20 Tutorials

This chapter presents information and sample problems to explore some of Thermal Desktop's features. Before beginning the tutorials, AutoCAD®, Thermal Desktop, and SINDA/FLUINT must be installed. Refer to the AutoCAD and Thermal Desktop installation instructions to complete these installations if the software is not already installed.

Tutorial files may be found in the \Tutorials\Thermal Desktop subdirectory which is located in the Thermal Desktop installation directory (usually C:\Program Files\Cullimore And Ring\Thermal Desktop). The \Tutorials\Thermal Desktop subdirectory includes additional subdirectories, one for each of the tutorials covered in the subsequent sections of this chapter. For some of the tutorials, a completed tutorial file is included in the \completed directory.

It is recommended that the \Tutorials\Thermal Desktop directory be *copied* to the user's own working area before beginning the tutorials. This ensures a copy of the original tutorial files will be available for use by other users at a later time.

The tutorials attempt to introduce a variety of concepts. Most often there is more than one way to accomplish the "task" of a tutorial. Different techniques are shown in an attempt to expose the user to the variety of features present in Thermal Desktop. In particular, there are many ways of selecting objects, and both noun-verb (object selection followed by a command) and verb-noun (a command followed by object selection) order is supported for most commands. Identical tasks may be performed differently to illustrate a particular feature.

It is also strongly suggested that each new user take the time to work through each of the tutorials (in this chapter as well as the other tutorial chapters: 21 RadCAD® Tutorials and 22 FloCAD® Tutorials). Although completing the tutorials requires time, doing so will provide the user with enough knowledge and skills to allow the user to quickly become effective and efficient while working in Thermal Desktop.

There are eleven tutorials in this chapter, as follows:

- 20.1 Getting Started
- 20.2 Setting Up a Template Drawing
- 20.3 Model Browser Example
- 20.4 Circuit Board Conduction Example
- 20.5 Beer Can Example
- 20.6 Conduction and Radiation Using Finite Elements
- 20.7 Mapping Temperatures From a Coarse Thermal Model to a Detailed NASTRAN Model
- 20.8 Contactor Example
- 20.9 Parameterizing for a Common Input
- 20.10 Dynamic SINDA Example

Functionality in the tutorials reflects the most recent version of Thermal Desktop and AutoCAD. Every effort has been made to present the user with the exact system prompts, messages and commands, and dialog boxes the user will see when using Thermal Desktop. Please note that occasionally minor changes may be made between application releases and, as such, small changes to wording in such items may be experienced. The "how to" functionality will not change.

Most of the tutorials are written in a two-column format. The left column provides specific instructions and the right column provides a general description of the steps, provides example views of the graphics area, and sometimes provides background information and alternative methods for accomplishing the steps. Reading through the information in the right column before completing the steps on the left will be much more beneficial to the user than autonomously completing the steps in the left column.

20.2 Setting Up a Template Drawing

Prerequisites:

- 20.1.1 Starting AutoCAD for the First Time

A *template* is something that can be used as a pattern for future projects. In Thermal Desktop, a *template drawing file* can be created and used as a foundation for creating new Thermal Desktop drawings. When creating a template it is important to consider how the template will be used, if the template will be used by only one user or by more than one, what preferences should be associated with the template and other such considerations.

In this tutorial, a template drawing file (file extension DWG) will be created. The template will be used in other tutorials and may be used as a permanent template for use in future thermal analysis tasks. Creating and then using a template drawing starts a new analysis with an environment set to pre-determined preferences, helping to keep drawings consistent and easy to use.

Before beginning a new thermal analysis task, **copy the template** to a directory chosen to store the work for that particular task or project. Once a copy is made, **rename the copied template** drawing to a meaningful name related to the thermal task or project. Then start Thermal Desktop by double-clicking on the renamed drawing file. Thermal Desktop will start with the preferences saved in the template, and the current working directory will be set to the directory which contains the drawing file. All database files created by Thermal Desktop for view factors, radiation conductors, and orbital heating rates will be placed in this working directory.

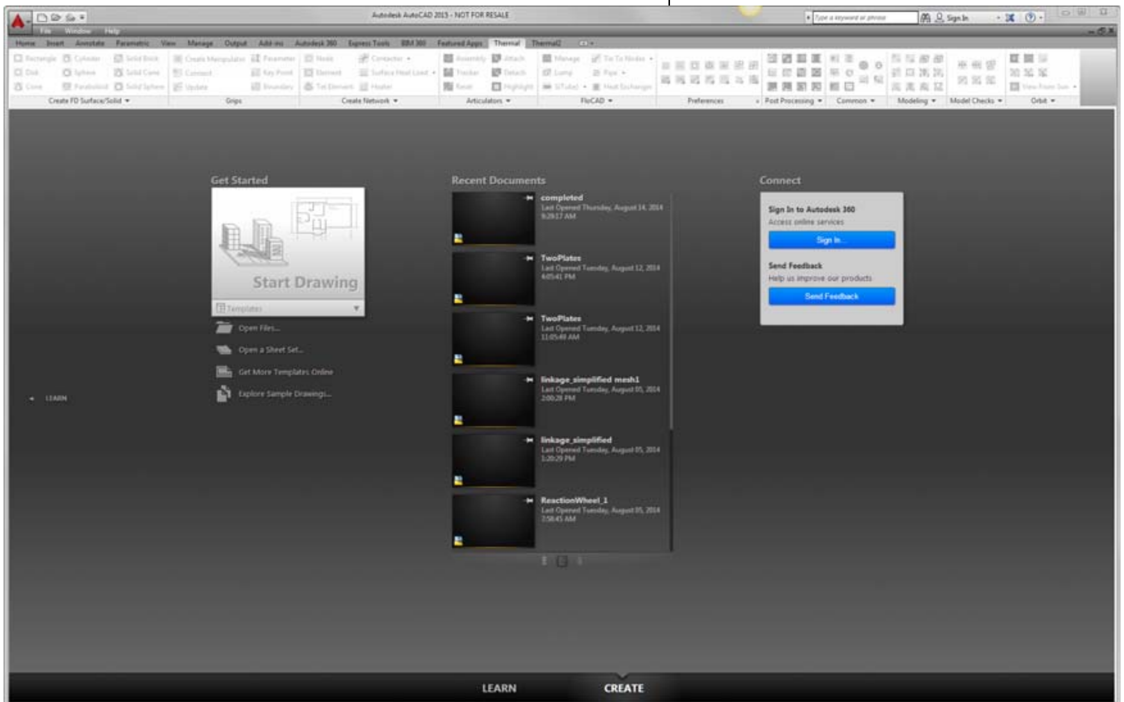
> Important: If Thermal Desktop is not launched using the drawing file for the task and instead is started from the AutoCAD Icon, the current working directory will be the user's Documents directory. This is not recommended, since multiple thermal analysis tasks may overwrite each other's database files.

The following exercise creates a template file with preferences set to be convenient for working with Thermal Desktop. The user is encouraged to experiment with the available options as a means to discover which settings are compatible to each individual's work style.

Setting Up a Template Thermal Desktop Drawing File
Important: The process described below should only be used when creating a template. In future tutorials or modeling when an existing model is not available, a template drawing file should be copied to the desired folder, renamed and the file icon double-clicked to start Thermal Desktop.
1. From the Windows desktop go to **Start** > **All Programs** and look for AutoDesk. Follow that path to the version of AutoCAD installed, and then follow that to the executable (i.e. **Start** > **All Programs** > **AutoDesk** > **AutoCAD 20xx - English** > **AutoCAD 20xx - English**).
Important: If it is not possible to complete the Getting Started drawing (drawing1.dwg) in the same sitting, exit Thermal Desktop (File > Exit) and respond No when prompted to save the drawing. To save the drawing, perform a Save As and change to a directory other than the initial system installation directory.

Setting Up a Template Thermal Desktop Drawing File

The AutoCAD window appears.



2. If the window looks like the image above, select **Start Drawing** under **Getting Started** to the upper left.

   The window changes to a grid and the toolbars and ribbons become active.

3. **View** > **Visual Styles** > **2D Wireframe** to be consistent with the images in this tutorial.

4. Select **File** > **Save As**.
5. Change the name of Drawing1.dwg to **thermal.dwg** in the **\Tutorials\Thermal Desktop\template** directory.
6. Select **Save**.

   The drawing area title bar is updated with the new name of the drawing, thermal.

Be sure to save the files in your copy of the tutorials directory.
This save occurs before any Thermal Desktop commands are issued, therefore, at this point the file is strictly an AutoCAD file.

Note: If your models may be opened with older versions of AutoCAD, choose the version predates the oldest version that may be used. Thermal Desktop 6.1 works with AutoCAD versions 2015 through 2019.

Setting Up a Template Thermal Desktop Drawing File

7. Select **Thermal** > **Utilities** > **Reset Thermal Desktop Interface**

8. In the **Thermal Desktop User Interface** dialog, verify that **All Ribbons**, **Standard Toolbar**, **AutoCAD Menus**, and **Ribbon/ Thermal Desktop Forms Reduction** are checked and other boxes are cleared.

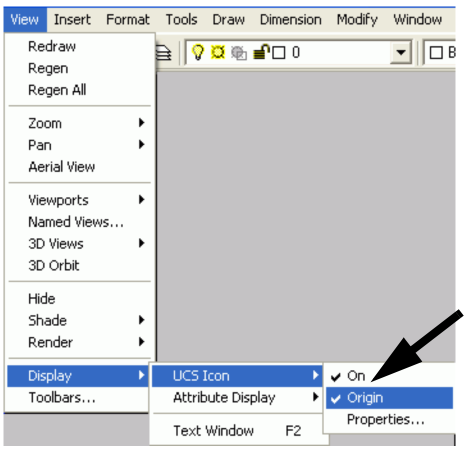
The Thermal Desktop splash screen will be displayed when the first Thermal Desktop command is issued.
If the model had contained Thermal Desktop objects, the splash screen would have appeared upon opening the file.

9. Select **View** > **3D Views** > **SW Isometric**.

   Note the UCS icon changes to reflect the view.

Selecting **View** > **3D Views** > **SW Isometric** here sets an isometric view parameter for the new template.

10. View the menu selection **View** > **Display** > **UCS Icon**> **Origin**.



11. Confirm a **check mark is displayed** next to **Origin**. If a check mark is not visible, click **Origin** to select the option. Otherwise click anywhere in the Thermal Desktop window or press <**ESC**>.

This step locates the User Coordinate System (UCS) display icon at the origin of the model, rather than in the lower left corner of the screen.
If the UCS origin is not in the display area, then the UCS will be displayed in the lower left corner.

Note: Depending upon the experience and preferences of the user, before saving the template, the toolbars on the main window may be rearranged. The default arrangement of toolbars reduces the horizontal size of the viewport. The toolbars can be moved to the sides of the viewport to produce a viewport with a more square aspect ratio. Put the cursor in the border of a toolbar (not on an icon) and drag by holding the left mouse button down to the new desired location. As a toolbar is moved close to the sides of the screen, the toolbar will automatically dock to that side.

The **Tools** > **Toolbars** menu choice may also be used to turn specific toolbars on and off and to customize the icons in each toolbar. See the AutoCAD help for more details on customizing toolbars. It is helpful to remove infrequently used icons from some toolbars so that more toolbars may be displayed without taking up excessive screen space.

<table>
<tr><th colspan="2">Setting Up a Template Thermal Desktop Drawing File</th></tr>
<tr><td>12. Select File > Exit.
13. Select Yes to save changes to the template file.
Thermal Desktop is closed.</td><td>Exiting AutoCAD without preceding the Exit command with a Save will bring up a dialog to save any changes.</td></tr>
<tr><td colspan="2">14. Open the \Tutorials\Thermal Desktop\template folder and examine the contents.
The \completed folder is a folder included in many tutorials. It contains the dwg file the user would have after finishing the tutorial.
The thermal.bak file is a backup file generated by AutoCAD when a Save is performed. This file is the previous version of the template.dwg file and can be used by changing the extension from .bak to .dwg in case the main file becomes corrupted.
The thermal.dwg file is a model file created by this tutorial. This file will be copied to other directories to use as a starting point for other tutorials and models.
The RcOptics.rco and TdThermo.tdp files are optical and thermophysical properties databases, respectively, that are automatically generated if they do not already exist, unless the model points to databases in specific file locations.</td></tr>
</table>

As experience is gained with Thermal Desktop and some of the advanced options, create a new template file with different parameters such as Units and other property settings as desired. A user may find it useful to have a template for SI units, a template for English units, or possibly a template with certain default settings (2.6 Defaults).

Important: Remember to create a copy and rename the template before beginning design work to insure the template remains in its original state. If copying the template at the directory/file level by dragging the template file icon to the working directory, be sure to hold the <Ctrl> key down when dragging the template drawing file so that the template is copied, rather than moved.

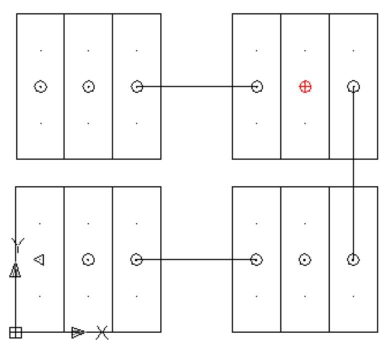
20.3 Model Browser Example

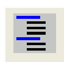
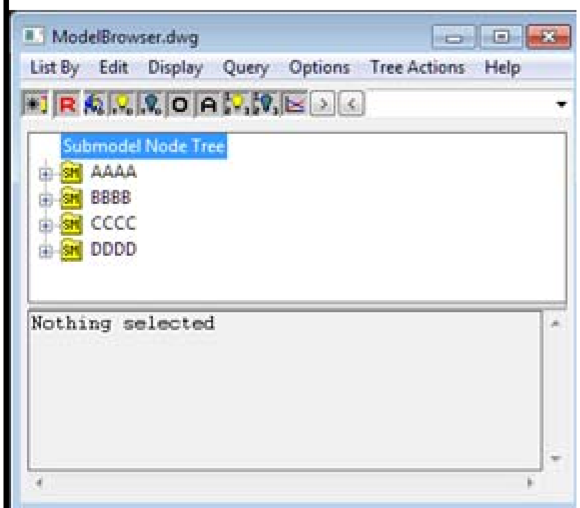
What will be learned:

- Overview of Thermal Desktop's Model Browser
- Using Model Browser for model troubleshooting and review
- Assigning material properties
- Basic use of Case Set Manager
- Obtaining steady state and transient solutions in Thermal Desktop
- Calculating heat flow between submodels
- Using Model Browser to create XY plots against time for transient solutions

This tutorial demonstrates some of the capabilities of Thermal Desktop's **Model Browser.** The example model for this tutorial is very simple but the capabilities extend very well to larger models.

Want to Learn More? Refer to 2.4 Model Browser *in the User's Manual for more information about the* **Model Browser**.

Model Browser Example
1. Double click on the file `ModelBrowser.dwg` located in the `Tutorials\Thermal Desktop\ModelBrowser` folder. Thermal Desktop opens with the ModelBrowser drawing on the screen. 
Thermal Desktop's **Model Browser** can be used to view information about a model. A modeless window (can be resized and minimized) will list model data based on the type of data to be selected. The window is divided into two frames: the tree frame and the output frame. The default is to list by Submodel and ID. The user may select the **List By** menu (within the Model Browser window) to see what types of objects are available for listing. The user can manipulate the AutoCAD graphics by simply making the graphics window active by clicking anywhere on the main AutoCAD window and then performing operations in that window.

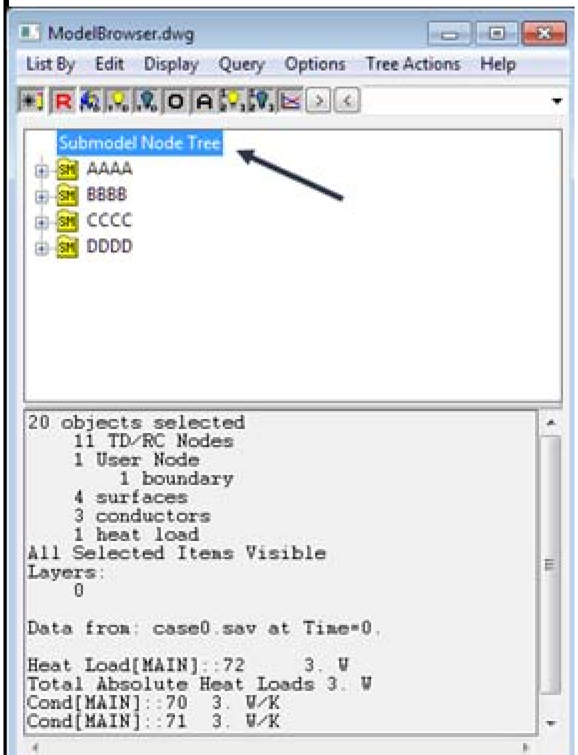
<table>
<tr><th colspan="2">Model Browser Example (Continued)</th></tr>
<tr><td colspan="2">The user can determine what has been selected by looking in the output frame. The output frame will detail how many items have been selected and their type, the visibility state, the layers that the objects reside on, and additional data for the selected items.
As items in the tree frame of the Model Browser are selected, additional information about the selected item is displayed in the lower portion of the window. Single clicking on an item results in high-level information about the selected item being displayed below. Double clicking on an item in the tree frame of the Model Browser results in the expansion of the data tree and more detailed information being displayed.</td></tr>
<tr><td>2.  or Thermal > Model Browser.
The Model Browser window appears on the left side of the screen.

</td>
<td>Display the Model Browser.

The tree display is the upper half.

The output frame is the lower half.</td></tr>
<tr><td>3. Click on AAAA in the tree.
The display at the bottom of the Model Browser changes to show only the objects associated with AAAA.</td>
<td>As individual items, in this case submodel AAAA, are selected, the display area at the bottom of the Model Browser changes to reflect the components of the selected item.

AAAA includes:
<code>5 objects selected</code>
<code>   2 TD/RC Nodes</code>
<code>   1 User Node</code>
<code>   1 boundary</code>
<code>   1 surface</code>
<code>   1 conductor</code></td></tr>
</table>

Model Browser Example (Continued)	

4. Click on Submodel Node Tree located in the main view area of the Model Browser.

 	
	When Submodel Node Tree is selected, all of the submodels that make up the model are selected, and displayed in the output frame.
	
	`20 objects selected`
	`11 TD/RC Nodes`
	`1 User Node`
	`1 boundary`
	`4 surfaces`
	`3 conductors`
	`1 heat load`
	
	Note: Use the scroll bar on the right of the output frame to view additional data.

5. Double click on submodel **AAAA**.	The tree expands to show nodes, 1, 2, and 3 are associated with submodel AAAA. An 'A', 'B', or 'D' in the icon for a node indicates that the node is Arithmetic, Boundary, or Diffusion, respectively. A circle indicates that the node definition is obtained from a surface, solid, or finite element.
6. Select node 1. **Note**: Select by single clicking on the object with the left mouse button.	Node 1 consists of a User node and a surface.

<table>
<tr><th colspan="2">Model Browser Example (Continued)</th></tr>
<tr><td>7. Double click on node 1.

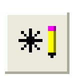
Note: Expand by double-clicking on the object name or clicking on the plus sign (+) to the left of the object.</td><td>The tree underneath node 1 is expanded and AAAA.1^::47 is displayed.

Items with the symbol :: (double colon) are graphical entities. The numbers after the :: are unique for each entity.

The ^ indicates that the node definition was originally defined by a surface, but the user overrode the definition.</td></tr>
<tr><td>8. Select AAAA.1^::47.</td><td>Only the node is selected. The surface, that is below the node is not selected. When an item with a :: is selected the objects below it in the tree are not selected, by default.

When an entity without a :: is selected all the objects below it are selected.</td></tr>
<tr><td>9. Double click on AAAA.1^::47.</td><td>The submodel AAAA tree expands again and Rect[MAIN]::45 is displayed underneath AAAA.1^::47.</td></tr>
<tr><td>10. Select Rect[MAIN]::45.</td><td>The output frame of the Model Browser shows that Rect[MAIN]::45 is a surface and is the only object selected.

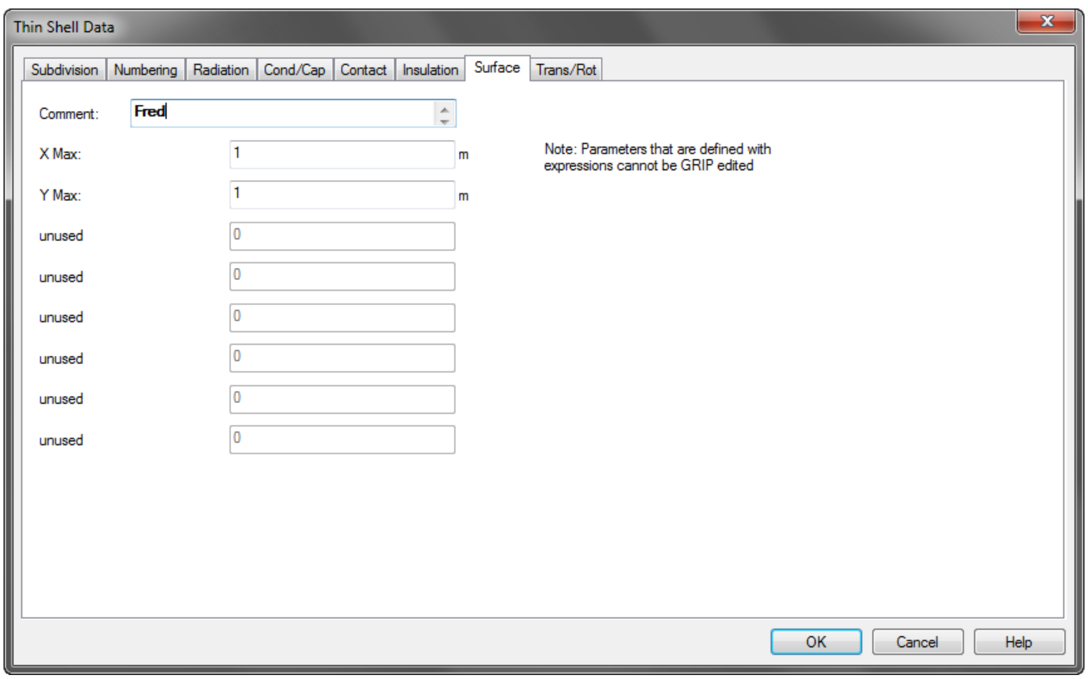
MAIN is the name of the submodel for the surface's conductors</td></tr>
<tr><td colspan="2">Being a separate window, the Model Browser has its own title bar, menu bar, tool bar icons and Windows control buttons. If items are selected in the Model Browser tree, use the icons and menus in the Model Browser.

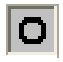
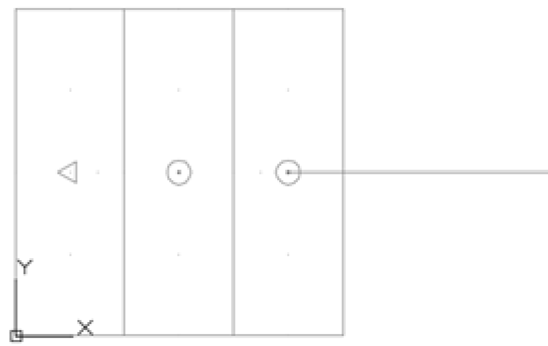
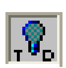
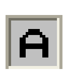
</td></tr>
<tr><td>11.  or Edit > Edit on the Model Browser menu bar.

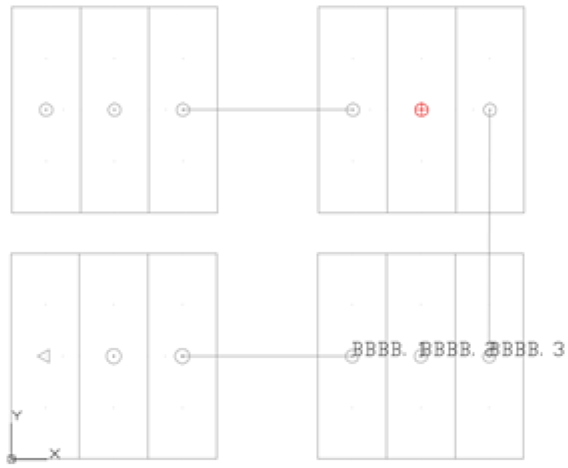
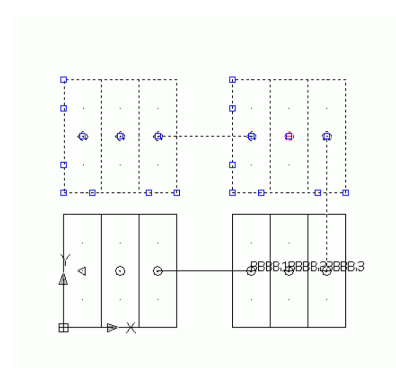
The Thin Shell Data dialog box appears.

Note: The Thin Shell Data dialog box for Rect::45 can also be displayed by double clicking on it.</td><td>Alternatively, you can right-click on Rect[MAIN]::45 and select Edit from the menu that appears</td></tr>
</table>

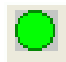
Model Browser Example (Continued)	
12. Select the **Surface** tab. 13. Type **Fred** in the Comment field, as shown below. 14. Select **OK**. 15.  to rebuild the data tree and deselect the Rect-Fred::45.	Once the comment is added and OK selected, the tree “flashes” and rebuilds itself. The rectangle is renamed Rect-Fred[MAIN]::45, incorporating the comment that was entered. This rebuilding capability is controlled with the Model Browser **Options** > **Auto Update** command. The Auto Update feature is useful with small models, but as models become larger, this can be time consuming. Deselect *Rect-Fred::45* and rebuild the tree.



Model Browser Example (Continued)	
16. Select submodel **AAAA**. 17.  **or Display** > **Only** on the Model Browser menu bar.	Notice the graphics in the main Thermal Desktop drawing area change to show only the selected items. Submodel AAAA is in the lower left of the drawing area. **Note**: It may be necessary to move the **Model Browser** out of the way, or minimize it, to view the drawing area. 
18. Select **Submodel Node Tree.**	The message `Mixed Visibility for selected items` appears in the lower portion of the **Model Browser.** This message means that although there are many objects in the submodel node tree (and they are listed in the output frame) some of the selected items in the drawing area cannot be seen by the user (not visible).
19. Select **Rect-Fred[MAIN]::45**. 20.  or **Display** > **Turn Visibility Off** in the Model Browser.	This turns visibility off for the selected item.
21.  or **Display** > **Undo Turn Visibility Off** in the Model Browser	This reverses the last visibility change. If the last change was to turn the visibility of an object(s) on, then after this command the visibility will be turned off or vice versa.
22. 	All the entities in the drawing are now visible. This command turns on the visibility of all items on visible layers.
23. Select submodel **BBBB.**	The lower portion of the **Model Browser** changes to display the objects associated with BBBB.

Model Browser Example (Continued)	
24.  or **Display** > **Turn Ids On** on the Model Browser menu bar.	The node IDs are displayed for submodel BBBB. 
25. Right-click on submodel **BBBB** and select **Send Selection Set to AutoCAD.**	Notice all items is submodel BBBB are highlighted.
26. Right-click on submodel **CCCC** and select **Send Selection Set to AutoCAD.**	The CCCC submodel is highlighted in the drawing area and grip points are dis-played. Also, these entities are now an AutoCAD selection set, so any command issued outside of the Model Browser, such as **Modify** > **Move**, will function on these selected objects after making the main window active.
27. Hold down <**Ctrl**> and select submodel **DDDD.** 28. Right-click on submodel **DDDD** and select **Send Selection Set to AutoCAD.**	Both CCCC and DDDD are selected. 

Model Browser Example (Continued)

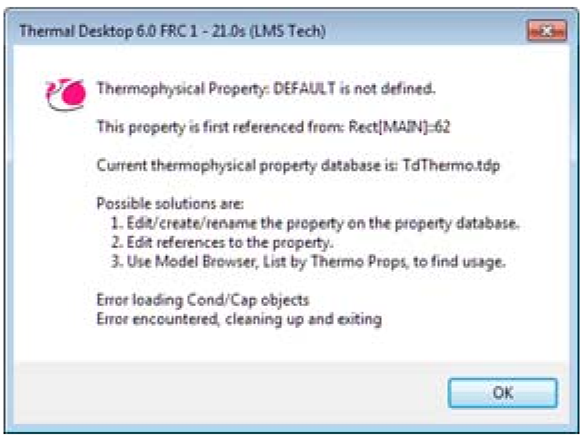
29.  or **Thermal > Case Set Manager** on the main Thermal Desktop menu/toolbar.

    The **Case Set Manager** dialog box appears.

30. Select **Case Set 0** if it is not already selected.

31. Select **Run 1 Selected Case.**

    A Thermal Desktop/AutoCAD dialog box appears with a message stating Thermophysical Property DEFAULT has not been defined.



32. Select **OK** to close the dialog box.

The Case Set Manager is the link between Thermal Desktop and SINDA/FLUINT. Under the Case Set Manager, the user can define different solution sets for the model.

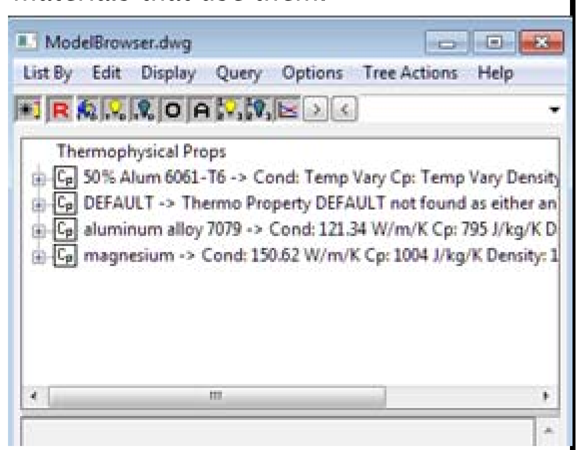
Case Set 0 is a simple steady-state analysis.

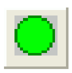
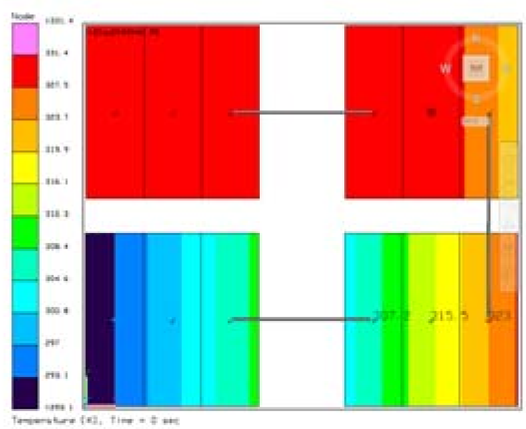
After a Case Set is defined and selected, the Run 1 Selected Case button writes out the SINDA/FLUINT input files, starts SINDA/FLUINT and brings the results back into Thermal Desktop for postprocessing.

The Thermophysical Property DEFAULT is used only as a name place holder and does have property definitions. Therefore, the model definition is incomplete and the solution cannot be started.

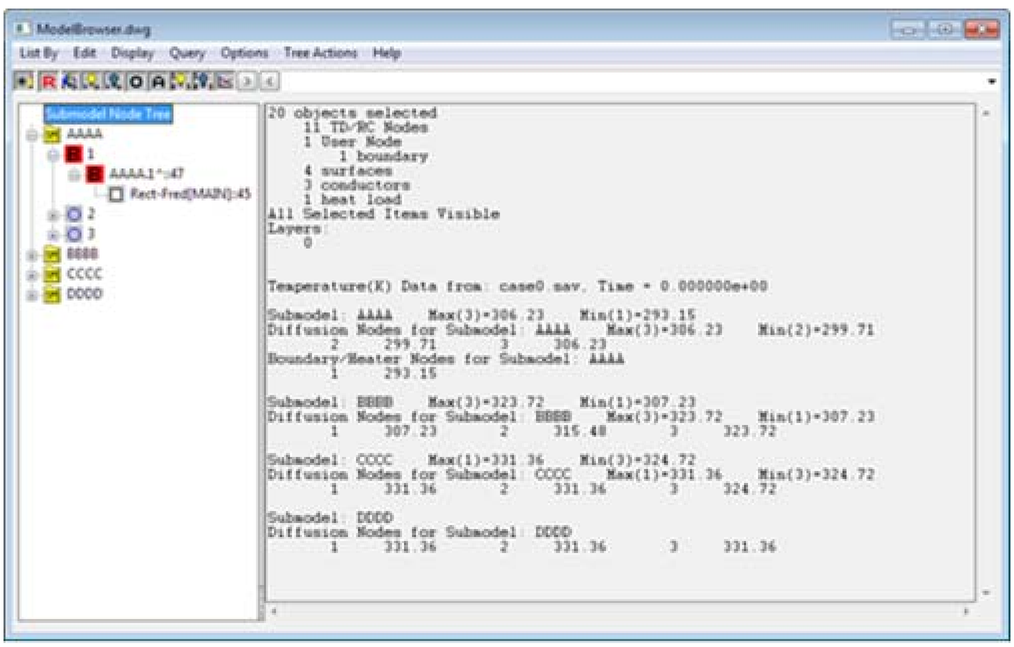
33. Select **List By > Thermo Props** on the Model Browser menu bar.

This command rebuilds the **Model Browser** so that objects are listed by the materials that use them.



Model Browser Example (Continued)	
34. Click on the + next to **DEFAULT** -> to expand the list. 35. Right-click on **Rect[MAIN]::62** (the surface) and select **Edit.** The Thin Shell Data dialog box appears. 36. Click on the **Cond/Cap tab.** 37. Click on the arrow next to the Material field and select **Stainless Steel** from the drop-down list. 38. Select **OK** to close the dialog box.	Change the DEFAULT material to Stainless Steel. Rebuilding the tree will verify that the DEFAULT material has been replaced.
39.  or **Thermal** >**Case Set Manager** on the main Thermal Desktop menu/toolbar. *Note: Rect::62 is still selected in the Model Browser.* The Case Set Manager dialog box appears. 40. Select **Run 1 Selected Case.** A Sinda/Fluint Run Status dialog box appears confirming the successful completion of the process. 41. Select **OK** to close the dialog box.	A solution is calculated for the default conditions: steady state with no radiation. If the image below does not appear, select a different Layout tab at the bottom left of the Thermal Desktop window. 

Model Browser Example (Continued)	
42. Select **List By** > **Submodel.Id** from the Model Browser menu bar. 43. Select **Submodel Node Tree**, if not already selected. 44. Scroll down the list in the output portion of the Model Browser look at the additional available information.	The submodel tree rebuilds and the window is back to its original form. In addition to the summary of the contents of the model, the output portion of the Model Browser includes the temperatures of the selected nodes, along with the Max and Min of the current selection set. These values are the current postprocessed data. If the current postprocessed data were heat rates, then these values would be heat rates. To increase the size of the output portion of the window, the divider between the windows can be dragged.
45. Select **Options** > **Temperatures** from the Model Browser menu bar. 46. View the **Output** area.	Scrolling in the text window, you'll see the output looks more like a SINDA TPRINT.
47. Select **Options** > **Output Window on Bottom** from the Model Browser menu bar to deselect it (remove the check mark).	This moves the output area to the right of the tree. The Model Browser window may be resized as desired. The example below shows the window resized horizontally and shortened.



Model Browser Example (Continued)	
48. Select Submodel **BBBB**. 49. View the **Output** area.	Selecting on a single submodel, such as BBBB will show the data only for that sub-model.
50. Select **Options** > **CSG** from the Model Browser menu bar. 51. Select **Submodel Node Tree**. 52. View the **Output** area.	Selecting **Options** > **CSG** shows the CSG of the selected nodes, sorted in lowest to highest order. The CSG is the capacitance of a node divided by the sum of all con-ductances attached to it; it directly affects the timestep of the model for transient runs. In order for the CSG to work, the SINDA save file must have capacitance and con-ductors saved on it. This is set on the Output Tab of the Case Set Manager.
53. Select **Options** > **Node Map** from the Model Browser menu bar. 54. View the **Output** area.	**Options** > **Node Map** shows a SINDA-like NODMAP capability in order for the user to determine how energy is transferred into a node.
55. Select **Options** > **Heat Map** in the **Model Browser**. 56. Select submodel **CCCC**. 57. View the **Output** area.	The heat map shows the energy summary for energy into and out of the selected nodes. Energy between the selected nodes is not in the tabulation. If submodel CCCC is selected, the heat map implies a load of 3W and has 3W leaving into submodel BBBB. If submodels BBBB and CCCC are selected the heat map output will show the energy going from submodel BBBB into submodel AAAA, along with the 3W heat source on submodel CCCC.

Model Browser Example (Continued)

58. Select **Options** > **Heat Flow Between Submodels** in the Model Browser window.

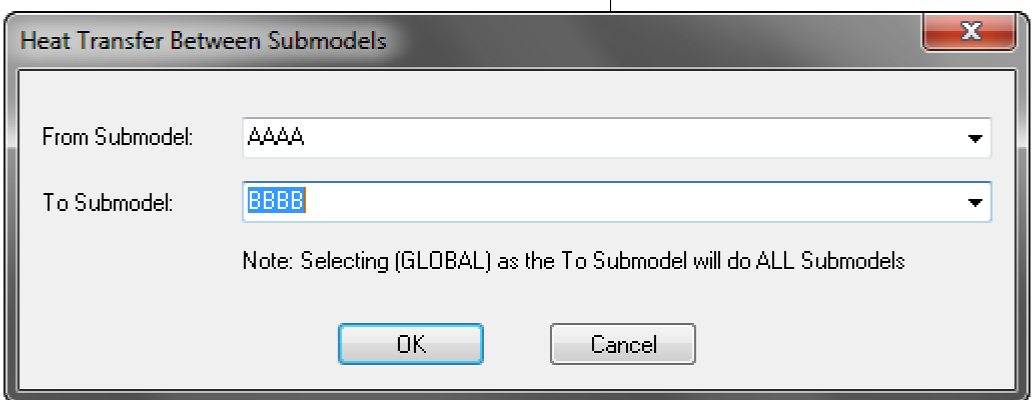
    The Heat Transfer Between Submodels dialog box appears.

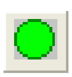
59. Click on the arrow next to the From Submodel field and select **AAAA** from the drop-down list.

60. Click on the arrow next to the To Submodel field and select **BBBB** from the drop-down list.

61. Select **OK**.

The results of the heat flow analysis is 3W going from submodel AAAA to submodel BBBB. The program cycles through all the nodes in submodel AAAA and sums the heat flows of all the conductors that connect to submodel BBBB.



62.  or **Thermal** >**Case Set Manager** on the main Thermal Desktop menu/toolbar.

    The Case Set Manager dialog box appears.

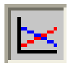
63. Select **Case Set 1**.

64. Select **Run 1 Selected Case**.

    A Sinda/Fluint Run Status dialog box appears confirming the successful completion of the process.

65. Select **OK** to close the SINDA/FLUINT RUN STATUS dialog box.

Case Set 1 is a transient analysis that solves for 10,000 seconds.

Model Browser Example (Continued)	
66. Select **Submodel Node Tree.** 67.  or **Query** > **Plot Selected** in the Model Browser menu. 68. Minimize or close the EZXY PLOTTER window.	With transient results, only the data for the currently postprocessed results are shown in the Model Browser output frame. The Plot command in the Model Browser creates a Data vs. Time plot of the post-processed variable for all selected items in the model tree.
69. Close the **Model Browser.** 70. Select **File** > **Exit.** A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes. 71. Select **Yes.**	Exit Thermal Desktop and save as prompted.

20.4 Circuit Board Conduction Example

What will be learned:

- Overview of how Thermal Desktop works
- Creating material properties
- Creating Thermal Desktop objects
- Changing object colors
- Applying an edge contactor
- Applying a face contactor
- Checking contactors
- Outputting SINDA/FLUINT conductance and capacitance data for review

Prerequisites:

- 20.2 Setting Up a Template Drawing

In this exercise, a circuit board will be mounted to an aluminum face. The circuit board will have a chip placed on it.

<table>
<tr><th colspan="2">Circuit Board Example</th></tr>
<tr><td colspan="2">1. Copy the template <code>thermal.dwg</code> file created in the first tutorial to the <code>\Tutorials\Thermal Desktop\board</code> directory.

Note: Be sure to hold the <Ctrl> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved.

2. Rename the copied template file to board.

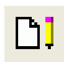
3. Start Thermal Desktop by double clicking on the board drawing file icon in the board directory.

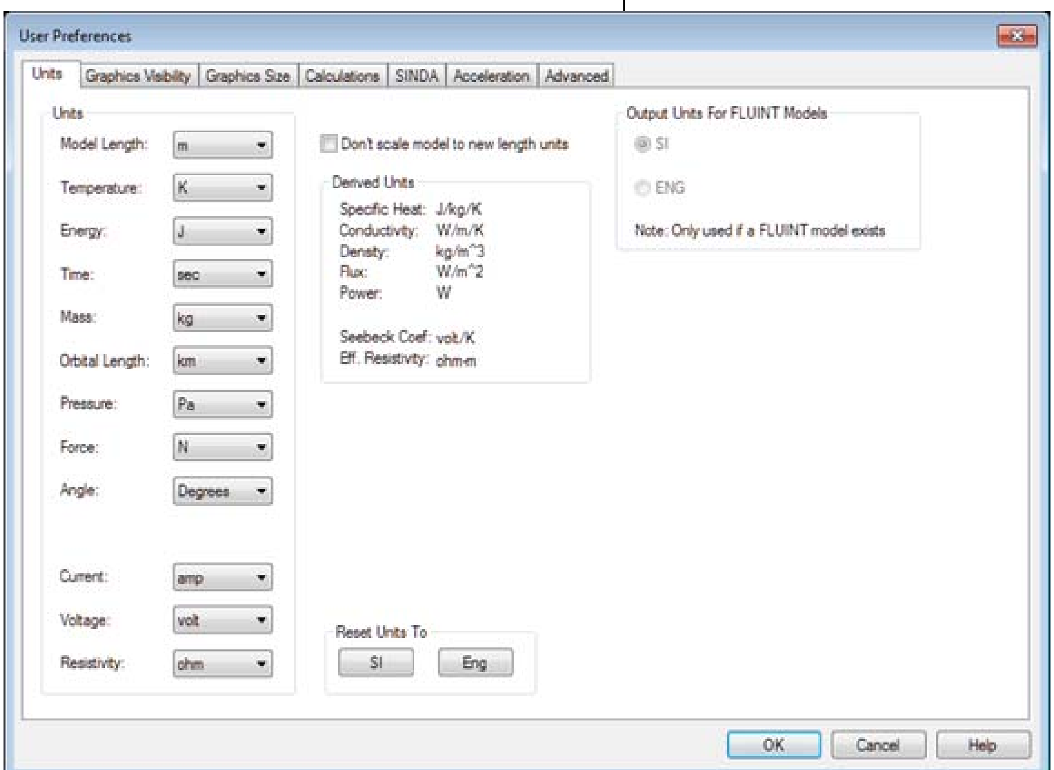
4.  or View > Visual Styles > 2D Wireframe to ensure consistency with the images in this tutorial.</td></tr>
<tr><td>5.  or Thermal > Thermophysical Properties > Edit Property Data.</td><td>This part of the exercise defines the thermophysical properties for aluminum, fr4, and the chip.

The default units for Thermal Desktop are SI. These properties have the units of:
■ W/m/K for conductance
■ J/Kg/K for specific heat
■ kg/m^3 for density.</td></tr>
</table>

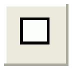
Circuit Board Example (Continued)	
6. Type **Aluminum** in the **New property to add** field. 7. Select the **Add** button. The Edit Thermophysical Properties - Aluminum dialog box appears. 8. Highlight the current value in the **Conductivity** field and type **237**. 9. Highlight the current value in the **Specific Heat** field and type **900**. 10. Highlight the current value in the **Density** field and type **2702**. 11. Select **OK**.	Define properties for Aluminum. Conductivity = 237 W/m/K Specific heat = 900 J/kg/K Density = 2702 kg/m^3
12. Type **fr4 2 oz copper** in the **New property to add** field. 13. Select the **Add** button. 14. Highlight the current value in the **Conductivity** field and type **17.7** 15. Highlight the current value in the **Specific Heat** field and type **0**. 16. Highlight the current value in the **Density** field and type **0**. 17. Select **OK**.	Define properties for fr4 2 oz copper. Conductivity = 17.7 W/m/K Specific heat = 0 J/kg/K Density = 0 kg/m^3 Zero values (0) for specific heat, density, or object thickness (covered later) will make the nodes arithmetic.
18. Type **chip** in the **New property to add** field. 19. Select the **Add** button. 20. Highlight the current value in the **Conductivity** field and type **0**. 21. Highlight the current value in the **Specific Heat** field and type **837.32** 22. Highlight the current value in the **Density** field and type **2000**.	Define properties for chip. Conductivity = 0 W/m/K Specific heat = 837.32 J/kg/K Density = 2000 kg/m^3 The chip will be a single node. Therefore the chip's internal conductivity is not used for these calculations, so any value can be used. If the chip were represented by multiple nodes, internal conduction could be ignored by either setting conductivity of the material to 0 or setting the object thickness to 0 (covered later). However, setting the thickness to 0 will also create arithmetic nodes. These values are approximations that are not really indicative of any particular chip.

Circuit Board Example (Continued)

23. Select **OK** to close the **Thermophysical Properties - chip** dialog box. 24. Select **OK** to close the **Edit Thermophysical Properties** dialog box.	Close the Thermophysical Properties dialog box.
25.  or **Thermal** > **Preferences**. 26. Select the **Units** tab if not already displayed. 27. Click on the arrow next to the **Model Length** field and select **in** (inches) from the drop-down list. 28. Select **OK** to close the **User Preferences** dialog box.	The model is to be built in inches. Notice that the energy units are in Joules, time in seconds, thus the energy rate units are Watts. The properties set earlier are automatically converted to inch (In) units.



Circuit Board Example (Continued)

29.  or **Thermal** > **Surfaces/Solids** > **Rectangle**.

    The Command line area should show:

    ```
    Command: _RcRectangle
    Origin point <0,0,0>:
    ```

30. Type **0,0**

Note: Remember, <**Enter**> is implied after typing input.

    The Command line should now read:

    ```
    Point for +X axis and X-size
    <@1,0,0>:
    ```

31. Type **6,0**

    The Command line should now read:

    ```
    Point to set XY plane and Y-size
    <@0,1,0>:
    ```

32. Type **0,3**

    The **Thin Shell Data** dialog box appears.

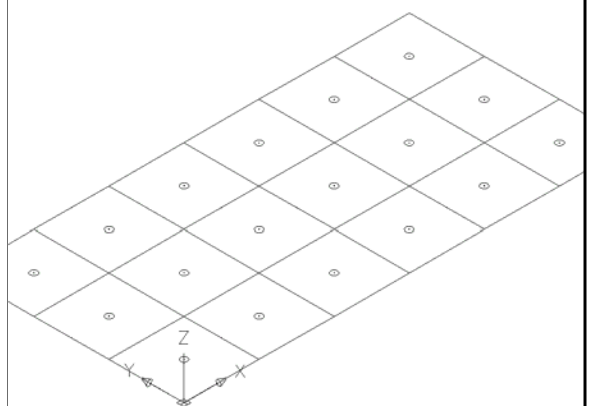
33. Click on the **Subdivision** tab if not already displayed.
34. Leave **Centered Nodes** selected.
35. Highlight the current value in the **X-direction Equal** field and type **6**
36. Highlight the current value in the **Y-direction Equal** field and type **3**

Input the appropriate data to make the aluminum plate with the proper nodalization. A rectangle will be created for the aluminum plate.

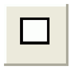
- Origin = 0,0
- X length = 6
- Y length = 3
- Centered nodes
- 6 subdivisions in the X direction
- 3 subdivisions in the Y direction

When values are typed, they will automatically be placed into the command line or next to the cursor; there is no need to physically click into the Command line area.

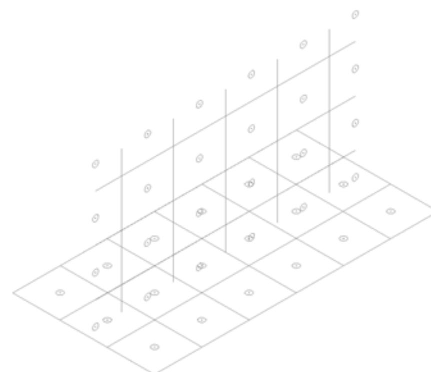
Inputs and/or prompts may appear next to the pointer when typed: this is referred to as Dynamic Input. This option can be changed with **Tools** > **Drafting Settings** on the Dynamic Input tab.

Circuit Board Example (Continued)	
37. Click on the **Cond/Cap** tab. 38. Click on the arrow next to the **Material** field and select **Aluminum** from the drop-down list.	Generate Nodes And Conductors must be selected to change the material and the thickness.
39. Highlight the current value in the **Thick-ness** field and type **0.05** 40. Select **OK** to close the Thin Shell Data dialog box.	Set the material as Aluminum with a thickness of 0.05 in. At the end of these steps, the screen should look similar to the view below:
41.  or **View** > **Zoom** > **Extents**.	 **Figure 20-7: Aluminum Plate** If OK is selected too soon, double-click the rectangle to re-enter the edit dialog.

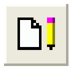
Circuit Board Example (Continued)	

42.  or **Thermal** > **Surfaces/Solids** > **Rectangle.**
The Command line area should show:
`Command: _RcRectangle`
`Origin point <0,0,0>:`
43. Type **0.25,1.5**
The Command line should now read:
`Point for +X axis and X-size <@1,0,0>:`
44. Type **@5.5,0**
The Command line should now read:
`Point to set XY plane and Y-size <@0,1,0>:`
45. Type **@0,0,3**
The Thin Shell Data dialog box appears.
	This part of the exercise creates the circuit board.
	The @ sign input tells the program to input a point relative to the last point input. In this example, the @5.5,0 is the same as typing 5.75,1.5.

46. Click on the **Cond/Cap** tab if not already displayed.	Additional information for the circuit board is to be added.
47. Highlight the current value in the **Cond Submodel** field and type **board.** 48. Click on the arrow next to the **Material** field and select **fr4 2 oz copper** from the drop-down list. 49. Highlight the current value in the **Thickness** field and type **0.03**	When inputting board for the Cond submodel, the word board must be typed in. In the next step, on in the Numbering tab, board will have been added to the pull-down list.
50. Click on the **Subdivision** tab. 51. Select **Edge Nodes** 52. Highlight the current value in the **X-direction Equal** field and type **6** 53. Highlight the current value in the **Y-direction Equal** field and type **4** 54. Click on the **Numbering** tab.	Thermal Desktop does not calculate the conductance from nodes to the edges of surfaces connected by contact or contactors. Therefore, that conductance should be included in the contact value or the better method, used here, is to use edge nodes.

Circuit Board Example (Continued)	
55. Click on the arrow next to the **Submodel** field and select **BOARD** from the drop-down list. 56. Select **OK** to close the Thin Shell Data dialog box. 57.  or **View** > **Zoom** > **Extents.**	When finished, the model should look similar to the drawing below (the colors will be changed shortly).  **Figure 20-8: Board on Aluminum Plate**
58. Type LTSCALE. The Command line should now read: `LTSCALE Enter new linetype scale factor <39.3701>:` 59. Type **0.5**	Change the Linetype scale factor so the edges of the new plate are more visible.

Circuit Board Example (Continued)

60.  or **Thermal** > **Preferences**.

    The User Preferences dialog box appears.

61. Select the **Graphics Visibility** tab.
62. Clear **TD/RC Nodes** (remove the check mark from the box).
63. Select **OK** to close the User Preferences dialog box.
64. The nodes are no longer displayed.

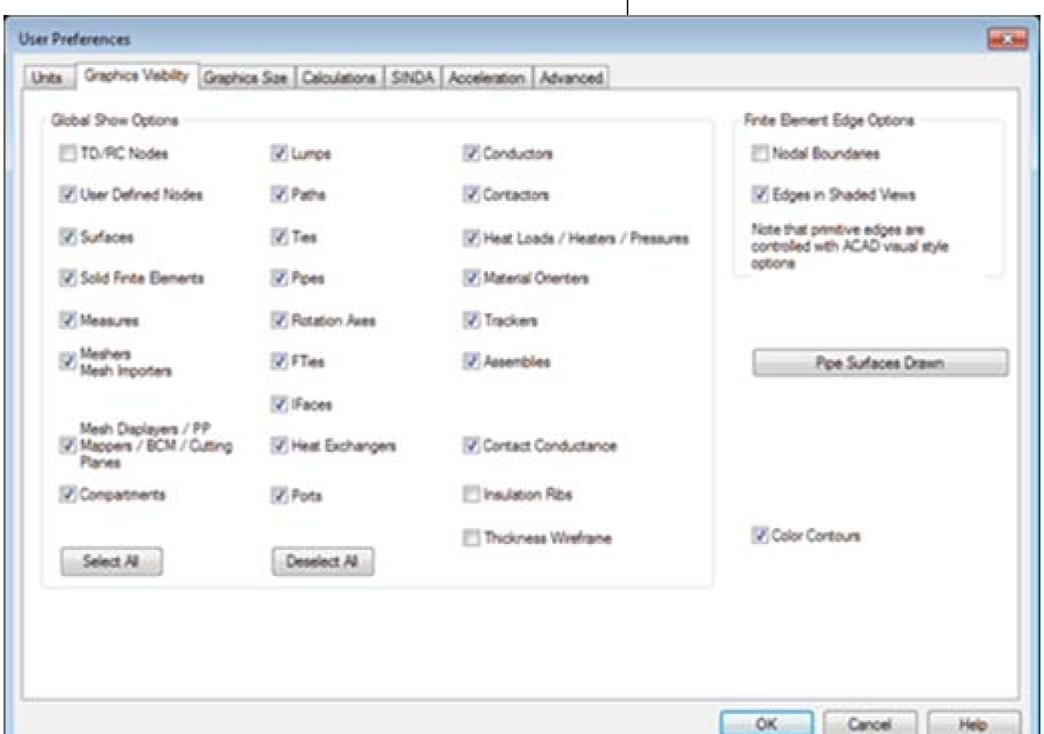
The rectangle displayed in the drawing area is divided into six units long the X axis and three along the Y axis. There are small circles in the center of each unit. The small circles are the nodes. The node display is to be turned off.

Notice that the nodes do not line up with the aluminum plate.

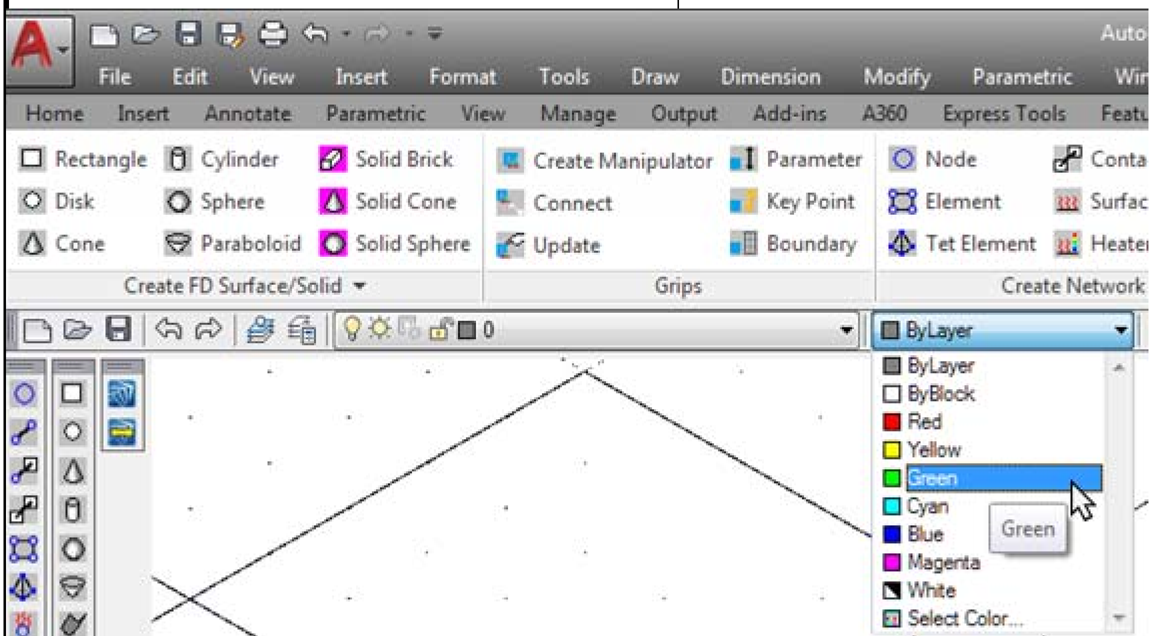
This step in the exercise turns off the node display starting from the Thermal menu.

 Note that a much faster way to toggle the display of nodes is to use the Toggle TD/RC Node Visibility icon located at the lower the right side of the screen.

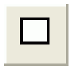
Note: Some tool bar icons may not be visible within the Thermal Desktop window borders. As with basic Windows functionality, tool bars can be moved and docked as desired by the user.



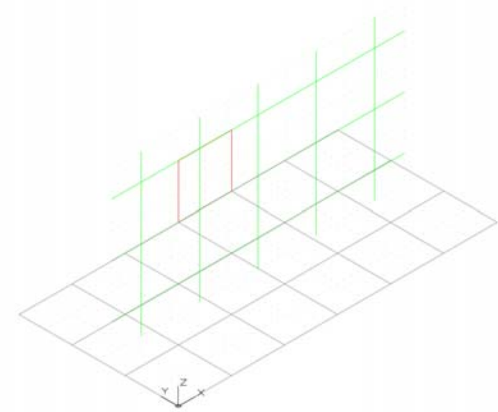
Circuit Board Example (Continued)	
65. Click on the newly created circuit board (the vertical rectangle) to select it. 66. Select the Color drop-down list (showing **ByLayer**) and select **Green**	Change the color of the circuit board. The default color of the circuit board is the color of the current layer (in this case the current color is white). The color is being overridden to change the circuit board to green.



Circuit Board Example (Continued)	

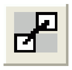
67.  or **Thermal** > **Surfaces/Solids** > **Rectangle.**
The Command line area should show:
`Command: _RcRectangle`
`Origin point <0,0,0>:`
68. Type **1.5,1.5,1.5**
The Command line should now read:
`Point for +X axis and X-size <@1,0,0>:`
69. Type **@1,0**
The Command line should now read:
`Point to set XY plane and Y-size <@0,1,0>:`
70. Type **@0,0,1**
The Thin Shell Data dialog box appears.
71. Click on the **Numbering** tab.
72. Highlight the current value in the **Sub-model** field and type **chip.**
73. Click on the **Cond/Cap** tab.
	Create a chip on the circuit board.
	
	The chip is purposely being placed so that it overlaps the nodes on the board.
	
	Instead of inputting the points, the points could be snapped to the drawing if desired.
	
	Hint: <Shift> and click the right mouse button to access a menu for different types of snap points: select Node to snap to the center of a surface.

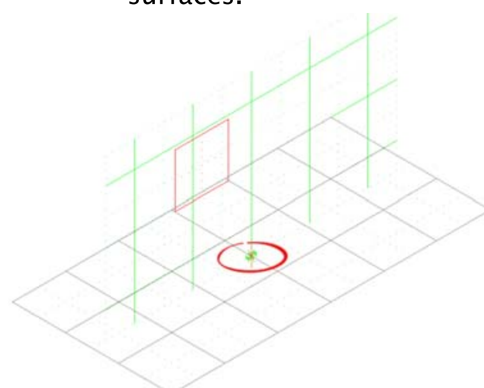
74. Click on the arrow next to the **Cond Submodel** field and select CHIP from the drop-down list. 75. Click on the arrow next to the **Material** field and select **CHIP** from the drop-down list. 76. Highlight the current value in the **Thick-ness** field and type **0.1** 77. Select **OK** to close the dialog box.	If CHIP is not in the drop-down list, then type it in.

Circuit Board Example (Continued)	
78. Click on the newly created chip to select it. 79. Select the Color drop-down list (showing ByLayer) and select **Red.**	Change the color of the chip to red. It may be necessary to ZOOM in to select the chip. When finished, the model should look similar to the drawing below.  **Figure 20-9: Chip on Circuit Board**

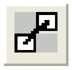
80. Select the newly created chip.
81.  or **Modify** > **Move**.
The Command line should now read:
`Specify base point or [Displacement] <Displacement>:`
82. Type **0,-0.065,0**
The Command line should now read:
`Specify second point or <use first point as displacement>:`
83. Press <**Enter**>.
The chip is moved away from the circuit board.
	It is good modeling practice to move the chip off of the board.
	
	The distance of the offset shall be set to half of the sums of the thickness of the board (0.03) and the chip (0.1), which equals 0.065. The direction is in the negative Y direction because that is the direction in the current Coordinate System (UCS).
	
	Remember that <**ENTER**> is assumed after typing into the command line. In actuality, <**ENTER**> is pressed twice.

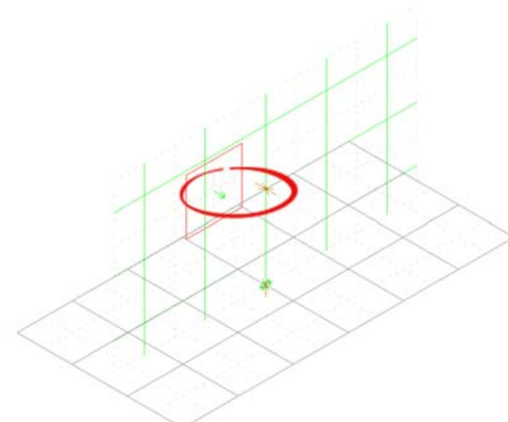
Circuit Board Example (Continued)	

84.  or **Thermal** > **FD/FEM Network** > **Contactor**.
The Command line should now read:
`Select faces contacting from or [MB]:`
85. Select the green circuit board in the drawing area.
The Command line should now read:
`Select faces contacting from or [MB]:`
86. Press <**Enter**>.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
87. Select horizontal surface in the drawing area.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
88. Press <**Enter**>.
The Contactor dialog box appears.
	The board needs to conduct to the base plate. This can be accomplished by using a contactor.

Circuit Board Example (Continued)	
89. Click on the arrow next to the **Conductor Submodel** field and select **BOARD** from the drop-down list. 90. Click on the **Contact From** drop-down and select **Edges.** 91. Highlight the current value in the **Conduction Coefficient** field and type **5** 92. Double-click **Rect[BOARD]::274** in the **From (1)**: list box. 93. In the **Select Edges** dialog, clear **Along Y at X=0** and check **Along X at Y=0** 94. Select **OK** to close the **Select edges** dialog. 95. Select **OK** to close the Contactor dialog box.	The Contactor dialog box is displayed. Once the changes have been made and OK is selected, the graphical image for a contactor is displayed. ■ The from object will be shown as a green arrow showing the edge selected for contact. ■ The to object will be shown in gold and will have arrows pointing to both sides of the surfaces.  **Figure 20-10: Edge Contactor** The edge for the contactor is determined by the creation of the BOARD surface: the X at Y=0 edge is the first edge specified after the origin.

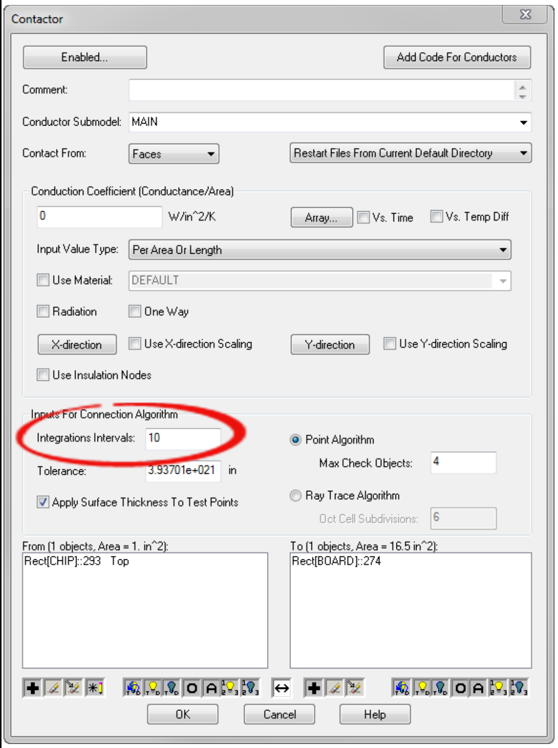
Circuit Board Example (Continued)	

96.  or **Thermal** > **FD/FEM Network** > **Contactor**.
The Command line should now read:
`Select faces contacting from or [MB]:`
97. Select the red chip in the drawing area.
The Command line should now read:
`Select faces contacting from or [MB]:`
98. Press <**Enter**>.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
99. Select green circuit board in the drawing area.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
100. Press <**Enter**>.
The Contactor dialog box appears.
	The chip needs to conduct to the board. This can be accomplished by using a contactor.

Circuit Board Example (Continued)	
101. Click on the arrow next to the **Conductor Submodel** field and select **CHIP** from the drop-down list. 102. Highlight the current value in the **Conduction Coefficient** field and type **5** 103. Select **Show Calcs** at the bottom right to close the Contactor dialog box and show the contactor markers.	The Contactor dialog box is displayed. Once the changes have been made and OK is selected, the graphical image for a contactor is displayed. ■ The from surface will be shown as a green arrow showing the side selected for contact. ■ The to surfaces will be shown in gold and will have arrows pointing to both sides of the surfaces.  **Figure 20-11: Face Contactor** Notice the green arrow points away from the circuit board. Since the default contactor has infinite tolerance and uses a point distance method, this will work fine for this case, but technically the contactor should be edited, and then the user should double click on the from surface to change the side of contact. If the surface had different nodes on each side, or if the ray trace algorithm was being used for the contactor, then this change would be mandatory in order to get the correct results.

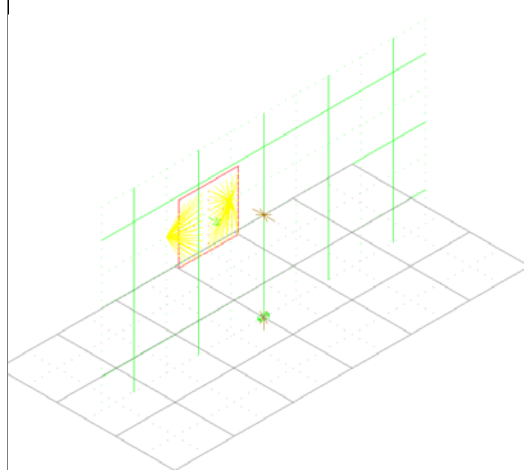
Circuit Board Example (Continued)

Yellow lines representing the contacts are displayed.



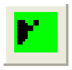
If the contactors dialog is closed without using Show Calcs, the contactor moarkers can be displayed using **Thermal** > **Model Checks** > **Show Contactor Markers** command, which shows the actual calculations for the contact.

Note: When items are connected, yellow lines are drawn from the contact point(s) to the node with which it is connected. The number



of test points is controlled in the **Contactor** dialog box, displayed earlier when the contactor was being created, as the Integration Intervals. A value of 10 integration intervals means the 10*10, or 100, points will be tested. In this model, notice that the chip is connected to 2 nodes.

If an object does not connect to a node, then it is displayed as a red point. To prevent an object from hooking up, the tolerance on the **Contactor** dialog box must be changed from the default of infinity to a smaller value.

Circuit Board Example (Continued)	
104.  or **Thermal** > **Model Checks** > **Clear Contact/or Markers** 105. Select **Thermal** > **Model Checks** > **Show Contactor Markers** and display markers for the other contactor. 106. Double-click the text **Double-click to Reset Graphics**	This command deletes the contactor marker lines. It actually deletes all the items on the "Radcad_rays" layer.
107. Select **Thermal** > **Cond/Cap Calculations** > **Output SINDA/FLUINT Cond/Cap.**	This part of the exercise demonstrates the Output SINDA/FLUINT Cond/Cap command. This command outputs the conductors and capacitance for the model to the file SINDA.CC located in the same directory as the drawing, in this case in the board directory. If the SINDA.CC file is opened in a text editor three different node blocks—MAIN (aluminum base), (circuit) BOARD, and CHIP—are detailed. In the conductor data for the chip, the chip is tied to two nodes on the circuit board representing the planar contact area. Likewise, the circuit board is tied to several nodes on the base representing the linear contact conductance.
108. Select **File** > **Exit.** A Thermal Desktop/AutoCAD dialog box appears asking if the user wants to save changes to the board.dwg. 109. Select **Yes.** The drawing is saved and Thermal Desktop is closed.	**Note**: It is good practice whenever working on a computer to periodically perform File > Save commands during the course of a session to help ensure work is not lost.

20.5 Beer Can Example

What will be learned:

- Overview of how Thermal Desktop works
- Creating material properties
- Creating Thermal Desktop objects
- Changing global visibility options
- Extruding planar objects into solid elements
- Surface coating free solid finite element faces
- Using model checks to verify model development
- Using of arbitrary nodes and conductors
- Using AutoCAD layers to control object visibility
- Using the Case Set Manager
- Parameterizing a model
- Creating XY time-dependent plots

Prerequisites:

- 20.2 Setting Up a Template Drawing

In this example, a beer can full of beer will be constructed. The initial temperatures will be set to something similar to a refrigerator temperature of 5C. Free convection heat transfer coefficients will be applied to the sides of the can and the top of the can.

Beer Can Example
1. Copy the template thermal.dwg file created in the first tutorial to the \Tutorials\Thermal Desktop\beercan directory. **Note**: Be sure to hold the <Ctrl> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved. 2. Rename the copied template file to beercan. 3. Start Thermal Desktop by double clicking on the beercan drawing file icon in the beercan directory.

Beer Can Example (Continued)

4.  or select **Thermal** > **Thermophysical Properties** > **Edit Property Data**.

   The Edit Thermophysical Properties dialog box appears.

5. Type Aluminum in the **New property to add** field.

Note: See comments in the right-hand column.

6. Select the **Add** button.

   The Thermophysical Properties dialog box appears.

This part of the exercise defines the thermophysical properties for aluminum and water.

If the tutorials are being performed in order from the beginning of the tutorial chapter, the user will have already defined Aluminum properties in the board model. Instead of redefining the properties, the user has two choices:

- First, it is possible to use the **Thermal** > **Thermophysical Properties** > **Open/Create Property DB...** command to open the database created in the board example. The Aluminum defined there could be used and the Water definitions added to that database. In the case, the material property will be stored in the Board tutorial folder.
- Second, the user can import the Aluminum properties from the Board tutorial database into the Beer tutorial database. Once the Edit Thermophysical Properties dialog is open, select the Import button. Open the database created in the board example and select Aluminum from the list of available properties. In the case, the material properties will be stored in the Beercan tutorial folder.

Note: If a Material is already listed in the **Edit Property Data** dialog box but one or more of the properties is different than what is needed, double click on the material of interest. The **Thermophysical Properties** dialog box will appear allowing changes to be made.

Beer Can Example (Continued)	
7. Highlight the current value in the **Conductivity k** field and type 237. 8. Highlight the current value in the **Specific Heat cp** field and type 900. 9. Highlight the current value in the **Density rho** field and type 2702. 10. Select **OK** to close the Thermophysical Properties dialog box. The Edit Thermophysical Properties dialog box reappears with Aluminum and the above values displayed in the main property/description field. 11. Type Water in the **New property to add** field. 12. Select the **Add** button. The Thermophysical Properties dialog box appears. 13. Highlight the current value in the **Conductivity k** field and type 0.6. 14. Highlight the current value in the **Specific Heat cp** field and type 4200. 15. Highlight the current value in the **Density rho** field and type 1000. 16. Select **OK** to close the Thermophysical Properties dialog box. The Edit Thermophysical Properties dialog box reappears with water and the above values displayed in the main property/description field. 17. Select **OK** to close the Edit Thermophysical Properties dialog box.	As in real projects, some assumptions must be made.

Beer Can Example (Continued)

18.  or **Thermal** > **Preferences**.

    The User Preferences dialog box appears.

19. Select the **Units** tab if not already displayed.

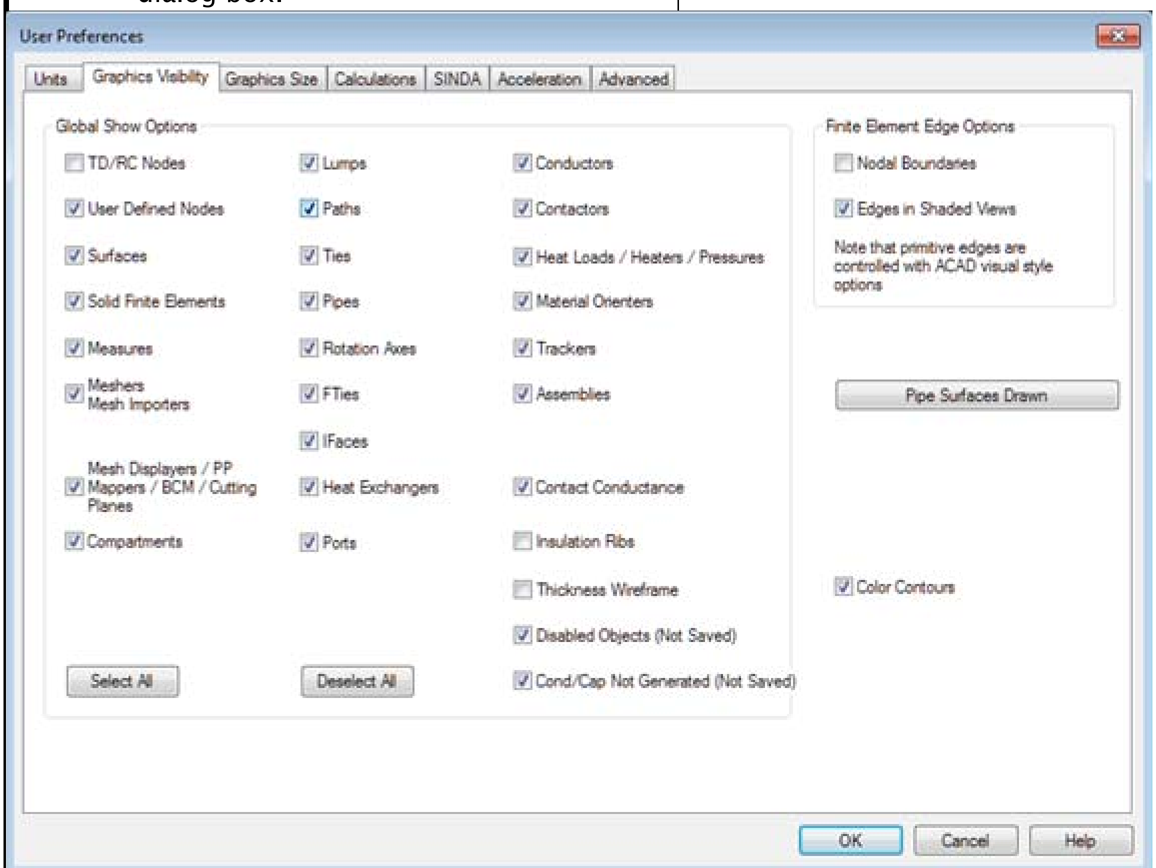
20. Click on the arrow next to the **Model Length** field and select **in (inches)** from the drop-down list.

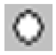
21. Select the **Graphics Visibility** tab.

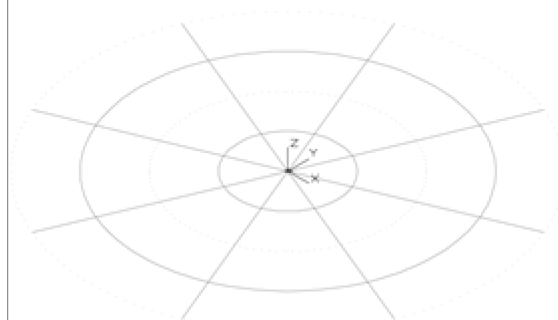
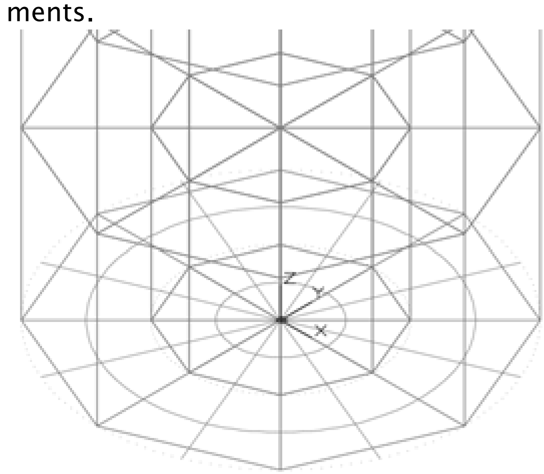
22. Clear **TD/RC Nodes** (remove the check mark from the box).

23. Select **OK** to close the User Preferences dialog box.

These steps change the units for the model to inches and turns off TD/RC node visibility for all the nodes that are attached to the surface.



Beer Can Example (Continued)	
24.  or **Thermal** > **Surfaces/Solids** > **Disk.** The Command line should now read: `Pick or enter point for center of disk <0,0,0>:`	Create the bottom of the aluminum can. Specify the origin.
25. Type 0,0 in the Command line. The Command line should now read: `Pick or enter point for +Z axis of disk <@0,0,1>:`	Define the axis direction.
26. Type 0,0,1 in the Command line. The Command line should now read: `Enter maximum radius or pick/ enter point <1.0>:`	
27. Type 1.3125 in the Command line. The Command line should now read: `Enter minimum radius or pick/ enter point <0.0>:`	
28. Press <Enter>. The Command line should now read: `Enter start angle or pick/enter point <0.0>:`	Using edge nodes is important since finite elements will be extruded from this disk. If centered nodes are used, then the finite elements will not fill the entire volume of the can.
29. Press <Enter>. The Command line should now read: `Enter end angle or pick/enter point <360.0>:`	
30. Press <Enter>. The Thin Shell Data dialog box appears displaying default values.	
31. Click on the **Subdivision** tab if not already displayed.	
32. Click on the radio button next to **Edge Nodes** to select it (display a dot in the circle).	

Beer Can Example (Continued)	
33. Highlight the current value in the **Angular Equal** field (subdivisions) and type 9. 34. Highlight the current value in the **Radial Equal** field (divisions) and type 3. 35. Click on the **Cond/Cap** tab. 36. Click on the arrow next to the **Material** field and select Aluminum from the drop-down list. 37. Highlight the current value in the **Thickness** field and type .05. 38. Select **OK** to close the Thin Shell Data dialog box. 39.  or **View** > **Zoom** > **Extents**.	When completed, this disk will represent the bottom of the beer can. This part of the exercise sets the disk properties. **Note**: If the **OK** button is accidentally selected before switching to the Cond/Cap tab, simply select the disk and select **Thermal** > **Edit** to get back to the form. 
40. Select the newly created disk. 41. Select **Thermal** > **FD/Fem Network** > **Extrude Normal To Planar Elements into Solids**. The Extrude/Revolve Planar Elements into Solids dialog box appears. (Next page)	The disk will be extruded into solid elements. 

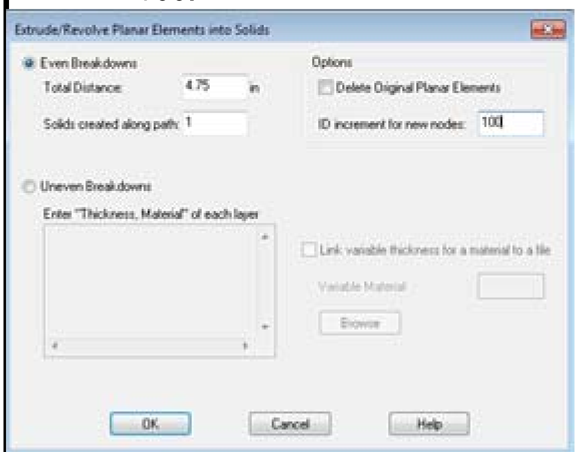
Beer Can Example (Continued)

42. Leave **Even Breakdowns** selected (dot in the circle).

43. Highlight the current value in the **Total Distance** field and type 4.75.

44. Highlight the current value in the **Solids created along path** field and type 4.

45. Highlight the current value in the **ID Increment for new nodes** field and type 100.



46. Select **OK** to close the dialog box.

Note: After the extrusion is completed, if the geometry looks like there is a hole in the middle of the extruded solids then Edge Nodes (Subdivision tab in the **Thin Shell Data** dialog box) was not selected when the disk was created. Perform the following steps to make the correction:

- Press <**Ctrl**>< **Z**> to undo the extrusion.
- Edit the disk to make the nodes edge nodes as follows:
- Select the disk in the drawing area.
- Select **Thermal** > **Edit.**
- In the **Thin Shell Data** dialog box, select the Subdivision tab and make the corrections. Click on **OK.**
- Return to Step 40.

Beer Can Example (Continued)

47.  or **Thermal** > **Edit.**

The Command line should now read:

`Select Objects or [Indiv MB]:`

48. Type all in the Command line.

The model in the drawing area is selected and `Select Objects or [Indiv MB]:` appears in the Command line area.

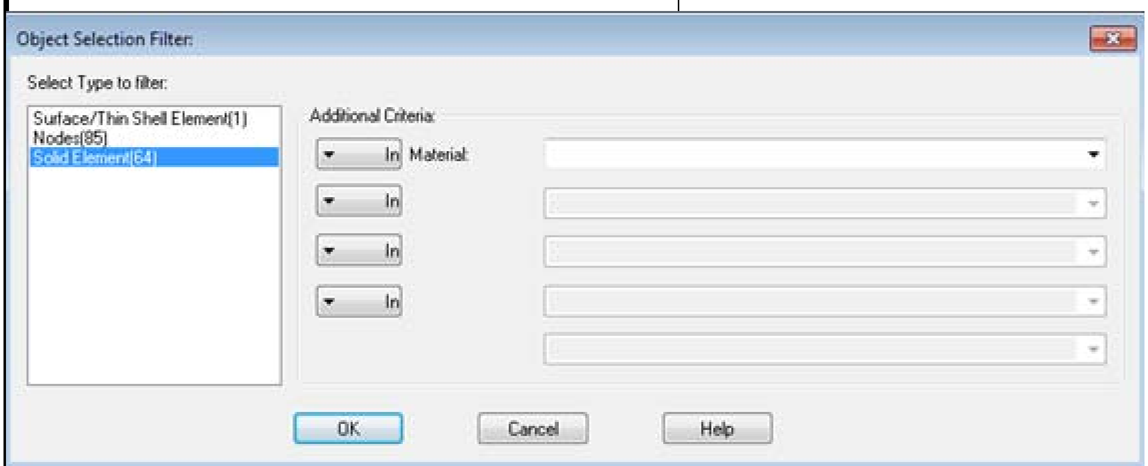
49. Press <Enter>.

The Object Selection Filter dialog box appears.

50. Click on **Solid Elements(64)** to select it.

The newly created solids must be edited to change their material to water. The properties of water are being used as an assumption of the properties of beer.

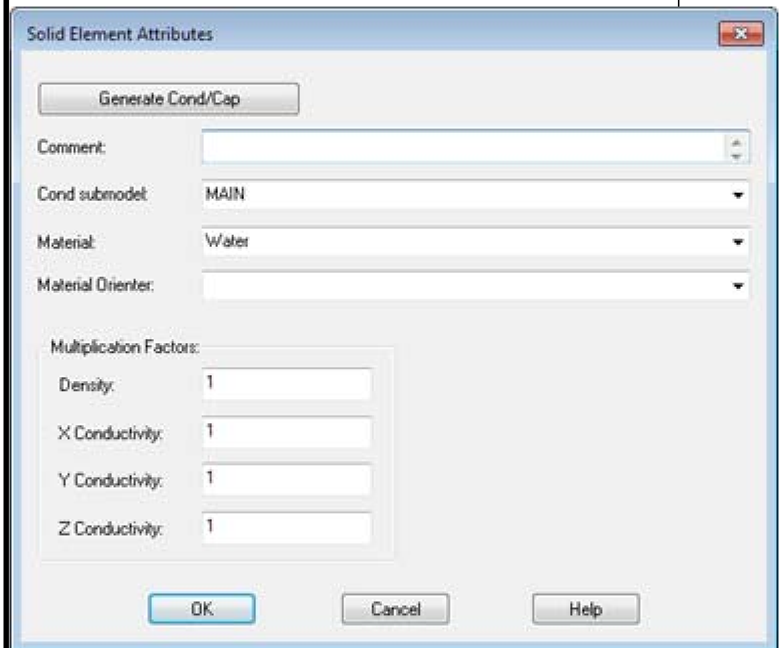
Only one type of object can be edited at a time. The Object Selection Filter makes it easy to select the desired object from the list.



Beer Can Example (Continued)

51. Select **OK** to close the selection filter and select the solid elements.

    The Solid Elements Attributes dialog box appears.



52. Click on the arrow next to the **Material** field and select Water from the drop-down list.
53. Select **OK** to close the dialog box.

    A Thermal Desktop/AutoCAD dialog box appears confirming the change.
54. Read the content of the dialog box and select **Apply Changes**.

Beer Can Example (Continued)

55. Select **Thermal** > **FD/Fem Network** > **Surface Coat Free Solid FEM Faces.**

    The Command line should now read:

    `Select the solids for free face calculations or [MB]:`

56. Type all in the Command line.

    The Command line should now read:

    `64 found`

    `Select the solids for free face calculations or [MB]:`

57. Press <Enter>.

    The Thin Shell Data - Multiple Surface/Element Edit Mode dialog box appears.

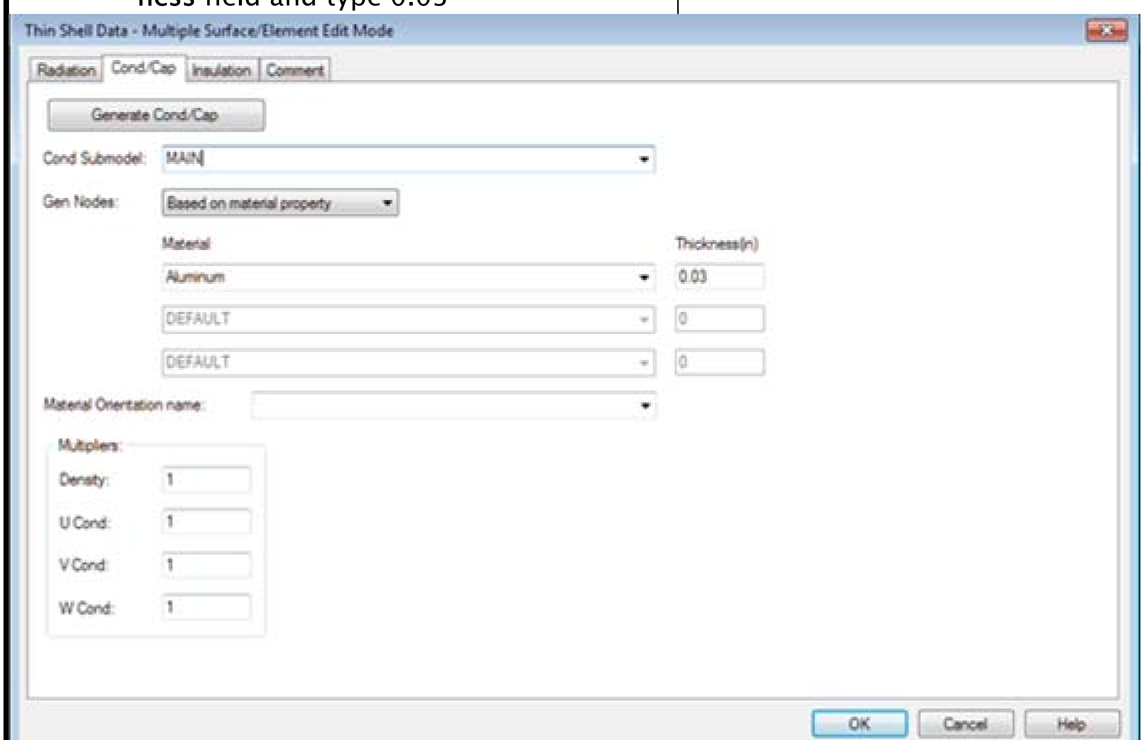
58. Click on the **Cond/Cap** tab.

59. Click on the arrow next to the **Material** field and select Aluminum from the drop-down list.

60. Highlight the current value in the **Thick-ness** field and type 0.03

Place the aluminum shell around the rest of the can. The solids will be surface coated to place the shell around the outer cylinder and the top.

Surface coating will place a planar element using the same nodes used by the solid elements. The command is smart enough to figure out that the outside faces are not hooked up to other solids (and creates the planar element there), while the inside faces are hooked to more than one solid, so those faces are not free.



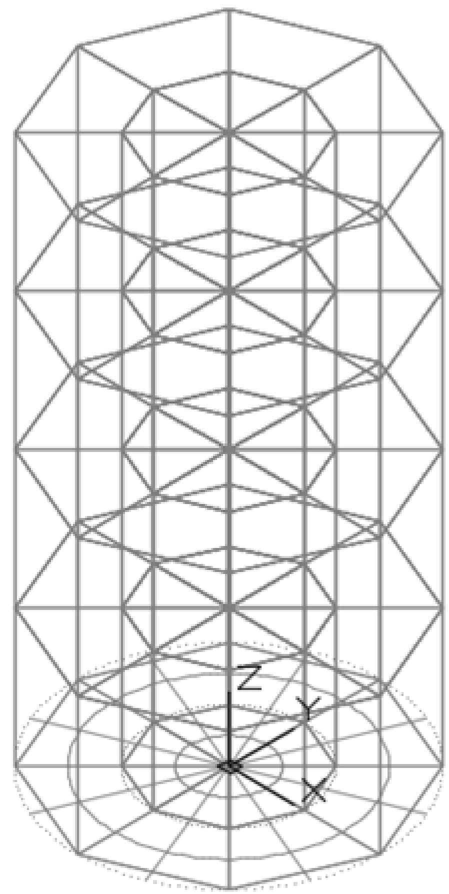
Beer Can Example (Continued)

61. Select **OK** to close the dialog box.

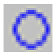
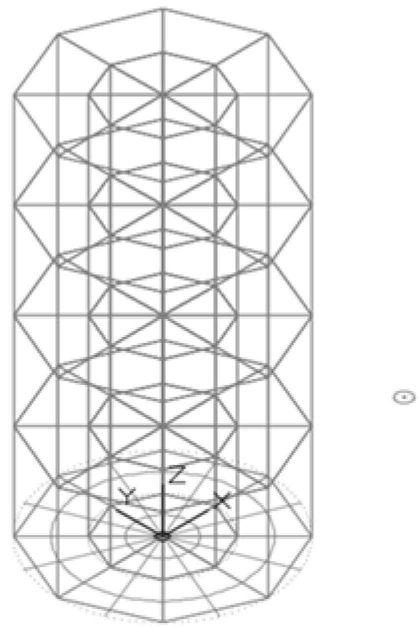
    A Thermal Desktop/AutoCAD dialog box appears confirming the change.

62. Read the content of the dialog box and select **Apply Changes**.

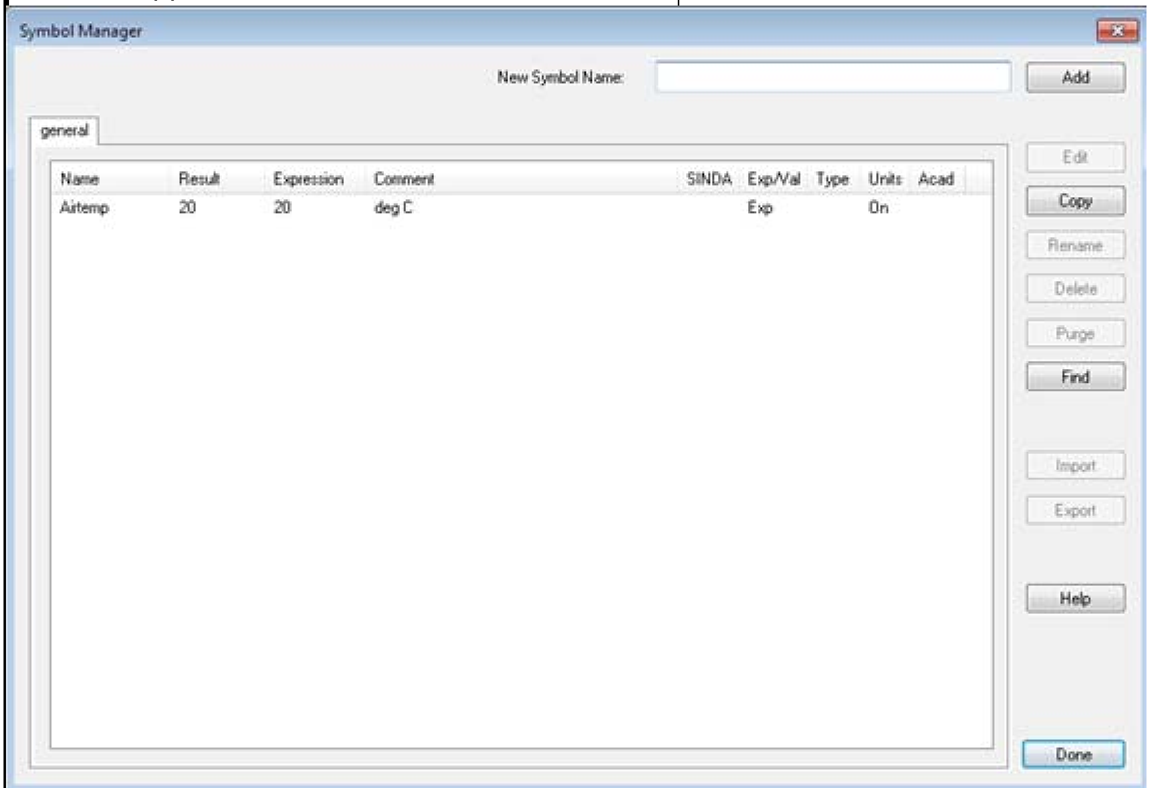
63.  or **View** > **Zoom** > **Extents**.



Beer Can Example (Continued)	
64.  or **Thermal** > **Edit.** `Select Objects or [Indiv MB]:` appears in the Command line area 65. Type all in the Command line. `198 found` `Select Objects or [Indiv MB]:` appears in the Command line area. 66. Press <Enter> to end the selection process. The Object Selection Filter dialog box appears. 67. Click on **Nodes(85)** in the **Select Type to Filter** field to select it. 68. Select **OK** to close the dialog box. The Node - Multi Edit Mode dialog box appears. 69. Highlight the current value in the **Initial temp** field and type 278.15. 70. Select **OK** to close the dialog box. A Thermal Desktop/AutoCAD dialog box appears confirming the change. 71. Read the content of the dialog box and select **Apply Changes.**	This part of the exercise edits all of the nodes to set their initial temperatures.

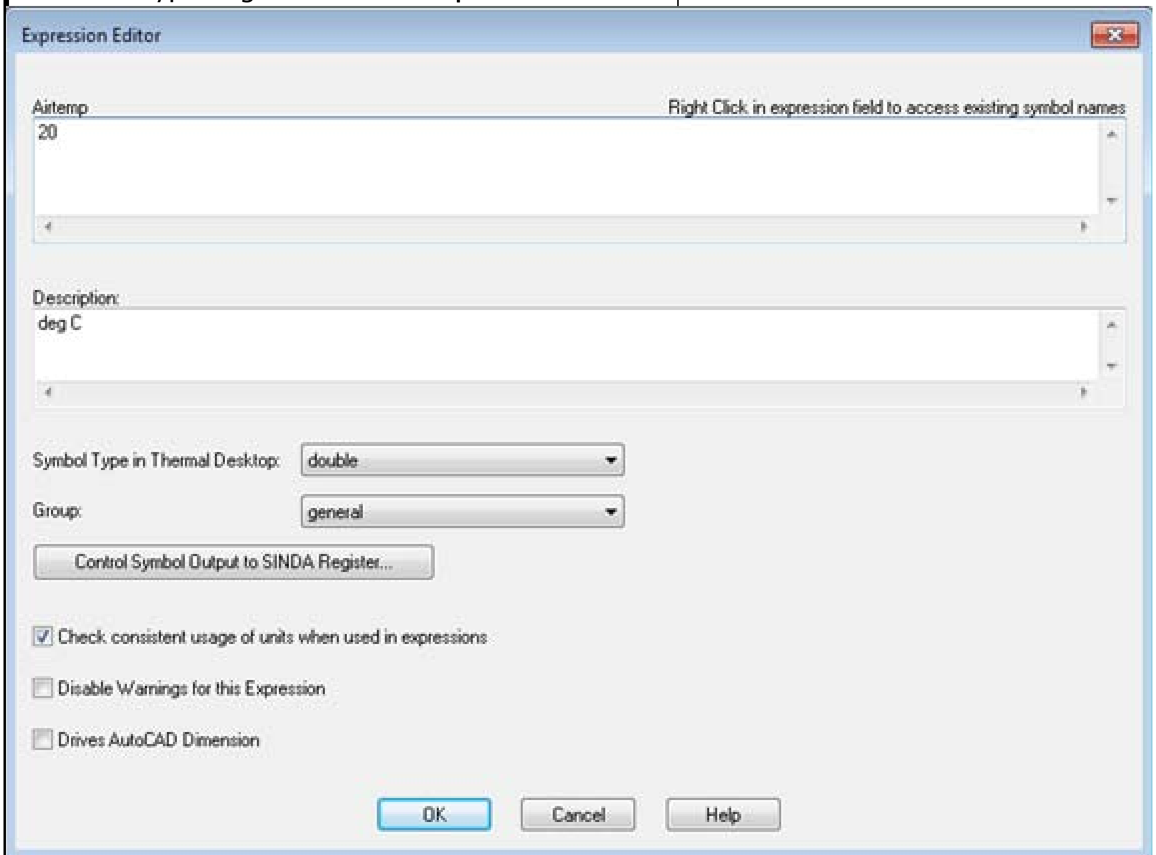
Beer Can Example (Continued)	
72.  or **Thermal** > **FD/Fem Network** > **Node.** The Command line should now read: `Enter location of node:` 73. Type 3,0,0 in the Command line. The node appears to the right of the model.	Create a node to connect to a convective conductor. This node will represent the ambient air temperature. 
74. Select the newly created node. 75.  or Thermal > Edit. The Node dialog box appears. 76. Highlight the current value in the **Sub-model** field and type Air. 77. Click on the radio button next to boundary in the **Type** field to select it (display a dot in the circle). 78. Double click in the **Initial temp** field. The Expression Editor dialog box appears.	The node will be edited to make it a boundary node and placed in submodel air. Notice that sometimes objects are selected before the command and some-times after the command. If objects are selected before the command, then the first operation of the command uses the “pre-selected” objects if they are the right type. If a command requires objects, but nothing is selected before the command, then the command line will query for the needed objects. The temperature of the node will be defined as a symbol, making it easy to set up a second case that has different air temperature. **Note**: The **Expression Editor** is dis-played when the mouse is double clicked in a field.

Beer Can Example (Continued)	
79. Select the **Add Symbol** button. The Symbol Manager dialog box appears.	Before using the symbol in the definition of the initial temperature, the symbol must be created.



80. Type Airtemp in the **New Symbol Name** field. 81. Select **Add.** An Expression Editor dialog box for Airtemp appears.	

Beer Can Example (Continued)	
82. Type 20 in the main entry field. 83. Type deg C in the **Description** field	



84. Select **OK** to close the Airtemp Expression Editor dialog box. The Symbol Manager dialog box is updated with the Airtemp information displayed.	**Note**: Symbols do not have units associated with them. When they are applied units are then assigned.

Beer Can Example (Continued)

85. Select **Done** in the Symbol Manager dialog.

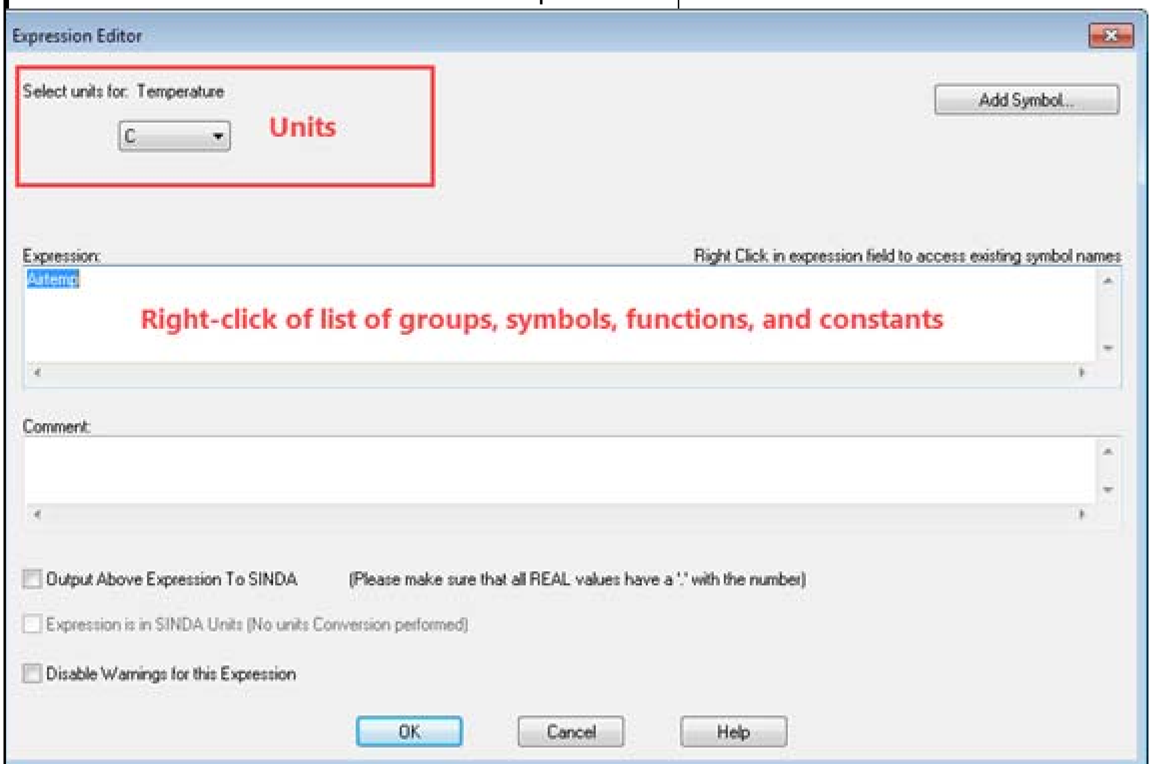
    The Expression Editor dialog box reappears.

86. Click on the arrow underneath **Select units for**: **Temperature** and select C from the drop-down list.

87. Right-click the **Expression** field, select **General** and then select Airtemp.

Now that the symbol for the air temperature has been defined, the expression for the temperature of the boundary node can be created.

Note: Symbols do not have units associated with them. When they are applied units are then assigned.



88. Select **OK** to close the Expression Editor dialog box.

Note: The **Initial Temp** value is now in bold type and should read **293.15**.

Beer Can Example (Continued)

89. Select **OK** to close the Node dialog box.

    The node’s shape is changed to reflect its designation as a boundary node.

90. Select **View** > **3D Views** > **Front**.

    The view changes. Note the UCS icon also moves to the lower left of the drawing area.

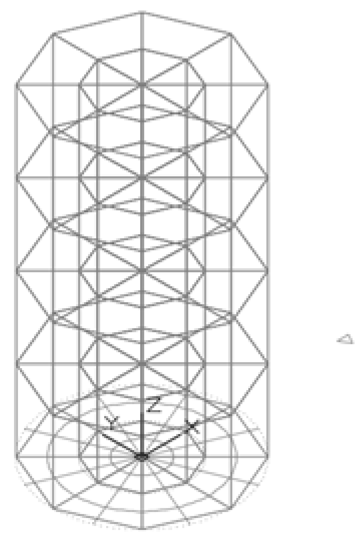
91. Type Zoom in the Command line.

    The Command line should now read:

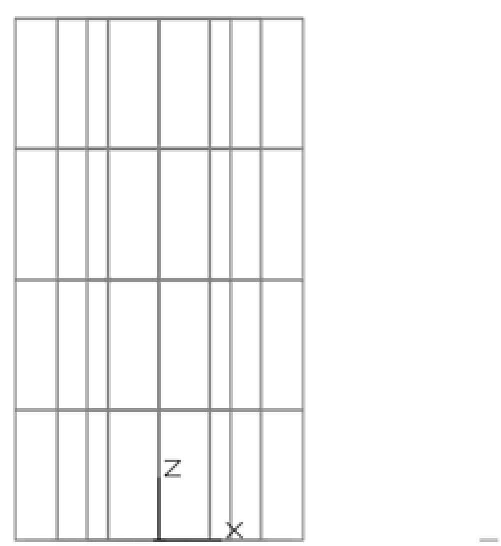
    ```
    All/Center/Dynamic/Extents/Previous/Scale/Window/Object <real time>:
    ```

92. Type 0.9x in the Command line.

Note: When this portion of the exercise is completed, the shape of the node changes to designate that it is now a boundary node.



Change the view from the current SW Isometric to a Front view.
The view should look as follows. Note the new node in the lower right-hand corner.



Beer Can Example (Continued)

A new layer is to be created for the conductors to reside on.

93.  or type layer in the Command line.

Note: The menu selection **Format** > **Layer** may also be used.

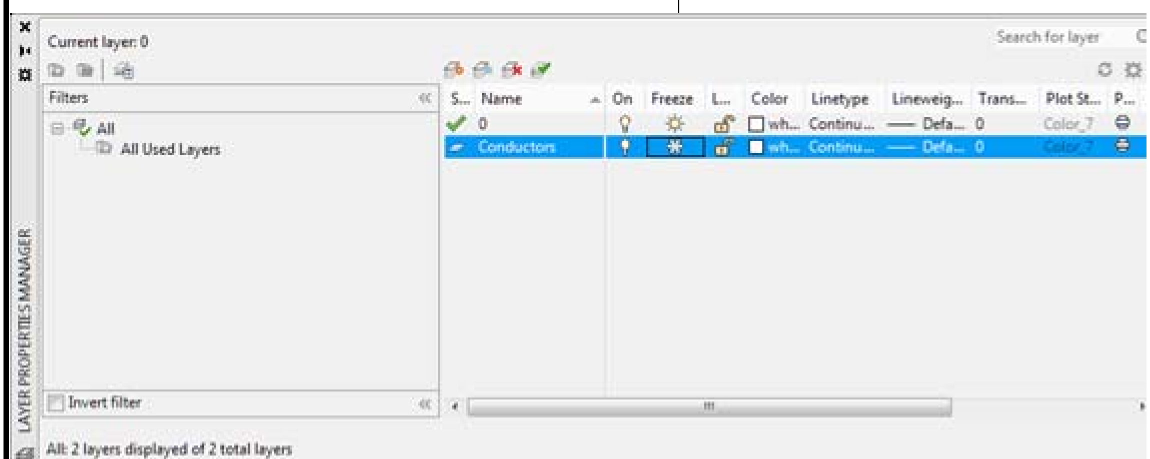
The Layer Properties Manager dialog box appears.

94.  to create a new layer.

A new layer named Layer1 is added.

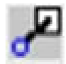
95. Highlight the name Layer1 (if not already highlighted) and type **Conductors** to change the name of this newly created layer.

96. Select the Freeze icon (sun) for the Conductors layer to freeze it (change the sun to a snowflake).



97. Close the Layer Properties Manager dialog box.

Beer Can Example (Continued)

98.  or **Thermal** > **FD/Fem Network** > **Node to Surface Conductor**.

    The Command line should now read:

    `Select node or [MB]:`

99. Click on the boundary node (lower right on the screen).

    The Command line should now read:

    `Select surfaces or [MB]:`

100. Select surfaces: Select from 1 to 2 as shown in to the right and as noted below:

     - Using the example to the right as a guide, click the left mouse outside and above the upper left corner of the surface area (1).

     The Command line should now read:

     `Specify opposite corner:`

     - Position the mouse outside and below the opposite, lower right corner of the surface area as shown in the example and click the left mouse button (2). Note that as the mouse is moved, a box is drawn around the area.

     The Command line should now read:

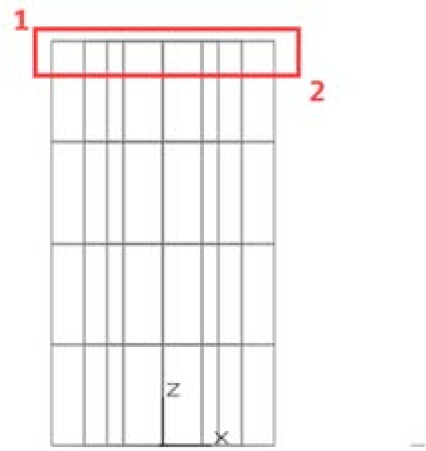
     `Select surfaces:`

101. Press <Enter>.
102. Select the new conductor.

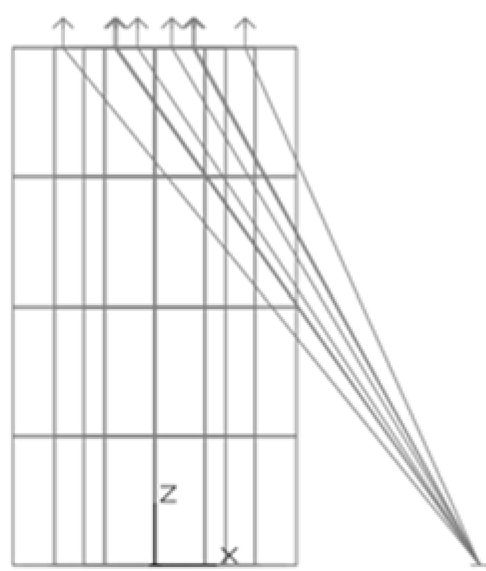
Note: The new conductor set can be selected by picking any line of the set.

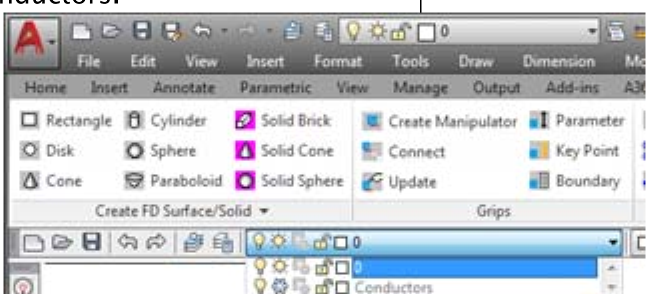
The next steps create the conductors and connect them to the surface.

When prompted to select the surface areas on the beer can, it is important to drag-select from the top left to bottom right since selecting in the reverse direction has a different meaning in AutoCAD (see example below).



A set of eight lines (representing the conductor) from the boundary node to the surface area are displayed.



Beer Can Example (Continued)	
103. **Thermal** > **Edit**. The Conductor dialog box appears. 104. Type Top Convection in the **Comment** field. 105. Click on the **Type** arrow and select **Natural Convection Horizontal Flat Plate Upside** from the drop-down list. 106. Select **Change Fluid**. 107. Expand Library and select **Air (Perfect Gas)** 108. Select **OK** to close the fluid selection. 109. Highlight the current value in the **Area/Perimeter** field and type 0.65625 110. Select **OK** to close the dialog box.	Edit the new conductor. For the disk: ■ radius = 1.3125. ■ area = pi*r^2 ■ perimeter = pi*r*2 ■ Area/Perimeter = radius/2 Change the fluid to be air
111. Select the new conductor. 112. Click on the Layer Control drop-down in the upper right toolbars, as shown, and select Conductors.  113. Select **Close** to confirm the change and close the dialog box. The conductor moves to the Conductor layer, which is frozen, and disappears from the screen.	This part of the exercise moves the conductor to the Conductor layer that was frozen in the previous step. Doing this will make the display less cluttered.

Beer Can Example (Continued)

114. **Thermal** > **FD/Fem Network** > **Node To Surface Conductor**.

     The Command line should now read:

     `Select node or [MB]:`

115. Select the boundary node (lower right).

     The Command line should now read:

     `Select surfaces or [MB]:`

116. Draw a selection box from points 1 to 2 as shown in figure to the right and as noted below:

     - Using the example to the right as a guide, click the left mouse at the lower right area of the surface area (1).

     The Command line should now read:

     `Specify opposite corner:`

     - Position the mouse on the opposite, upper left of the surface area as shown in to the right and click the left mouse button (2). Note that as the mouse is moved, a box is drawn around the area.

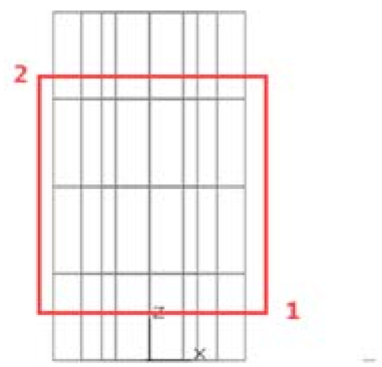
     The Command line should now read:

     `Select surfaces or [MB]:`

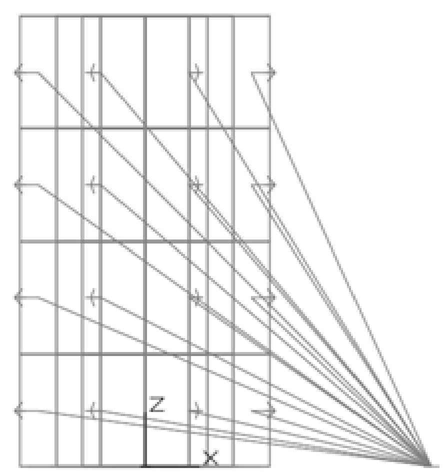
117. Press <Enter>.

     A set of sixteen lines (representing the conductor) from the boundary node to the surface area are displayed.

The drawing below shows the correct point selection order to be used for the next steps. **It is important to begin in the lower right area of the beer can (first point, 1), as shown, and move the mouse to the upper left area (second point, 2).** When selecting from the bottom right to the top left, any entity that is fully or partially enclosed will be included in the selection set. If the selection order is changed (point 2 and then point 1) only the items that are fully included in the box will be included in the selection set.



When these steps are completed, the screen should appear similar to the example below.

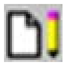


Beer Can Example (Continued)	
118. Select the new conductor. 119. **Thermal** > **Edit**. The **Conductor** dialog box appears. 120. Type Side Convection in the **Comment** field. 121. Click on the Type arrow and select Natural Convection Vertical Cylinder - Isothermal from the drop-down list. The content of the Conductor dialog box changes to reflect the selection. 122. Select **Change Fluid.** 123. Expand **Library** and select Air (Perfect Gas) 124. Select **OK** to close the fluid selection. 125. Highlight the current value in the **Height** field and type 4.75. 126. Highlight the current value in the **Diameter** field and type 2.625. 127. Select **OK** to close the Conductor dialog box.	
128.  or select **Thermal** > **Model Browser**. The Model Browser appears. 129. Select **List By** > **Conductors** in the Model Browser. The Model Browser tree displays the Conductor Tree 130. Right-click on **Cond-Side Convection** under **MAIN** and select **Change Layer** > **Conductors.** The conductor moves to the Conductor layer, which is turned off, and disappears from the screen. 131. Close or minimize the Model Browser window.	As with the first conductor, this new conductor will be moved from layer 0 to the layer Conductor so that it does not clutter up the display.

Beer Can Example (Continued)	

132. Select **Thermal** > **Model Checks** > **List Duplicate Nodes**.
The Command area should now show:
`Listing of duplicate nodes`
`No duplicate nodes were found`
Note: If the statement does not appear in the command line, press <F2> to view the complete Command line comments.
	Before any geometry building is complete, it is important to look for duplicate nodes. If any duplicate nodes are found, it may be necessary to use the Resequence IDs command to renumber them.
	Note: See 7.3 Resequence IDs.
	Look at the output and see if any are found.

Beer Can Example (Continued)

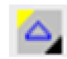
133.  or **Thermal** > **Preferences**.

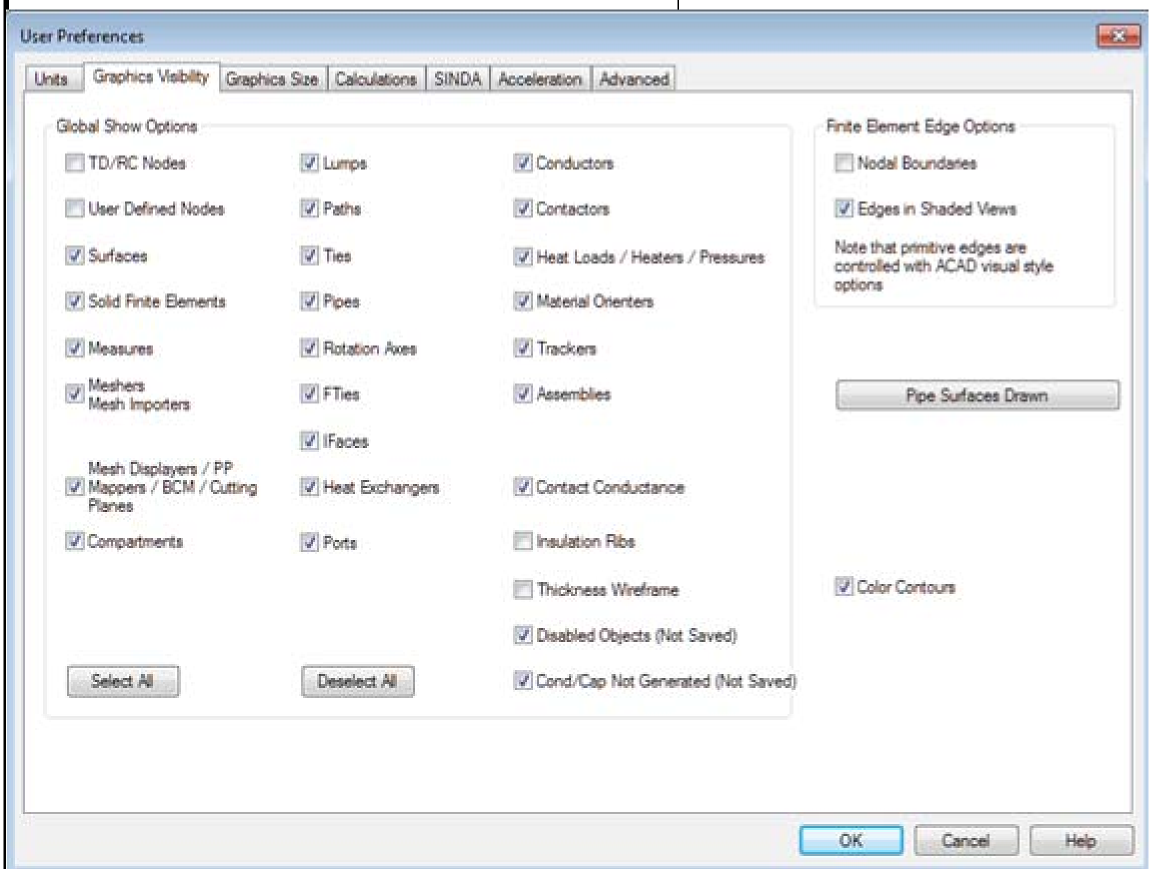
The User Preferences dialog box appears.

134. Select the **Graphics Visibility** tab if not already displayed.

135. Click on **User Defined Nodes** to clear it (remove the check mark from the box).

Turn off the display of the air node. Alternatively, the visibility of some objects can be toggle off and on using icons in the toolbars. For User Defined Nodes, it is the icon:





136. Select **OK** to close the dialog box.

The air node disappears from the drawing area.

Beer Can Example (Continued)	
137.  or type layer in the Command line. The Layer Properties Manager dialog box appears. 138.  to create a new layer. A new layer named Layer1 is added to the existing layers.	Create a new layer called RightSide, which is where the right side of the beer can will be placed.
139. Highlight the name Layer1 if not already highlighted. Type **RightSide** to change the name of this newly created layer. 140. Select the Freeze icon (sun) for the RightSide layer to turn freeze it (change the sun to a snowflake), if it is not already frozen. 141. Close the Layer Properties Manager dialog box.	New layers use the settings of the selected layer when they are created. If the Conductor is selected when the new layer is created, then the new layer will be frozen.

Beer Can Example (Continued)

142. Select the right side of the beer can by drawing a selection box from points 1 to 2 as shown on the right and as noted below:

    - Using the example to the right as a guide, click the left mouse at the lower right area of the surface area (1).

    `Specify opposite corner:` appears in the Command line area.

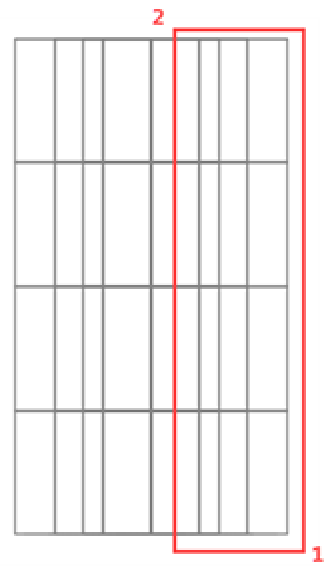
    - Position the mouse as shown in to the right (above and to the right of the middle line) and click the left mouse button (2). Note that as the mouse is moved, a box is drawn around the area.

    The right side of the can is selected in the drawing area.

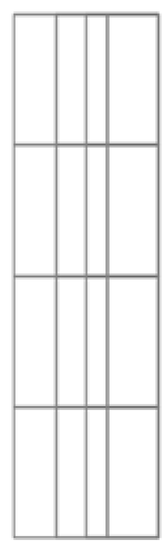
143. Click on the **Layer Control** drop-down in the upper right toolbars, as shown, and select **RightSide**.

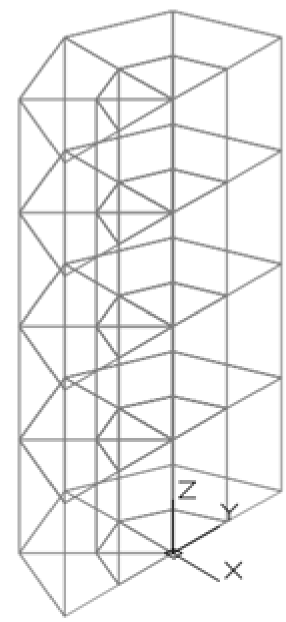
144. Select **Close** to confirm the change and close the dialog box.

Split the beer can into two sides so that the temperatures in the middle of the beercan can be determined later in the exercise. \



Once the selected right side is moved to the RightSide layer, the drawing area should look similar to the example below:



Beer Can Example (Continued)	
145. Select **View** > **3D Views** > **SE Isometric.** The new view appears in the drawing area.	The new view should now look as follows. 

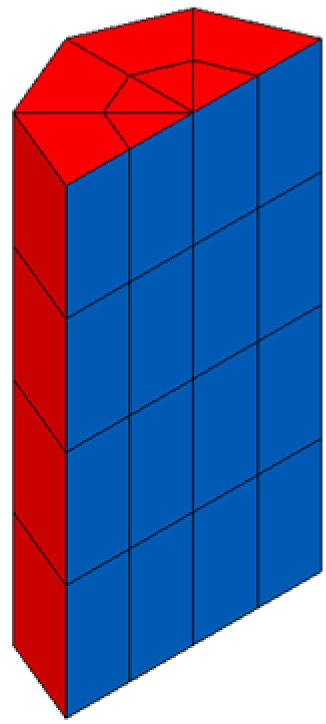
Beer Can Example (Continued)

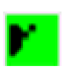
146.  or **Thermal** > **Model Checks** > **Color by Property Value** > **Conductivity.**

Note: If the blue is a little dark, feel free to rotate a little bit to see if better. This is also a good time to review graphics settings (19.4 Graphics Settings).

147. Select **View** > **3D Views** > **SE Isometric.**

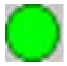
This command verifies that the materials are set correctly. The picture should look similar to the view below (you may need to rotate the model) with the aluminum being about 6 and the water being about 0.01. If the values are not right, edit the material property of the incorrect entities. Some elements may appear wrong, but this is likely a graphics issue.



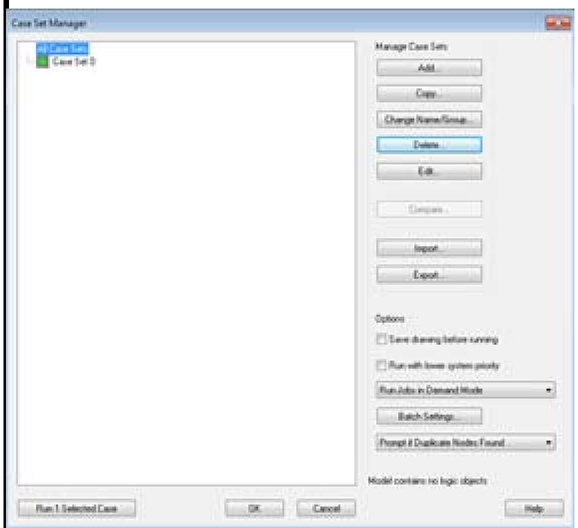
148.  or **Thermal** > **Model Checks** > **Color by Property Value Off.**

The model reverts back to the previous wireframe view.

Beer Can Example (Continued)

149.  or **Thermal** > **Case Set Manager**.

The Case Set Manager dialog box appears.



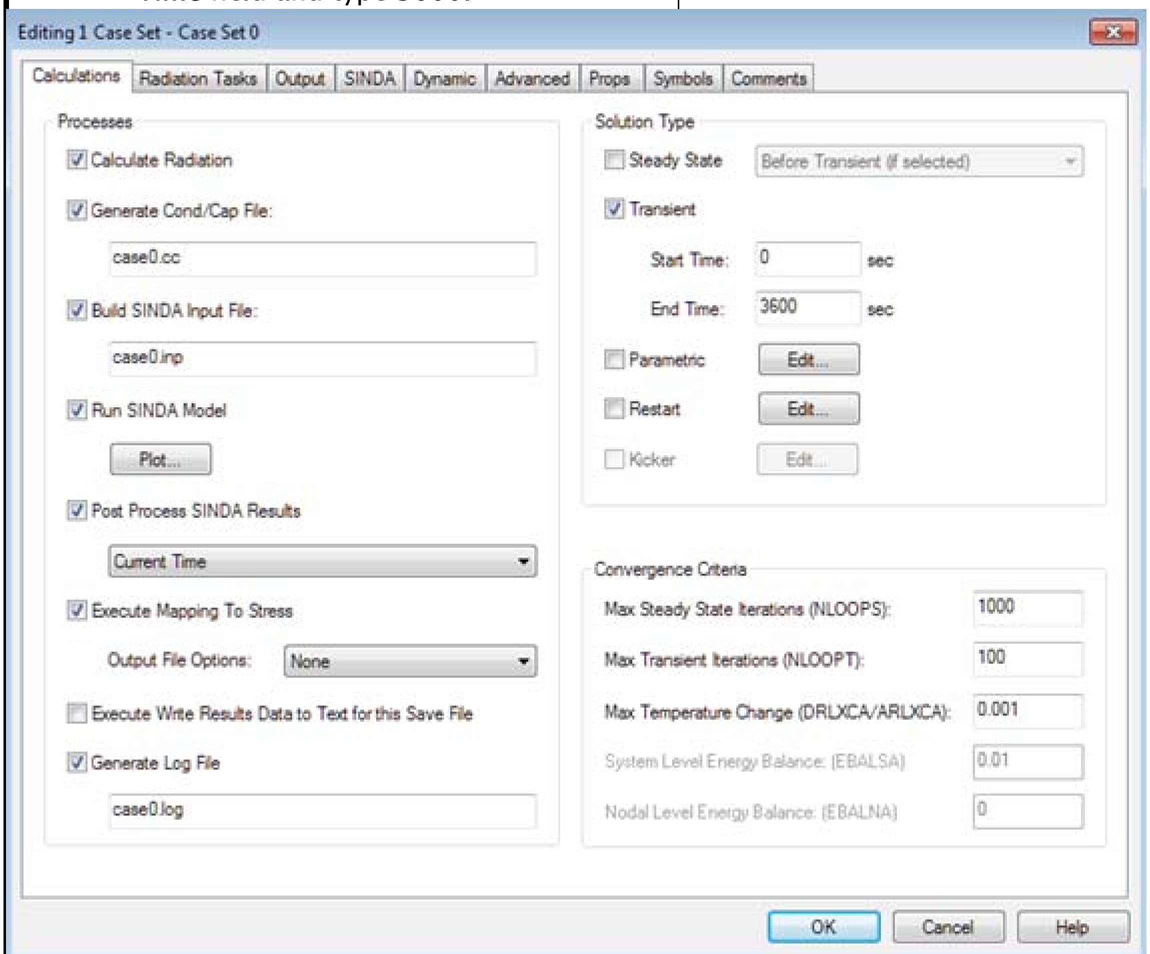
150. Select **Edit**.

The Case Set Information dialog box appears. *(next page)*

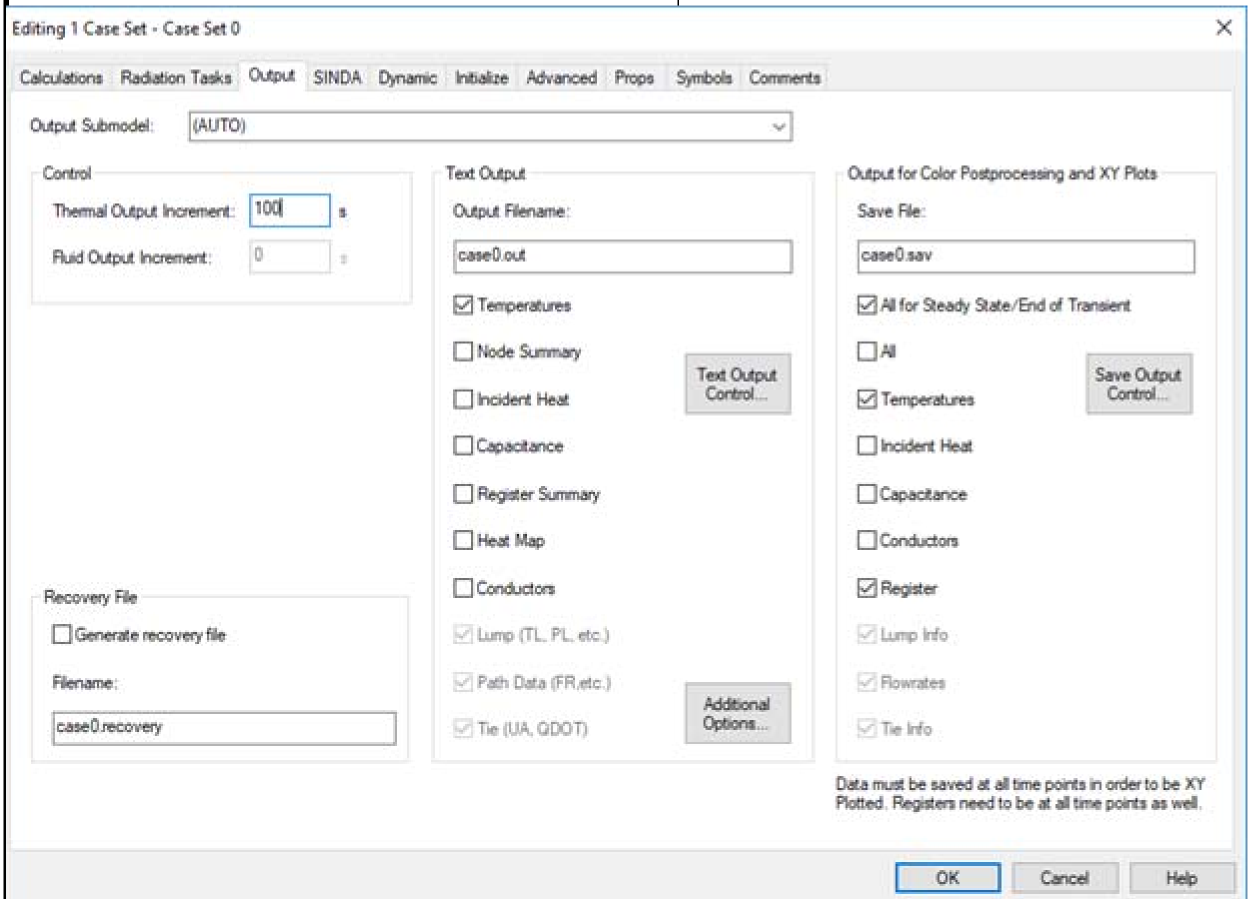
The **Case Set Manager** changes the view from the geometric model to temperatures with the click of a button.

The default process is to run a steady state case, but a transient run is what is needed here.

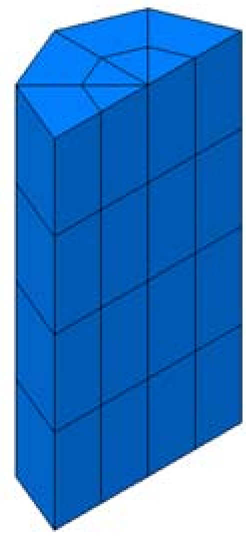
Beer Can Example (Continued)	
151. Select the **Calculations** tab. 152. Click on **Steady State** in the **Solution Type** field to clear it (remove check mark from the box). 153. Select **Transient** in the **Solution Type** field to select it (place a check mark in the box). 154. Highlight the current value in the **End Time** field and type 3600.	On the Calculations tab, the Solution type is chosen and, since a transient analysis is desired, an end time is set.



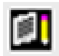
Beer Can Example (Continued)	
155. Select the **Output** tab. 156. Highlight the current value in the **Thermal Output Increment** field and type 100.	The Output Increment defines how often during the solution the chosen values will be written to the output files.



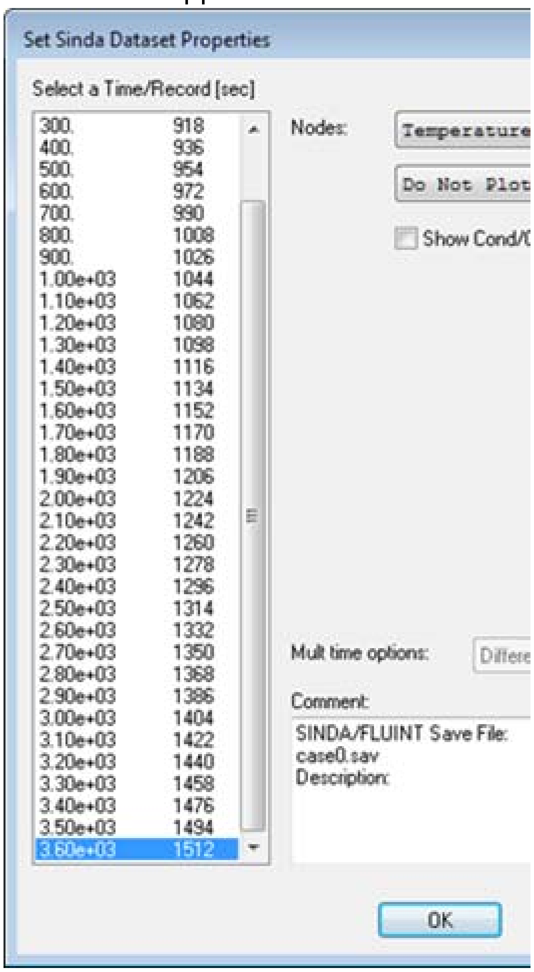
157. Select **OK**. The Case Set Manager dialog box is on the screen.	

Beer Can Example (Continued)	
158. Select **Run 1 Selected Case**. A SINDA/Fluint Run Status dialog box appears stating the successful completion of the processor. 159. Select **OK** to close the dialog box. The model changes from the geometric view.	When the run is complete the temperature view should look similar to the following.  If the solution fails, please check the air node temperature. If it is accidentally input as 20K, the solution will fail.

Beer Can Example (Continued)

160.  or **Thermal** > **Post Processing** > **Edit Current Dataset.**

The Set SINDA Dataset Properties dialog box appears.

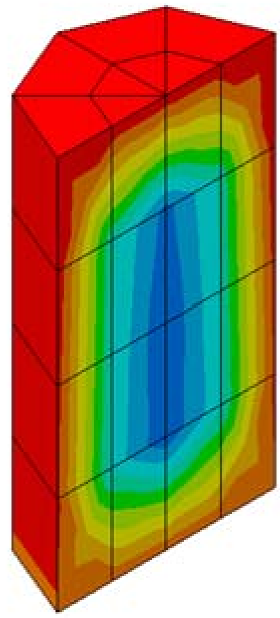


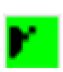
161. Scroll down the list in the **Select a Time/ Record [sec]** field and select 3600 (3.60e+003).

162. Select **OK**.

After the solve is completed, the initial temperatures are displayed on the model in the postprocessing state.

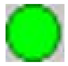
Note: If the colors do not look right, please 19.4 Graphics Settings.



163.  or **Thermal** > **Post Processing** > **Post Processing Off.**

The model returns to the geometric view in the drawing area.

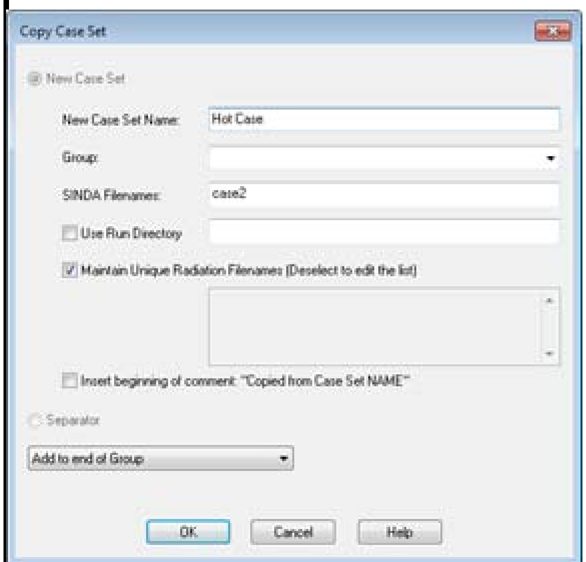
Beer Can Example (Continued)

164.  or **Thermal** > **Case Set Manager**.

The Case Set Manager dialog box appears.

165. Select **Copy**.

The **Copy Case Set** dialog box appears.



166. Highlight the current value in the in the **New Case Set Name** field and type Hot Case.

167. Select **OK** to close the dialog box.

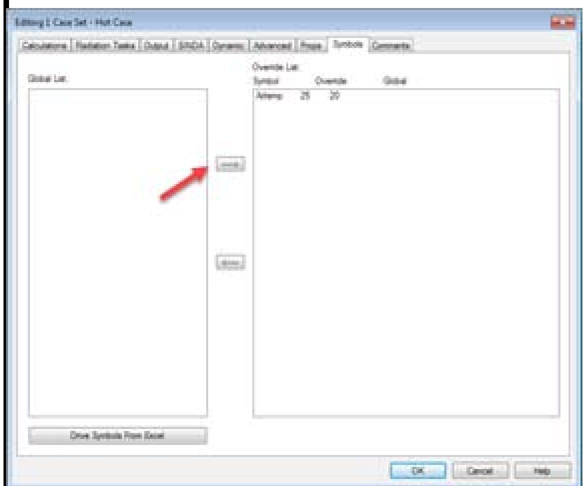
The Case Set Manager dialog box updates to reflect Hot Case in the Case Sets field.

168. Select **Hot Case**.

169. Select **Edit**.

The Editing 1 Case Set - Hot Case dialog box appears.

Create a hot case where the air temperature is 25 °C.
By overriding the global definition of 20 °C with 25 °C, the new case can be run quickly and it will be able to go back to it at a later time.
When this case is run, all the SINDA files will go to case1.*
Once the run is finished, edit the post-processing dataset to change to the end time.

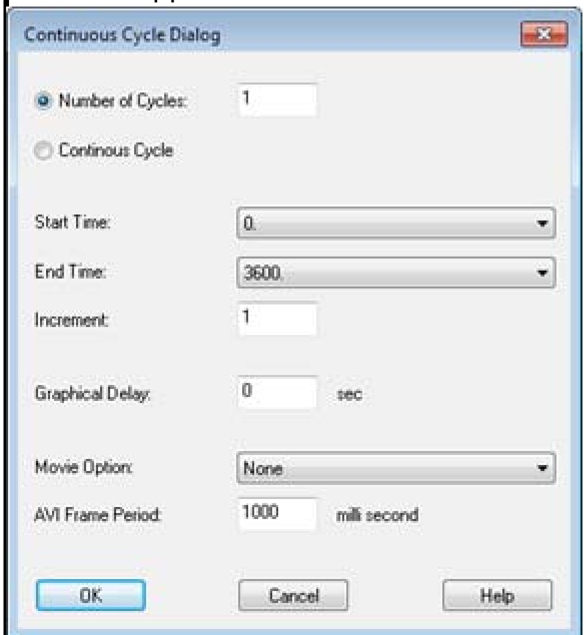
Beer Can Example (Continued)	
170. Select the **Symbols** tab. 171. Select **Airtemp** in the **Global List** field to highlight it. 172. Click on the right arrow located in the center of the dialog box.  The Expression Editor (Airtemp) dialog box appears. 173. Highlight the current value in the main entry field (20) and type 25 174. Select **OK** to close the dialog box. The Case Set Information - Hot Case dialog box displays the change.	

Beer Can Example (Continued)	
175. Select **OK** to close the Case Set Information - Hot Case dialog box and re-display the Case Set Manager dialog box. 176. Select **Run 1 Selected Case**. A SINDA/Fluint Run Status dialog box appears stating the successful completion of the processor. 177. Select **OK** to close the dialog box. The model changes from the geometric view. 178.  or **Thermal** > **Post Processing** > **Edit Current Dataset.** The Set SINDA Dataset Properties dialog box appears. 179. Scroll down the list in the Select a Time/Record [set] field and select 3600 (3.60e+003). 180. Select OK.	
181.  or **Thermal** > **Utilities** > **Capture Graphics Area**.	The **Thermal** > **Utilities** > **Capture Graphics Area** will save the current graphics window to ScreenCapture1.bmp. The program determines the lowest ScreenCapture# that it can use so as to not overwrite an existing file. For example, a second command would save to ScreenCapture2.bmp. To verify the graphic is saved, open the beercan directory folder and ScreenCapture1 will be included.

Beer Can Example (Continued)

182.  or **Thermal** > **Post Processing**> **Animate Through Time.**

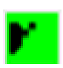
The Continuous Cycle Dialog dialog box appears.



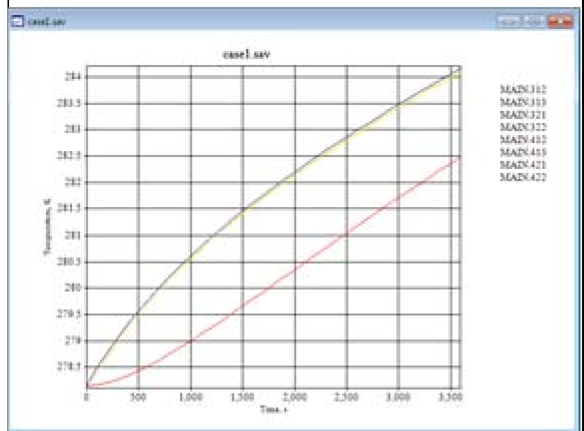
183. Select **OK**.

184. View the screen.

This command will animate through all the times on the postprocessing file.

185.  or **Thermal** > **Post Processing** > **Post Processing Off.**

The model returns to the geometric view in the drawing area.

Beer Can Example (Continued)	
186. Select an element in the drawing 187.  or select **Thermal** > **Post Processing** > **X-Y Plot Data vs. Time.** 188. View the results.	This command will bring up the external XY Plotting program. This program will plot the transient for nodes of the element that have been selected.  *Note: The results will be different depending upon what was selected in the drawing area.* The user can change the nodes displayed by selecting the Edit > Add/Edit menu command in EZXY. The nodes being displayed and any plot customization can be saved to a file that can then be brought up external to Thermal Desktop.
189. Select **File** > **Exit.** A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes. 190. Select **Yes**.	Exit Thermal Desktop and save as prompted.

20.6 Conduction and Radiation Using Finite Elements

What will be learned:

- Creating planar finite elements
- Extruding and revolving planar elements into solid finite elements
- Verifying proper connections of elements
- Surface coating finite element solids for radiation, area contact, or insulation

Prerequisites:

- 20.2 Setting Up a Template Drawing

In this example, a finite element model will be created with fixed temperatures at either end. The process begins by creating a single quad element. The AutoCAD *array* command will be used to create a grid of elements. The planar quad elements will be extruded and revolved into 3D solid elements. The solid elements will be surface-coated with zero-thickness planar elements to be used in a later tutorial for assigning radiation properties. Temperature boundary conditions will be applied and the model will be solved by SINDA. Radiation will be optionally added to the analysis at the end of the tutorial.

Finite Element Example
1. Copy the template thermal.dwg file created in the first tutorial to the \Tutorials\Thermal Desktop\finiteElement directory.
Note: Be sure to hold the <Ctrl> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved.
2. Rename the copied template file to fe1.dwg.
3. Start Thermal Desktop by double clicking on the fe1 drawing file icon in the finiteElement directory.

Finite Element Example (Continued)

4.  or select **Thermal > Thermophysical Properties > Edit Property Data.**

   The Edit Thermophysical Properties dialog box appears.

5. Type Aluminum in the New property to add field.

6. Select the Add button.

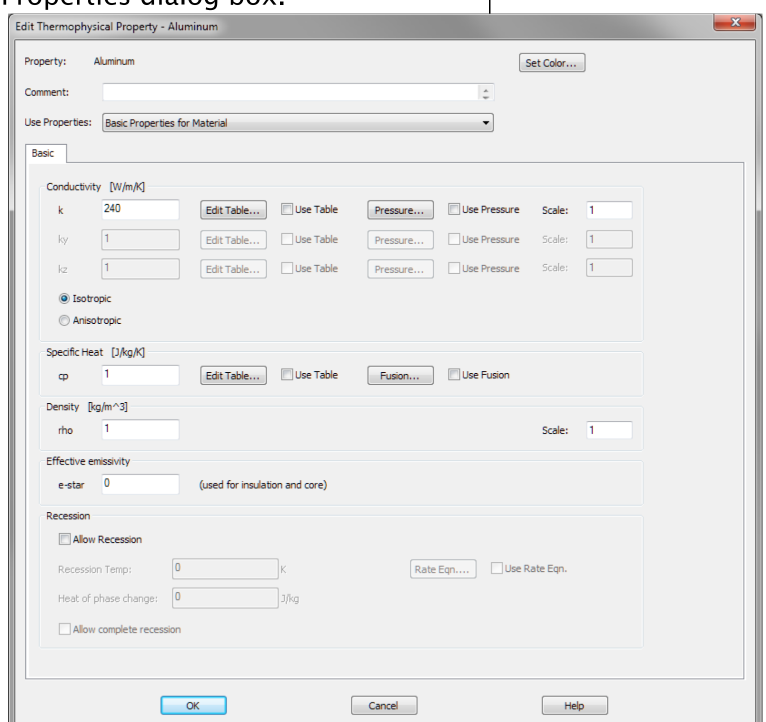
   The Thermophysical Properties dialog box appears.

7. Highlight the current value in the Conductivity k field and type 240.

8. Select OK to close the Thermophysical Properties dialog box.

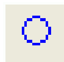
   The Edit Thermophysical Properties dialog box reappears with the updated Aluminum value displayed in the main property/description field.

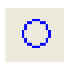
9. Select OK to close the Edit Thermophysical Properties dialog box.

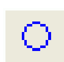


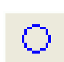
Create the property aluminum with a conductivity of 240. Only the conductivity will be entered/updated. This will be a steady state example.

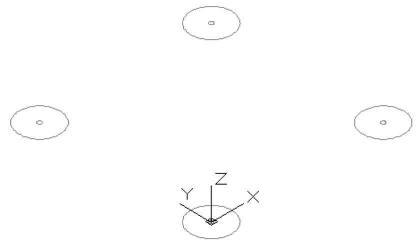
Depending upon where the user started the tutorials, Aluminum may already exist. If so, perform Step 4, double click on Aluminum in the **Edit Thermophysical Properties** dialog box and move to Step 7.

<table>
<tr><th colspan="2">Finite Element Example (Continued)</th></tr>
<tr>
<td>
10.  or Thermal > FD/FEM Network > Node.
The Command line should now read:
<code>Enter location of node:</code>
11. Type 0,0 in the Command line.
The first node is created at the origin.

12.  or Thermal > FD/FEM Network > Node.
The Command line should now read:
<code>Enter location of node:</code>
13. Type 1,0 in the Command line.
The second node is created.

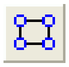
14.  or Thermal > FD/FEM Network > Node.
The Command line should now read:
<code>Enter location of node:</code>
15. Type 1,1 in the Command line.
The third node is created.

16.  or Thermal > FD/FEM Network > Node.
The Command line should now read:
<code>Enter location of node:</code>
17. Type 0,1 in the Command line.
The fourth node is created.

18.  or View > Zoom > Extents.
</td>
<td>
This part of the exercise creates 4 nodes. When finished, the model should look similar to the example below.


To repeat a command, the user can press <ENTER> or right click.
</td>
</tr>
</table>

Finite Element Example (Continued)

19.  or **Thermal > FD/Fem Network > Element.**

    The Command line should now read:

    `Select nodes for linear element or [MB]:`

20. Select node 1, the node at the axis of the UCS icon.

    The Command line should now read:

    `Select nodes for linear element or [MB]:`

21. Select node 2, the node to the left of the first node.

    The Command line should now read:

    `Select nodes for linear element or [MB]:`

22. Select node 3, the node above the first node.

    The Command line should now read:

    `Select nodes for linear element or [MB]:`

23. Select node 4, the node to the right of the first node.

    The Command line should now read:

    `Select nodes for linear element or [MB]:`

24. Press <Enter> to end the selection process.

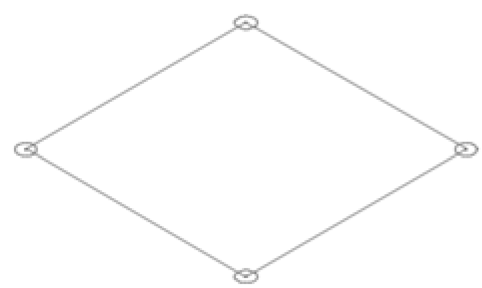
    Lines appear on the screen connecting the four nodes.

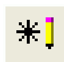
A quad element is being created from the four new nodes.

The order in which the nodes are picked is extremely important. Refer to the drawing below to select the nodes.

The order follows the right hand rule to determine which side is up. For example, picking the nodes in the order 1,2,4,3 would produce a quad where the diagonals would cross.

When the element is created, the view should be similar to the example below:

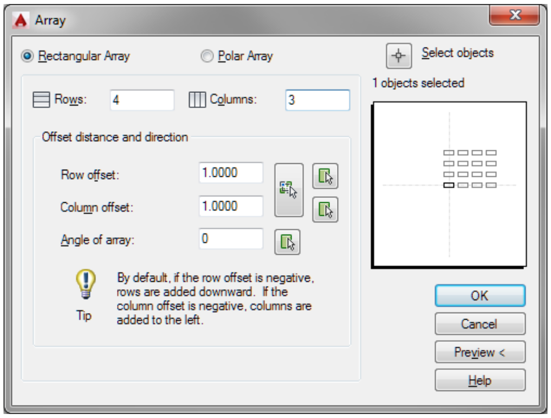


Finite Element Example (Continued)	
25. Select the new quad element to highlight it. 26.  or **Thermal** > **Edit.** The Thin Shell Data dialog box appears. 27. Select the Cond/Cap tab. 28. Click on the arrow next to the Material field and select Aluminum from the drop-down list. 29. Select OK to close the dialog box.	The element is being edited to apply the material property created earlier. The thickness does not matter since this element is used for an extrusion.

Finite Element Example (Continued)

30. Select the new quad element.
31. Type ARRAYCLASSIC.

    The Array dialog box appears.

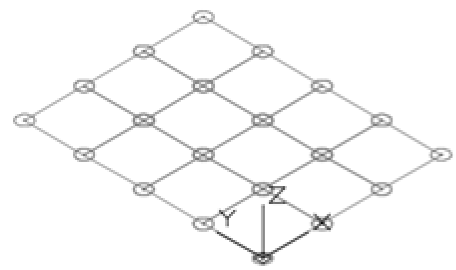


32. Select Rectangular Array if not already selected (display a dot in the circle).
33. Highlight the current value in the Rows field and type 4 if a different value is displayed.
34. Highlight the current value in the Columns field and type 3.
35. Highlight the current value in the Row Offset field and type 1 if a different value is displayed.
36. Highlight the current value in the Column Offset field and type 1 if a different value is displayed.

Note: The Row Offset and Column Offset fields display as 1.0000.

37. Select OK to close the dialog box.
38.  or **View** > **Zoom** > **Extents**.

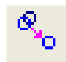
This part of the exercise uses the Array command to create a 4x3 grid of quad elements. When completed, the model should look similar to the view below.



Finite Element Example (Continued)	

39. Select **Thermal** > **Model Checks** > **Show Free Edges**.
The Command line should now read:
`Select the elements for free edge calculations or [MB]:`
40. Type all in the Command line.
The Command line should now read:
`Select the elements for free edge calculations or [MB]:`
41. Press <Enter>.
The grid lines turn red and the Command line area should show:
`48 individual edges found`
`48 free edges found`
	The next steps use the Show Free Edges command to determine if these nodes are properly connected.
	
	Once the Show Free Edges command is executed, notice that red lines cover the whole grid of the model. What has happened is that the Array command has copied the nodes as well as the elements and, therefore, the nodes are lying on top of each other. If the user output the model at this point (after resequencing the nodes), there would be no conduction between the elements.

Finite Element Example (Continued)

42.  or **Thermal** > **FD/Fem Network** > **Merge Coincident Nodes.**

    The Command line should now read:

    `Select nodes to be merged or [MB]:`

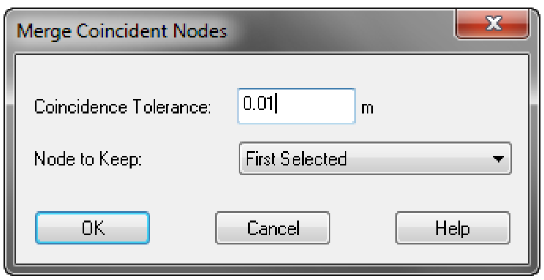
43. Type all in the Command line.

    The Command line should now read:

    `Select nodes to be merged or [MB]:`

44. Press <Enter>.

    The Merge Coincident Nodes dialog box appears.



45. Highlight the current value in the Coincidence Tolerance field and type .01.

46. Select OK to close the dialog box.

    A Thermal Desktop/AutoCAD dialog box appears asking for confirmation of the merge.

47. Select Yes.

This group of steps merges the coincident nodes.

Finite Element Example (Continued)	
48.  or type regen in the Command line. The array turns white. 49. Select **Thermal** > **Model Checks** > **Show Free Edges**. The Command line should now read: `Select the elements for free edge calculations or [MB]:` 50. Type all in the Command line. The Command line should now read: `Select the elements for free edge calculations or [MB]:` 51. Press <Enter>. The outside edge of the array turns red. 52.  or type regen in the Command line. The array turns white.	The regen command is performed here to clear the screen from the previous Show Free Edges and Merge Coincident Nodes commands. The free edges are checked again and now only the outlying edges are drawn in red.

Finite Element Example (Continued)

53. Select **Thermal** > **FD/FEM Network** > **Extrude Planar Elements into Solids**.

    The Command line should now read:

    `Select Planar Elements/Edge Conics for Revolve/Extrude or [MB]:`

54. Type all in the Command line.

    The Command line should now read:

    `12 found`

    `Select Planar Elements/Edge Conics for Revolve/Extrude or [MB]:`

55. Press <Enter>.

    The Command line should now read:

    `Select point to extrude from:`

56. Type 0,0 in the Command line.

    The Command line should now read:

    `Select point to define extrude vector/distance:`

57. Type 0,0,5 in the Command line.

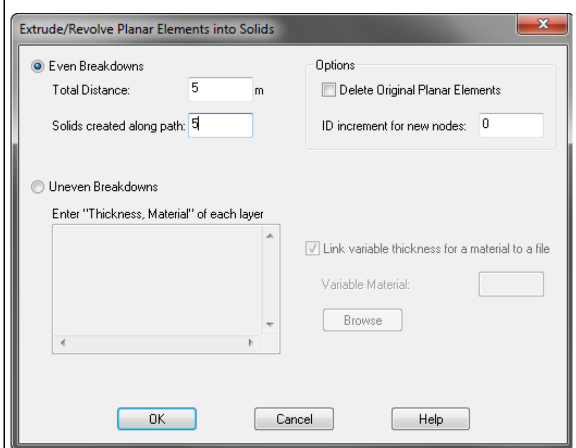
    The Extrude/Revolve Planar Elements into Solids dialog box appears.

58. Leave Even Breakdowns selected (dot in the circle).

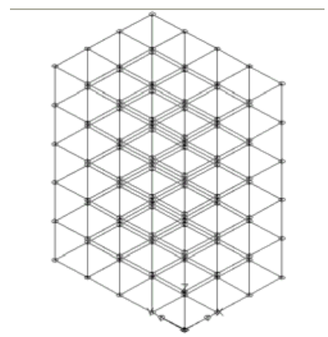
59. Highlight the current value in the Solids Created along path field and type 5.

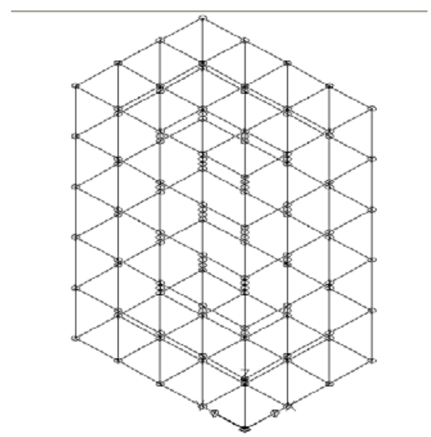
60. Select OK to close the dialog box.

The planar elements are extruded to make bricks. The vector given provides the distance of the extrusion.



Finite Element Example (Continued)	

61. Type zoom in the Command line.
The Command line should now read:
`Specify corner of window, enter a scale factor (nX or nXP), or`
`[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:`
62. Type extents in the Command line.
The view shifts to show the full array.
63. Type –vpoint in the Command line.
The Command line should now read:
`Specify a view point or [Rotate] <display compass and tripod>:`
64. Type –1,–1,0.9 in the Command line.
The view of the extruded model is rotated.
	The view of the model is zoomed to the extents of the drawing area and then rotated a little bit to move the model off of the isometric view. The model should look similar to the view below.
	

Finite Element Example (Continued)	
65.  or **Thermal** > **FD/Fem Network** > **Hide Solid Interior Faces**. The view in the drawing area shifts.	Since the model was rotated off the normal, notice that there are many double lines that can be seen. These are the edges of the interior of the model. These lines can clutter up the model, especially if the model is the meshed. The next step turns off these lines. By turning off these lines redisplays, rotations, and post-processing of the model will process faster. The calculation to hide the interior lines is only made when the user selects the command. Thus, if more geometry is added, or deleted, the user may need to re-execute the command to get the proper view. 

Finite Element Example (Continued)

66. Select **Thermal** > **FD/Fem Network** > **Revolve Planar Elements into Solids**.

    The Command line should now read:

    `Select Planar Elements/Edge Conics for Revolve/Extrude or [MB]:`

67. Type all in the Command line.

    The Command line area should show:

    `12 found`

    `Select Planar Elements/Edge Conics for Revolve/Extrude or [MB]:`

These steps revolve the planar elements.

"All" can be used in the Command line for selection purposes since the nodes and the solids will be filtered out. Remember, only the bottom face has planar elements.

68. Press <Enter>.

    The Command line should now read:

    `Select base point to revolve from:`

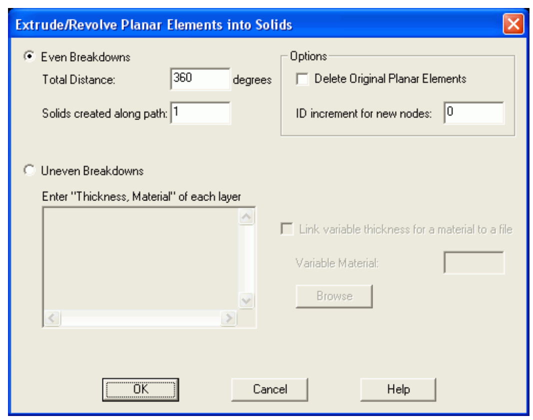
69. Type –3,0 in the Command line.

    The Command line should now read:

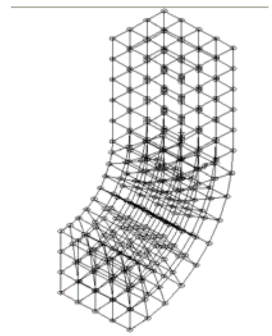
    `Select point to define revolve axis:`

70. Type –3,3 in the Command line.

    The Extrude/Revolve Planar Elements into Solids dialog box appears.



71. Leave Even Breakdowns selected.
72. Highlight the current value in the Total Distance field and type 90.
73. Highlight the current value in the Solids created along path field and type 9.
74. Select OK to close the dialog box.

Finite Element Example (Continued)	
75.  or **View** > **Zoom** > **Extents**.	Use the Hide Interior Faces command to clean up the display as needed. The model should look similar to the drawing below. 

Finite Element Example (Continued)

76. Select **Thermal > Modeling Tools > Make AutoCAD group > From Thermal Objects.**

    The Command line should now read:

    `Select entities to make into a group or [MB]:`

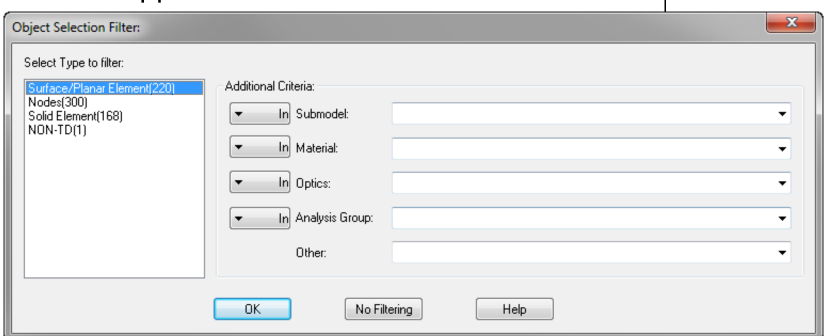
77. Type all in the Command line.

    The Command line should now read:

    `Select entities to make into a group or [MB]:`

78. Press <Enter>.

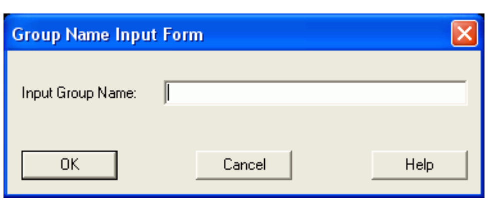
    The Object Selection Filter dialog box appears.



    Leave the current selections as they appear: Surfaces/Planar Elements(12).

79. Select OK to close the dialog box.

    The Group Name Input Form dialog box appears.



80. Type plane in the **Input Group Name** field.

81. Select OK to close the dialog box.

The model now consists of planar element and solids. The planar elements must be deleted because their function is complete.

The objects will be put into a group and then the AutoCAD Delete function to delete that group will be used.

Once all the object are selected, the filter appears and allows the planar objects to be filtered from the solids and nodes.

Finite Element Example (Continued)	

82.  or select **Modify** > **Erase**.
The Command line should now read:
`Select objects:`
83. Type group in the Command line.
The Command line should now read:
`Enter group name:`
84. Type plane in the Command line.
The Command line should now read:
`Select objects:`
85. Press <Enter>.
	These steps will delete the planar elements. Note that the letter 'g' could be used instead of the word "group".

Finite Element Example (Continued)

86. Select **Thermal > FD/Fem Network > Surface Coat Free Solid Faces.**

    The Command line should now read:

    `Select the solids for free face calculations or [MB GRP]:`

87. Type all in the Command line.

    The Command line should now read:

    `Select the solids for free face calculations or [MB GRP]:`

88. Press <Enter>.

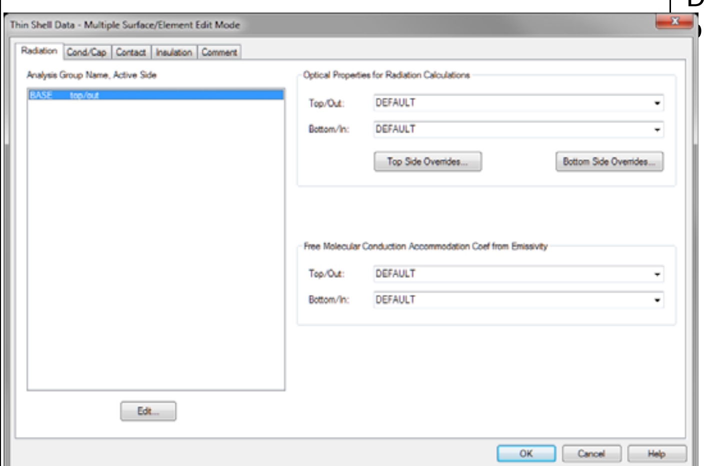
    The Command line area should show:

    `0 free tri faces found`

    `220 free quad faces found`

    and the Thin Shell Data - Multiple Surface/Element Edit Mode dialog box also appears.

89. Review the Radiation tab.



90. Select the Cond/Cap tab.

91. Click on the arrow next to the Material field and select Aluminum from the drop-down list.

92. Select OK to close the dialog box.

Finite element solids must be surface coated to allow the definition of radiation, area contact or contactors, or insulation.

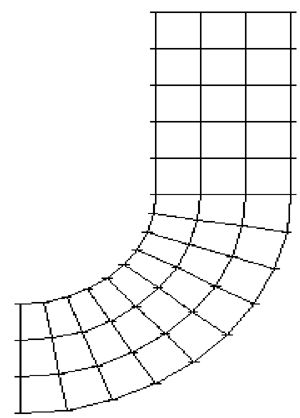
The Surface Coat Free Solid Faces command will calculate all of the solid free faces and place a planar element on that face so that the active side is top.

When the Thin Shell Data dialog box opens the Radiation tab is active. The optical property DEFAULT is defined as a black-body: the emissivity and absorptivity are both unity.

The thickness of the planar element is set to zero so that it does not affect capacitance and conductance calculations. With zero thickness the material properties will not be used, however, a material must still be selected since the material DEFAULT is undefined, unlike the optical property.

A Thermal Desktop/AutoCAD dialog box appears asking to confirm the change.

93. Select Apply Changes to close the dialog box.

Finite Element Example (Continued)	
94. Select **View** > **3D Views** > **Front.** 95. Type zoom in the Command line. The Command line should now read: `Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:` 96. Type .8x in the Command line.	Change the view to make it easier to select the nodes on the ends. The nodes will be selected and changed to boundary nodes. 

Finite Element Example (Continued)

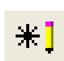
97. Select from points 1 to 2 as shown in Figure 20-21 and as noted below:

- Using the example to the right as a guide, click the left mouse above the upper left corner of the surface area (1).

The Command line should now read:

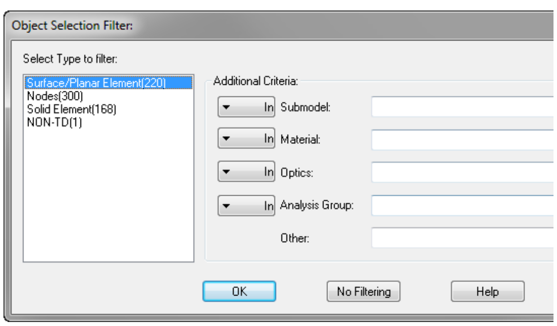
`Command: Specify opposite corner:`

- Position the mouse on the opposite, lower right corner of the selection area as shown in the example and click the left mouse button (point 2). Note that as the mouse is moved, a box is drawn around the area.

98.  or **Thermal > Edit.**

The Object Selection Filter dialog box appears.

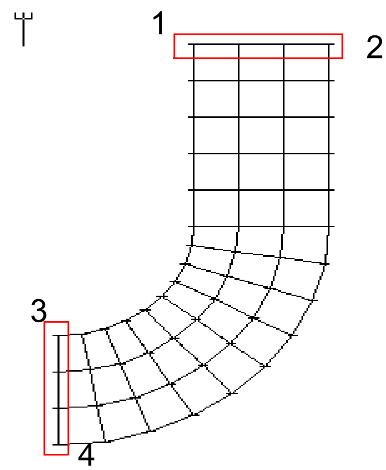
99. Select Nodes[20] in the Select Type to filter field.



100. Select OK to close the dialog box.

The nodes on the ends of the object must be edited. to apply the boundary temperatures.

Refer to the example shown below (Figure 20-21) when selecting the nodes, starting the selection box at point 1. After the first point is selected, select point 2.



When points 1 and 2 are selected, not only are the nodes selected but the surface coated planar elements that were created earlier are also selected. When the Edit function is selected, the function determines that more than one type of entity has been selected. The **Object Selection Filter** dialog box is displayed.

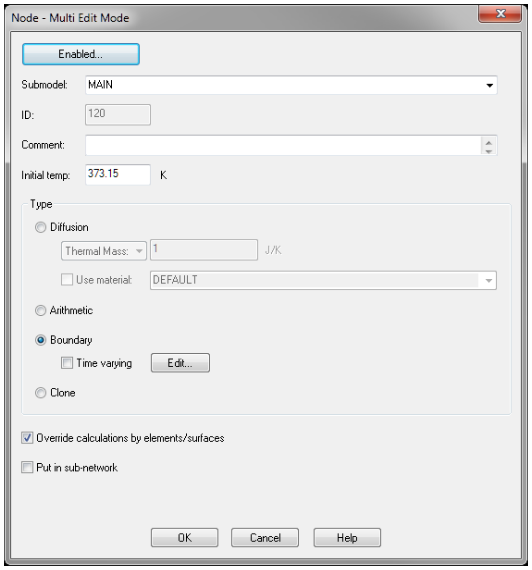
Note: The Object Selection Filter dialog box can also be displayed by selecting **Thermal > Modeling Tools > Toggle Filter On.**

Finite Element Example (Continued)

The Node – Mulit Edit Mode dialog box appears.

101. Click on Override calculation by ele– ments/surfaces to place a check mark in the box.

The Type frame activates.



102. Click on boundary to place a dot in the circle.
103. Highlight the current value in the Initial temp field and type 373.15.
104. Select OK to close the dialog box.

A Thermal Desktop/AutoCAD dialog box appears asking for confirmation of the changes.

105. Confirm the changes to close the dialog box.

The selected nodes are changed to boundary nodes and the temperature is set. When nodes are part of an element (or surface), the definition of those nodes are obtained from the associated element (or surface). Overriding the calculation by the element (or surface) allows the user to provide a new definition for the selected node or nodes.

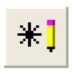
Finite Element Example (Continued)

106. Select from points 3 to 4 as shown in Figure 20-22 *and as noted below:*
    - Click the left mouse at point 3 as shown in Figure 20-22.

    The Command line should now read:

    `Command: Specify opposite cor-ner:`

    - Click the left mouse at point 4 as shown in Figure 20-22.

107.  or **Thermal** > **Edit.**

    The Object Selection Filter dialog box appears.

108. Select Nodes[20] in the Select Type to filter field.
109. Select OK to close the dialog box.

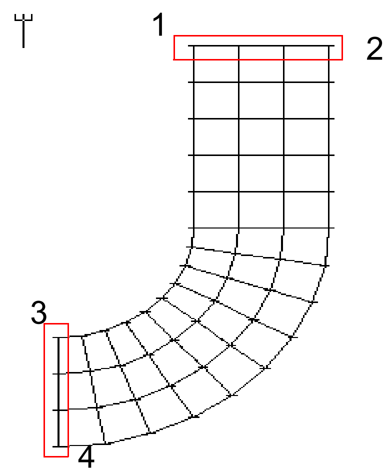
    The Node - Multi Edit Mode dialog box appears.

110. Click on Override calculation by elements/surfaces to place a check mark in the box.

    The Type field activates.

111. Click on Boundary to place a dot in the circle.
112. Highlight the current value in the Initial temp field and type 273.15.
113. Select OK to close the dialog box.
114. Confirm the changes to close the Multi Edit Dialog box.

Again, the selected nodes are changed to boundary nodes and the temperature is set. When nodes are part of an element (or surface), the definition of those nodes are obtained from the associated element (or surface). Overriding the calculation by the element (or surface) allows the user to provide a new definition for the selected node or nodes.



Finite Element Example (Continued)

115.  or select **Thermal** > **Modeling Tools** > **Resequence IDs.**

The Command line should now read:

`Select entity(s) for Node ID Resequencing or [MB GRP]:`

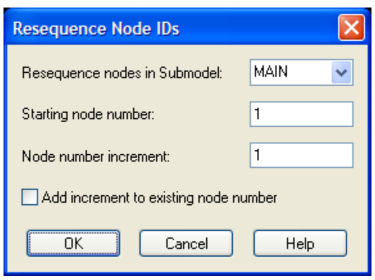
116. Type all in the Command line.

The Command line should now read:

`Select entity(s) for Node ID Resequencing or [MB GRP]:`

117. Press <Enter>.

The Resequencing Node IDs dialog box appears.

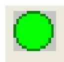


118. Leave the default values and select OK to close the dialog box.

The Command line area should show:

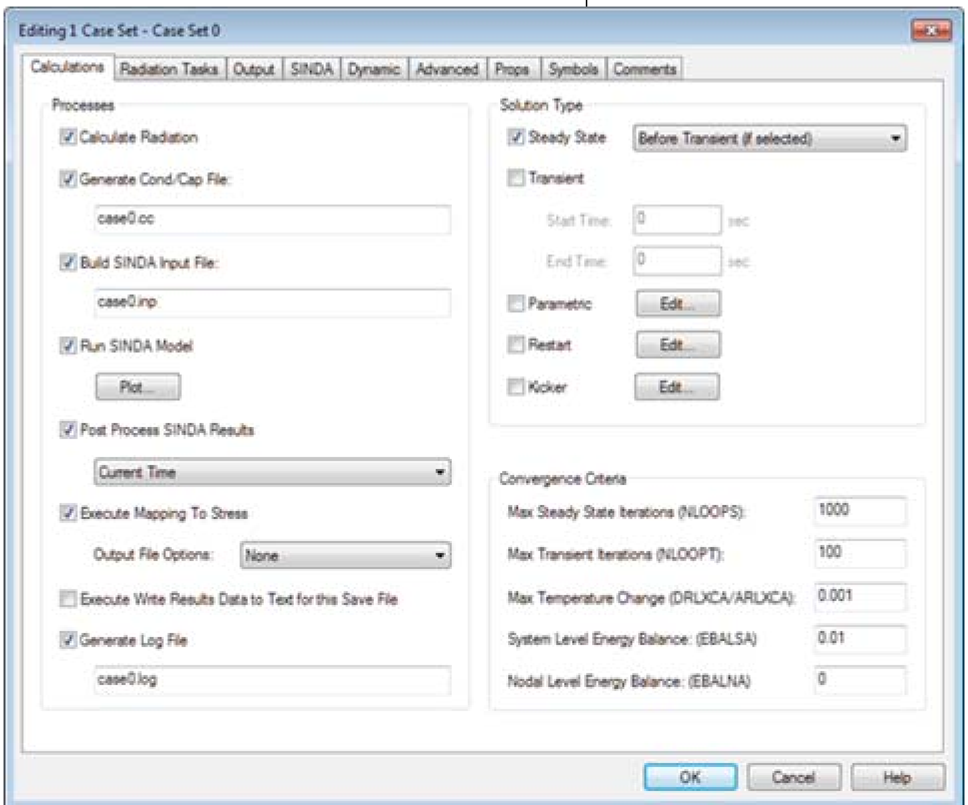
`300 nodes were changed`

This part of the exercise resequences the node IDs so all nodes have a unique number.

Finite Element Example (Continued)	
119.  or **Thermal** > **Case Set Manager**. The Case Set Manager dialog box appears. 120. Click on Edit.	The **Case Set Manager** is the link from Thermal Desktop to SINDA/FLUINT. The Case Set Manager allows the user to set up different thermal analysis cases, each with its own radiation calculations and parameters. Once the cases are defined, the user can create SINDA models, run the analyses, and post-process the results with the click of a single button. Once parameters are set and the **Run 1 Selected Case** button is clicked, Thermal Desktop will calculate any radiation conductors and heating rates for all of the tasks set up for the current Case Set. Nodes and conductors are then computed and output. A SINDA/FLUINT model is then built and run. And finally, the temperature results are displayed mapped onto the thermal model in color. Editing a case allows the case to be defined beyond the default settings.

Finite Element Example (Continued)

The Case Set Information - Case Set 0 dialog box appears.



121. Examine the tabs and the information on each tab, but do not change anything at this time.

122. Select OK to close the dialog box.

The Case Set Manager dialog box reappears.

By selecting **OK**, the case is allowed to use the default settings: no radiation with a steady state analysis. Although Calculate Radiaiton is checked, not radiation tasks were set up.

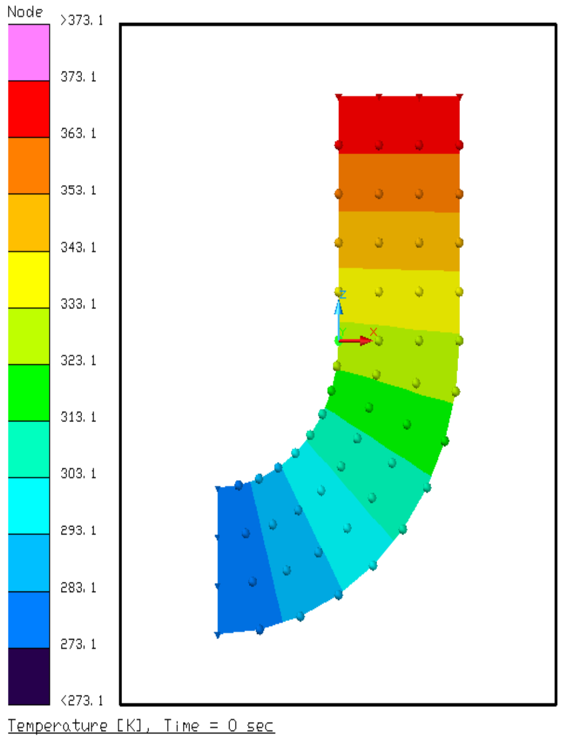
123. Click on Run 1 Selected Case.

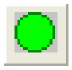
The Case Set program is run.

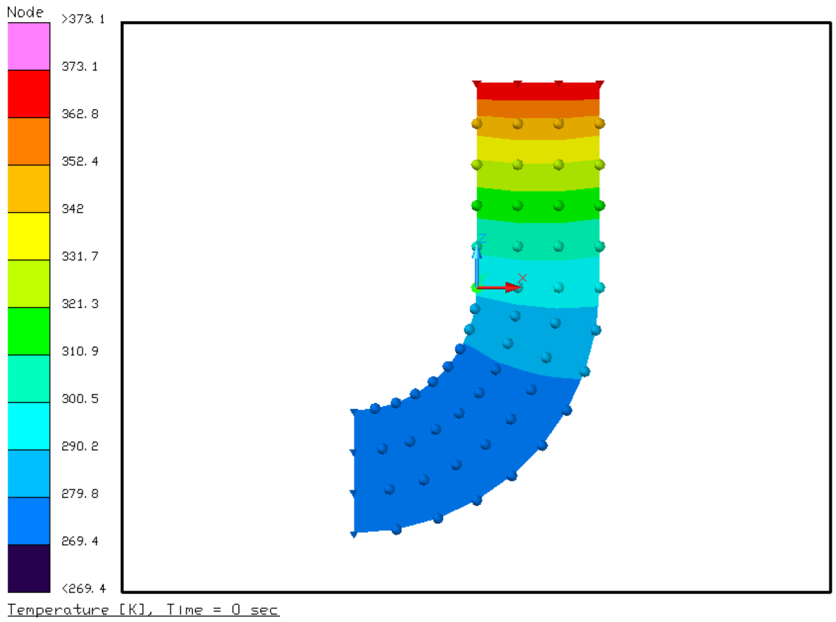
When complete, the graphic output appears in the drawing area. the SINDA/ Fluint Run Status dialog box appears on top of the graphic output confirming successful completion of the process.

Selecting **Run 1 Selected Case** will allow the **Case Set Manager** to output the conductors and capacitance, build the SINDA model, solve the SINDA model, and display the steady state temperatures back on the model.

124. Select OK to close the dialog box.

Finite Element Example (Continued)	
125. Select **View > 3D Views > Front.**	
 	
126. Select File > Save	Save the model before adding radiation.
If you have access to a RadCAD license, you may continue; otherwise, skip ahead to 141 Select File > Exit..	

Finite Element Example (Continued)	
127.  or **Thermal** > **Case Set Manager**. The Case Set Manager dialog box appears. 128. Click on **Copy**. The Add New Case Set dialog box appears. 129. Type Radiation for the **New Case Set Name** 130. Select OK to close the dialog box. The Case Set Manager dialog box reappears. 131. Select **Radiation** Case Set and click on Edit. The Case Set Information - Radiation dialog box appears.	
132. Select the **Radiation Tasks** tab, if it is not already selected. 133. Select the **Add** button. The **Radiation Analysis Data** dialog box appears. 134. Confirm **Radks** is selected in the **Calculation Type** region. 135. Confirm **BASE** is in the Analysis Group field. 136. Confirm **Monte Carlo** is selected in the **Calculation Method** region. 137. Select OK to close the dialog box. The Case Set Information - Radiation dialog box reappears. 138. Select OK to close the dialog box. The Case Set Manager dialog box reappears. 139. Click on Run 1 Selected Case. The Case Set program is run. When complete, the graphic output appears in the drawing area. the SINDA/Fluint Run Status dialog box appears on top of the graphic output confirming successful completion of the process. 140. Select OK to close the dialog box.	An Analysis Group is a user-defined group of objects which will exchange energy though radiation. The group BASE is a default and all surfaces are included unless otherwise specified. To learn more about Analysis Groups 4.1 Radiation Analysis Groups. To learn more about radiation analyses and calculations 10.1 Radiation Calculations and Output to SINDA/FLUINT.

Finite Element Example (Continued)	
The radiation calculation includes a sink temperature of 2.73 K by default, explaining the cooler temperatures. 	
141. Select **File** > **Exit.** A Thermal Desktop/AutoCAD dialog box appears asking if the user wants to save the changes made to the drawing. 142. Select Yes. The drawing is saved and Thermal Desktop closes.	Exit Thermal Desktop. **Note**: Be sure to save the changes to the file since it will be used as a starting point for another tutorial.

20.7 Mapping Temperatures From a Coarse Thermal Model to a Detailed NASTRAN Model

What will be learned:

- Importing a finite element mesh created outside of Thermal Desktop.
- Mapping temperatures from a Thermal Desktop model to another type of model.

Prerequisites:

- 20.2 Setting Up a Template Drawing

This tutorial maps temperatures from a coarse model of a plate with two holes in it to a different mesh of the same model that has much finer detail. This situation is very common when the stress analyst has a very detailed model for a part, but the thermal model must be made coarse so as to get the part integrated into the entire thermal model.

CRTech would like to give a special thanks to Jim Braley for providing the sample NASTRAN models for this tutorial.

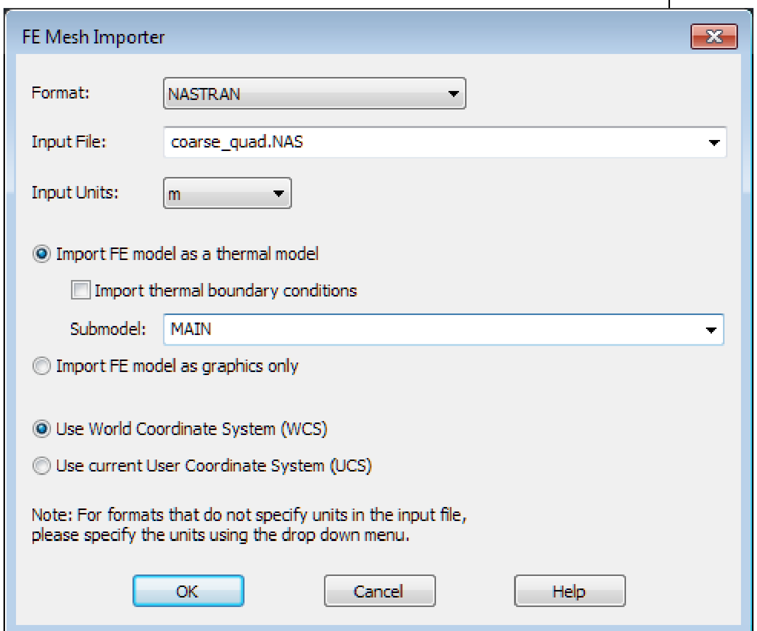
Mapping Example	
1. Copy the template thermal.dwg file created in the first tutorial to the \Tutorials\Thermal Desktop\mappingExample directory. **Note**: Be sure to hold the <Ctrl> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved. In addition to the copied template drawing, there are two existing files in the mappingExample folder: ▪ coarse_quad.nas ▪ fine_quad.nas. The two files will be imported into the model during the exercise. 2. Rename the copied template file to coarse. 3. Start Thermal Desktop by double clicking on the coarse drawing file icon in the mappingExample directory.	
4. Select **View** > **3D Views** > **Top**. The UCS icon reflects the new orientation.	Change the view in the drawing area to the top view.

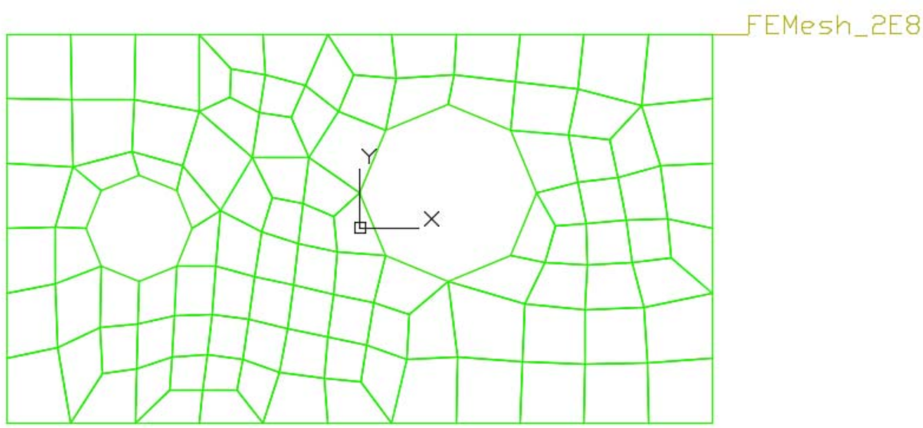
Mapping Example (Continued)

5. Select **Thermal** > **Import** > **Create FE Mesh Importer.**
6. The FE Mesh Importer dialog box appears.
7. Change the input file name to be coarse_quad.NAS.
8. Change the submodel to **MAIN.**
9. Select OK to close the dialog box.

The next step imports the coarse_quad.NAS file into the drawing. When the import process is complete, a model of a plate with two holes in it will be displayed.

The model should look similar to the example below.





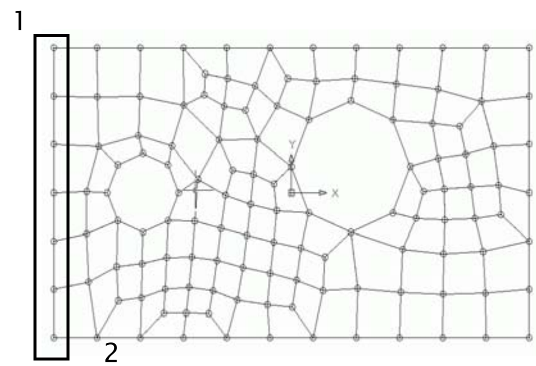
Mapping Example (Continued)

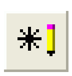
10. Select **Thermal** > **Preferences**
11. Select Graphics Visbility tab
12. Check the TD/RC Nodes box
13. Select OK to close the User Preferences dialog.

14. Select the nodes on the left edge of the model by drawing a selection box from points 1 to 2, as shown in the example to the right.

Note: In this case, selection order is not important in that only the nodes are to be selected rather than the nodes and any other objects that may be partially enclosed by the selection box. Remember that selecting objects from top to bottom will only select those items fully enclosed by the selection box whereas selecting objects from bottom to top will include items not fully enclosed by the selection box into the selection set.

Some boundary conditions must be created so the model can be run and some gradients obtained. The nodes on the left edge of the model are to be designated as boundary nodes.



15.  or **Thermal** > **Edit.**

    The Node - Mulit Edit Mode dialog box appears. (next page)

16. Select Override calculations by elements/surfaces to place a check mark in the box.

    The Type fields activate.

17. Select boundary (place a dot in the circle).
18. Select OK to close the dialog box.

    A Thermal Desktop/AutoCAD dialog box appears asking for confirmation of the node changes.

19. Confirm the changes to close the dialog box.

    The node shapes change to show their new designation

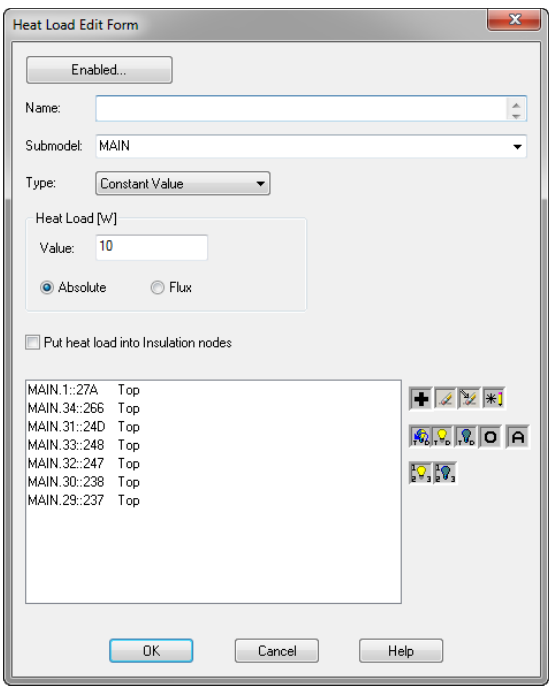
Mapping Example (Continued)

20. Select the nodes on the right edge of the model by drawing a selection box from points 3 to 4, as shown in the example to the right.

Note: As in the selection of the nodes on the left edge, selection order (top to bottom versus bottom to top) is not important in this case as only the nodes on the right edge are being selected in this step.

21.  or select **Thermal > FD/Fem Network > Heat Load on Nodes.**

The Heat Load Edit Form dialog box appears.



22. Highlight the current value in the Heat Load [W] Value field and type 10.

23. Select OK to close the dialog box.

A heat load of 10 watts is to be applied to the nodes on the right edge of the model.

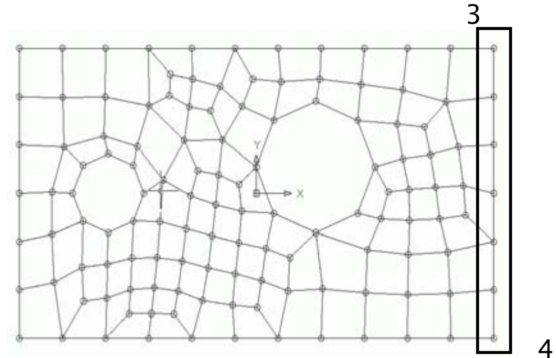
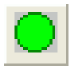


Figure 20-26: Selection Points

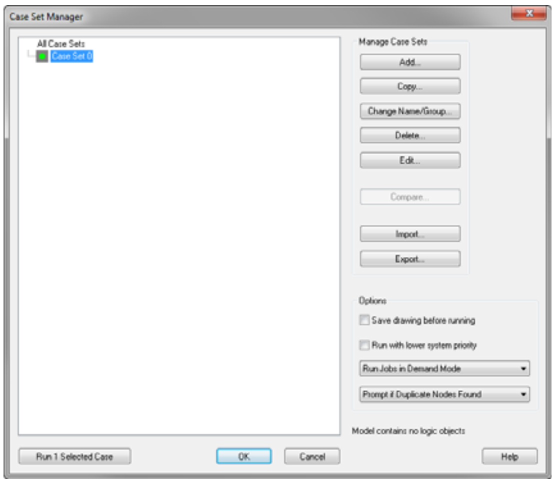
Note: Because of the top view of the model, once the heat load has been applied it appears the nodes change color from green to red. If the model is rotated to look at from another angle, small red arrows are displayed which give the illusion of the red nodes from the top view.

If the model is rotated back to a top view, the nodes appear white in color again, with small spots of red.

Mapping Example (Continued)

24.  or **Thermal** > **Case Set Manager**.

The Case Set Manager dialog box appears.



25. Click on Edit.

The Case Set Information dialog box appears.

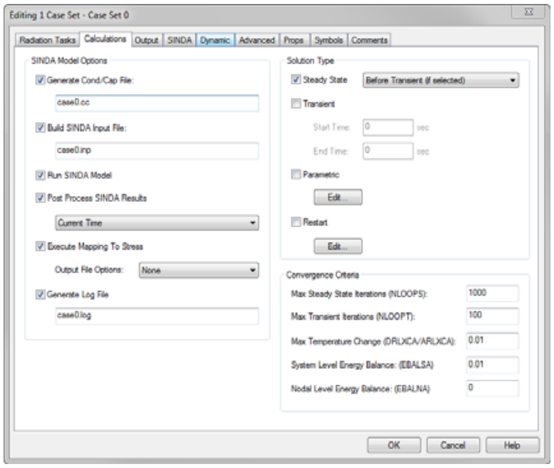
Use the **Case Set Manager** to process a steady state solution on the problem.

The primary purpose of the **Case Set Manager** is to allow the user to set up different thermal analysis cases and to have the calculations made: from doing radiation calculations to creating and running the SINDA model to post-processing temperatures with the click of a single button. When the Run 1 Selected Case button is clicked, Thermal Desktop will first calculate the radiation conductors and heating rates for all of the tasks set up for the current Case Set. Nodes and conductors are then computed and output. A SINDA model is then built and run. And finally, the temperature results are displayed mapped onto the thermal model in color.

The user may set up different Case Sets to be steady state or transient analyses. Each Case Set may have different start and stop times for transient runs. The user can also have different SINDA Logic, property databases or aliases, or even different symbol values

Mapping Example (Continued)

26. Click on the Calculations tab.



Steady State in the Solution Type field is already selected.

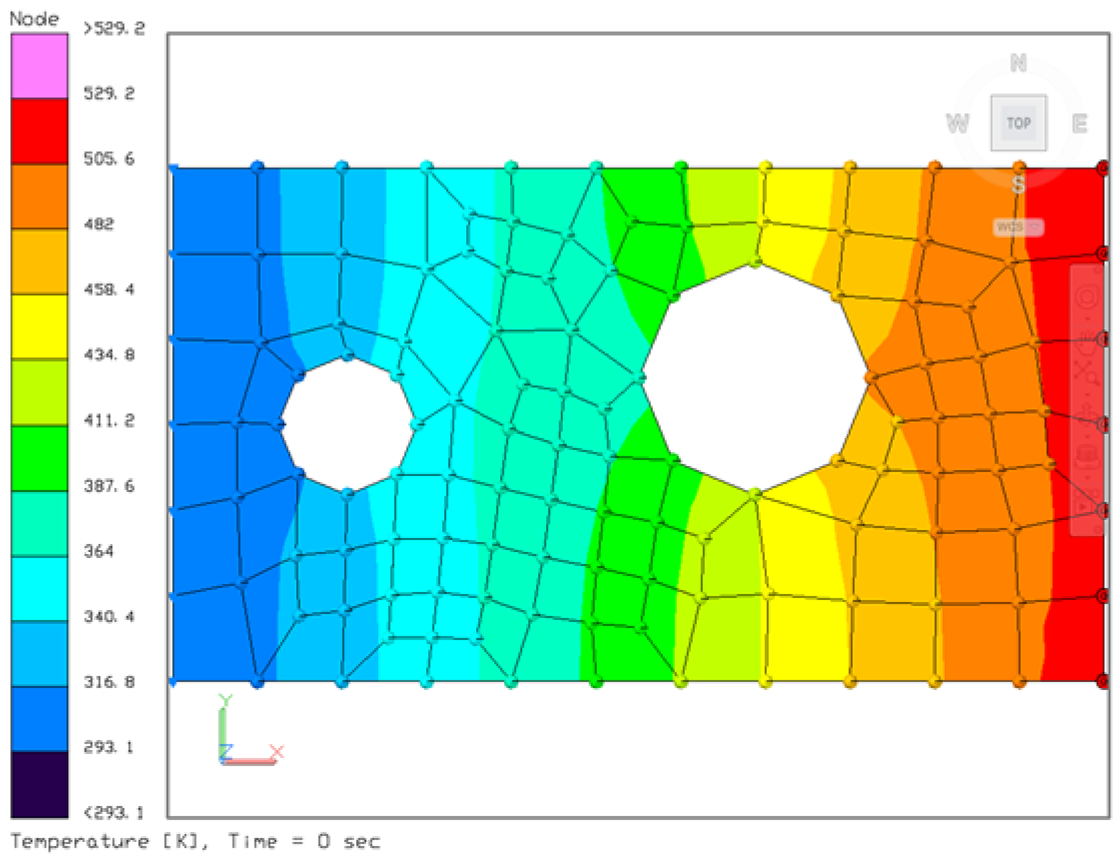
27. Select OK to close the dialog box and return to the Case Set Manager dialog box.

28. Click on Run 1 Selected Case.

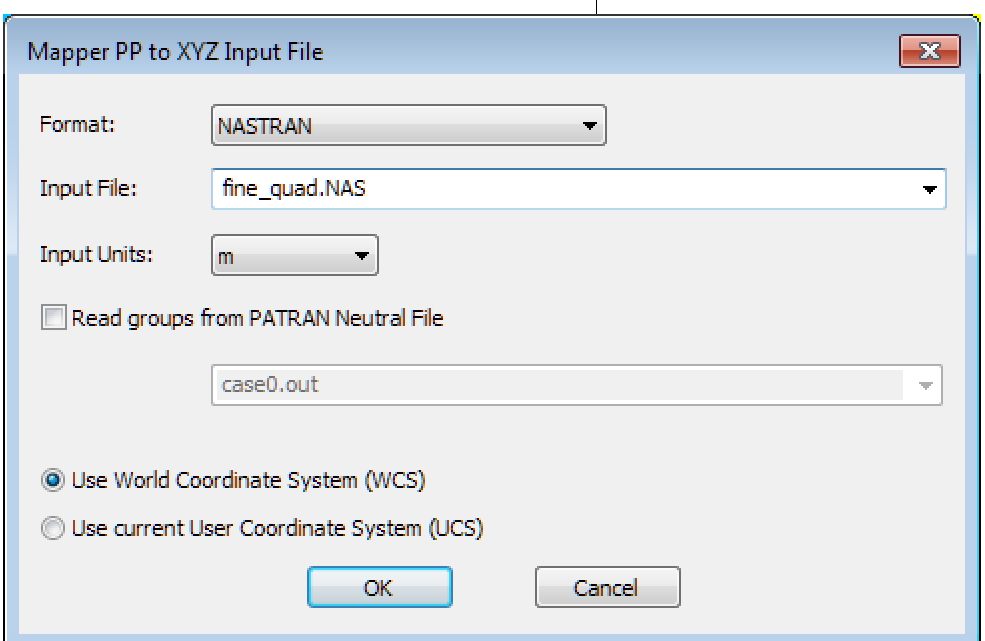
Mapping Example (Continued)

The solution is processed. When complete, the data displays on the screen along with a SINDA/FLUINT Run Status dialog box confirming successful completion of the run.

29. Click on OK to close the dialog box.



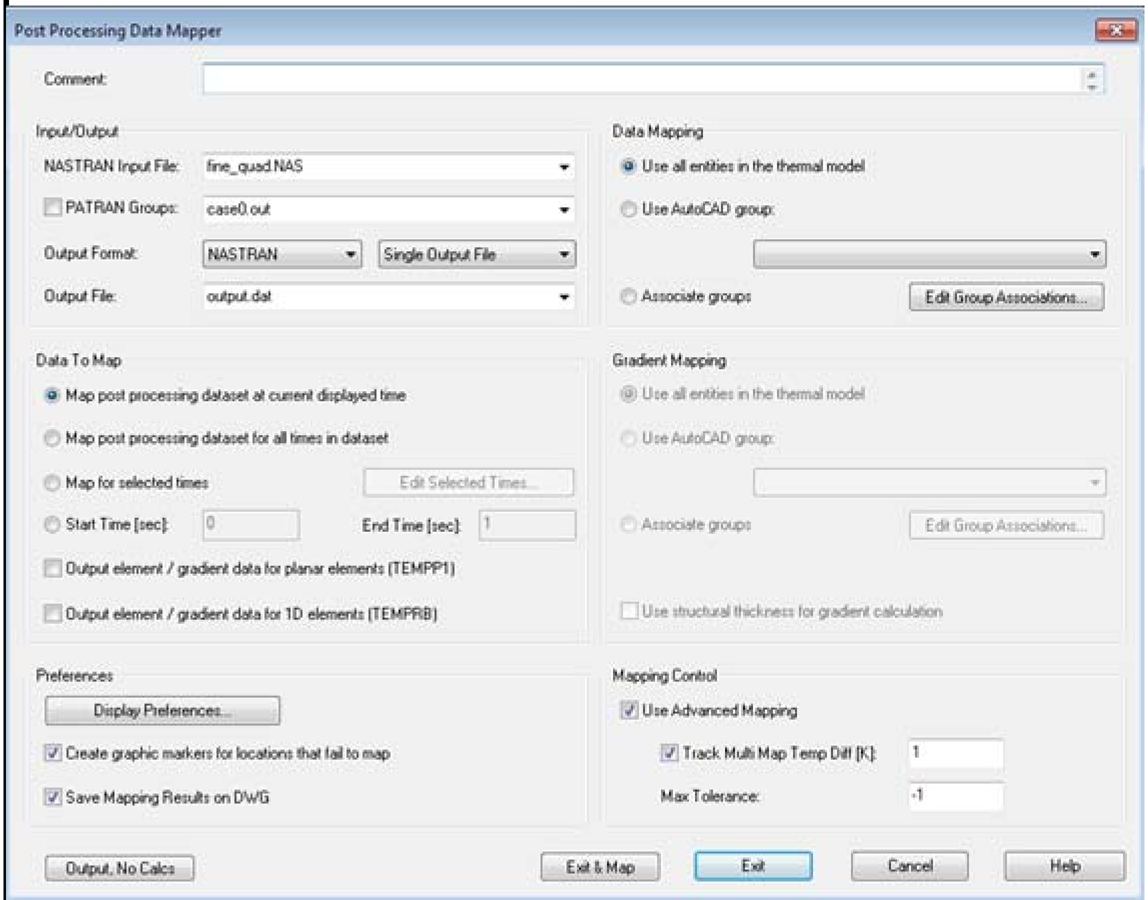
Mapping Example (Continued)	
30. Select **Thermal** > **Export** > **Post Processing Data Mapper.** The Mapper PP to XYZ Input File dialog box appears. 31. Choose **NASTRAN** in the Format field. 32. Choose fine_quad.nas in the Input File field drop down menu. 33. Ensure that **Use World Coordinate System (WCS)** is selected	The stress analyst has added a more detailed model titled fine_quad.nas. and needs the temperatures for the nodes in this model in order to perform thermal stress calculations. The Map Object allows the Thermal Desktop user to preview the stress model in side Thermal Desktop, align the two models, and map the postprocessed data to the stress model. The available format options are NASTRAN, ANSYS, FEMAP, and I-deas. The user may choose to browse if the desired input file is not in the current directory. The user may choose to align the stress model to the World Coordinate System (WCS) or the User Coordinate System (UCS)



Mapping Example (Continued)

34. Select OK to close the dialog box.

    The **Post Processing Data to XYZ Mapper** dialog box appears.



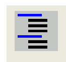
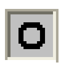
35. Highlight the current value in the Output File field and type temps.out.

36. Select Exit & Map to close the dialog box and map the data.

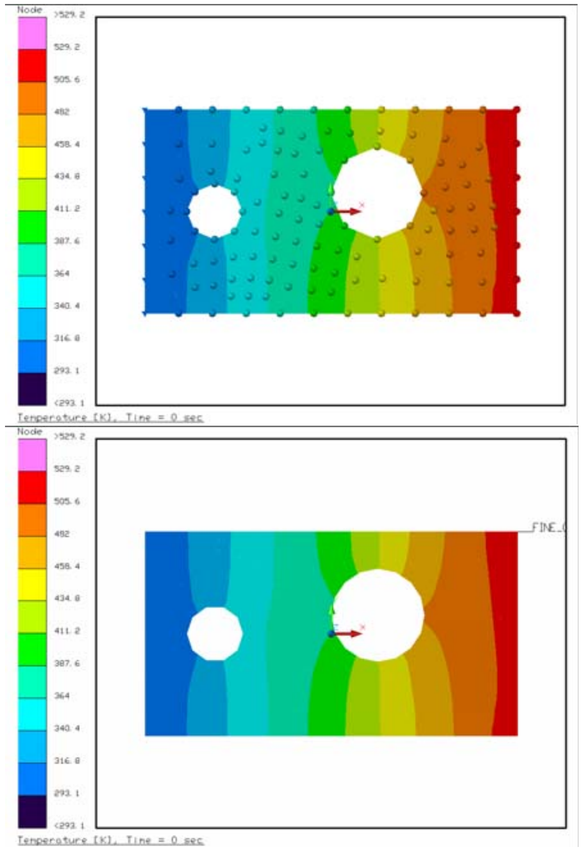
The Use Advanced Mapping option automatically finds the best mapping. If more than one object is found for a location, then a notice will be provided if the temperature difference is greater than the Track Multi Map Temp Dif value.

The user can choose to map the data immediately using Exit & Map, or may simply Exit to manipulate the mapper graphical object (e.g. align the mapper to the thermal model)

The file temps.out contains mapped temperatures in the format necessary for them to be included into the Stress model. The engineer must cut and paste these into the proper place in the NASTRAN model in order to perform the thermal stress calculations.

Mapping Example (Continued)	
37.  or **Thermal** > **Model Browser**. The Model Browser window appears on the left side of the screen. 38. Choose **List**>**Mesh Displayers**/**PP Mappers**/**BCM**/**Cutting Planes** from the Model Browser menu bar. 39. Select **MapperPP** from the Mesh Displayers tree. 40.  or **Display** > **Turn Visibility Off**. The mapper turns off and the coarse finite element model is visible. Note the shape of the contours and the holes. 41.  or **Display** > **Only** on the Model Browser menu bar. The finite element model has been turned off (notice the nodes are gone) and the mapper is turned back on. Note that the shape of the holes has been refined, but the contours are the same.	After mapping is complete, the mapped data is displayed on the mapper. Any locations in the external file (the stress model) that did not get mapped will be displayed as grey.

Mapping Example (Continued)



Finite element thermal model based on coarse mesh

Mapper based on fine mesh

Figure 20-25: Compare Mapper to

42. Select **File** > **Exit**.

Note: Thermal Desktop can also be exited using the Windows Close button (X) in the upper right corner of the screen.

A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes.

43. Select Yes.

20.8 Contactor Example

What will be learned:

- Using the Model Browser
- Using layers to control object visibility
- Using edge Contactors
- Using face Contactors
- Using material-based Contactors
- Checking contactors
- Using Symbols and Expressions

This tutorial demonstrates some of the capabilities of Thermal Desktop's Contactors. The example model is a pipe with circular, plate fins and a small block-shaped sensor. The plate fins will be connected to the pipe by edge contactors that could represent a braze or weld. The sensor is connected to the pipe by an area contactor that could represent a gasket or insulation.

Contactors, like contact created with the contact tab in the object edit window, are used to thermally connect two objects. However, contactors provide more functionality than contact such as:

- allowing a gap between the objects being connected
- allowing material, radiation, one-way, and insulation defined connections
- allowing the user to select which objects will be included in the connection tests.

Note: This tutorial is meant to show different capabilities of contactors. To accomplish this, the best modeling options may not have been used. For example, using solid disks (with only one subdivision in the thickness direction) instead of planar disks for the fins would have eliminated having to specify the thickness of the fin for the contactor, but would have eliminated the practice with edge contactors.

Contactor Example

1. Double click on the file Contactors.dwg located in the Tutorials\Thermal Desktop\Contactors folder.

   Thermal Desktop opens with the Contactors drawing on the screen.

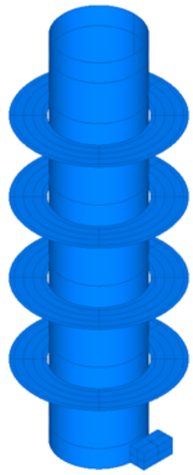
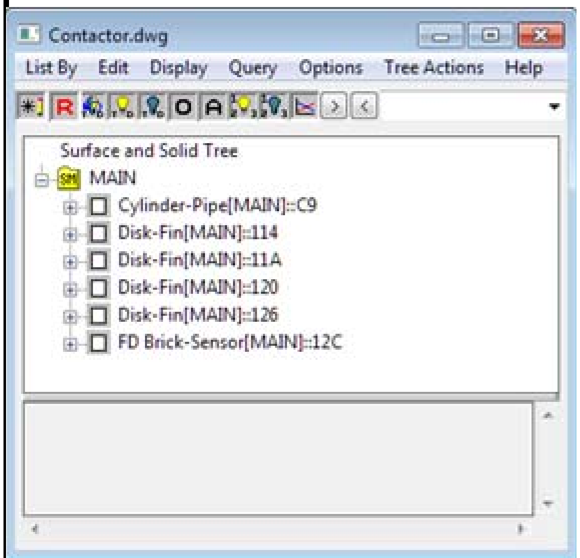


Figure 20-28:

Contactor Example (Continued)

2.  or **Thermal** > **Model Browser.**

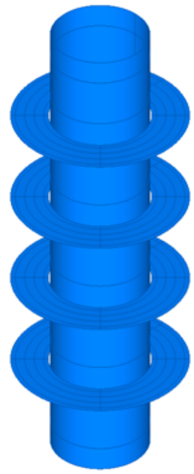
   The Model Browser window appears on the left side of the screen.



3. Using the Model Browser menu bar click on **LIST** to confirm that Surfaces/Solids is selected (check mark).
4. If necessary, expand the Main tree by clicking on the **+** sign in front of the folder.
5. Click on **FD Brick-Sensor.**
6.  or **Display** > **Turn Visibility Off** in the Model Browser toolbar.
7. Minimize the **Model Browser** window or move so it does not block the drawing.
8. Press <**ESC**> to be sure the selection set is clear.

Use model browser to set selection options and turn off objects that will not be modified during upcoming steps.

The drawing should now appear similar to the view below:



Contactor Example (Continued)

9.  or **Thermal** > **FD/FEM Network** > **Contactor**.

   The Command line should now read:

   `Select faces contacting from or [MB]:`

10. Select each of the four fins in the drawing area.

    The Command line should now read:

    `Select faces contacting from or [MB]:`

11. Press <**ENTER**>.

    The Command line should now read:

    `Select surfaces contacting to or [MB]:`

12. Select the pipe in the drawing area.

    The Command line should now read:

    `Select surfaces contacting to or [MB]:`

13. Press <**ENTER**>.

    The Contactor dialog box appears. (next page)

14. Type Fins to pipe in the comment field.
15. From the drop-down list in the Contact From field select Edges.
16. Select the Use Material check box.

    The Use Material field becomes active.

17. From the drop-down list in the Use Material field select Aluminum, 2024-T6.

Create edge contactors to thermally connect the fins to the pipe. The connection is assumed to be an aluminum weld of thickness WeldThickness.

Important: Contactors do not calculate the conductance from the node to the edges of surfaces connected by contactors. Therefore, that conductance should be included in the contactor coefficient or the better method is to use edge nodes. In this model, edge nodes are used for the fins and sensor.

Important: In most cases, surfaces with smaller nodal areas should be chosen as the From surfaces. All integration points on a From surface are used in the testing algorithm.

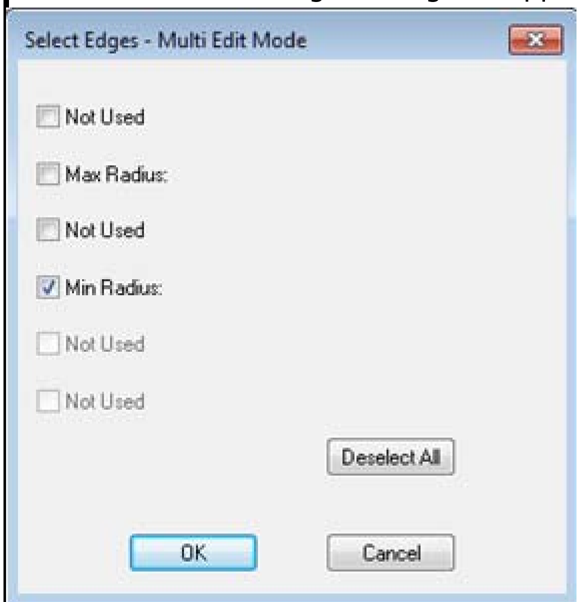
Contactor Example (Continued)	
18. Double click in the Coeff field. The Expression Editor dialog box appears. Using right-click to select the symbol names from the general list, enter the following expression in the expression field: `FinThickness/WeldThickness` 19. Click OK to close the dialog box. The Contactor dialog box updates to show 6 in the Coef field in bold type.	The SINDA conductor value resulting from a contactor is either kA/L, hA, or a Radk (εi*Bij*Ai). When Use Material is selected for an edge contactor, the conductivity is obtained from the chosen material property and the length of the surface edge is taken from the geometry. Therefore, to complete kA/L, the Coeff value must be the thickness of the edge (FinThickness) divided by the distance in the direction of the heat flow (WeldThickness). To see definitions of the symbols being used, select the **Add Symbol** button while in the Expression editor.

Contactor Example (Continued)

20. Select all four fins in the From: field.

21.  **Edit From Object** icon at the bottom of the Contactor dialog box.

22. The Select Edges dialog box appears.



23. Select the **Deselect All** button

24. Check Min Radius so the box is checked.

25. Click OK to close the dialog box.

    The Contactor dialog box updates to show Min Radius beside each object in the From field.

26. Click Show Calcs to close the Contactor dialog box and show the contactor markers.

When the Contactor dialog box closes, the graphical image for the contactors are displayed:

- The **from** surfaces have **green** arrows on the edges (or faces) included in the contactor.
- The **to** surfaces have **gold** arrows at the centroid of each face.

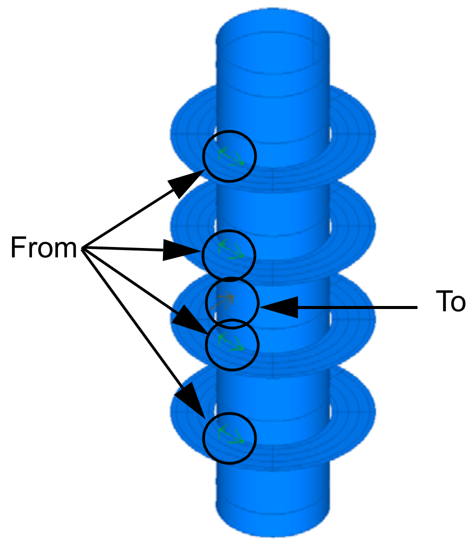


Figure 20-29:

Contactor Example (Continued)

Yellow lines connecting the edges of the disks to nodes of the pipe appear.

Changing to wireframe, , will make the markers more visible.

27.  toggles node visibility on.

28.  or **Thermal** > **Model Checks** > **Clear Contact/or Markers**

    The contactor markers are cleared.

29.  toggles node visibility off.

Use contactor markers to verify connection of the Fins to Pipe contactor.

The contactor markers connect the integration points of the From edges to the nodes of the To surfaces.

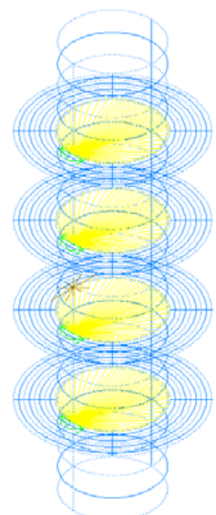


Figure 20-30: Contactor markers

Note: If the Coef field is 0, then the contactor markers will not be drawn.

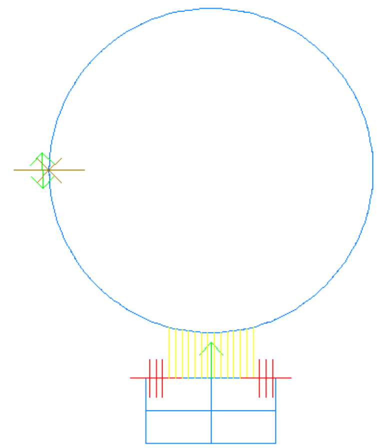
Ten contactor markers are drawn from the inner edges of each fin (Figure 11-58). The markers show which integration points have met the requirements of the testing algorithm. Any integration point not within the tolerance of the To surface would be marked with a red cross.

When nodes are visible, note that not all pipe nodes are connected, but all inner edges of the fins are connected. Also note that To surface nodes can be connected multiple times, but test points are connected only once.

Contactor Example (Continued)	
30. Restore the Model Browser Window. 31. Select the four fins in the Model Browser tree.	Create area contactor to thermally connect the sensor to the pipe. The connection is a per-area coefficient.
32.  **Turn Visibility Off** icon in the Model Browser toolbar. The disks representing the fins disappear from the graphics window. 33. Select the Sensor.	The Model Browser typically minimizes to the upper left corner of the display.
34.  **Turn On Visibility** icon in the Model Browser toolbar. The brick representing the sensor appears in the graphics window. 35. Right-click the sensor and choose **Send Selection Set to AutoCAD.**	The Contactor command accepts preselected items (the sensor in this case) as the from object(s) and moves on to selecting To surfaces.

36.  or **Thermal** > **FD/FEM Network** > **Contactor.**
The Command line should now read:
`Select surfaces contacting to or [MB]:`
37. Type MB
A new Model Browser window opens.
38. Select **Cylinder-Pipe** and close the new Model Browser Window.
The Command line should now read:
`1 object selected`
`1 added`
`1 found`
`Select surfaces contacting to or [MB]:`
39. Press **<ENTER>**.
The Contactor dialog box appears.
	The MB option allows selection from a model browser window. Whatever is selected when the newly opened Model Browser is closed is added to the selection set.

Contactor Example (Continued)	
40. Type Sensor to pipe in the comment field. 41. Highlight the current value in the Tolerance field and type 0.008. 42. Highlight the current value in the Conduction Coefficient field and type 5. 43. Choose **Ray Trace Algorithm** in the Inputs for Connection Algorithm Section.	For the ray trace algorithm, a ray of length Tolerance is shot normal to the From surface at each test point. If the ray intersects with a To surface, the ray terminates and a connection is made. By contrast, the point algorithm checks for surfaces within tolerance in all directions.
44. Double-click the sensor in the From field. 45. Deselect all check-boxes except **YMAX**. 46. Select OK to close the **Select Faces** dialog. 47. Select Show Calcs to close the Contactor dialog box and show the contactor markers. A text window will appear showing that the contactor did not make 100% connection due to the tolerance used. The contactor markers will show gold (connected) and red (outside of tolerance). 48. Close the text window.	The contactor is limited to only the side where the actual contact is made. Using ray tracing and tolerance would also eliminate the non-contact sides, but calculation time would be required to determine this. Selecting only certain sides, prevents unnecessary calculations. Use contactor markers to verify connection of the Sensor to Pipe contactor.

Contactor Example (Continued)	
49. **View** > **3D Views** > **Top.** The markers can be seen more easily. 50.  or **Thermal** > **Model Checks** > **Clear Contact/or Markers** The contactor markers are cleared. 51. **View** > **3D Views** > **SW Isometric.** 52. Restore the Model Browser Window. 53.  **Display All** icon in the Model Browser toolbar. The fins appear in the graphics window. 54. Minimize the **Model Browser** window.	When the view is changed to Top, the graphics display should look like this:  Since the ray trace algorithm was chosen, the markers appear as rays normal to the From surface. Test points that are not within the tolerance are shown in red.

Contactor Example (Continued)	
55. Using the Layer Drop Down Menu, select **boundary** as the current layer. The Layer Drop Down Menu originally says Objects and is located next to the  button. The drop down menu is accessed by clicking on the down-arrow to the right of the word Objects. 56.  or **Thermal** > **Surfaces/Solids** > **Rectangle** `Origin point <0, 0, 0>` appears in the Command Line area. 57. Type -0.1,0,0 in the Command Line, then return. `Point for +X and X-size <@1, 0, 0>` appears in the Command Line area. 58. Type @0.01,0,0 in the Command Line then return. `Point to set XY plane and Y-size` `<@0,1, 0>` appears in the Command Line area. 59. Type @0,0.01,0 in the Command Line then return. The Thin Shell Data dialog box appears. 60. On the **Numbering** tab, type AMBIENT into submodel. 61. Click on the **Cond/Cap** tab. 62. Select AMBIENT for **Cond Submodel**. 63. Click on Generate Cond/Cap button to open the Expression Editor 64. Type 0 into the Expression. 65. Click OK to close **Expression Editor**.	Create a small surface to represent the ambient air so contactors can be created from the fins to the air. Ambient conditions will typically be represented by a user defined node, but conductors between nodes and surfaces do not allow edge conduction. Therefore, a small surface will be created to represent convection from the edges of the fins and allow a contactor. Other options would be to use FD Solid Cylinders for the fins or ignore the convection from the edge of the fin. A value of zero in the expression prevents the surface properties from being generated for a SINDA file. If radiation analyses were to be performed, "Not in analysis group" should be selected for all analysis groups.
66. Click OK to close the **Thin Shell Data** dialog box. 67.  or **View** > **Zoom** > **Extents**	

Contactor Example (Continued)	
68. Restore the Model Browser Window. 69. Choose **List By** > **Submodel.ID** 70. Right-click **AMBIENT** and select **Edit** 71. In the Object Selection Filter, select Nodes(1) and OK. 72. Select **Override calculations by elements/surfaces** (place a check mark in the box). 73. Select Boundary (place a dot in the circle). 74. Type Ambient in the Comment field. 75. Double-click in the **Submodel** field to highlight MAIN. 76. Type AMBIENT into the **Submodel** field. 77. Select OK to close the dialog box.	Override the surface node to create a boundary node. If User Node Visibility is on, a triangle appears at the center of the surface. Even though the ambient node will not be generated as part of the surface, it will now be generated as a user node.

Contactor Example (Continued)

78.  or **Thermal** > **FD/FEM Network** > **Contactor**.

    The Command line should now read:

    `Select faces contacting from or [MB]:`

79. Select all four fins in the drawing area.

    The Command line should now read:

    `Select faces contacting from or [MB]:`

80. Press <**ENTER**>.

    The Command line should now read:

    `Select surfaces contacting to or [MB]:`

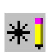
81. Select the newly created small surface in the drawing area.

    The Command line should now read:

    `Select surfaces contacting to or [MB]:`

82. Press <**ENTER**>.

    The Contactor dialog box appears.

83. Type Fin faces to ambient in the comment field.
84. Highlight the current value in the Conduction Coefficient field and type 50.
85. Select all four fins in the From: field.
86.  **Edit From Object** icon near the bottom of the Contactor dialog box.

    The Select Faces dialog box appears.

87. Select both top and bottom (check marks should be in both boxes).
88. Click OK to close the dialog box.
89. Click Show Calcs to close the Contactor dialog box and display the contactor markers.

Create convection from faces of fins using contactors.

Note new Contactor graphical objects when complete.

Here the order of selecting **From** and **To** is important. The area of the contact is calculated using the **From** surfaces.

Contactor Example (Continued)

90. Read the warning message that appears. This is not a problem in this situation since the fin surface areas are needed and the area of the boundary surface is not used.
91. Close the **Warning** window.
92. **View** > **3D Views** > **Front.**
93.  or **Thermal** > **Model Checks** > **Clear Contact/or Markers**

    The contactor markers are cleared.
94. **View** > **3D Views** > **SW Isometric.**

Use contactor markers to verify connec–tion of the Fin faces to Ambient contactor.

Notice the separation between the ends of the contact markers and the fins in the view below:

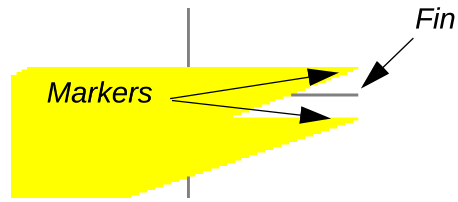


Figure 20-31: Contactors with surface thickness

This happens because the test points are allowing for the thickness of the fins.

Contactor Example (Continued)	

95.  or **Thermal** > **FD/FEM Network** > **Contactor**.
The Command line should now read:
`Select faces contacting from or [MB]:`
96. Select all four fins in the drawing area.
The Command line should now read:
`Select faces contacting from or [MB]:`
97. Press <**ENTER**>.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
98. Select the small surface in the drawing area.
The Command line should now read:
`Select surfaces contacting to or [MB]:`
99. Press <**ENTER**>.
The Contactor dialog box appears.
100. Type Fin edges to ambient in the comment field.
101. From the drop-down list for the Contact From field select Edges.
102. Double-click in the Conduction Coefficient field.
The Expression Editor dialog box appears,
103. Right-click to select the symbol names in the **general** list and enter the following expression in the Expression field
	Create convection from edges of fins using contactors.

`FinThickness*50` 104. Click OK to close the dialog box. The Conduction Coefficient field now shows 0.3.	Again, the **From** objects are used for the geometry. For an edge contact, the length of the edge is calculated so the thickness of the edge must be included in the coefficient.

Contactor Example (Continued)	
105. Select all four fins in the From: field. 106.  **Edit From Object** icon. The Select Edges dialog box appears. 107. Deselect all checked boxes. 108. Select Max Radius so the box is checked. 109. Click OK to close the dialog box. 110. Click OK to close the Contactor dialog box.	
111. Select **File** > **Exit.** A **Thermal Desktop**/**AutoCAD** dialog box appears asking to save the drawing changes. 112. Select Yes.	Exit Thermal Desktop and save as prompted.

Running the model requires a heat load or a fixed boundary temperature (different than the boundary node already defined). The user can add a heat load to the pipe surface or change the pipe nodes to boundary nodes and obtain a solution. The node IDs must also be resequenced.

If the user runs a transient solution, the time step taken by SINDA/FLUINT will likely be very small. This is caused by the large conductance at the base of the fins: G = k*A/L. If the user adds a factor to the coefficient of the fin-to-pipe contactor, he or she can see how the conductance of the contactor can affect the solution time step.

20.9 Parameterizing for a Common Input

What will be learned:

- Inserting a model into another drawing file
- Importing and renaming symbols

Prerequisites:

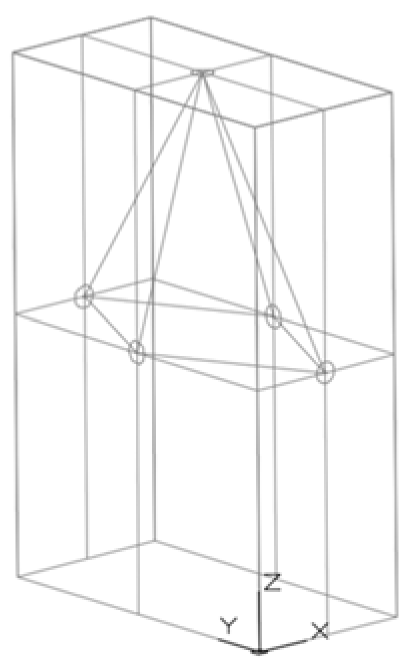
- 20.2 Setting Up a Template Drawing

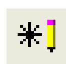
Sometimes it is common that the same geometry is used over and over in the same model. This geometry can be as simple as a five or six-sided box or can be much more complicated. In this example, a simple five sided-box with centered nodes that has conductors between each of the faces has been created. The geometry has been parameterized so that the user can simply change the x, y, and z sizes so that the box will automatically update when the user changes the symbols.

Parameterized Box Example

1. Double click on the file box.dwg located in the Tutorials\Thermal Desktop\Parameterized Box For Insert folder.

   Thermal Desktop opens with the drawing on the screen.



Parameterized Box Example (Continued)	
2. Select **Thermal** > **Symbol Manager**. The Symbol Manager dialog box appears. 3. Select Done to close the dialog box.	Take a few moments to examine the model. Bring up the **Symbol Manager**. Notice the parameters for x, y, and z sizes. There is also a parameter for the thickness of the faces on the box.
4. Click on the top of the box to select it. 5.  or **Thermal** > **Edit**. The Thin Shell Data dialog box appears. 6. Select the Surface tab. 7. Look at the X Max and Y Max fields.	Look at some of the data that make up the top of the box. After selecting the top, utilize Thermal Desktop's Edit function to display the **Thin Shell Data** dialog box. Notice the lengths of the X and Y axes are parameterized (Surface tab) and that the Z translation is also programmed (Trans/Rot tab). Take a moment to select some of the other surfaces and conductors to get an idea of how they are programmed.
8. Select the Trans/Rot tab. 9. Look at the Translation Z field. 10. Select OK to close the dialog box. 11. Repeat the process for some of the other surfaces and conductors as desired.	
12. Select **File** > **Exit**. A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes. 13. Select Yes.	Close box.dwg saving it to the current AutoCAD version and exit Thermal Desktop.
14. Copy the template thermal.dwg file created in the first tutorial to the \Tutorials\Thermal Desktop\Parameterized Box For Insert directory. **Note**: Be sure to hold the <Ctrl> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved. 15. Rename the copied template file to parameter. 16. Start Thermal Desktop by double clicking on the parameter drawing file icon in the Parameterized Box For Insert directory.	

Parameterized Box Example (Continued)

17. Select **Insert** > **Block**.

    The Insert dialog box appears.

18. Select Browse.

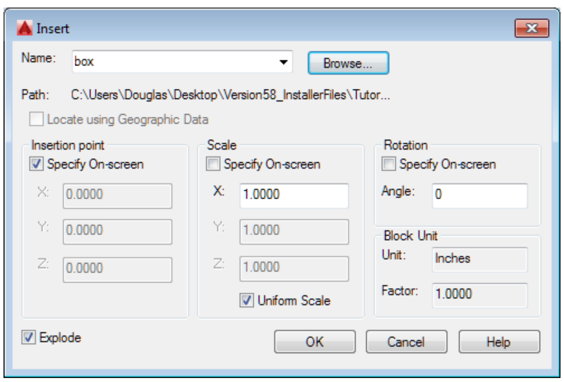
    The Select Drawing File dialog box appears with two drawings displayed in the drawing field.

19. Select box.dwg to highlight it.

20. Select Open.

    The Insert dialog box reappears with box displayed in the Name field.

21. Select Specify On-Screen in the Insertion field (place a check mark in the box) if not already selected.



22. Check Explode

23. Select OK to close the dialog box.

    The Command Line should now read:

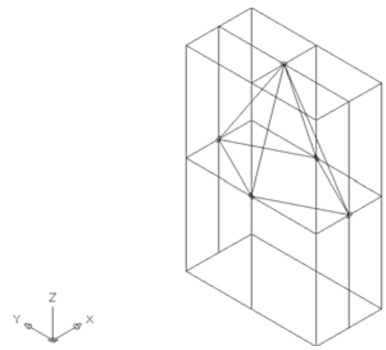
    `Specify insertion point or [Base-point/Scale/X/Y/Z/Rotate]:`

24. Click at any point on the screen to place the box. (Some versions of AutoCAD may prompt for scaling factors at this point)

    A close view of a box corner appears in the drawing area.

Bring the box drawing into the template copy.

Each user will have a somewhat different image appear in the drawing area because of the insertion point selections but, in general, the drawing should appear similar to the drawing below once zoomed extents is performed.



Parameterized Box Example (Continued)

25.  or **Zoom** > **Extents**.
26. Click on the top of the box.

    Notice that only the top is selected.
27. Select other surfaces as desired and press <Esc> to deselect when finished.

If the entire box is selected then the block was not exploded. Explode the box so that it is no longer an AutoCAD Block, but individual Thermal Desktop entities. Just type EXPLODE when the entire box is selected.

Once Explode is performed, individual components of the box can be selected rather than only the whole box.

28. Select **Thermal** > **Symbol Manager**.

    The Symbol Manager dialog box appears and the dialog box's fields are empty.
29. Select Import.

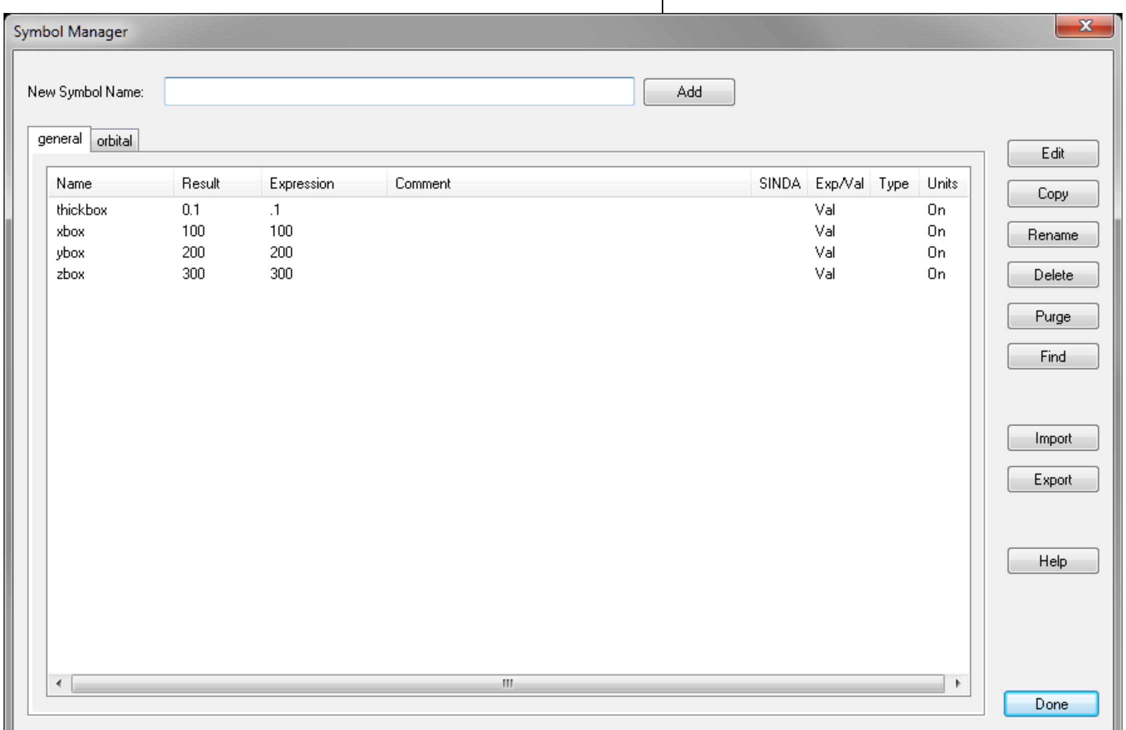
    The Open dialog box appears.
30. Select box.dwg to highlight it and then select Open.

    The Import dialog box appears.
31. Select **thickbox, xbox, ybox and zbox** and select **Import**.

    The Symbol Manager dialog box reappears with the symbols displayed.

Use the **Symbol Manager** Import command to import the file boxSymbols.sym. Xbox, ybox, zbox and thickness symbols are imported.

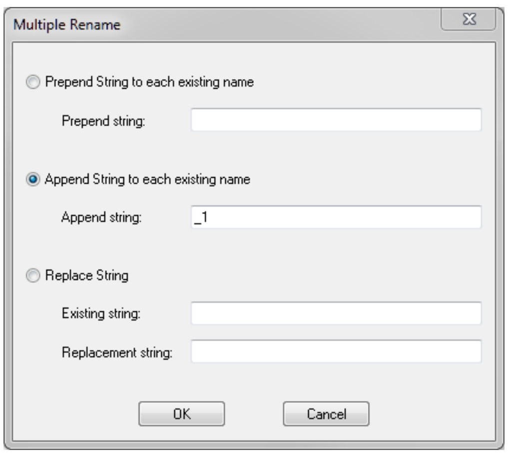
Symbols can be exported into SYM files from the symbol manager. Symbols can be imported either from a SYM file or directly from a DWG file.



Parameterized Box Example (Continued)

32. Select the 4 symbols:
    - Click on thickbox to highlight it and then, hold down the <Shift> key and click on zbox.
33. Select Rename on the right side of the form.

    The Multiple Rename dialog box appears.
34. Select the radio button beside Append String to each existing string.
35. Type _1 in the **Append string** field.



36. Select OK.

    The Symbol Manager dialog box reappears displaying the new symbol names.

The box is to be imported multiple times, so the imported symbols must be renamed. Append _1 (underscore 1) to each of the symbol current names.

Parameterized Box Example (Continued)

37. Select xbox_1.
38. Select Edit.

    The Expression Editor dialog box appears with the current xbox_1 information.

Note: Double clicking on a symbol also displays the Expression Editor.

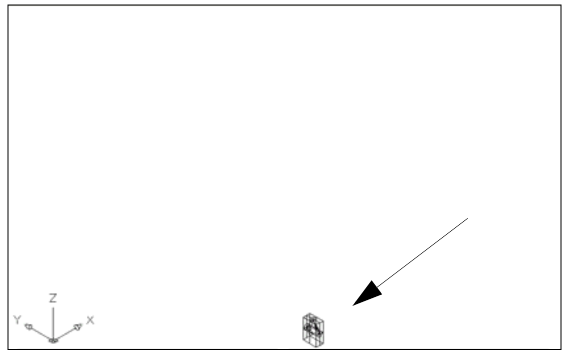
39. Highlight the current value in the Expression field if not already highlighted and type 10.
40. Select OK.

    The changed parameters for xbox_1 are reflected.
41. Repeat the process for ybox and zbox, changing the values to 1/10 of the current value.

    The Symbol Manager dialog box displays the new values.
42. Select Done to close the Symbol Manager dialog box.

Edit the symbols for xbox, ybox, and zbox to be one tenth of their original values.

The box changes in the drawing area to reflect the changes in size.



43. Select **Insert** > **Block.**

    The Insert dialog box appears with box in the Name field highlighted.

    Specify On-Screen in the Insertion field is already selected from the previous insertion.
44. Select OK to close the dialog box.

    A box is attached to the cursor waiting for insertion.

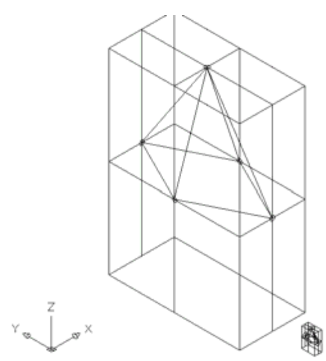
    The Command Line should now read:

    `Specify insertion point for block:`
45. Click at any point on the screen to place the new box.

    The new box appears on the screen. Notice the difference in the sizes of the two boxes—the first box's size was changed to 1/10 of its original size (xbox_1).

Insert another box. Use the **Insert** > **Block** command, but this time it is not necessary to reselect the box.dwg file, simply change the Name: pull down to box and select OK.

Select any point on the screen to place the box, and then explode the box.



Once the second box is inserted, explode it so the individual entities.

Parameterized Box Example (Continued)

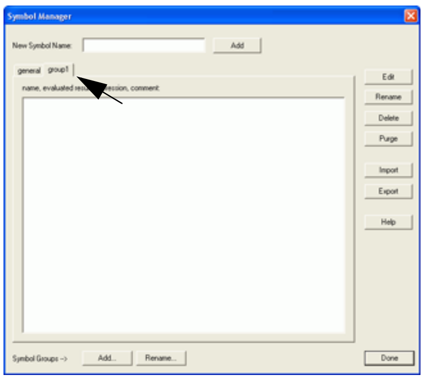
46. Click on various parts of the box as desired to confirm “explosion” and press <Esc> when finished.

47. Select **Thermal** > **Symbol Manager**.

    The Symbol Manager dialog box reappears with the symbols renamed earlier displayed.

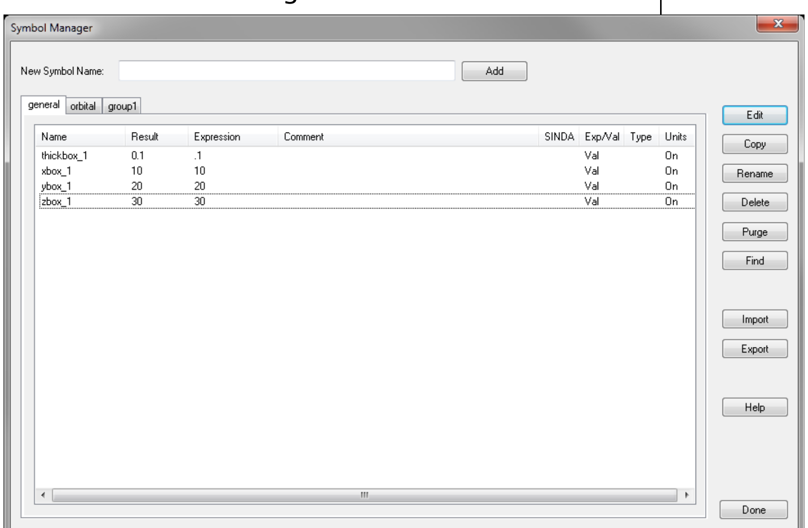
48. Right-click the **general** tab and select Add New Group to create a new Symbol Group.

The symbols listed in the **Symbol Manager** belong to a group named general. Add a second symbol group named group1.



A new tab named group1 is displayed.

49. Click on the general tab.

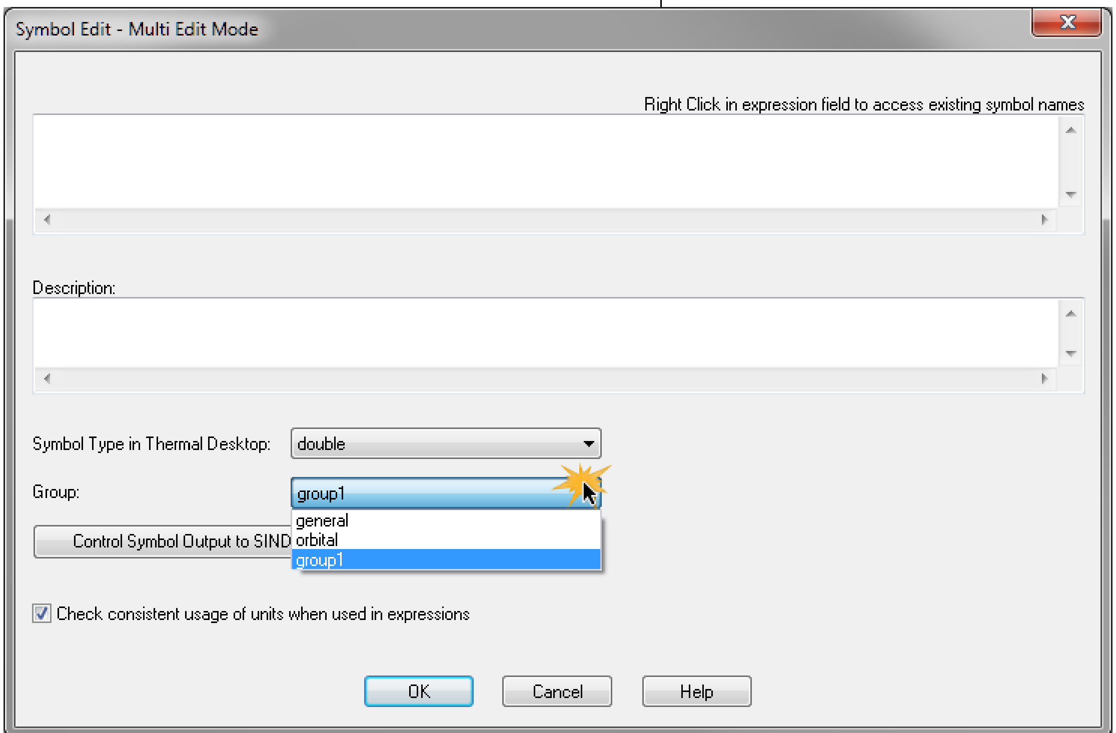


50. Select all 4 symbols.

51. Select Edit.

Parameterized Box Example (Continued)

The Symbol Edit – Multi Edit Mode dialog box appears.



52. Click on the arrow next to the Group field and select group1 from the drop-down list.

53. Select OK to close the Symbol Edit – Multi Edit Mode dialog box.

    The Symbol Manager dialog box shows the symbols are removed from the general symbol group and moved to group1. The general tab is empty.

Note: Click on the group1 tab to verify the move

Parameterized Box Example (Continued)

54. Select the general tab if not already selected.

55. Select Import.

    The Open dialog box appears.

56. Select box.dwg to highlight it and then select Open.

    The Import dialog box appears.

57. Select **thickbox, xbox, ybox and zbox** and select **Import**.

    Another set of the original symbols are imported and display in the general tab.

58. Select the 4 newly imported symbols.

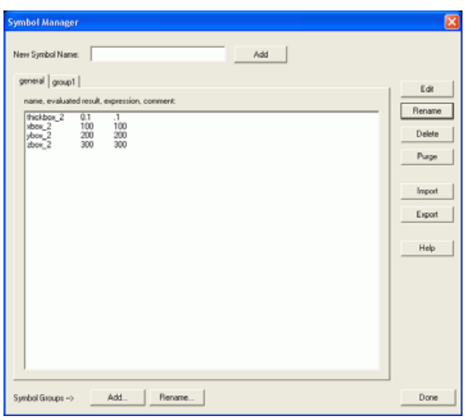
59. Select Rename.

    The Multiple Rename dialog box appears.

60. Type _2 in the **Append string** field and verify that **Append String to each existing name** is selected.

61. Select OK.

    The Symbol Manager dialog box reappears displaying the new symbol names.



62. Select xbox_2.

Add another set of symbols and append the names of the entities with _2.

Symbols can be imported directly from DWG files as well as exported symbol files.

Also change the values of xbox, ybox and zbox to 1/2 of the current values.

<table>
<tr><th colspan="2">Parameterized Box Example (Continued)</th></tr>
<tr><td>

63. Select Edit.

    The Expression Editor dialog box appears with the current xbox_2 information.

64. Highlight the current value in the Expression field if not already highlighted and type 50 (1/2 of the current value of 100).

65. Select OK to close the Expression Editor dialog box.

    The edited value displays in the Symbol Manager.

66. Repeat this process for ybox and zbox, changing the current values by 1/2.

    The Symbol Manager dialog box reflects the changes.

67. Select Done to close the Symbol Manager.

</td><td>

The second box reflects the change in size.

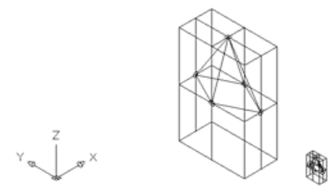


Figure 20-34: Second Box Edited

</td></tr>
<tr><td colspan="2">These steps may be repeated for as many boxes that are in the model.</td></tr>
<tr><td>

68. Select **File** > **Exit.**

    A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes.

69. Select Yes.

</td><td>

Exit Thermal Desktop and save as prompted.

</td></tr>
</table>