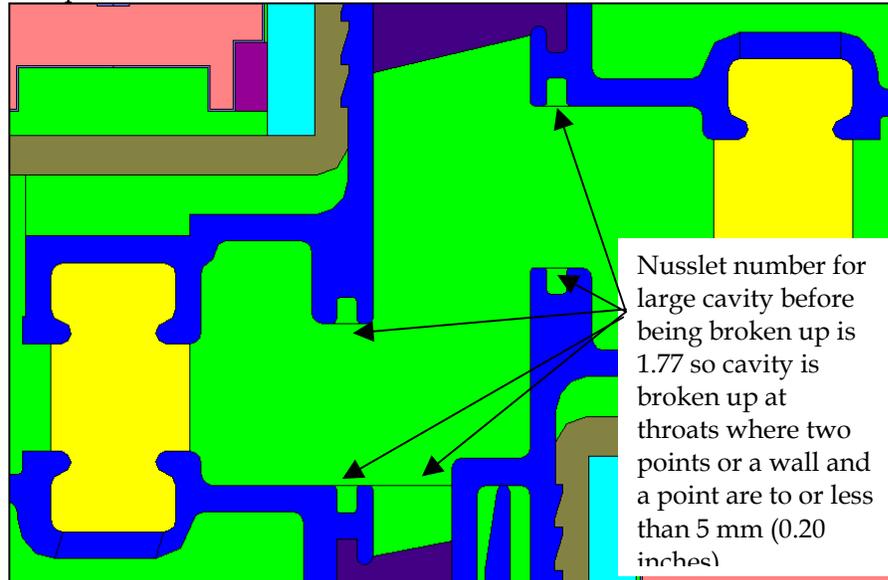


Figure 6-8. Breaking up frame cavities.

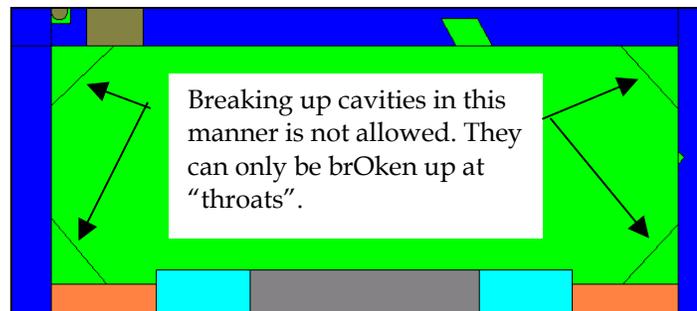


Figure 6-9. Breaking up frame cavities in this manner is not allowed.

6.3.7. Slightly Ventilated Exterior Cavities

For NFRC simulations, 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, shall be modeled according to *ISO 15099, Section 6.7.1* which states that cavities greater than 2mm but equal to or less than 10 mm shall be modeled as slightly ventilated air cavities. The THERM Material Library has a default material for this case, called “Frame Cavity Slightly Ventilated NFRC 100-2001”, which will be used to fill the entire cavity. The figure below illustrates this.

Model a cavity open to the exterior environment as slightly ventilated when:

$$d \geq b$$

and

$$2 \text{ mm} < b \leq 10 \text{ mm}$$

For cavities $< 2 \text{ mm}$, use the standard “Frame Cavity NFRC 100-2001”

For cavities $> 10 \text{ mm}$, do not model a frame cavity

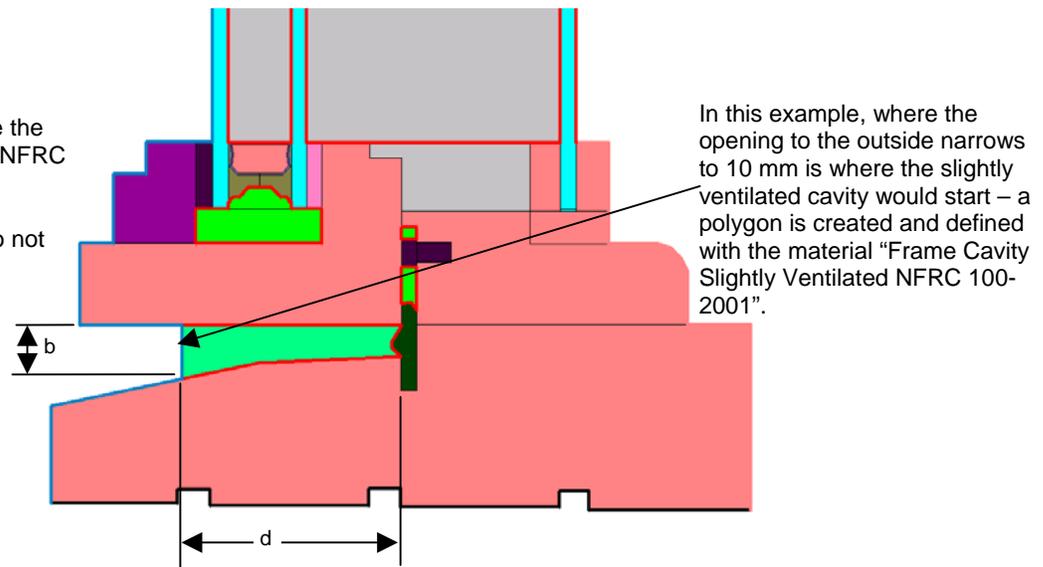


Figure 6-10. Slightly Ventilated exterior cavities .

If during testing, the product is sealed at the inside surface with tape, those cavities must be simulated with the standard “Frame Cavity NFRC 100-2001” material.

Open cavities, which would be sealed during testing, must be filled with a polygon of material **Frame Cavity NFRC 100-2001**.

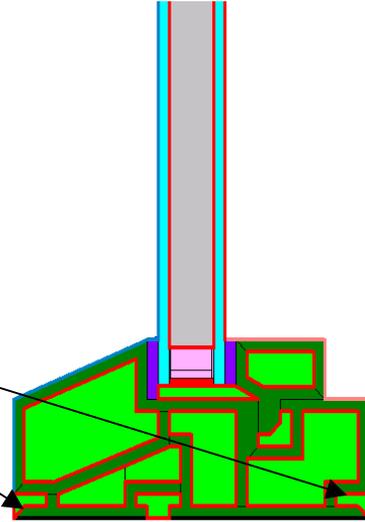


Figure 6-11. Cavities open to the interior are filled with Frame Cavity NFRC 100-2001 material .

6.3.8. 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 10, "Interpreting Arcs" in 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 must 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 must be not greater than 2.5 mm (ISO 15099 Standard).

6.3.9. Modeling Sloped Sills

A product with a sloped sill must be modeled in the same way that it was 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 must 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.

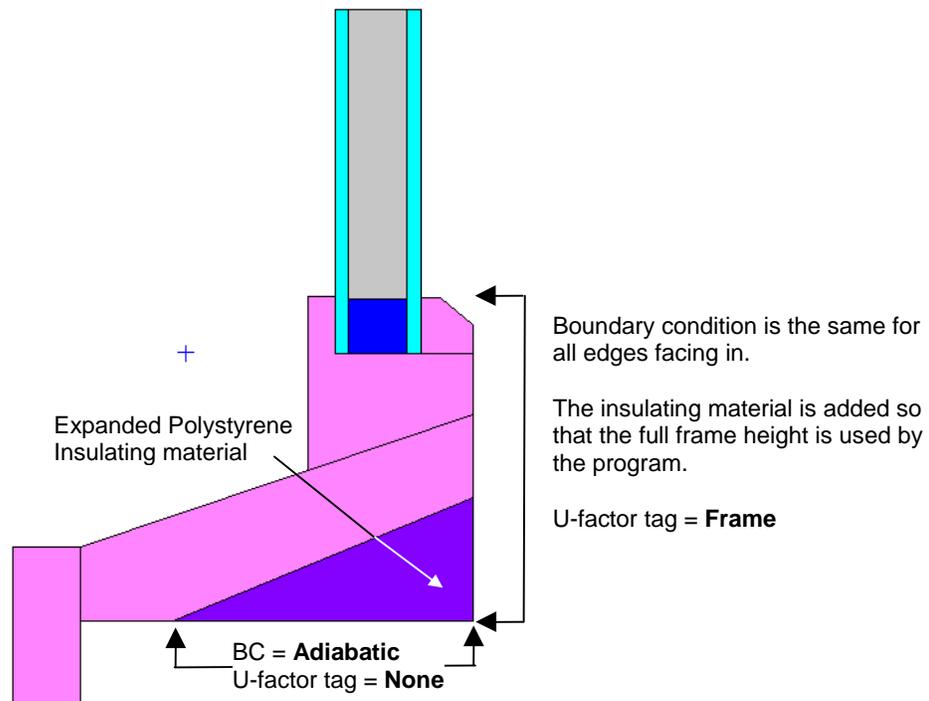


Figure 6-12. Model the sloped sill as it was tested, with insulating material under the sill.

6.4 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 5.9, "Inserting a Glazing System" in the *THERM User's Manual* contains detailed information about this step. The discussion in this manual is added information about glazing systems.

6.4.1. Overview

For NFRC simulations, a glazing system shall 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

As discussed in Section 6.5, 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.

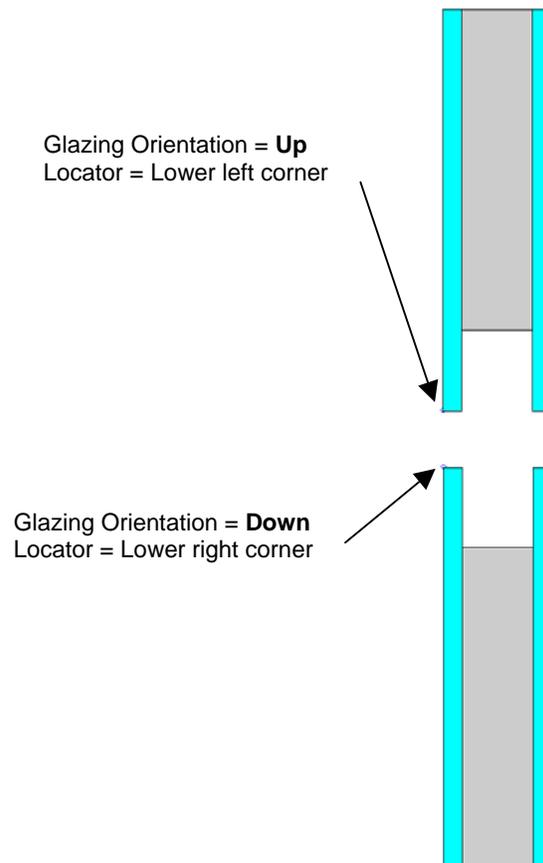


Figure 6-13. The Locator position when importing glazing systems in different orientations.

6.4.2. Inserting Glazing Systems

The *THERM User's 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.

Step 1:
Select the Glazing System choice from the Library menu

Step 2:
Select the Glazing System from a WINDOW glazing system library

Check the "Use nominal glass thickness" to turn on this feature. This may be useful for multiple glazing option calculations to ensure that all glazing systems have the same thickness

Step 3:
Enter the appropriate values;

- CR cavity height = value from Table 6-2 based on product height (see Section 6.4.5)
- Edge of glass = 63.5 mm
- Glazing system height = 150 mm

Check the Use CR Model for Window Glazing Systems if appropriate

Step 4:
Enter the appropriate **Gap Properties**. For NFRF simulations, "Default", which uses the values directly from WINDOW, is the correct setting

Step 5:
Define the Boundary Conditions:

- Click on "Use U-factor values"
- Exterior Boundary Conditions:
 - Set to "Use existing BC from Library", and set the value to "NFRF 100-2001 Exterior"
- Interior Boundary Condition:
 - Set to "Use convection plus enclosure radiation", which sets the boundary condition by default to "AutoEnclosure"

Figure 6-14. Inserting a Glazing System into a cross section.

The NFRC modeling requirements when inserting a glazing system are:

- **CR cavity height:** See table 6-2, Section 6.4.5 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 100-2001 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*.

6.4.3. Adding the Radiation Enclosure Model

In accordance with NFRC 100-2001, the THERM radiation enclosure model shall be used for all simulations.

In THERM 5.0, *it is not necessary to draw the enclosure geometry*. If the glazing system was imported with the boundary condition set to "Use convection plus enclosure radiation", THERM will automatically assign boundary conditions to the glazing system that have the Radiation Model set to "Autoenclosure". This type of boundary condition models the radiation enclosure without the enclosure geometry being drawn. A more detailed description is found in Section 6.5, "Boundary Conditions" in this manual.

See the *THERM User's Manual* for more details about the Autoenclosure feature, and information about drawing the enclosure geometry for the case where more than one radiation enclosure is desired.

6.4.4. 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 must 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 that must 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 case" file, and will contain no results once 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.

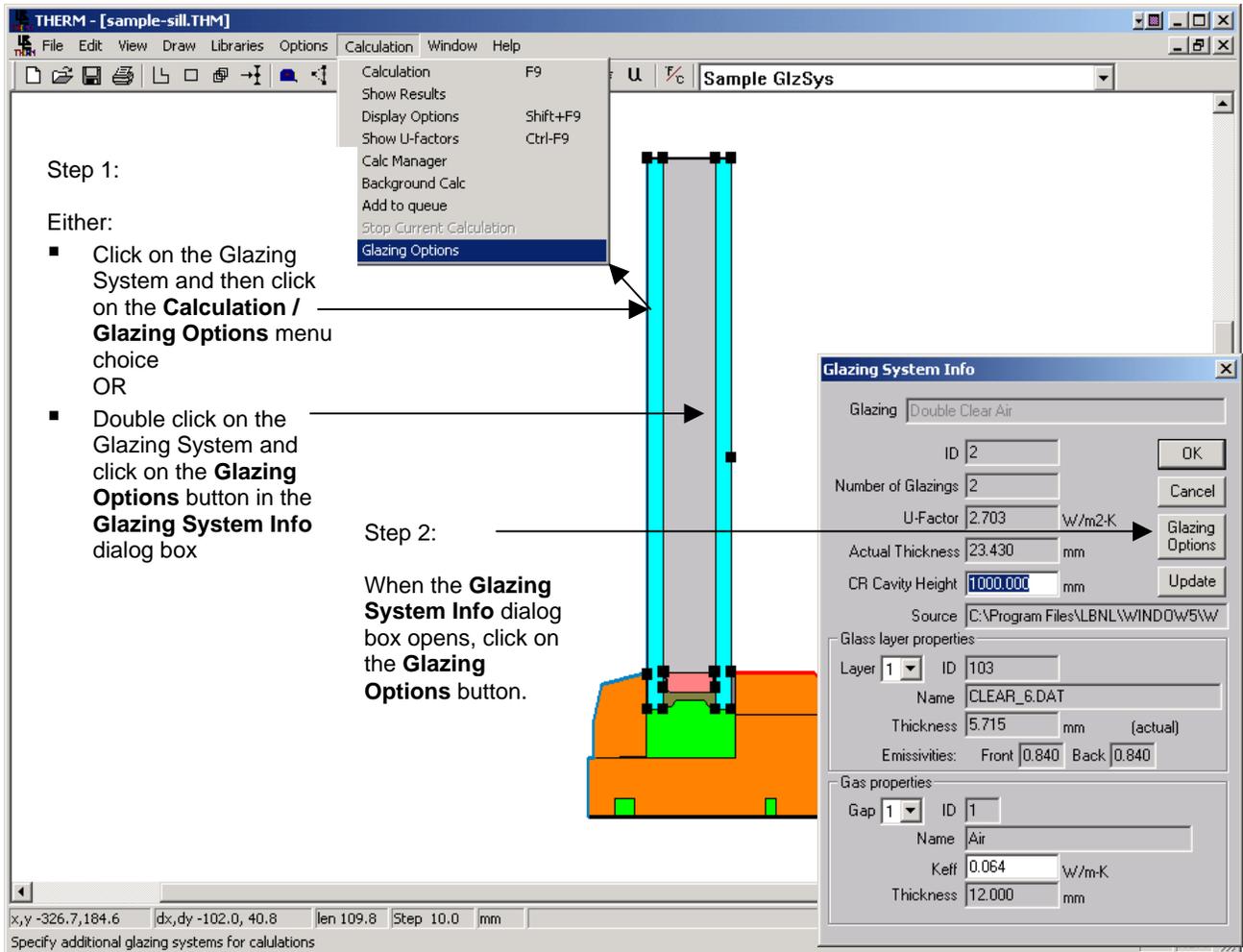
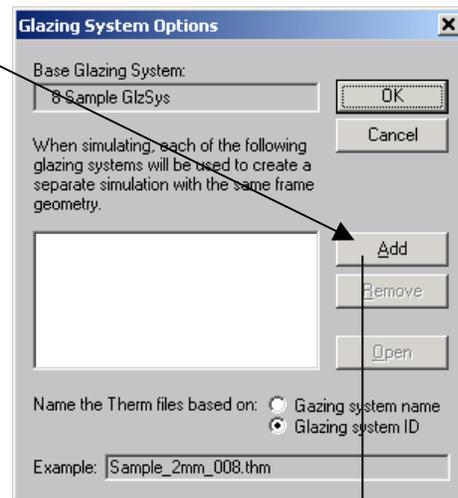


Figure 6-15. Define multiple glazing options.

- 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.
- 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.
 - Boundary Conditions:** THERM will automatically calculate the boundary conditions for each separate glazing system option.

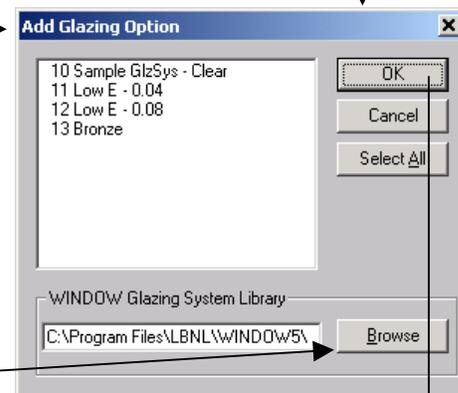
Click on the **Add** button in the **Glazing System Options** dialog box.



The **Add Glazing Option** dialog box shows all the glazing systems which match the Base Glazing System in terms of glass layer thickness and overall thickness.

Select individual glazing systems (use **Shift-click** and **Ctrl-click**) or click on **Select All**, and click the **OK** button.

Click on the **Browse** to select a different WINDOW Glazing System Library.



The selected **Glazing Systems** appear in the **Glazing System Options** dialog box.

Click on the radio button which indicates to the program how to name the individual THERM files for each glazing system option. Either the **Glazing System ID** or **Name** will be appended on to the base case.

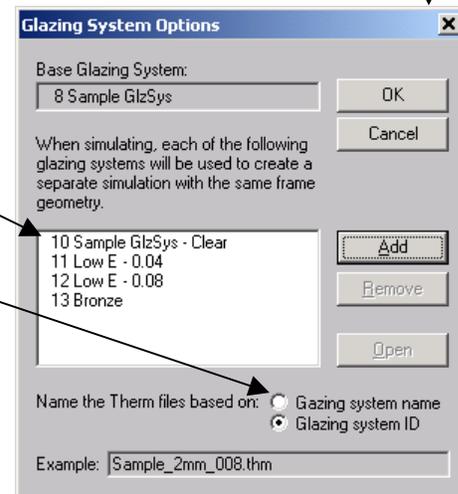


Figure 6-16. Select the glazing systems to be associated with the base case file.

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 6.6, "Calculating the Results"*.

6.4.5. Condensation Resistance

There are two ways to activate the Condensation Resistance model in THERM:

- When importing the glazing system, check the “Use CR Model for Window Glazing System” checkbox.
- In the **Options** menu, **Preferences** choice, **Therm File Options** tab, check the “Use CR Model for Glazing Systems”

When inserting the glazing system into the model, as discussed in Section 6.4.2, the value for the “CR cavity height” shall come from the table below.

Table 6-2. Default Values for Actual Glazing Height Modeling for Condensation Resistance

Real Product Height	Default Glazing Height for Condensation Modeling (input as the “CR 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 Resistance 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 CR temperature data that will be used, when this profile is imported into WINDOW, to generate the overall Condensation Resistance 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 CR results.

The Condensation Resistance model is not used for vertical cross sections, such as a Jamb, a Vertical meeting rail, or a Vertical Divider. Even if the CR model is activated (i.e., the “Use CR Model for Window Glazing System” is checked) for these Cross Section Types, THERM will not perform the CR simulation for that file.

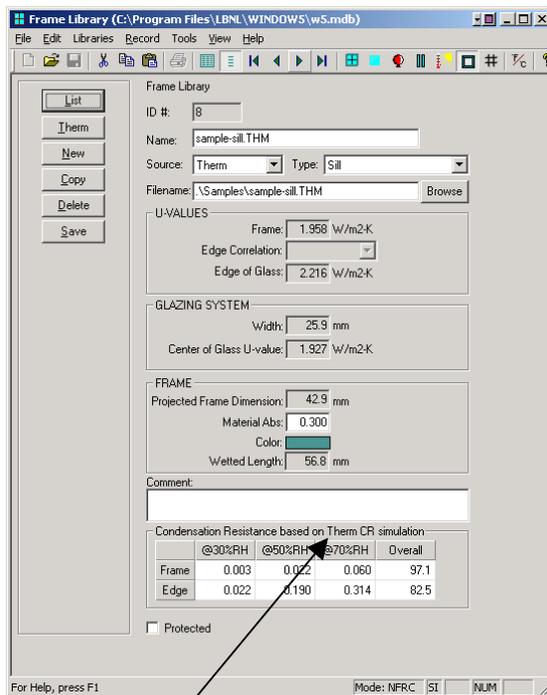
THERM generates temperature data from both the CR and U-factor simulations which WINDOW uses to calculate the frame CR values (shown in the Frame Library Detailed View) and also the whole product CR values (shown in the Window Library Detailed View). When the CR model is not appropriate for the whole product CR calculations, WINDOW will use the temperatures from the THERM U-factor calculation instead, as shown below.

Product Type (From THERM)	WINDOW 5 Whole Product Tilt (input in Window Library Detailed View)	Temperatures Used by WINDOW5 for whole product CR calculations
Horizontal elements: Sill, Head, Horizontal Meeting Rail, Horizontal Divider	90	CR
Horizontal and Vertical elements: <ul style="list-style-type: none"> ▪ Sill, Head, Horizontal Meeting Rail, Horizontal Divider; ▪ Jamb, Vertical Meeting Rail, Vertical Divider 	$\geq 0, \leq 20$	U-factor
Vertical elements: Jamb, Vertical Meeting Rail, Vertical Divider	90	U-factor
Horizontal and Vertical elements: <ul style="list-style-type: none"> ▪ Sill, Head, Horizontal Meeting Rail, Horizontal Divider; ▪ Jamb, Vertical Meeting Rail, Vertical Divider 	$> 20, < 90$	CR not calculated

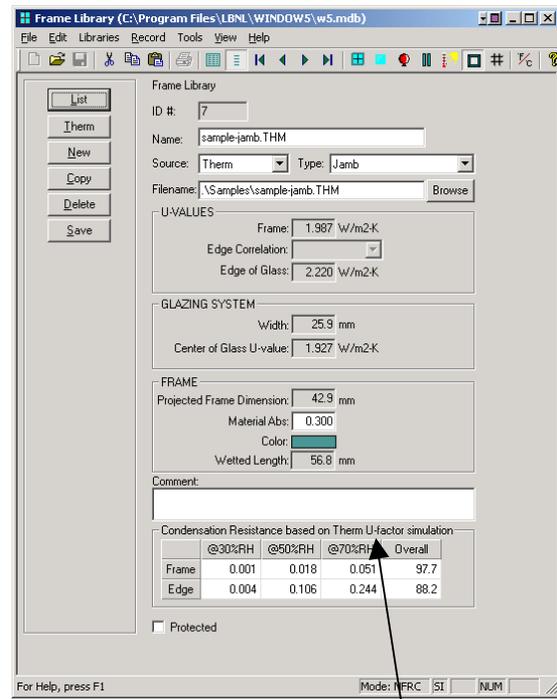
The application of these rules happens when WINDOW 5 performs a calculation in the Window Library. It is up to the simulator to have provided WINDOW with the appropriate simulations in THERM in order for WINDOW to be able to calculate the result. For example, if there are no CR results for a Sill or Head THERM file modeled in a window with a 90° tilt, then WINDOW will not calculate a whole product CR value because the needed CR temperature data does not exist. A message, shown below, will indicate which cross sections are missing the CR data.



Figure 6-17. WINDOW5 will display a message if it determines that CR temperature data is not available from the THERM file.



For Product Type = Sill, not rotated, the CR calculation will be based on the THERM CR simulation



For Product Type = Jamb, not rotated, the CR calculation will be based on the THERM U-factor simulation

Figure 6-18. WINDOW5 Frame Library Detailed View displays the type of THERM temperature data to be used with the CR calculation, either U-factor or CR.

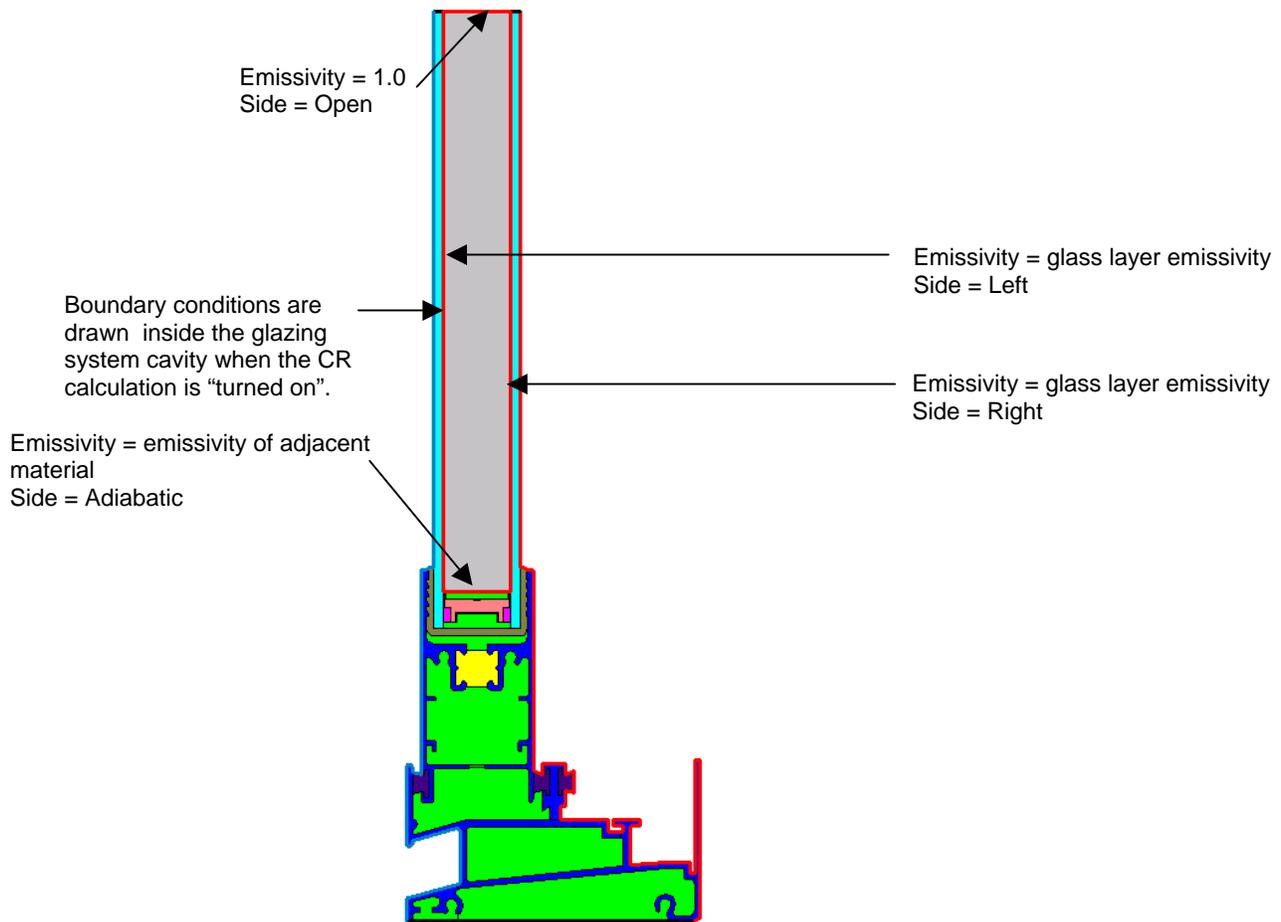


Figure 6-19. Modeling the profile with the CI model turned on.

6.5 Defining Boundary Conditions

Chapter 6 of the *THERM User's Manual* contains a detailed explanation of how to define boundary conditions in a model. The information found in this manual is supplemental to that discussion.

6.5.1. Overview

Boundary conditions must be defined for all the surfaces on the perimeter of the models, as well as at the surfaces adjacent to the 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 (which is required by NFRC for all simulations) 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.
 - **For Frame Elements:** Set all non-glazing system interior boundary conditions according to the material, from the following predefined 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.)If there are frame materials that fall into more than one category, apply the boundary condition of the predominant material to all the frame elements):
 - 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)
- **Exterior:** The "NFRC 100-2001 Exterior " exterior boundary condition is 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 film coefficient of 26 W/m²-°C (4.58 Btu/h-ft²-°F).
- **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.

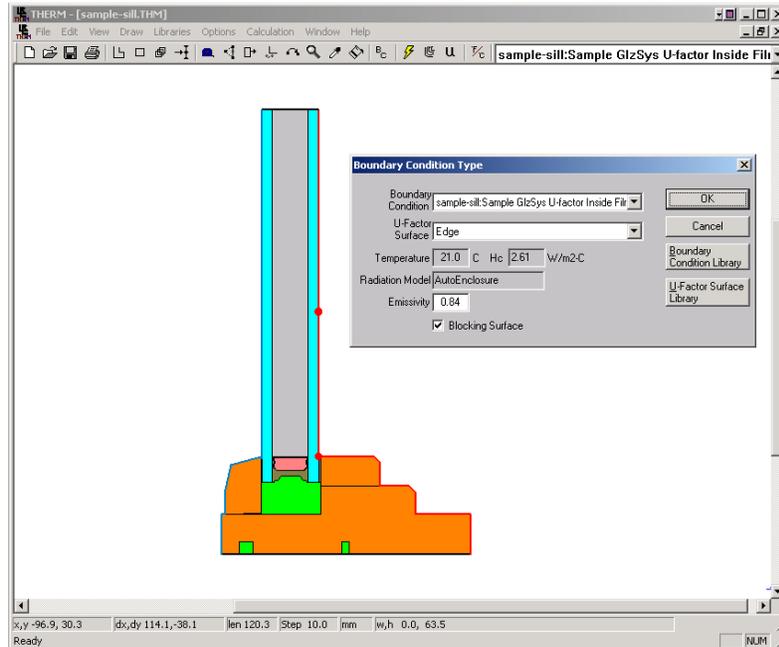


Figure 6-20. Double click on a boundary condition segment to see its characteristics..

THERM has a **Boundary Condition Library** (accessed from the **Library/Boundary Conditions** menu, or by double clicking on a boundary condition) which has the standard boundary conditions defined by NFRC, shown in Table 6-3, as well as **Adiabatic**. By default, the exterior boundary conditions are blue, interior boundary conditions are red, and the adiabatic boundary condition is black.

Table 6-3. 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 100-2001 Exterior	Blackbody	26	4.578	26	4.578
Interior Aluminum Frame (convection only)	AutoEnclosure	3.29	0.579	4.94	0.869
Interior Thermally Broken Frame (convection only)	AutoEnclosure	3.00	0.528	4.38	0.771
Interior Thermally Improved Frame (convection only)	AutoEnclosure	3.12	0.549	4.60	0.81
Interior Wood/Vinyl Frame (convection only)	AutoEnclosure	2.44	0.429	3.38	0.595
WINDOW Glazing System boundary condition <filename>:<glazing system name> U-factor Inside Film	AutoEnclosure	Depends on the WINDOW calculations for the imported glazing system			

6.5.2. 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 must be assigned in most cases.

- **NFRC 100-2001 Exterior** for the entire exterior surface of the glazing system and frame
- **Adiabatic** for bottom of the frame
- **Interior Frame Components:** The appropriate interior convection-only boundary condition for all other interior surfaces, from the choices in Table 6-3, found in the THERM Boundary Condition Library.

The technique for doing this is described in detail in Chapter 6, "Defining Boundary Conditions" of the *THERM User's Manual*.

In THERM, in addition to assigning boundary conditions to a boundary segment, the **U-factor Surface Tags** must 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. See *NFRC 100*, "Figure 2, Fenestration Product Schematic -- Vertical Section". 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. It is also possible to change the boundary condition definition for boundary segments using the **Select Material/BC** toolbar button. Click on the **Eye Dropper** tool, click on a boundary segment whose definition is to be duplicated to other segments, then click on the boundary segment(s) which need that boundary condition definition. The program will

assign both the boundary condition and U-factor tag from the copied segment. There is also one level of "undo" for boundary conditions, accessed through the **Edit/Undo** file menu choice.

If 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 Condition 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.

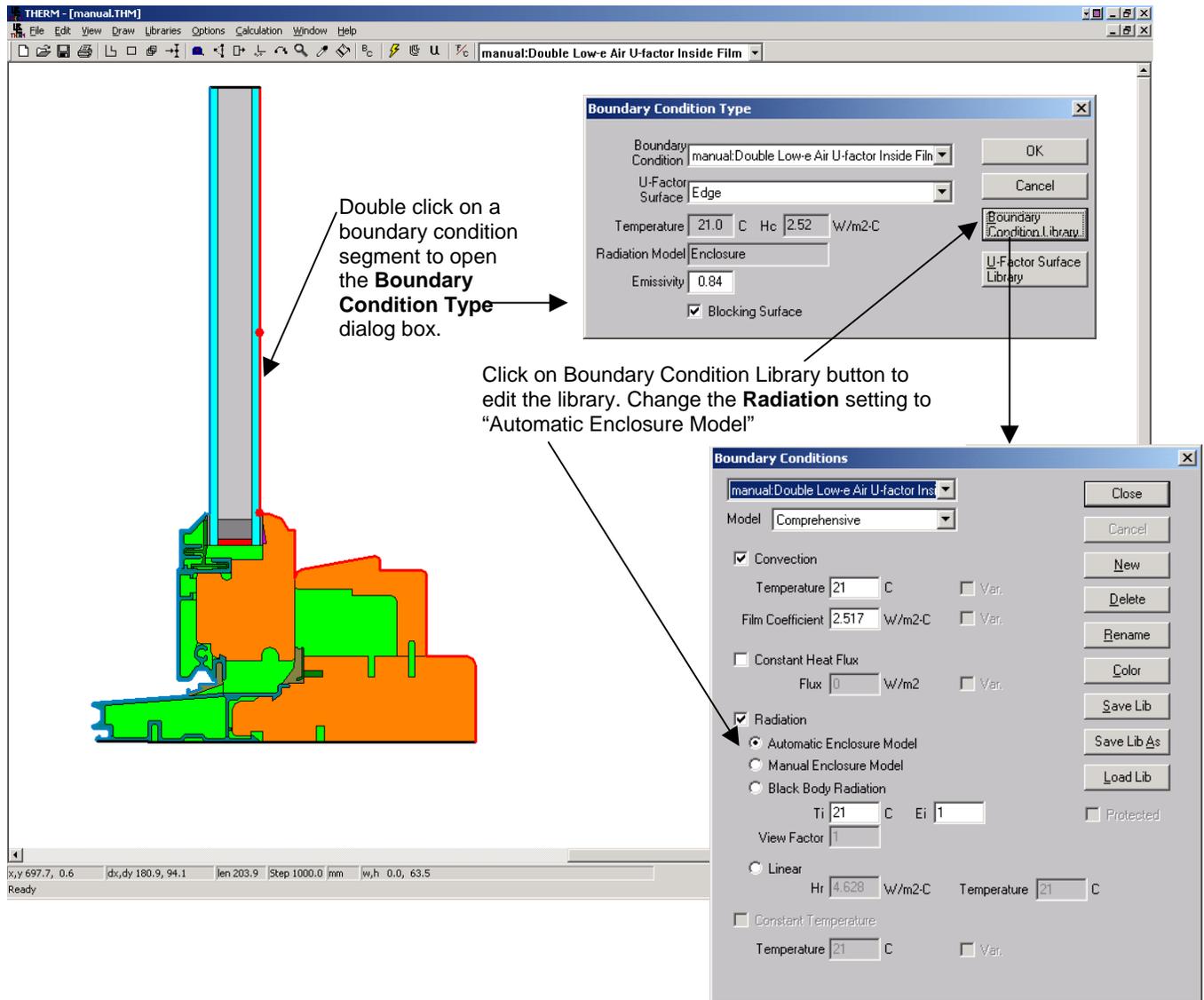


Figure 6-21. Make sure that the interior boundary conditions have set to "AutoEnclosure".

When boundary conditions are generated for a model, the program keeps them in the model until they are regenerated. 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.

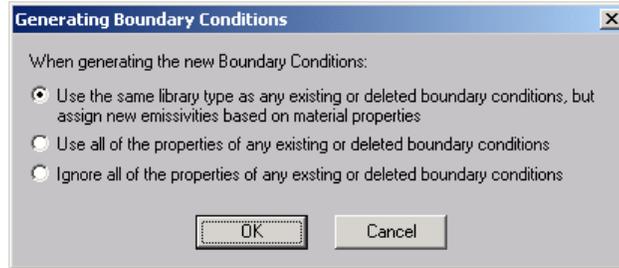


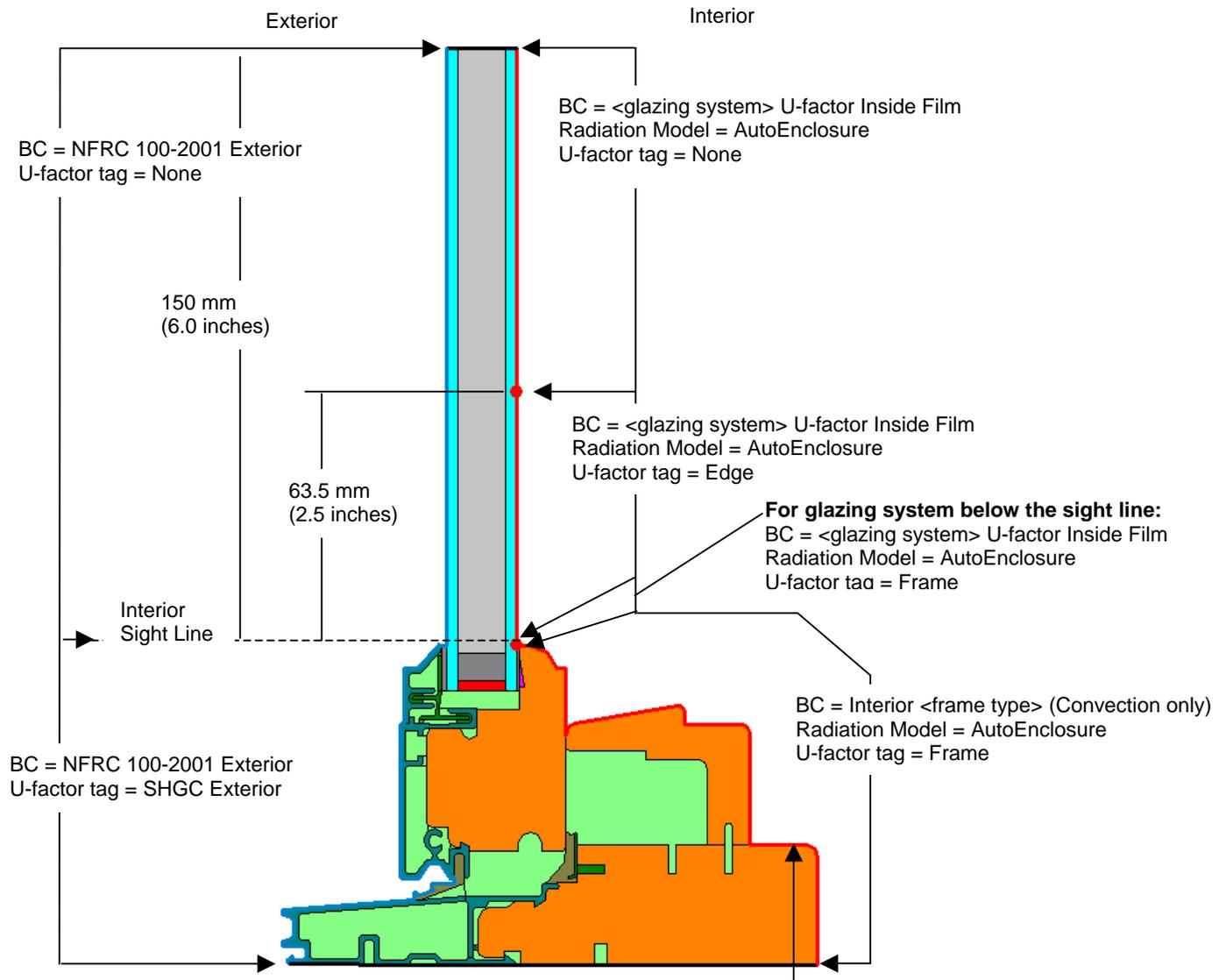
Figure 6-22. The dialog box displayed when boundary conditions are regenerated.

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 the *THERM User's 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 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 the enclosure geometry being drawn, and the convection-only film coefficients for these boundary conditions will be calculated and assigned automatically by the program

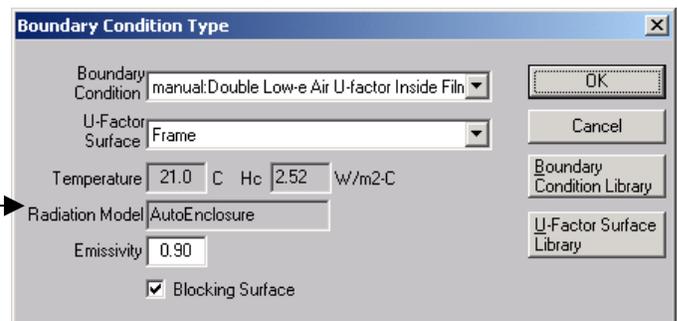


Figure 6-23. Standard boundary condition assignments for most models.

6.5.3. 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. See Section 6.3, "Error Detection in THERM" in the *THERM User's Manual*.

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 **View Voids and Overlaps** (from the **Draw** 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 Figure 6-20. In general, select the option **Automatically adjust points within tolerance option**, in which case THERM will fix all of the points automatically. This now 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, and if there are still "bad points", select the **Automatically adjust points within tolerance option**, in which case THERM will fix all of the points automatically. (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.)

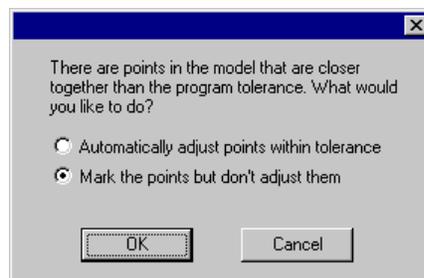


Figure 6-24. Message when trying to define boundary conditions indicating that there is a problem in the geometry.

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 all the adjacent polygons, 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 with bad points, 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.

6.6 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**.

See Chapter 7 in the *THERM User's Manual* for detailed information about the calculation procedures, and the results produced by THERM.

6.6.1. 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 the *THERM User's Manual*.

The one variable in the mesh generation that the user can control is the **Quadtree 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 NFRC required default value for the mesh parameter is 6, as shown in Table 6-4.

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 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.

6.6.2. 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 the *THERM User's 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 **Percent Error Energy Norm** in THERM is 10%, as shown in Figure 6-19. 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.

6.6.3. Required Settings for NFRC Simulations

In the **THERM File Options** tab found in the **Options/Preferences** menu, the following values must be set:

Table 6-4. Options/Preferences settings for THERM NFRC modeling

Setting	Value for NFRC Modeling
Quad Tree Mesh Parameter	6 or greater
Maximum % Error Energy Norm	10%
Run Error Estimator	must be checked

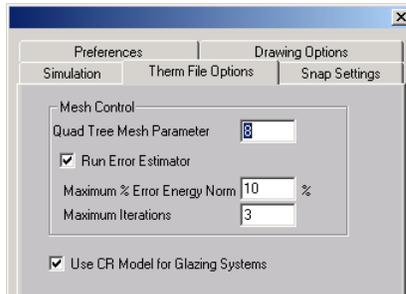


Figure 6-25. NFRC required THERM File Options settings.

In the **Simulation** tab found in the **Options/Preferences** menu, the Relaxation Parameter can be set to **1.0**, and the feature to “Automatically adjust relaxation parameter” should be set, as shown in the figure below.

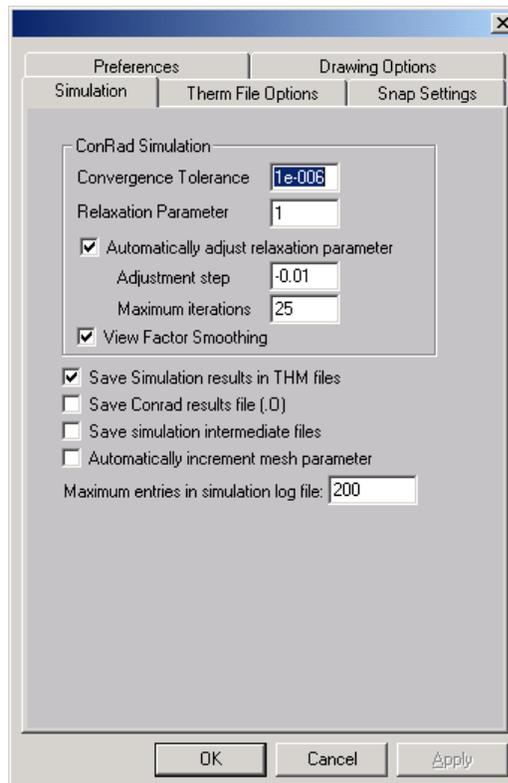


Figure 6-26. Simulation option settings.

6.6.4. Calculations

When a calculation is started, THERM can do the calculations in the background while another model is being worked on. There is also 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. Section 7.2, "Calculations," in the *THERM User's Manual* contains detailed information about the Calc Manager.

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 Figure 6-20:

- **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.

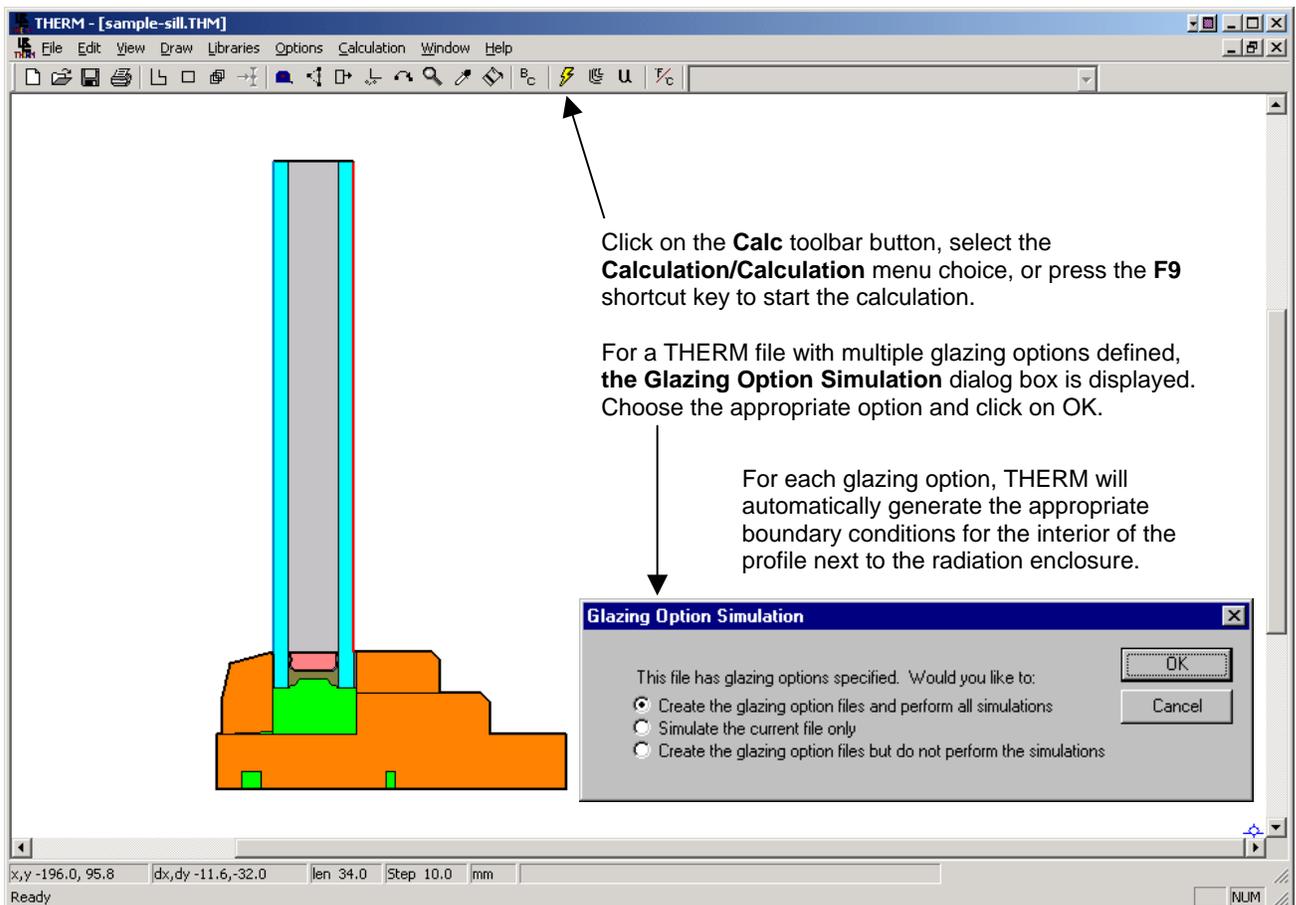


Figure 6-27. When a file with multiple glazing options is calculated, THERM gives several calculation options.

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.

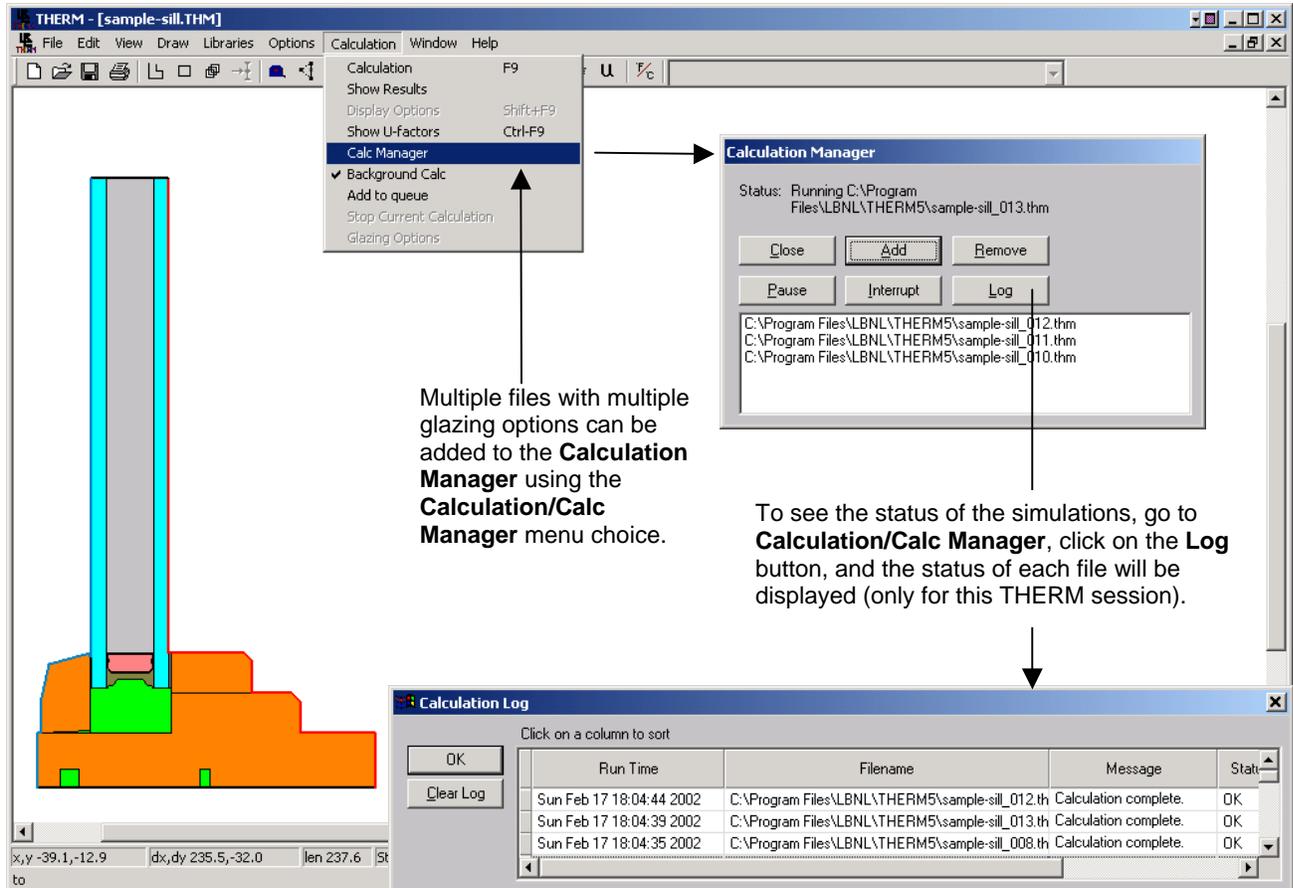


Figure 6-28. The **Calc Manager** can be used to simulate multiple files with multiple glazing options in each file.

6.6.5. 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. 

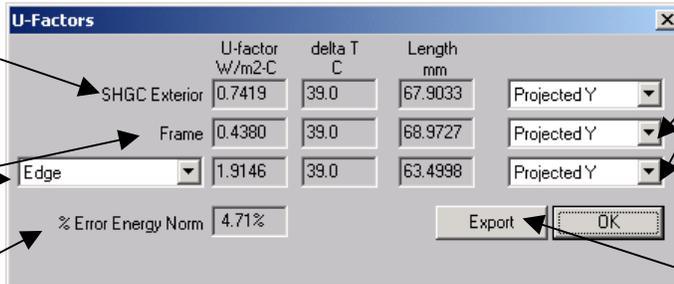
In addition, the graphic display of the results can be controlled using either the **Calculation/Show Results** menu choice or the **Show Results** toolbar button. 

The U-factors can be viewed for **Projected X**, **Projected Y**, **Total Surface Length**, or a **Custom Frame Length**. U-factors are generally calculated in the **Projected Y** direction, but there are a few special cases where the **Custom Frame Length** feature must be used for an NFRC simulation. (See the following discussion in *Section 6.6.6, "Custom Frame Length"* for an example using **Custom Frame Length**.)

The SHGC Exterior U-factor is meaningless – it is a result of the SHGC Exterior tag, which is used by WINDOW to calculate the frame exterior wetted length for the ISO 15099 SHGC value.

Frame and Edge U-factors are reported. These values will be used by WINDOW to calculate the total product U-factor.

% Error Energy Norm must be less than 10%.



	U-factor W/m ² -C	delta T C	Length mm	
SHGC Exterior	0.7419	39.0	67.9033	Projected Y
Frame	0.4380	39.0	68.9727	Projected Y
Edge	1.9146	39.0	63.4998	Projected Y
% Error Energy Norm	4.71%			

U-factors are generally calculated in the Projected Y dimension.

Use the Export button to generate a comma delimited file with the results.

Figure 6-29. U-factor results from the **Calculation/Show U-factors** menu choice.

Local temperatures can be displayed at the cursor (from the **View/Temperature at Cursor** menu choice) and 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.

THERM files (*.THM) can be saved without the detailed temperature data by selecting the **Options/Preferences** menu, going to the **Simulation** tab, and *unchecking* **Save Simulation results in THM files**. The THERM file will then contain only the geometry and boundary conditions used to run the simulation, the U-factor results, and the report, thus making the file much smaller.

6.6.6. 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.

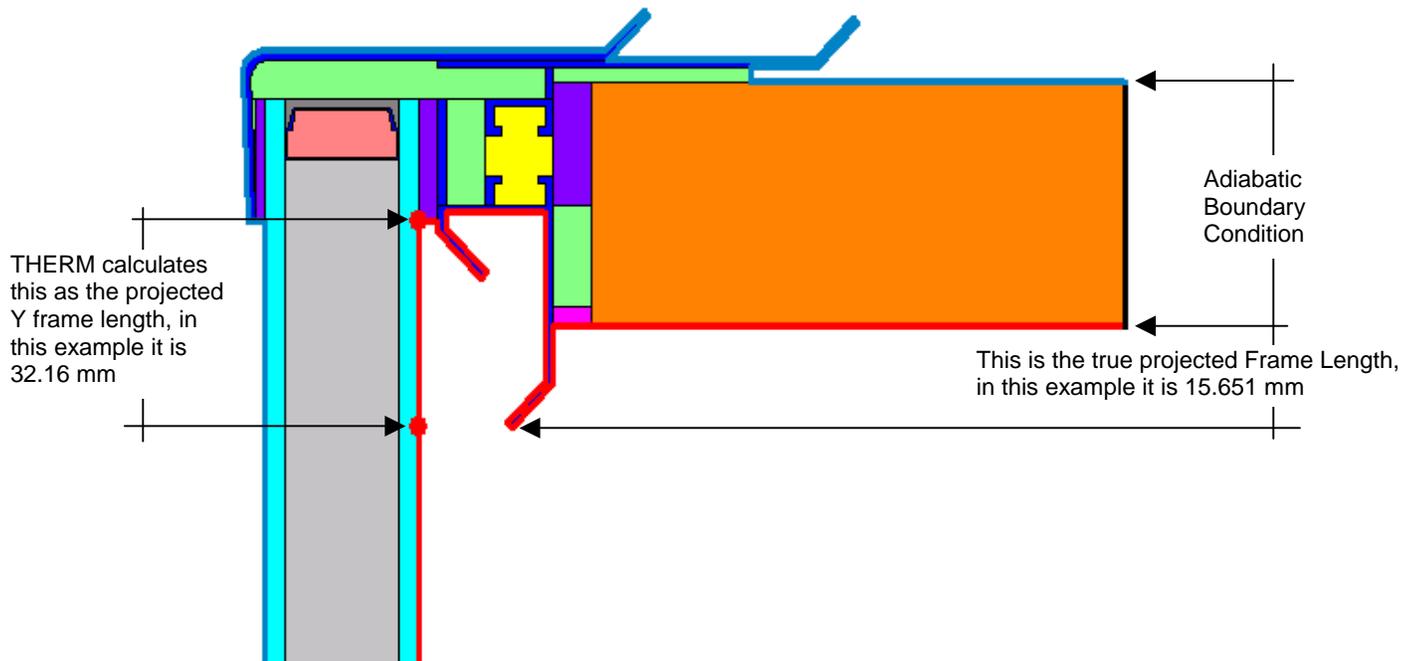


Figure 6-30 Example of a case where THERM does not calculate the projected Y frame length correctly because of an adiabatic boundary condition that "overlaps" an interior boundary condition.

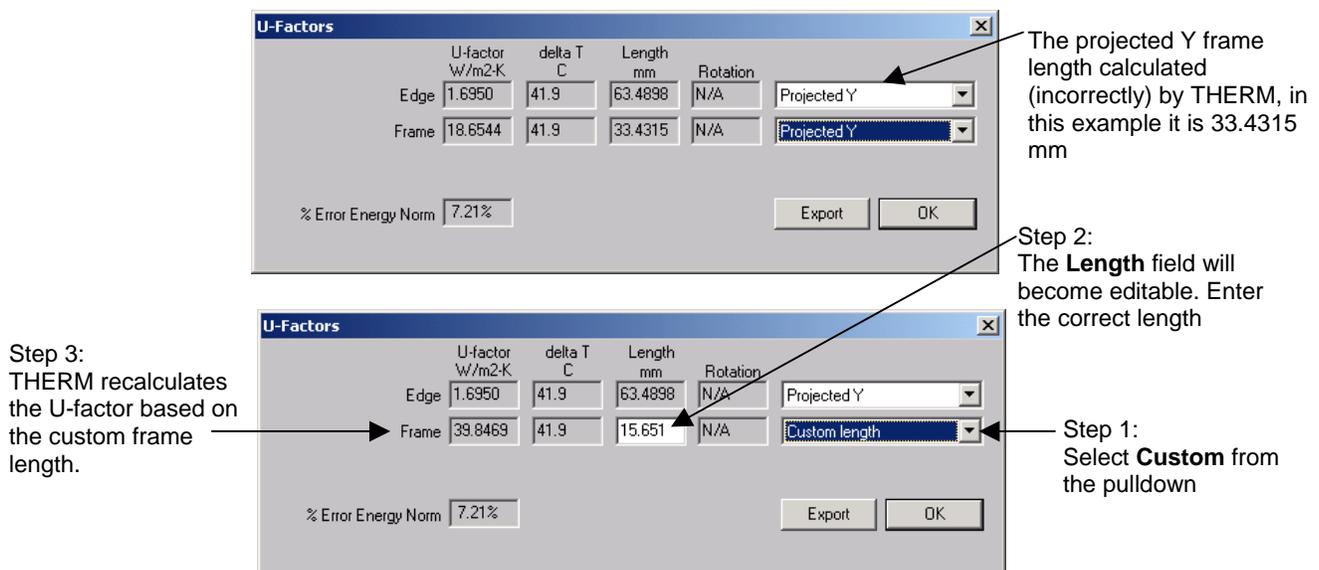


Figure 6-31 The Custom Frame Length feature allows the correct frame length to be entered.

6.6.7. 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 *WINDOW User's Manual*, Section 4.7.3, "Importing THERM Files". 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 frame **Absorptivity** to 0.30 or 0.50, according to the NFRC 200 specifications.

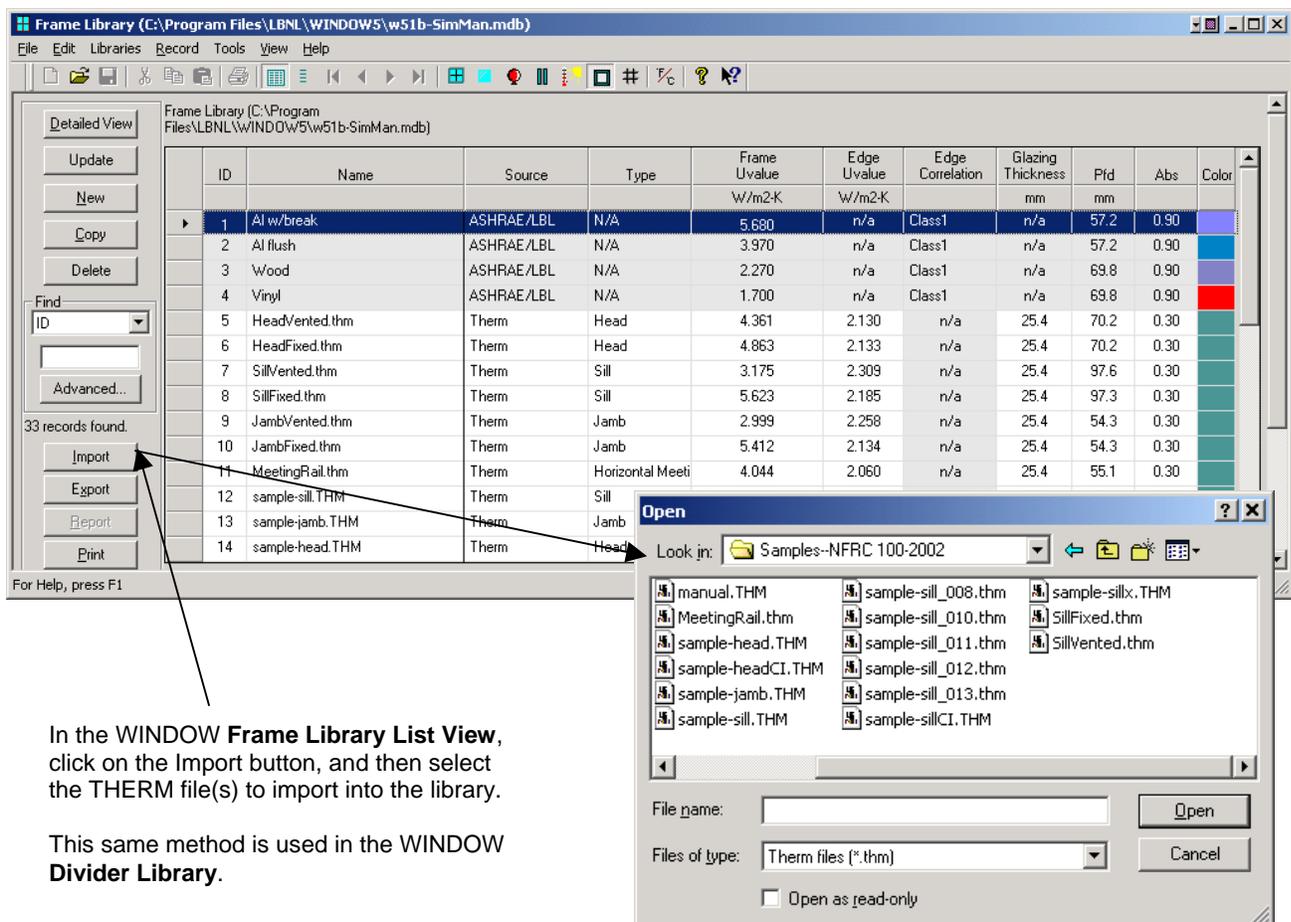


Figure 6-32 Importing THERM files into the WINDOW **Frame Library**.