

7. CALCULATING RESULTS

7.1. Overview of THERM Calculation Methods

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

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

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

7.2. Calculations

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

7.2.1. Calculation Menu

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

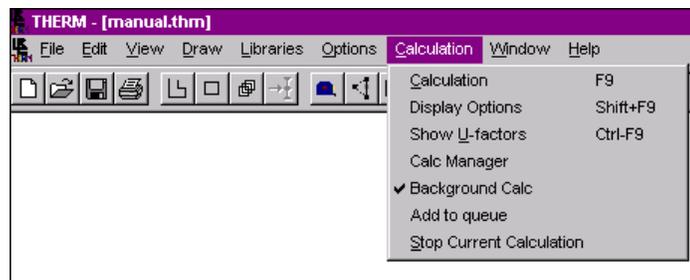


Figure 7-1. The **Calculation** menu choices

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

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

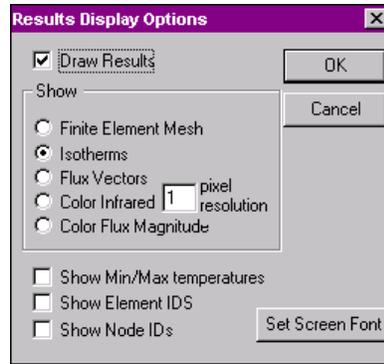


Figure 7-2. The **Results Display Options** dialog box

Show U-factors
(**Ctrl-F9**)

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

Calc Manager

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

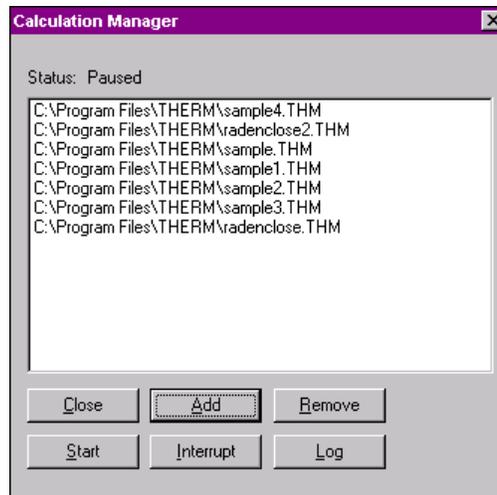


Figure 7-3. Selecting the **Calculation/Calc Manager** menu choice opens the **Calculation Manager** dialog box which shows a list of THERM files that can be run.

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

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

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

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

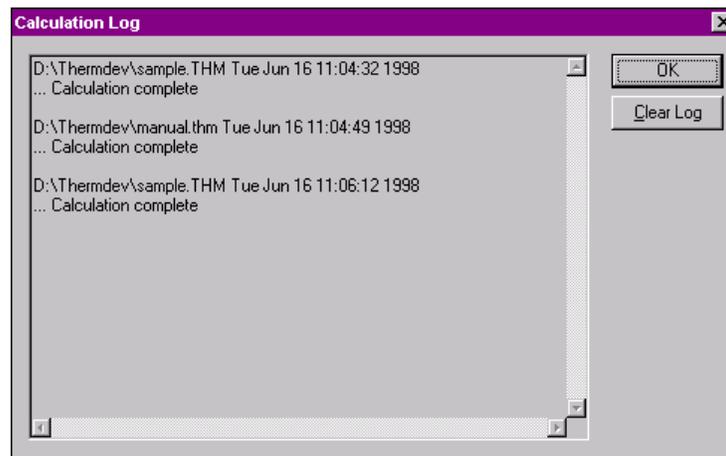


Figure 7-4. Calculation Log accessed from Calc/Calc Manager.

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

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

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

7.2.2. Calculating Results for a THERM file

To start calculating a THERM file, do the following:

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

2. Turn on **Background Calc** from the **Calculation** menu. When it is on, a check mark will appear to the left of the **Calculation/Background Calc** menu choice.
3. Press the **Calculation** toolbar button, , use the **Calculation/Calculation** menu item, or press **F9** to start the calculation.
4. The program will show the calculation steps being performed (because **Background Calc** is turned on). Depending on the complexity of the cross section and the fineness of the mesh as well as the CPU speed and amount of memory of your computer, calculation steps can take several seconds to a few minutes.

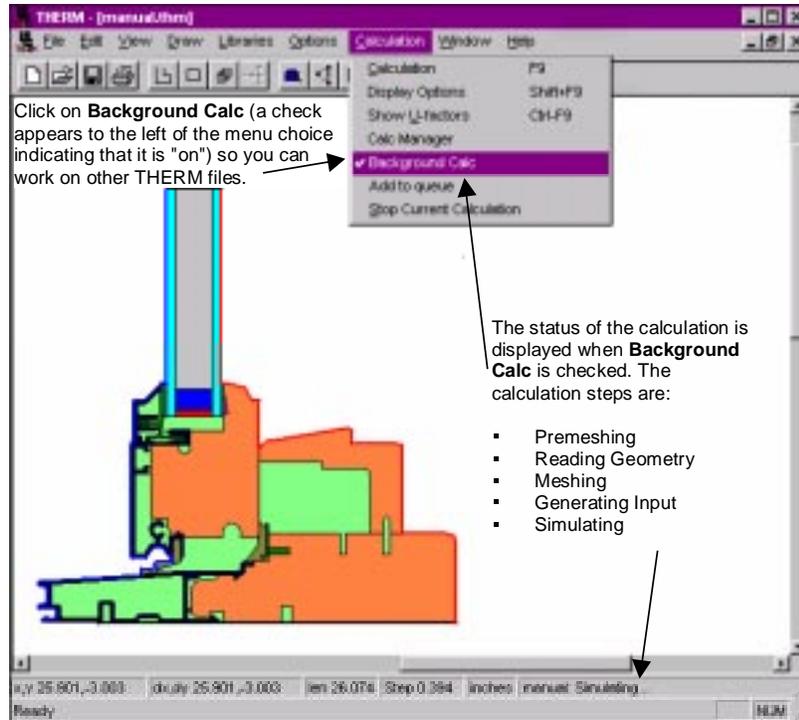


Figure 7-5. THERM displays the steps in the calculation when the **Calculation/Background Calc** menu option is on.

7.2.3. Calculation Options

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

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

*Quad Tree Mesh
Parameter*

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

Run Error Estimator

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

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

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

Default: checked.

Maximum % Error Energy Norm

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

Maximum Iterations

This box specifies the maximum number of times the program will modify the mesh and resimulate if the **Run Error Estimator** box is checked. THERM will stop simulating and

display a warning message when this value is exceeded even if the target value for the percent error energy norm has not been reached. The last set of results will be displayed. **Default:** 3.

Use CI Model for Window4 Glazing Systems

A check box indicates that you want to use the **Condensation Index Model** when WINDOW4 glazing systems are imported. **Default:** unchecked

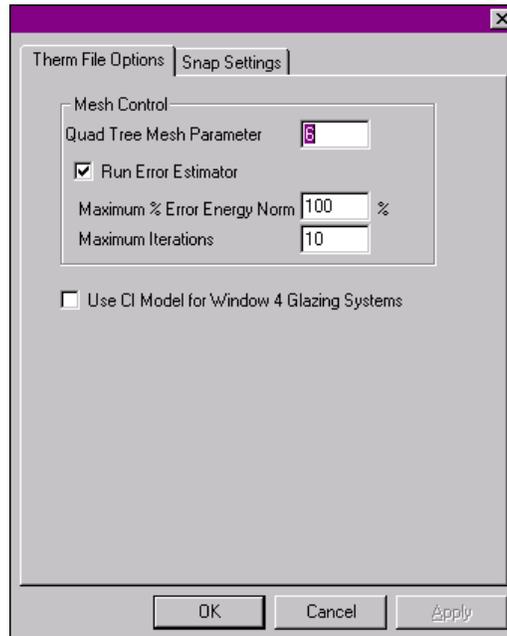


Figure 7-6. **THERM File Options** affect how THERM calculates results.

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

Convergence

Tolerance

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

Relaxation

Parameter

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

View-Factor

Smoothing

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

Save Simulation

Results in THM

Files

This check box determines how the results are saved. You can save results with the THERM file, which creates a large file containing all of the mesh, temperature, and U-factor results for a cross section. If this box is not checked, the files are much smaller (1K) and only contains the U-factor results; you will be able to view the geometry of the

cross section, the U-factors, and the report but not the mesh or temperature results. In this case, the results are current but not available and the **Calc/Display Options** menu choice will be inactive (grayed out). **Default:** checked

*Save CONRAD
Results file (.O)*

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

*Save simulation
Intermediate files*

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

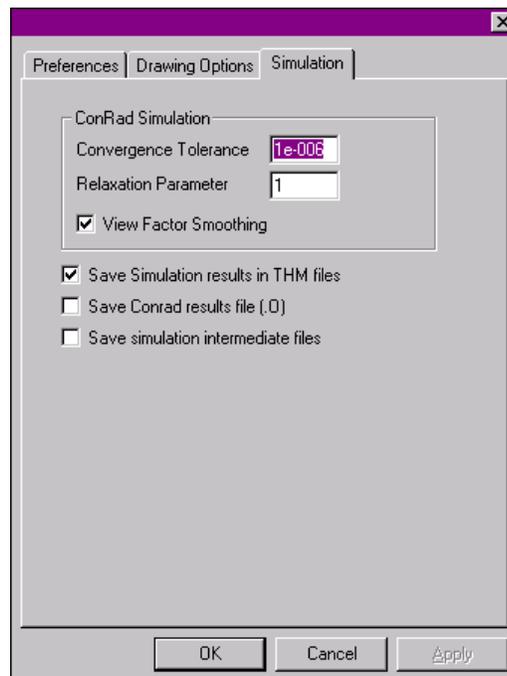


Figure 7-7. The **Simulation** preferences accessed from the **Options/Preferences** menu

7.3. Feedback and Troubleshooting During the Calculation Process

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

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

- Error Estimator (optional)

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

7.3.1. Premeshing

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

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

- (1) When very fine details are collapsed. This often happens with acute angles where one of the end points is less than 0.01 mm away from a line. If this is the case, another point will be added to the line when the boundary conditions are assigned and the two points will then be merged. The acute angle will be turned into a zero area extension and with the two points lying on top of one another.
- (2) When the **Automatically adjust points within tolerance** option is chosen during the generation of boundary conditions. In this case, points within the checking tolerance (a distance of 0.1 mm) are merged automatically by the program and the possibility of creating an invalid polygon is very high. This is why it is recommended that you do not automatically adjust bad points without first inspecting them.

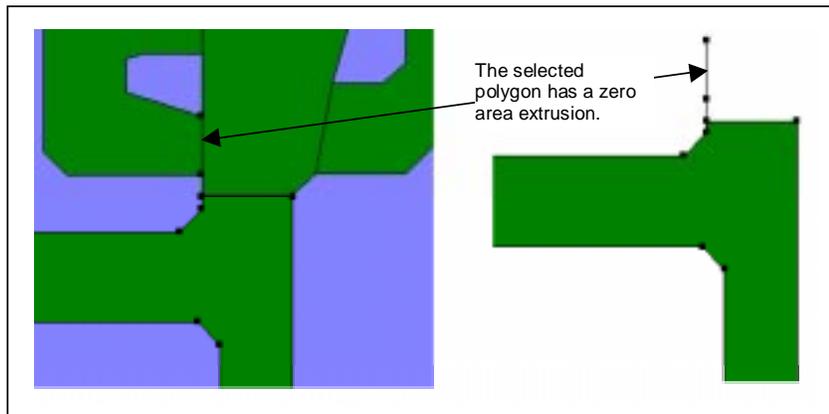


Figure 7-8. An example of a zero area extrusion which will result in an error message of "Invalid Polygon found, Stopping Simulation".

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

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

7.3.2. Reading Geometry

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

7.3.3. Meshing

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

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

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

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

7.3.4. Generating Input

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

7.3.5. Simulating

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

7.3.6. View Factors (optional)

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

7.3.7. Simulating (optional)

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

7.3.8. Preparing Input (optional)

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

7.3.9. Error Estimator (optional)

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

7.4. Viewing the results

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

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

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

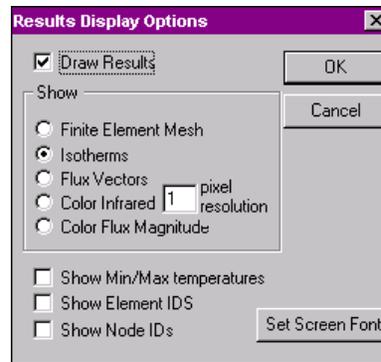


Figure 7-9. The **Results Display Options** dialog box allows you to choose the results that are displayed.

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

- | | |
|----------------------------------|---|
| <i>Draw Results</i> | This box must be checked in order for the graphic results to be drawn. Default: checked. |
| <i>Show</i> | The Show section has the following choices, only one of which can be active at a time. The setting will determine what graphic results are drawn by the program when you click the OK button, as long as the Draw Results option is checked. These choices are discussed in detail later in Section 7.4.5, "Graphic Results." <ul style="list-style-type: none"> ▪ <i>Finite Element Mesh</i> ▪ <i>Isotherms</i> ▪ <i>Flux Vectors</i> ▪ <i>Color Infrared:</i> Drawing these results may take several minutes. You can press the Esc key if you want to cancel the drawing of these results. <ul style="list-style-type: none"> ▪ <i>Pixel resolution:</i> this value controls how finely the color infrared is drawn. If you choose one pixel, THERM draws the image one pixel at a time, which can be slow but gives the smoothest image. If you choose a higher number, the program draws in blocks of pixels; the number you have chosen determines how many pixels make up the sides of the block. The image is drawn faster than using the one-pixel setting but does not appear as smooth. ▪ <i>Color Flux Magnitude:</i> Drawing these results may take several minutes. You can press the Esc key if you want to cancel the drawing of these results. |
| <i>Show Min/Max temperatures</i> | Check this box in order to display the minimum and maximum temperatures in the simulation. This is discussed in more detail in the <i>Min/Max Temperature</i> section. |
| <i>Show Element Ids</i> | Check this box to see the numerical ID values that have been assigned to each mesh element. This is primarily a research feature. Default: unchecked. |
| <i>Show Node Ids</i> | Check this box to see the numerical ID values that have been assigned to each mesh node. This is primarily a research feature. Default: unchecked. |
| <i>Set Screen Font</i> | Click this button to access a Font dialog box where you can select a font for the THERM graphic display from the list of fonts that are installed on your computer. This option can be useful if, for example, the labels of the isotherms do not display well on your monitor; changing the font may make them more readable. |

7.4.1. U-factors

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

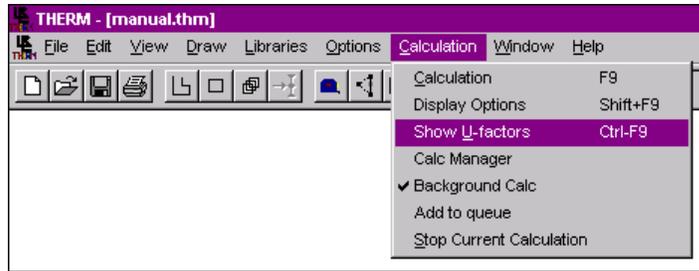


Figure 7-10. Select the **Calculation/Show U-factors** menu to see the U-Factor results.

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

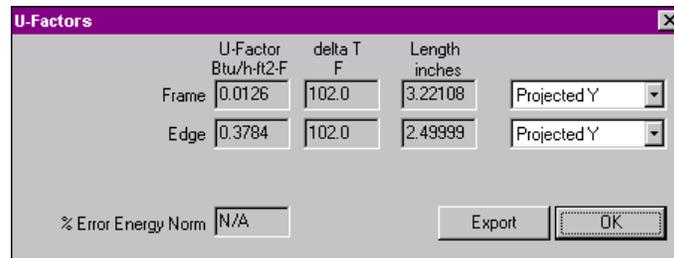


Figure 7-11. The U-factor results dialog box

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

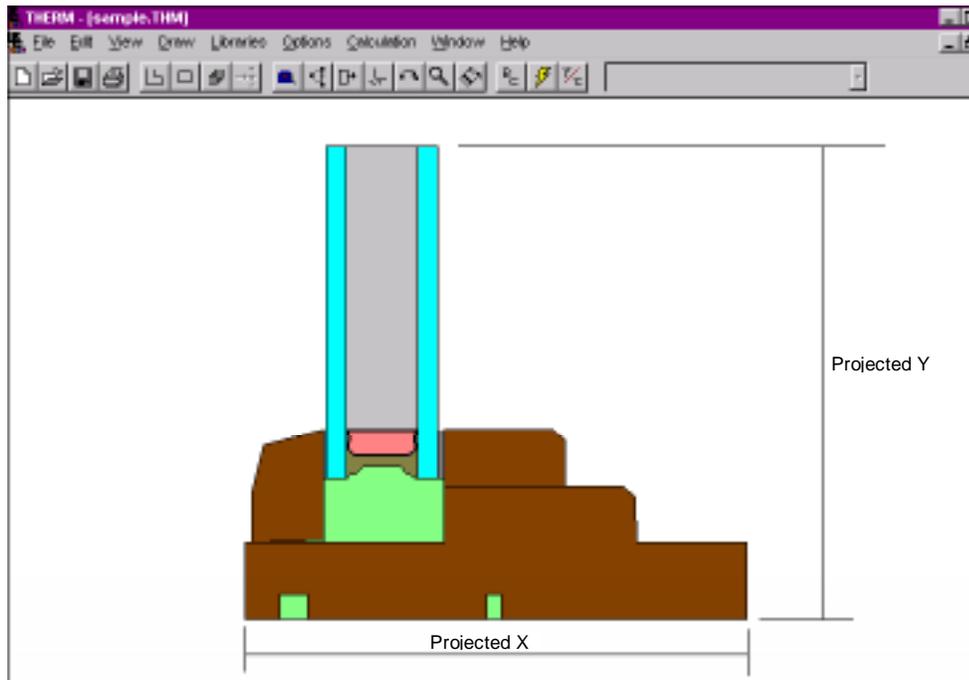


Figure 7-12. Projected X and Projected Y

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

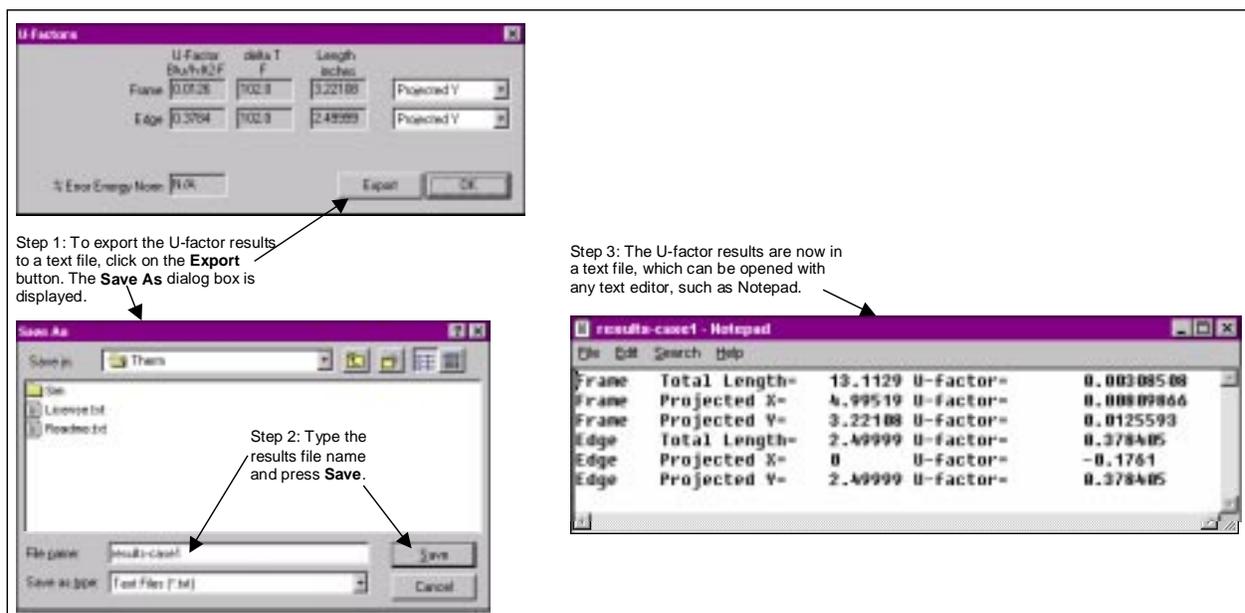


Figure 7-13. You can export the U-factor results to a text file using the **Export** button on the **U-factors** dialog box.

7.4.2. Min/Max Temperature

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

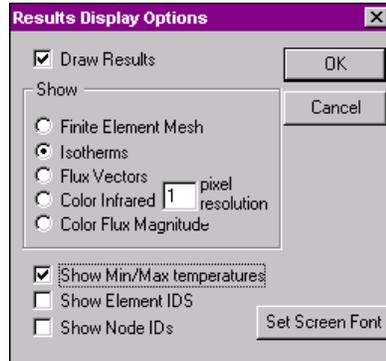


Figure 7-14. Check the **Show Min/Max temperatures** box in **Results Display Options** dialog box.

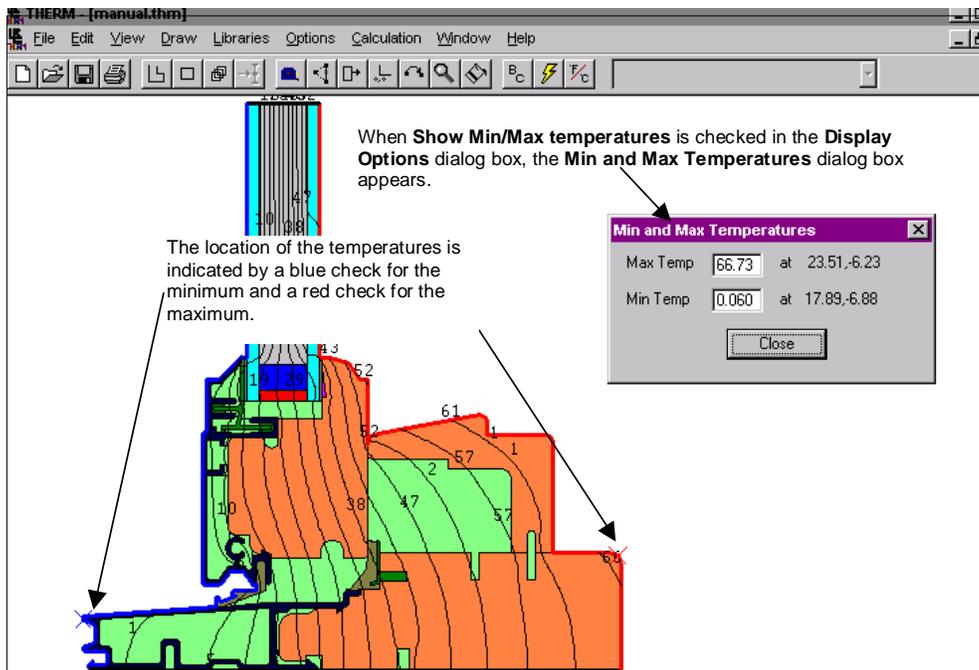


Figure 7-15. Display of minimum and maximum temperatures

7.4.3. Temperature at Cursor

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

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

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

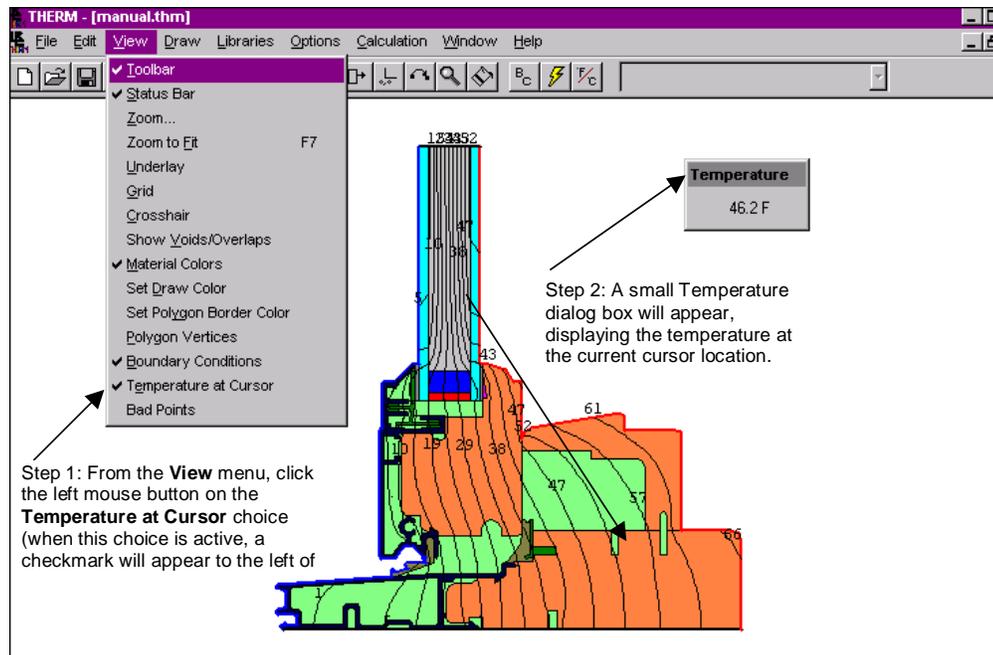


Figure 7-16. Display of the temperature at the cursor location

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

7.4.4. Average Temperature with Tape Measure

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

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

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

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

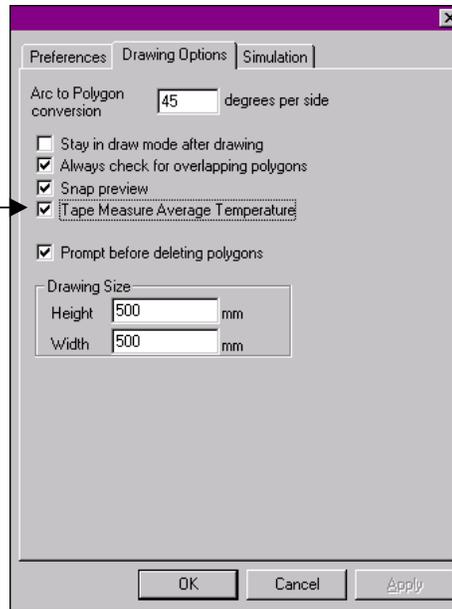


Figure 7-17. Turn on the **Tape Measure Average Temperature** feature from the **Options/Preferences** menu in order to display the temperature at a specific point.

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

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

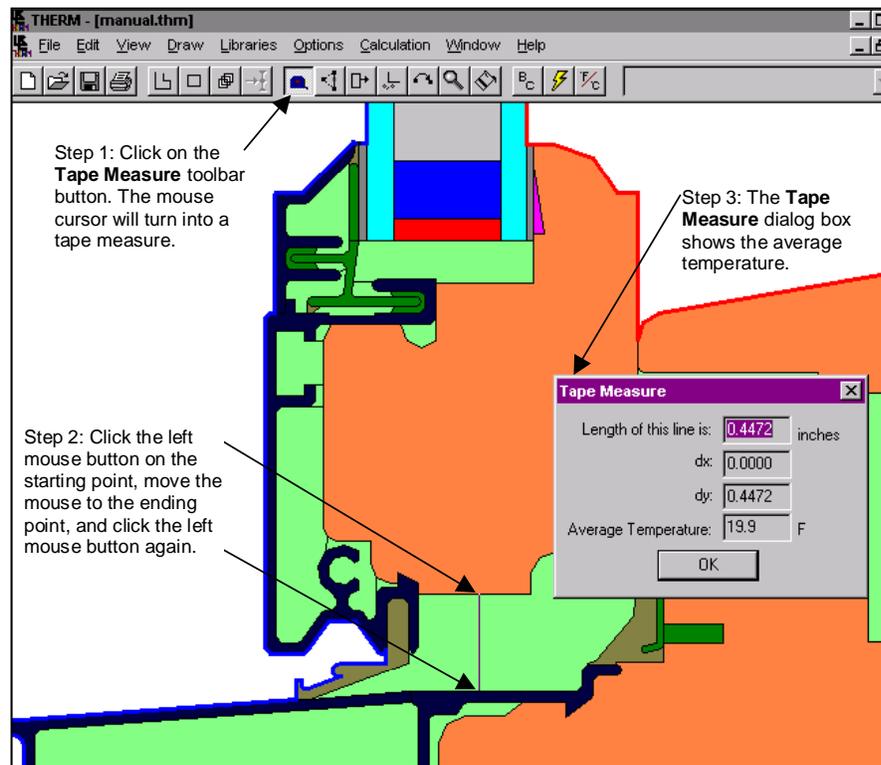


Figure 7-18. Use the Tape Measure to determine the average temperature across a particular surface.

7.4.5. Graphic Results

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

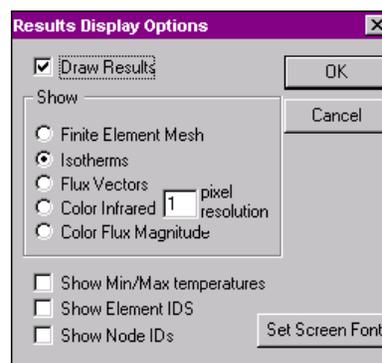


Figure 7-19. Graphic results display options

7.4.6. Finite-Element Mesh

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

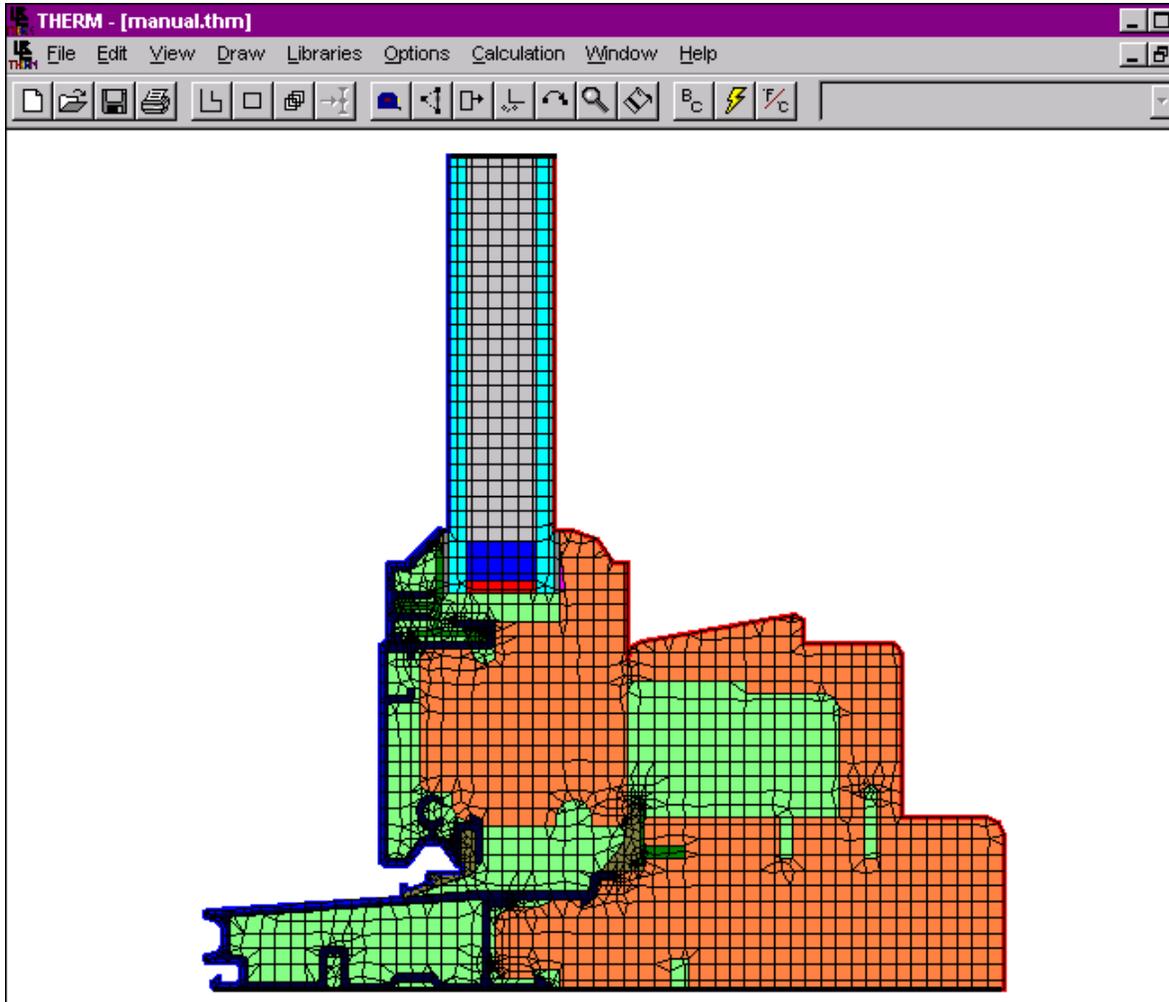


Figure 7-20. Graphic results: Finite-element mesh

7.4.7. Isotherms

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

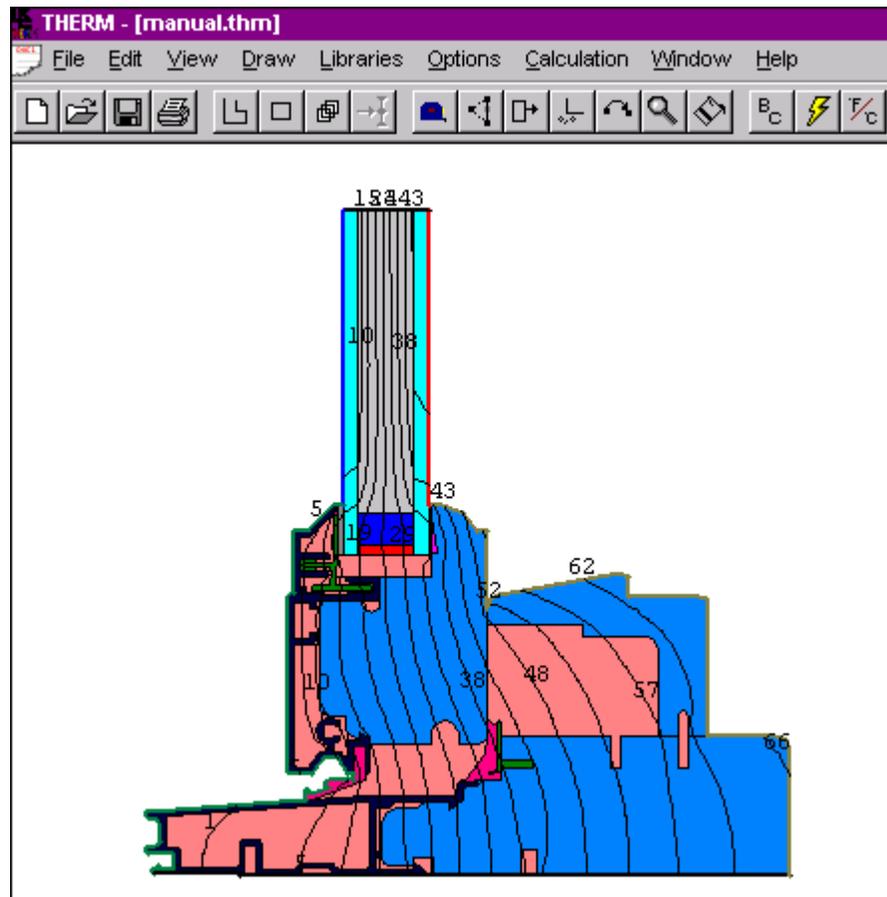


Figure 7-21. Graphic results: Isotherms

7.4.8. Flux Vectors

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

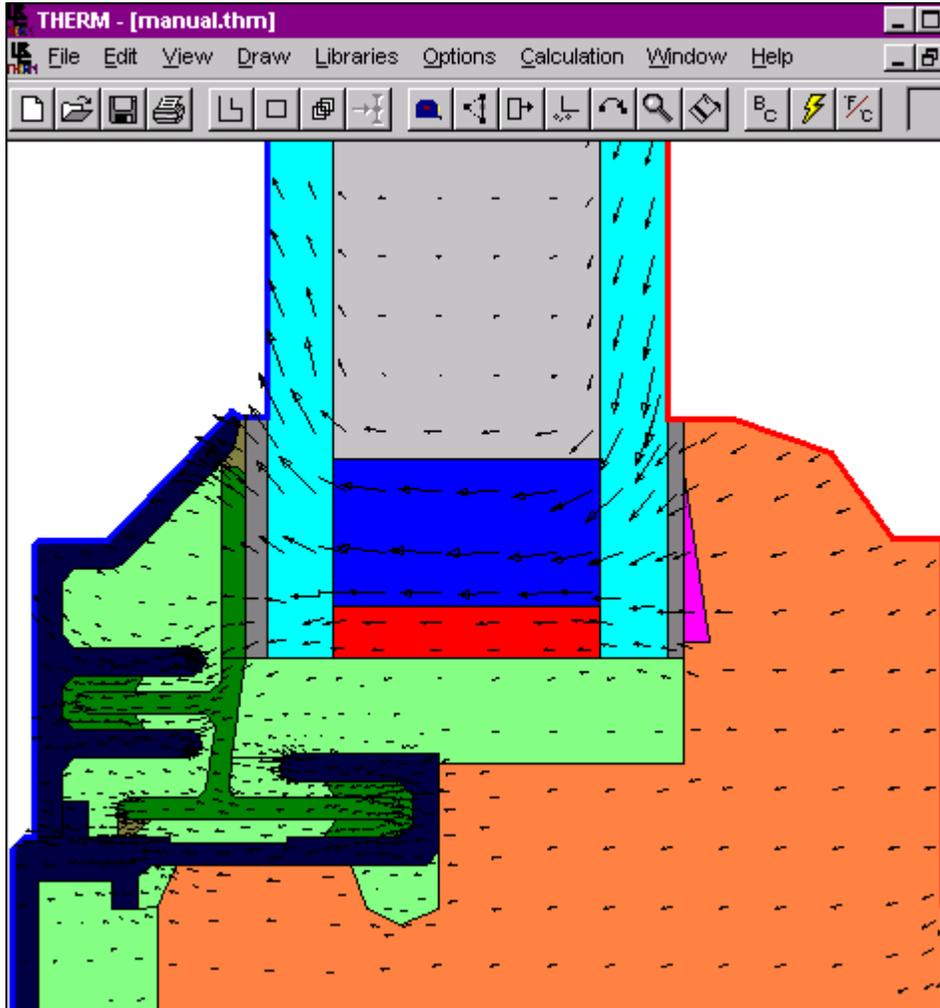


Figure 7-22. Graphic results: Flux vectors

7.4.9. Color Infrared

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

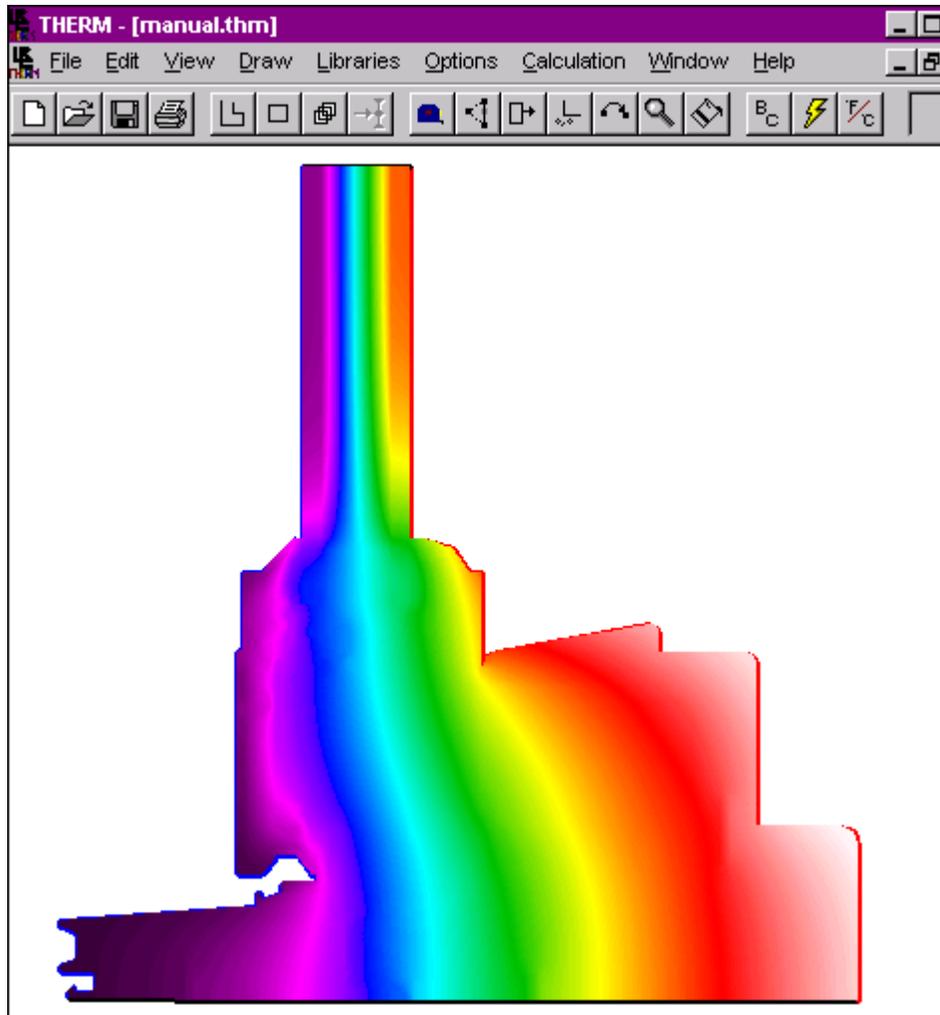


Figure 7-23. Graphic results: color infrared

7.4.10. Color Flux Magnitude

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

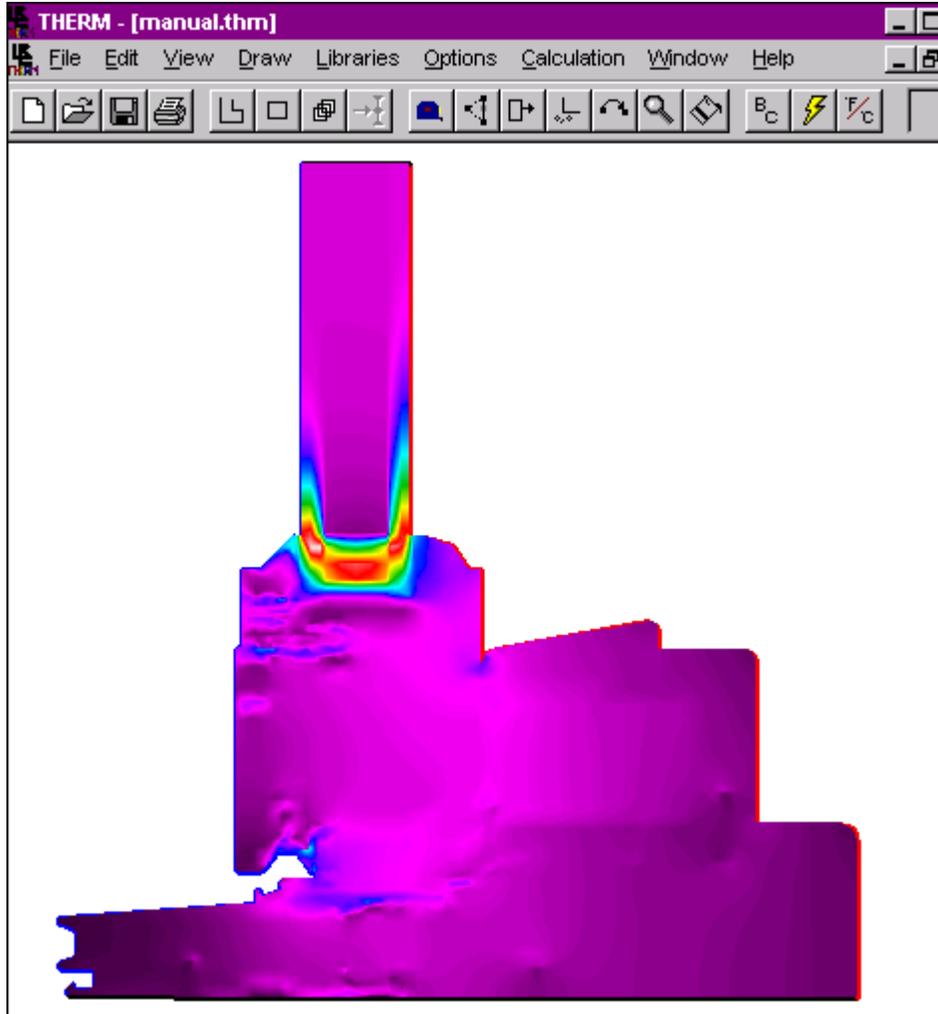


Figure 7-24. Graphic results: Color flux magnitude

7.4.11. Report

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

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

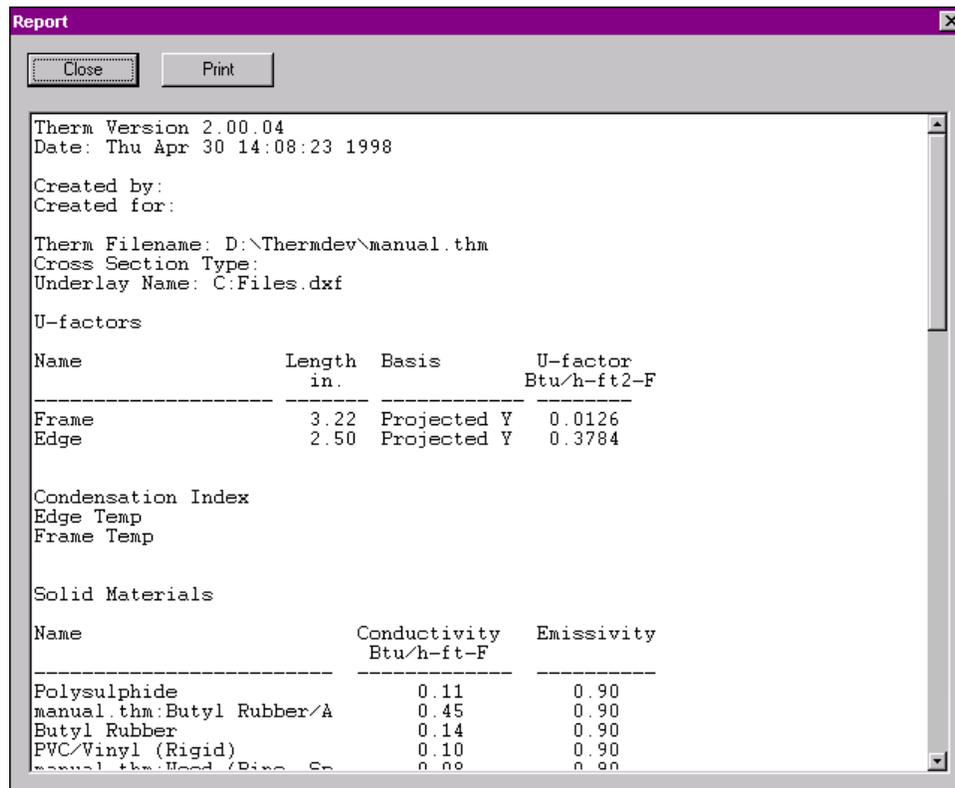


Figure 7-25. You can display the THERM report by selecting the **File** menu and **Report** menu choice.

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

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

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

- Divider
- Head
- Jamb
- Meeting Rail
- Sill

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

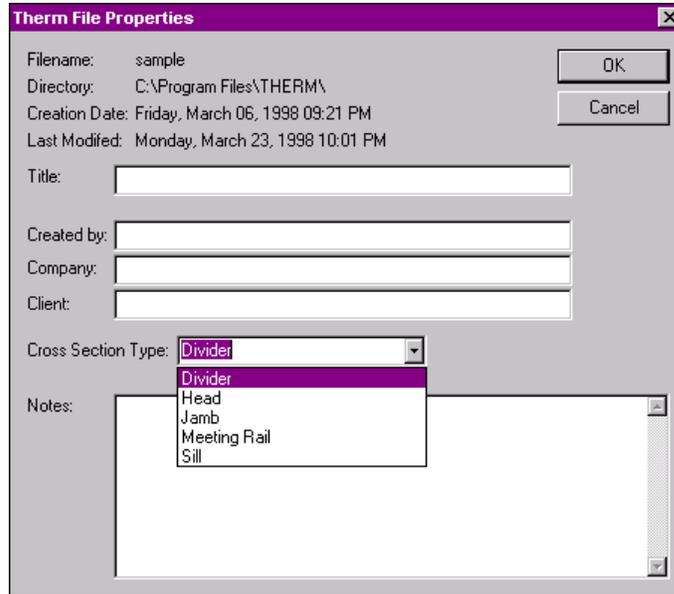


Figure 7-26. The File menu, Properties choice allows you to input information about the THERM file for the report.

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

Created by:
Created for:

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

U-factors

Name	Length in.	Basis	U-factor Btu/h-ft ² -F
Frame	3.22	Projected Y	0.0126
Edge	2.50	Projected Y	0.3784

Condensation Index
Edge Temp
Frame Temp

Solid Materials

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

Cavities

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

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

Glazing Systems

Name	COG U-factor Btu/h-ft ² -F	Overall Thickness in.	Cavity Height in.
Dbl, low-e, Ar	0.30	0.75	39.37

Standard Boundary Conditions

Name	Temperature F	Film Coefficient Btu/h-ft ² -F
Exterior Surface	-0.00	5.112
Interior Wood/Vinyl	70.00	1.340
Dbl, low-e, Ar U-fact	70.00	1.353
Dbl, low-e, Ar U-fact	0.01	5.050

Calculation Specifications

Mesh Parameter: 7
Estimated Error: Not Calculated

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

7.4.12. Export to WINDOW

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

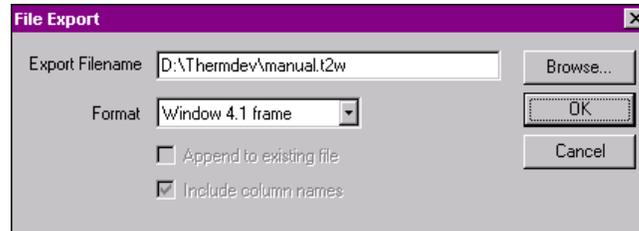


Figure 7-28. Exporting a THERM file to a WINDOW file format

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