1.4 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:

- 1.1 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 - legacy\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.

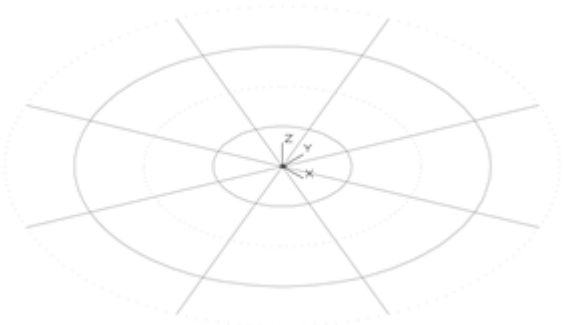
2. Rename the copied template file to `beercan.dwg`.
3. Start Thermal Desktop by double clicking on the `beercan.dwg` drawing file icon in the beercan directory.

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. Select **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**. 23. Select **OK** to close the **User Preferences** dialog box.	These steps change the units for the model to inches.

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>:`	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.
28. Press <**Enter**>. The Command line should now read: `Enter start angle or pick/enter point <0.0>:`	
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 `option` button next to **Edge Nodes** to select it.	

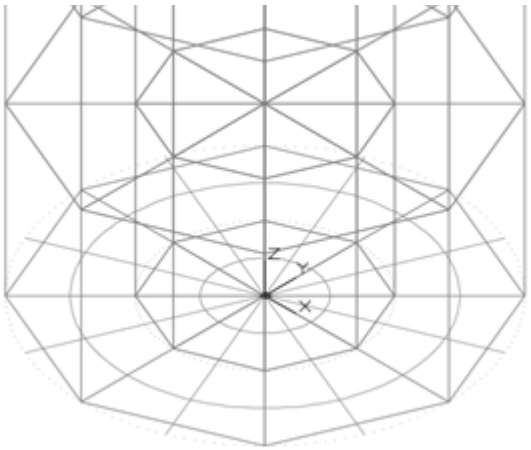
Beer Can Example (Continued)	
33. Highlight the current value in the **Angular Equal** field (subdivisions) and type **9**. 34. Highlight the current value in the **Radial Equal** field (divisions) and type **3**. 35. Click on the **Cond/Cap** tab. 36. Click on the arrow next to the **Material** field 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.	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.
39. Select **Thermal** > **Preferences**. The **User Preferences** dialog box appears. 40. Select the **Graphics Visibility** tab. 41. Clear **TD/RC Nodes**. 42. Select **OK** to close the **User Preferences** dialog box.	These steps turn off TD/RC node visibility for all the nodes that are attached to surfaces or solids.
43.  or **View** > **Zoom** > **Extents**.	

Beer Can Example (Continued)

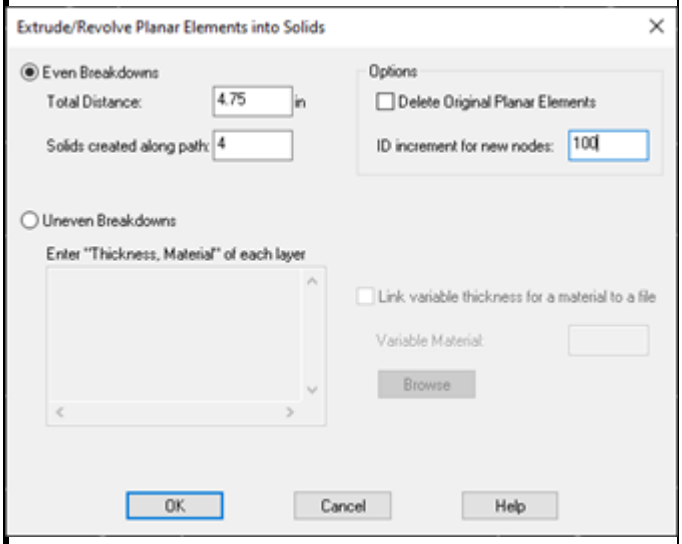
44. Select the newly created disk.
45. Select **Thermal** > **FD/Fem Network** > **Extrude Normal To Planar Elements into Solids**.

    The **Extrude/Revolve Planar Elements into Solids** dialog box appears. (Next page)

The disk will be extruded into solid elements.



46. Leave **Even Breakdowns** selected.
47. Highlight the current value in the **Total Distance** field and type **4.75**
48. Highlight the current value in the **Solids created along path** field and type **4**
49. Highlight the current value in the **ID Increment for new nodes** field and type **100**



50. 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)

51.  or **Thermal** > **Edit**.

    The Command line should now read:

```
Select Objects or [Indiv MB]:
```

52. Type **all** in the Command line.

    The model in the drawing area is selected and the message below appears in the Command line area.

```
Select Objects or [Indiv MB]:
```

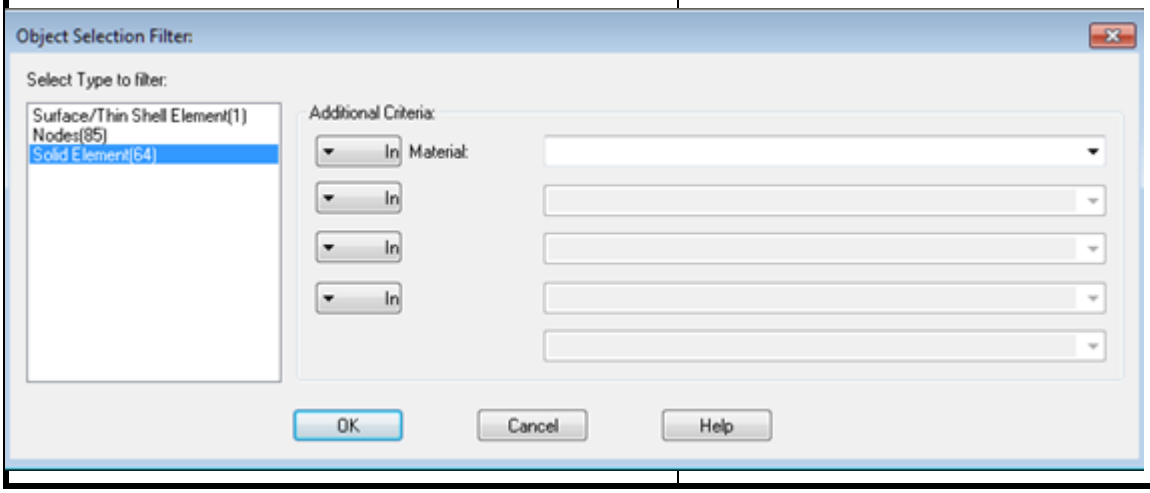
53. Press <**Enter**>.

    The **Object Selection Filter** dialog box appears.

54. 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.

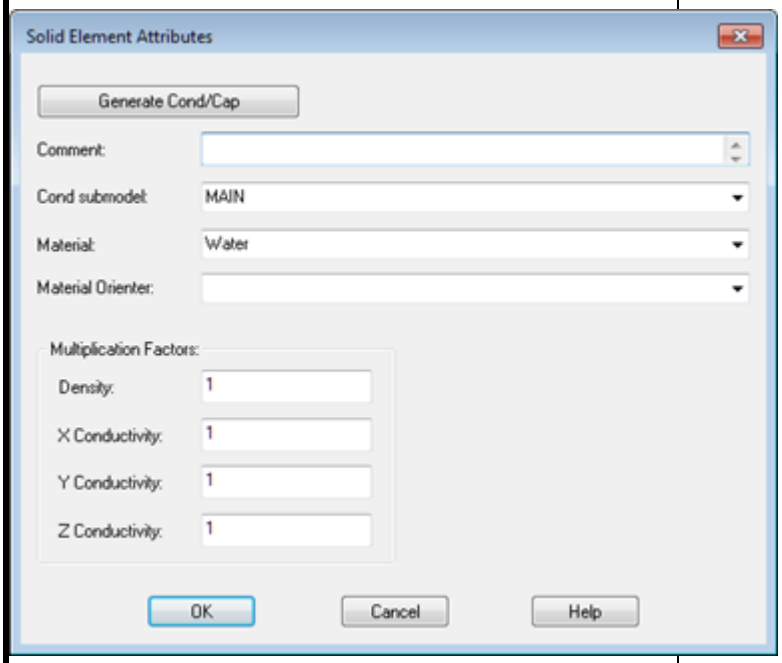
Only one type of object can be edited at a time. The Object Selection Filter makes it easy to select the desired object from the list.



Beer Can Example (Continued)

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

    The **Solid Elements Attributes** dialog box appears.



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

    A **Multi Edit Dialog** window appears confirming the change.
58. Read the content of the dialog box and select **Apply Changes**.

Beer Can Example (Continued)

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

    The Command line should now read:

```
Select the solids for free face calcula-
tions or [MB]:
```

60. Type **all** in the Command line.

    The Command line should now read:

```
64 found
Select the solids for free face calcula-
tions or [MB]:
```

61. Press <**Enter**>.

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

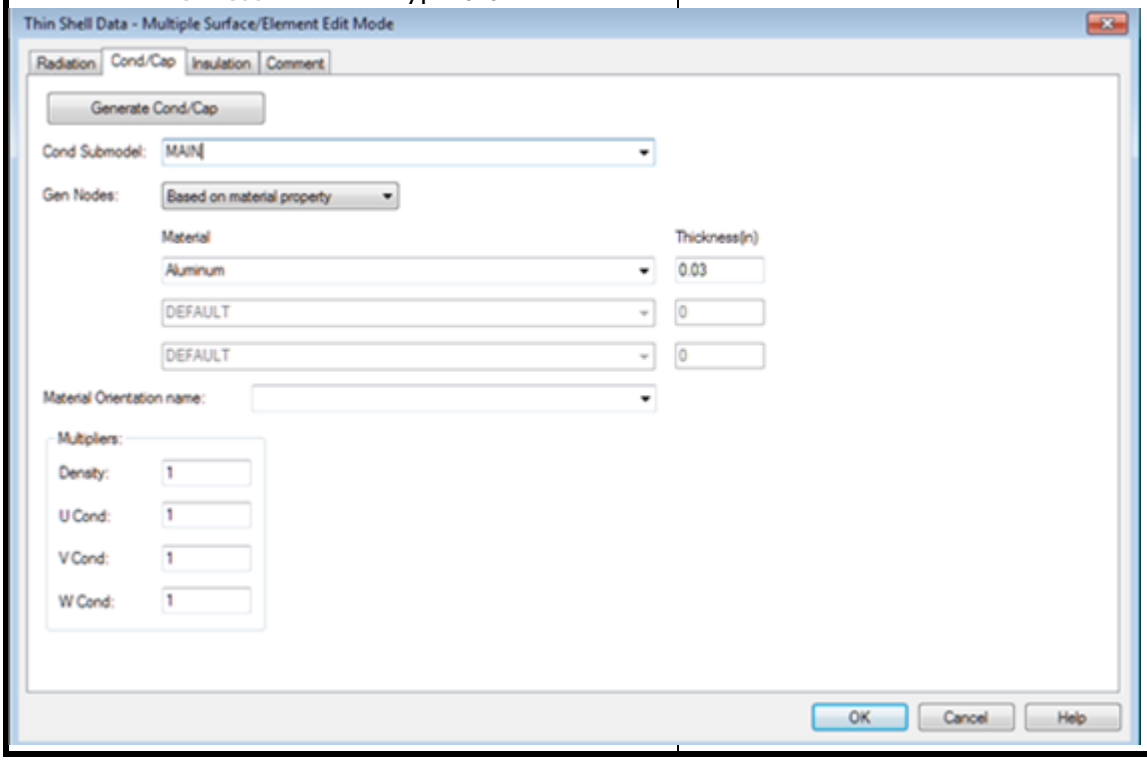
62. Click on the **Cond/Cap** tab.

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

64. Highlight the current value in the **Thickness** field and type **0.0**3

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)

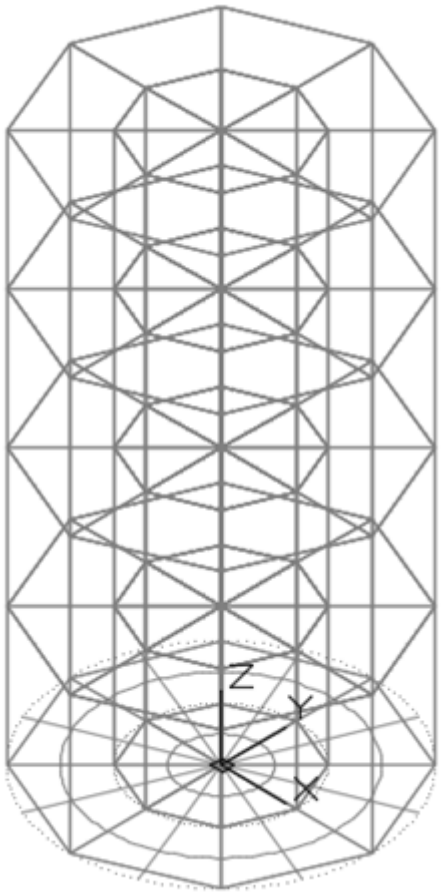
65. Select **OK** to close the dialog box.

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

    A text file indicating the number and type of surface elements that were created by the surface coat operation will also appear. You can close this.

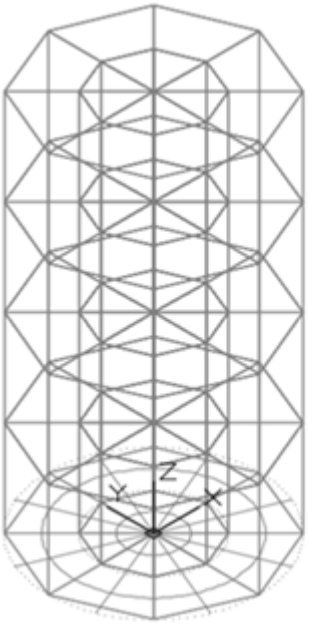
66. Read the content of the dialog box and select **Apply Changes**.

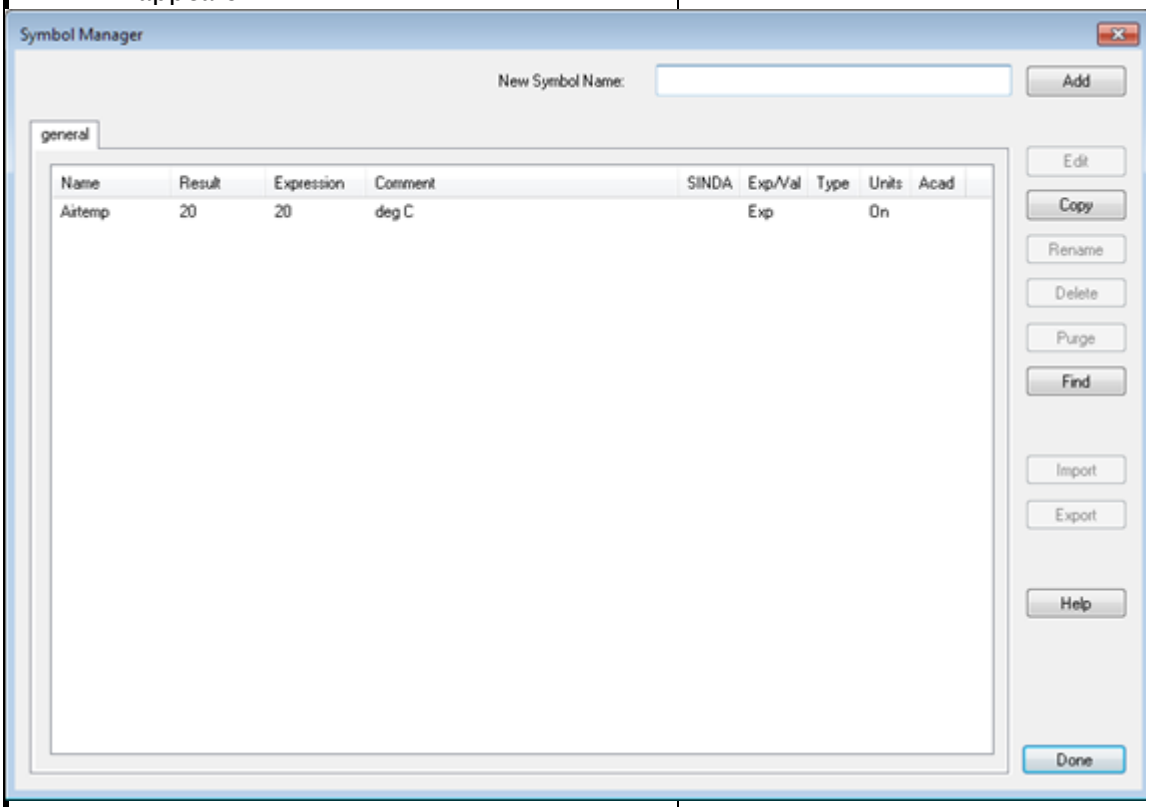
67.  or **View** > **Zoom** > **Extents**.



Beer Can Example (Continued)	

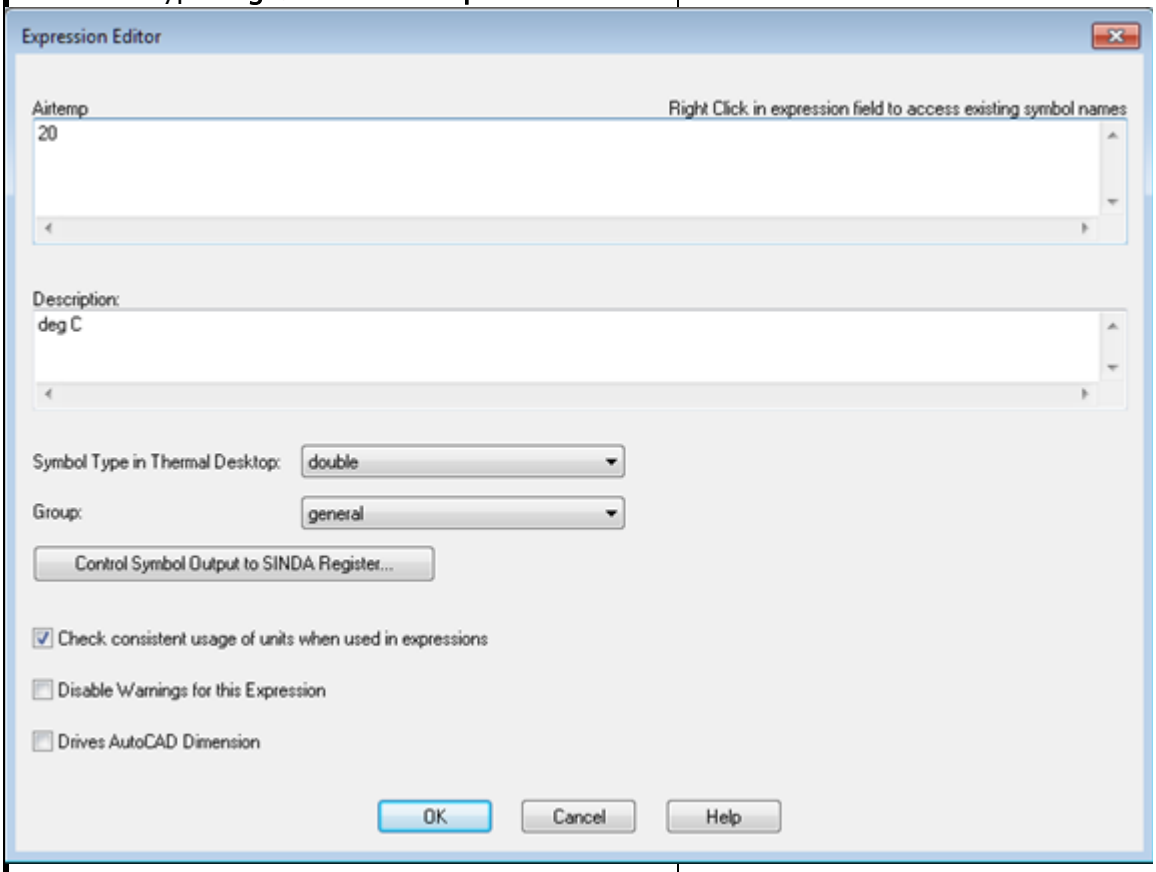
68.  or **Thermal** > **Edit**.
`Select Objects or [Indiv MB]:`
appears in the Command line area
69. Type **all** in the Command line.
The command line should read:
`198 found`
`Select Objects or [Indiv MB]:`
70. Press <**Enter**> to end the selection process.
The **Object Selection Filter** dialog box appears.
71. Click on **Nodes(85)** in the **Select Type to Filter** field to select it.
72. Select **OK** to close the dialog box.
The **Node - Multi Edit Mode** dialog box appears.
73. Highlight the current value in the **Initial temp** field and type **278.15**.
74. Select **OK** to close the dialog box.
A **Thermal Desktop/AutoCAD** dialog box appears confirming the change.
75. 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)	
76.  or **Thermal** > **FD/Fem Network** > **Node**. The Command line should now read: `Enter location of node:` 77. 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. 
78. Select the newly created node. 79.  or **Thermal** > **Edit**. The **Node** dialog box appears. 80. Highlight the current value in the **Sub-model** field and type **Air**. 81. Click on the `option` button next to boundary in the **Type** field to select it. 82. 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 sometimes 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 displayed when the mouse is double clicked in a field.

Beer Can Example (Continued)	
83. 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.
	
84. Type **Airtemp** in the **New Symbol Name** field. 85. Click **Add**. An **Expression Editor** dialog box for Airtemp appears.	

Beer Can Example (Continued)

86. Type **20** in the **main entry** field.
87. Type **deg C** in the **Description** field



88. 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)

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

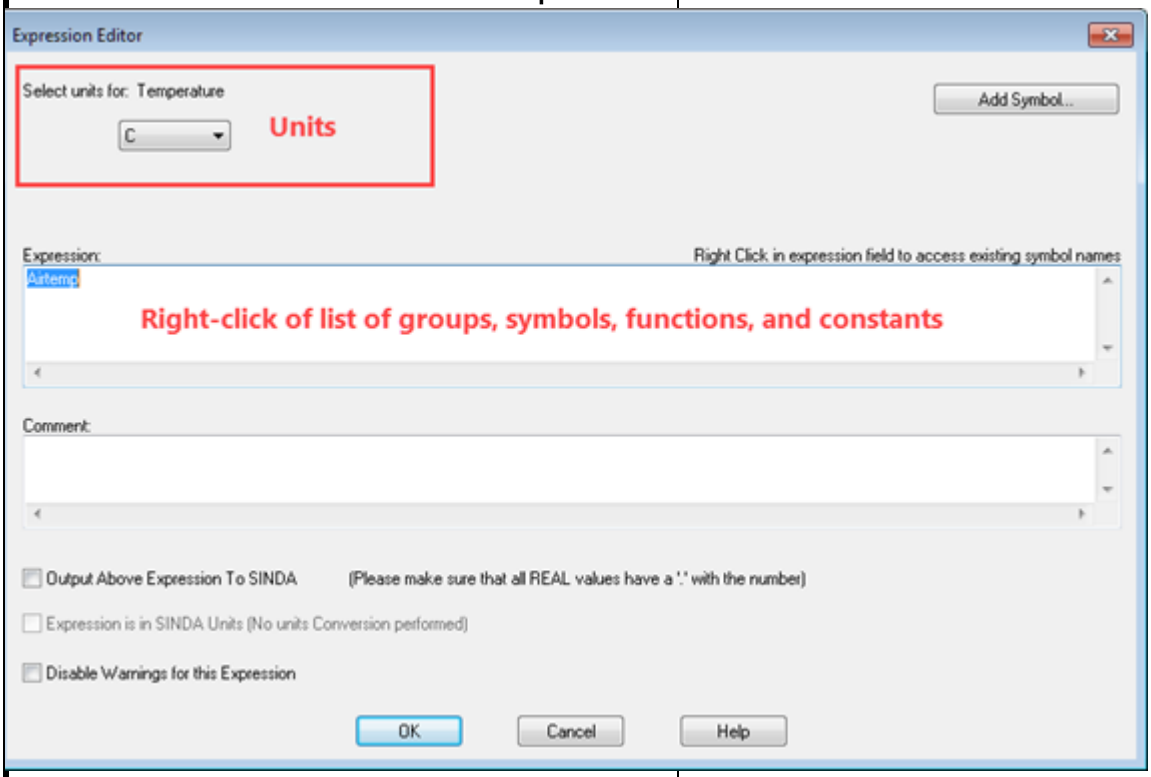
    The **Expression Editor** dialog box reappears.

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

91. 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.



92. 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)

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

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

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

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

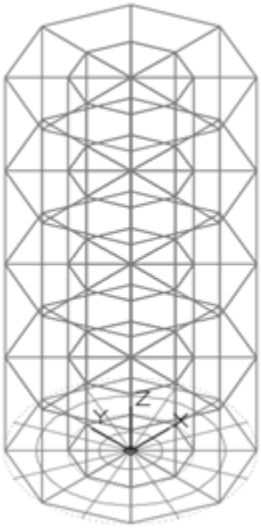
95. Type **Zoom** in the Command line.

    The Command line should now read:

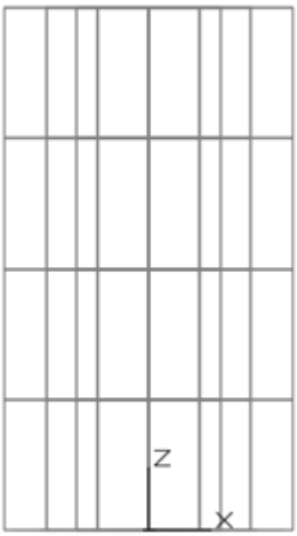
```
All/Center/Dynamic/Extents/Previous/
Scale/Window/Object <real time>:
```

96. 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)

97.  or type **layer** in the Command line.

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

The **Layer Properties Manager** dialog box appears.

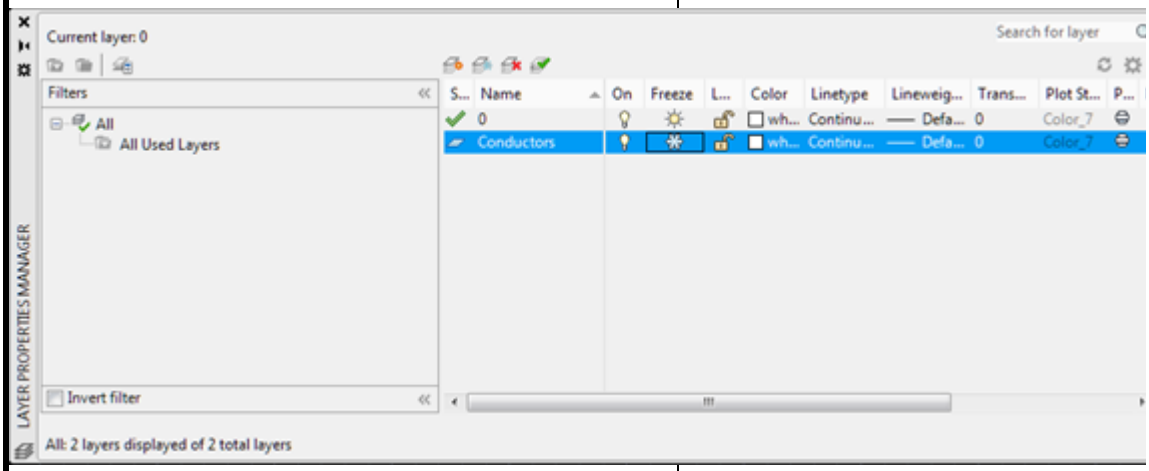
98.  to `create a new layer.`

A new layer named **Layer1** is added.

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

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

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



101. Close the **Layer Properties Manager** dialog box.

Beer Can Example (Continued)

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

The Command line should now read:

```
Select node or [MB]:
```

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

The Command line should now read:

```
Select surfaces or [MB]:
```

104. 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:
```

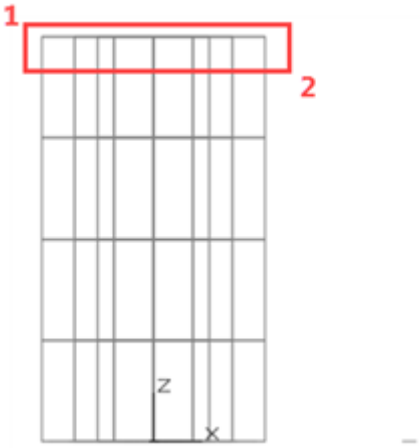
105. Press <**Enter**>.

106. Select the new conductor.

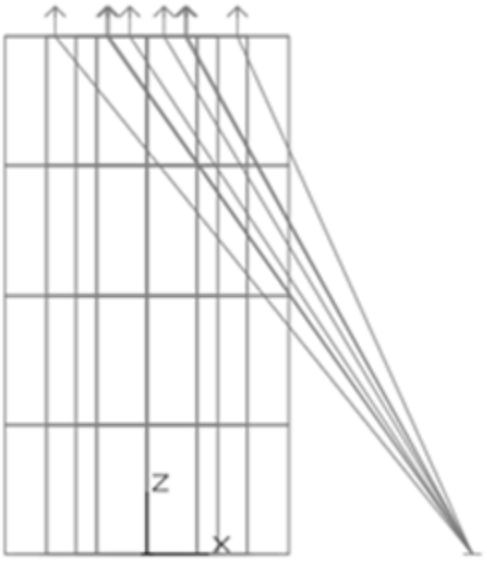
Note: The new conductor set can be selected by picking any line of the set.

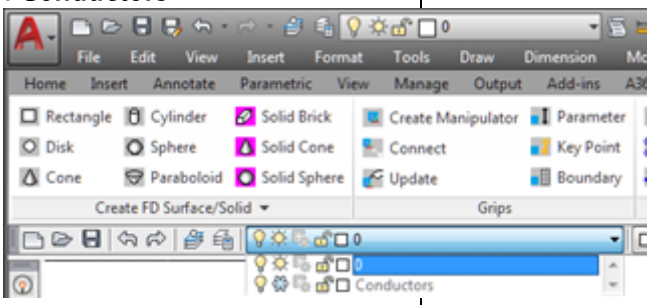
The next steps create the conductors and connect them to the surface.

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)	
107. **Thermal** > **Edit**. The **Conductor** dialog box appears. 108. Type **Top Convection** in the **Comment** field. 109. Click on the **Type** arrow and select **Natural Convection Horizontal Flat Plate Upside** from the drop-down list. 110. Select **Change Fluid**. 111. Expand Library and select **Air (Perfect Gas)** 112. Select **OK** to close the fluid selection. 113. Highlight the current value in the **Area/Perimeter** field and type **0.65625** 114. 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
115. Select the new conductor. 116. Click on the **Layer Control** drop-down in the upper right toolbars, as shown, and select **Conductors**.  117. Select **OK** 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)

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

The Command line should now read:

```
Select node or [MB]:
```

119. Select the boundary node (lower right).

The Command line should now read:

```
Select surfaces or [MB]:
```

120. 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.

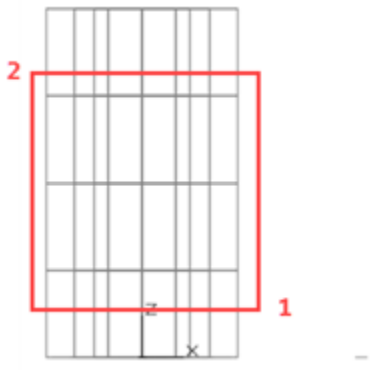
The Command line should now read:

```
Select surfaces or [MB]:
```

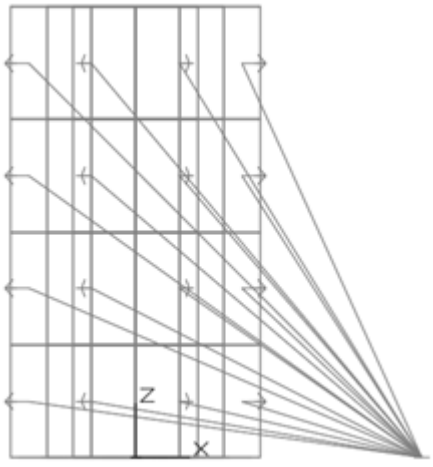
121. Press <**Enter**>.

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)	
122. Double-click the new conductor. The **Conductor** dialog box appears. 123. Type **Side Convection** in the **Comment** field. 124. 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. 125. Select **Change Fluid**. 126. Expand `Library` and select **Air (Perfect Gas)** 127. Select **OK** to close the fluid selection. 128. Highlight the current value in the **Height** field and type **4.75**. 129. Highlight the current value in the **Diameter** field and type **2.625**. 130. Select **OK** to close the **Conductor** dialog box.	
131.  or select **Thermal** > **Model Browser**. The Model Browser appears. 132. Select **List By** > **Conductors** in the Model Browser. The Model Browser tree displays the Conductor Tree 133. 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. 134. 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)	

135. 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.
	
	Look at the output and see if any are found.

Beer Can Example (Continued)

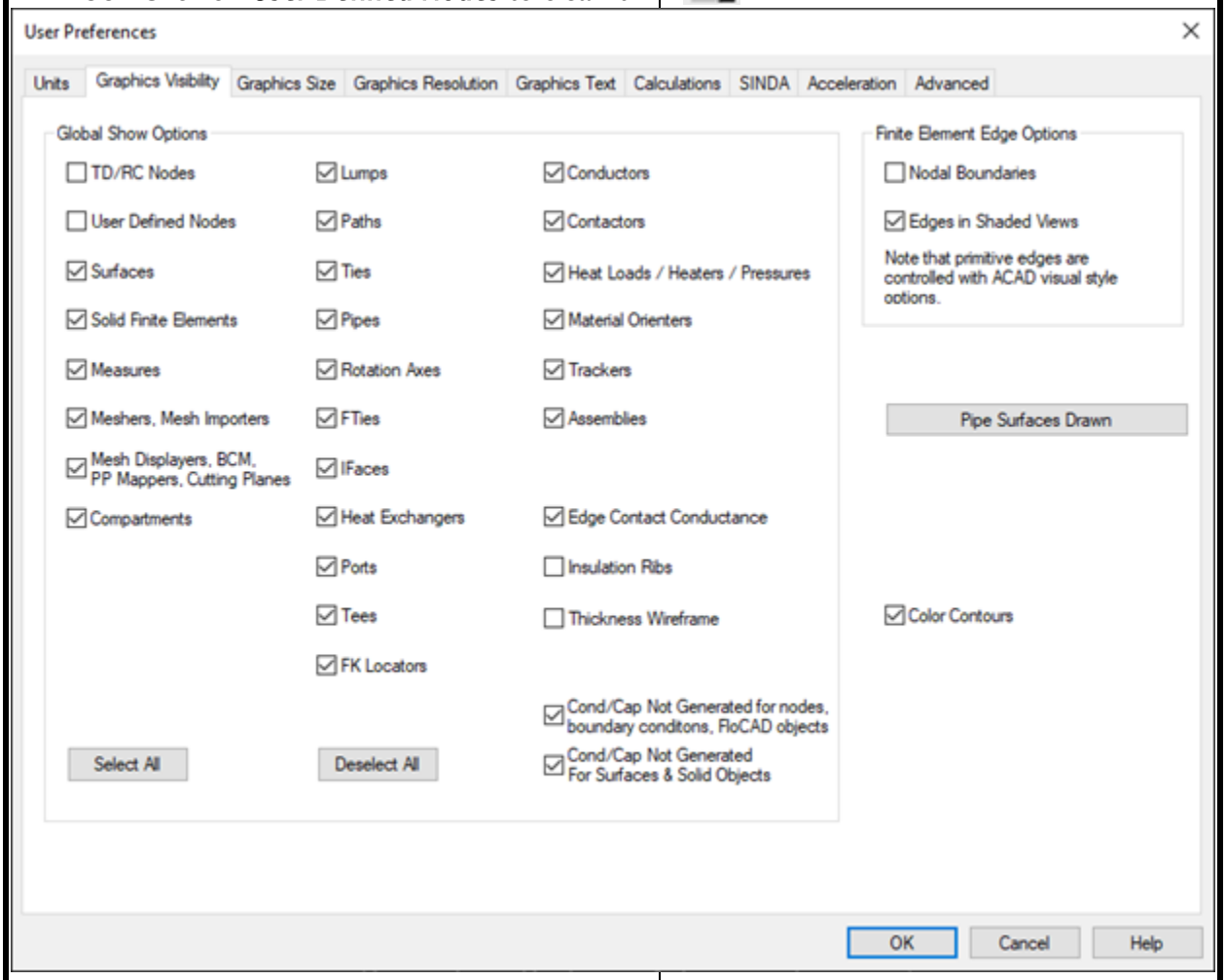
136. Select **Thermal** > **Preferences**.

     The **User Preferences** dialog box appears.

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

138. Click on **User Defined Nodes** to clear it.

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 or ribbon. For User Defined Nodes, it is the icon:



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

     The air node disappears from the drawing area.

Beer Can Example (Continued)	
140.  or type **LAYER** in the Command line. The **Layer Properties Manager** dialog box appears.	Create a new layer called RightSide, which is where the right side of the beer can will be placed.
141.  to create a new layer. A new layer named **Layer1** is added to the existing layers.	
142. Highlight the name **Layer1** if not already highlighted. Type **RightSide** to change the name of this newly created layer. 143. 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. 144. 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)

145. 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.

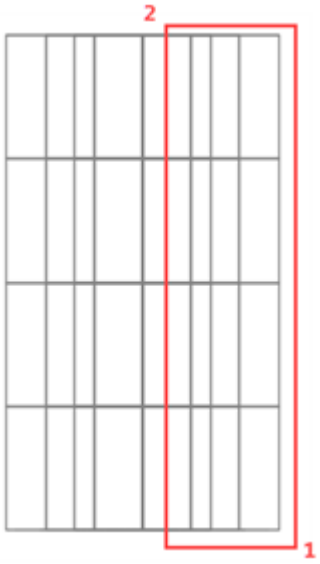
- Position the mouse as shown in to the right (above and to the right of the middle line) and click the left mouse button (2). Note that as the mouse is moved, a box is drawn around the area.

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

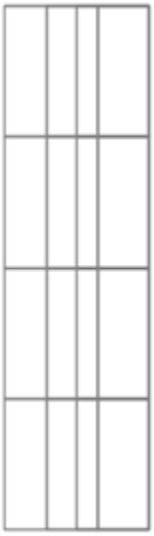
146. Click on the **Layer Control** drop-down in the upper right toolbars, as shown, and select **RightSide**.

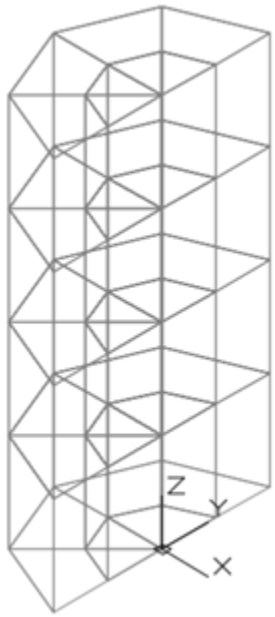
147. Select **OK** to confirm the change and close the dialog box.

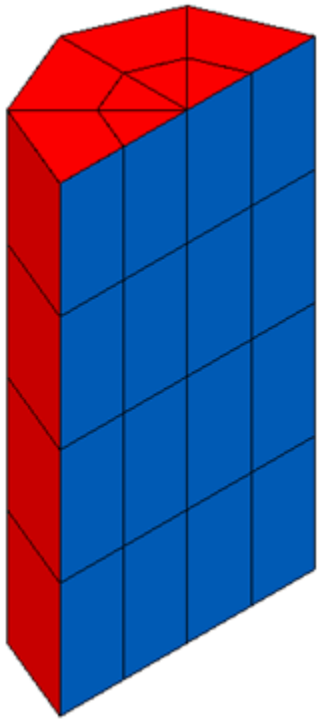
Split the beer can into two sides so that the temperatures in the middle of the beercan can be determined later in the exercise. \

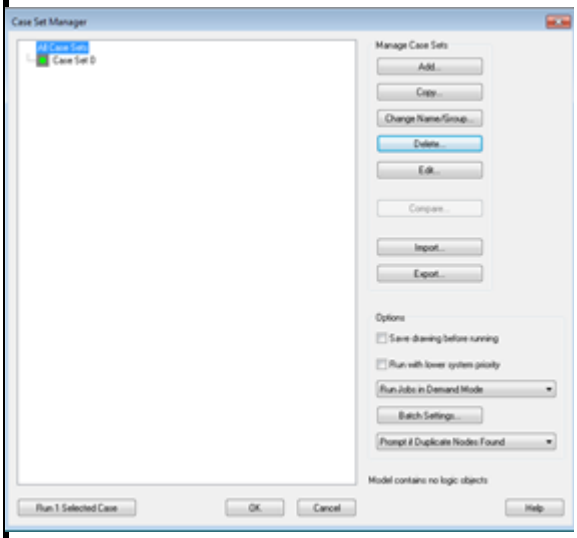


Once the selected right side is moved to the RightSide layer, the drawing area should look similar to the example below:



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

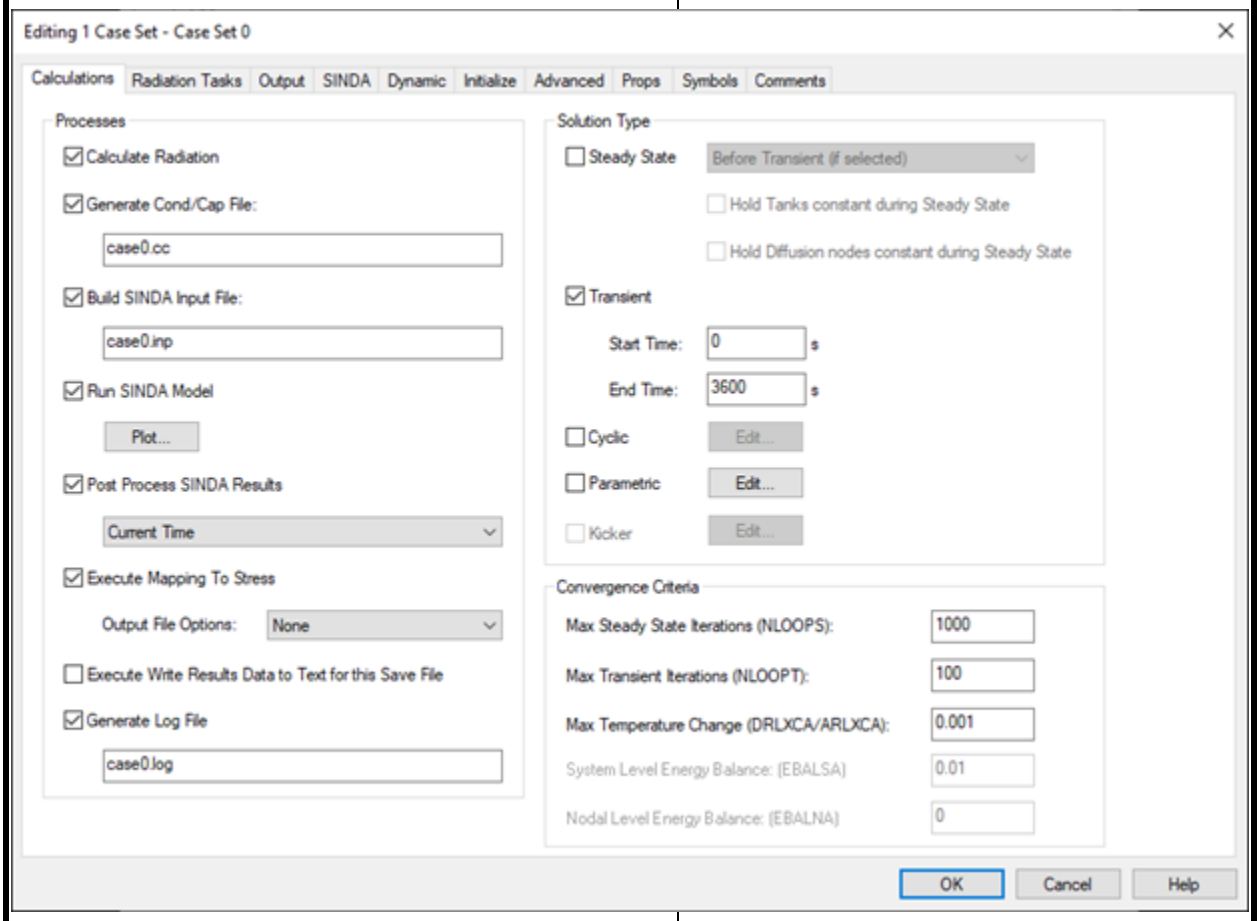
Beer Can Example (Continued)	
149.  or **Thermal** > **Model Checks** > **Color by Property Value** > **Conductivity**. **Note:** If the blue is a little dark, feel free to rotate a little bit to see if better. This is also a good time to review graphics settings 150. Select **View** > **3D Views** > **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 correct, edit the material property of the incorrect entities. Some elements may appear wrong, but this is likely a graphics issue. 
151.  or **Thermal** > **Model Checks** > **Color by Property Value Off**. The model reverts back to the previous wireframe view.	

Beer Can Example (Continued)	
152.  or **Thermal** > **Case Set Manager**. The **Case Set Manager** dialog box appears.  153. Select **Edit**. The **Case Set Information** dialog box appears.	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)

154. Select the **Calculations** tab.
155. Uncheck **Steady State** under Solution Type.
156. Check **Transient** under Solution Type.
157. 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.

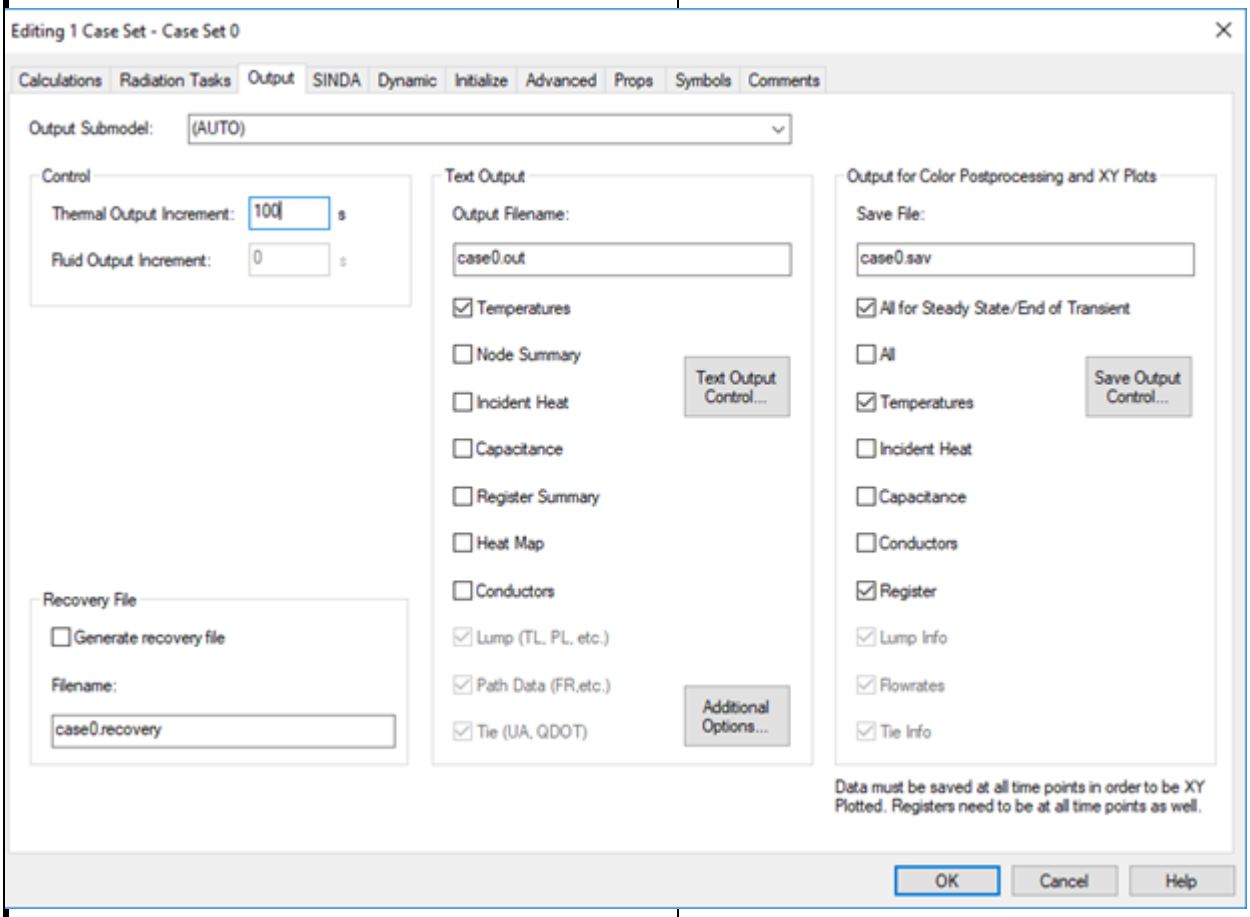


Beer Can Example (Continued)

158. Select the **Output** tab.

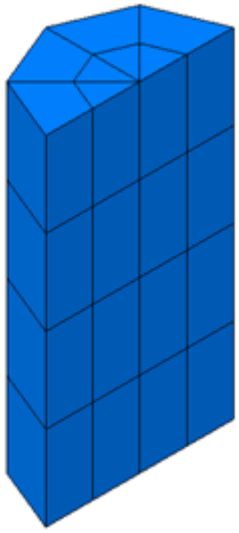
159. Highlight the current value in the **Thermal Output Increment** field and type **100**.

The Output Increment defines how often during the solution the chosen values will be written to the output files.



160. Select **OK**.

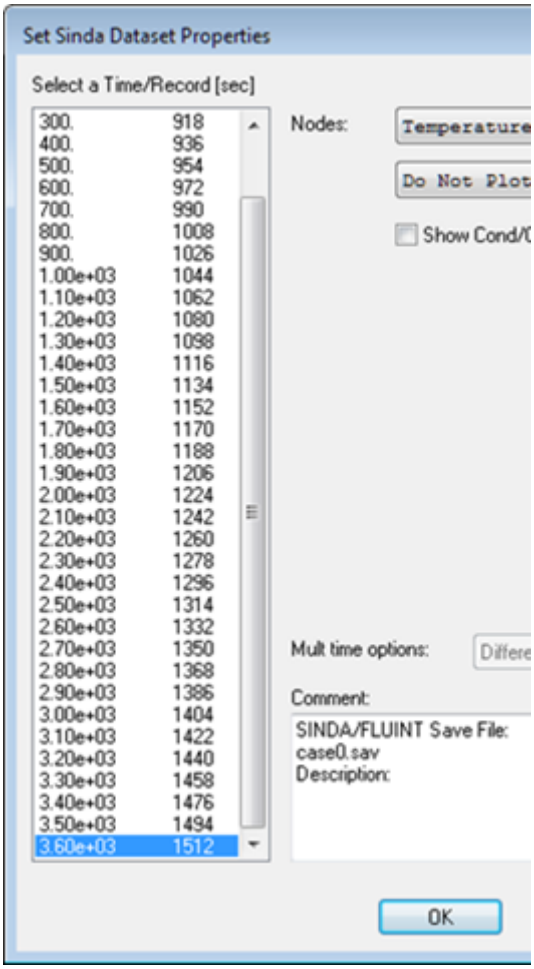
The **Case Set Manager** dialog box is on the screen.

Beer Can Example (Continued)	
161. Select **Run 1 Selected Case**. A **SINDA/Fluint Run Status** dialog box appears stating the successful completion of the processor. 162. 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. If you don't see the Run Status Window, check behind your Thermal Desktop window.

Beer Can Example (Continued)

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

The **Set SINDA Dataset Properties** dialog box appears.

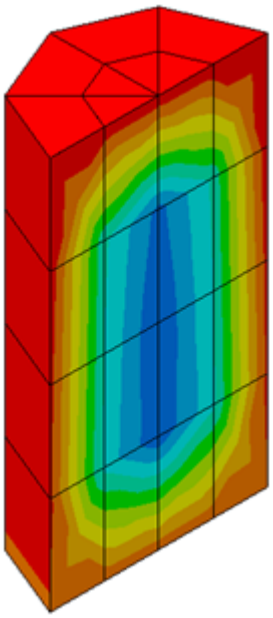


164. Scroll down the list in the **Select a Time/Record [sec]** field and select **3600 (3.60e+03)**.

165. 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, check graphics settings.



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

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

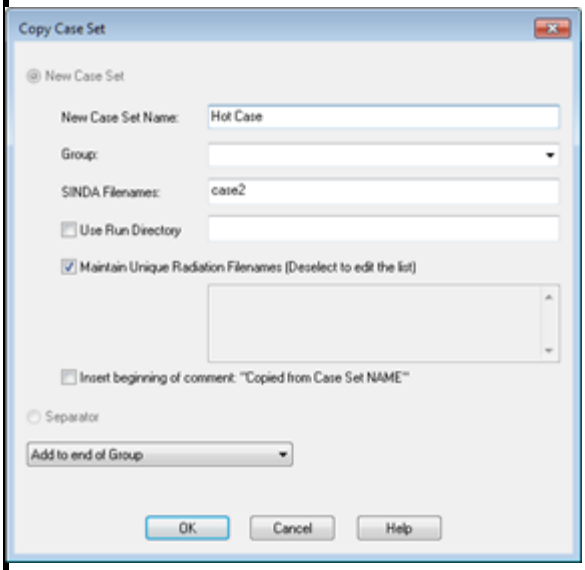
Beer Can Example (Continued)

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

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

168. Select **Copy**.

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



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

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

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

171. Select **Hot Case**.

172. Select **Edit**.

The **Editing 1 Case Set – Hot Case** dialog box appears.

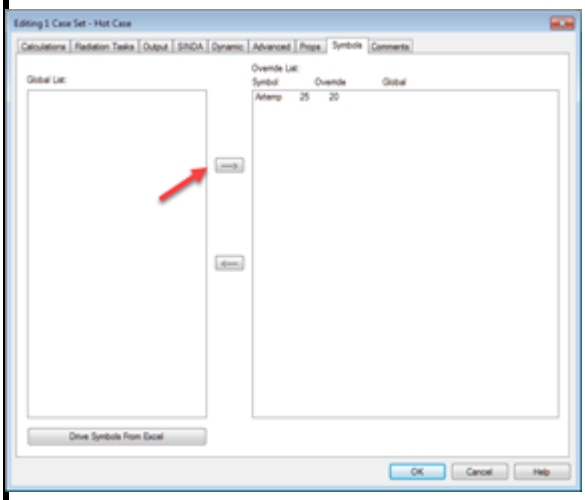
Create a hot case where the air temperature is 25 °C.
By overriding the global definition of 20 °C with 25 °C, the new case can be run quickly and it will be able to go back to it at a later time.
When this case is run, all the SINDA files will go to case1.*
Once the run is finished, edit the post-processing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

Beer Can Example (Continued)

173. Select the **Symbols** tab.

174. Select **Airtemp** in the **Global List** field to highlight it.

175. Click on the right arrow located in the center of the dialog box.

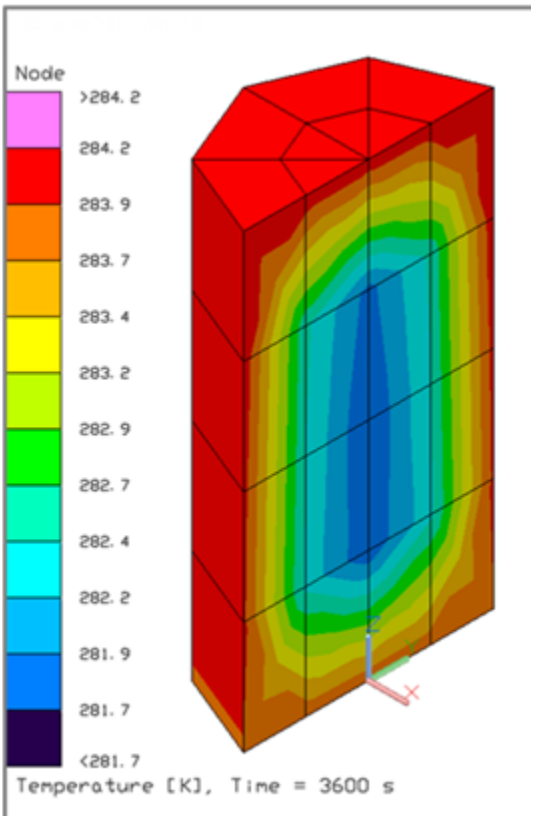


The **Expression Editor (Airtemp)** dialog box appears.

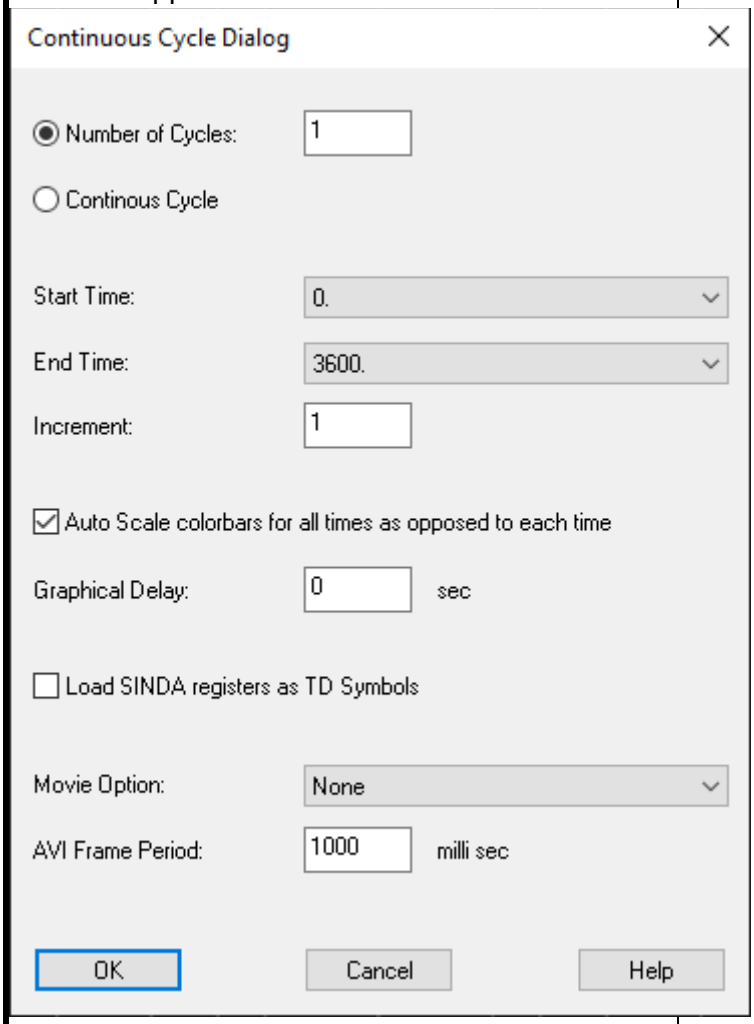
176. Highlight the current value in the **main entry** field (20) and type **25**

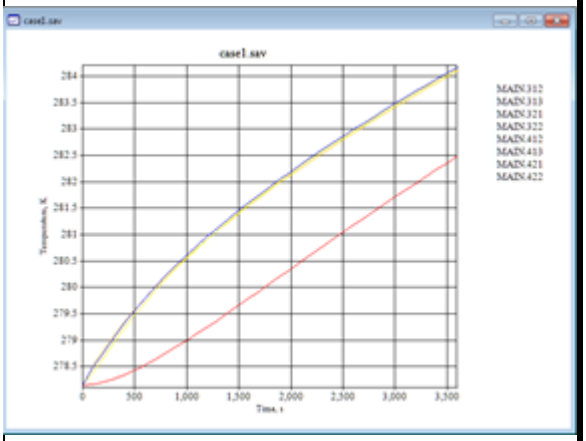
177. Select **OK** to close the dialog box.

The **Case Set Information – Hot Case** dialog box displays the change.

Beer Can Example (Continued)	
178. Select **OK** to close the **Case Set Information - Hot Case** dialog box and redisplay the **Case Set Manager** dialog box. 179. Select **Run 1 Selected Case**. A **SINDA/Fluint Run Status** dialog box appears stating the successful completion of the processor. 180. Select **OK** to close the dialog box. The model changes from the geometric view. 181.  or **Thermal** > **Post Processing** > **Edit Current Dataset**. The **Set SINDA Dataset Properties** dialog box appears. 182. Scroll down the list in the **Select a Time/Record [set]** field and select **3600 (3.60e+03).** 183. Select **OK**.	
184.  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)	
185.  or **Thermal** > **Post Processing**> **Animate Through Time**. The **Continuous Cycle** Dialog dialog box appears.	This command will animate through all the times on the postprocessing file.
186. Select **OK**. 187. View the screen.	
188.  or **Thermal** > **Post Processing** > **Post Processing Off**. The model returns to the geometric view in the drawing area.	



Beer Can Example (Continued)	
189. Select an element in the drawing 190.  or select **Thermal** > **Post Processing** > **X-Y Plot Data vs. Time**. 191. 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.
192. Select **File** > **Exit**. A **Thermal Desktop/AutoCAD** dialog box appears asking to save the drawing changes. 193. Select **Yes**.	Exit Thermal Desktop and save as prompted.