1.5 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:

■ 1.1 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 `\Tutori-als\Thermal Desktop - legacy\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`.

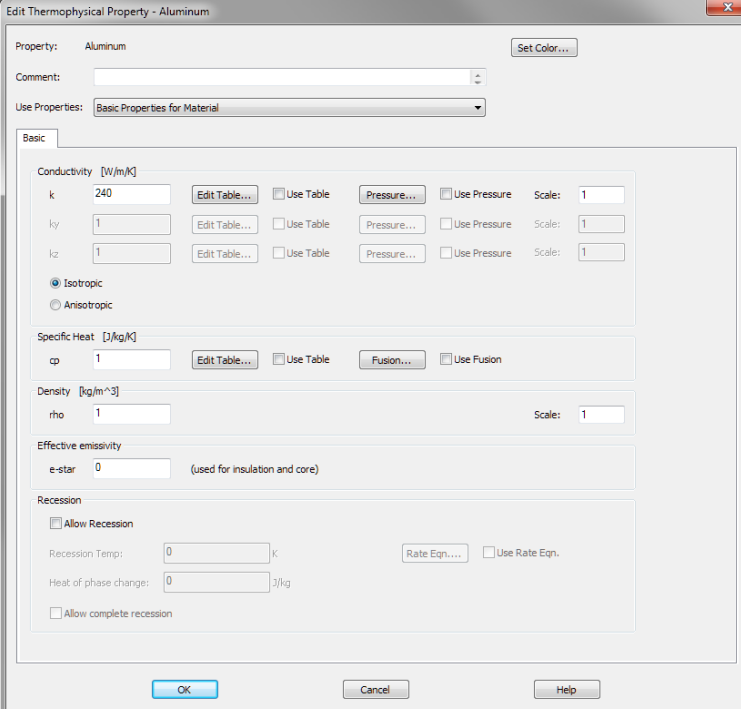3. Start Thermal Desktop by double clicking on the `fe1.dwg` file icon in the `finiteElement` directory.

---

4. Move the UCS for easier visibility by doing one of the following:

   ■ Click **View > Display > UCS Icon > Origin** to turn off that option.

   ■ Right-click on the UCS icon and select **UCS Icon Settings > Show UCS Icon at Origin** to turn of the option.

Finite Element Example (Continued)

5. ![Cp icon] or select **Thermal > Thermophysical Properties > Edit Property Data**.

   The **Edit Thermophysical Properties** dialog box appears.

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

7. Select the **Add** button.

   The **Thermophysical Properties** dialog box appears.

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

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

   The **Edit Thermophysical Properties** dialog box reappears with the updated Aluminum value displayed in the property columns.

10. 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.

Finite Element Example (Continued)

11.   ⊙ or **Thermal > FD/FEM Network > Node**.

The Command line should now read:

`Enter location of node:`

12.   Type **0,0** in the Command line.

The first node is created at the origin.

13.   ⊙ or **Thermal > FD/FEM Network > Node**.

The Command line should now read:

`Enter location of node:`

14.   Type **1,0** in the Command line.

The second node is created.

15.   ⊙ or **Thermal > FD/FEM Network > Node**.

The Command line should now read:

`Enter location of node:`

16.   Type **1,1** in the Command line.

The third node is created.

17.   ⊙ or **Thermal > FD/FEM Network > Node**.

The Command line should now read:

`Enter location of node:`

18.   Type **0,1** in the Command line.

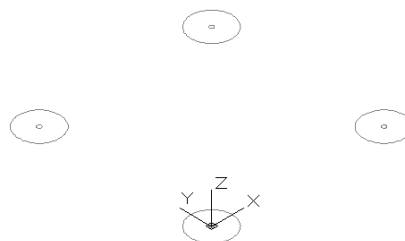The fourth node is created.

19.   ⊠ or **View > Zoom > Extents**.

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.

Finite Element Example (Continued)

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

    The Command line should now read:

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

21. 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]:1 found`
    `Select nodes for linear element or [MB]:`

22. 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]:1 found, 2 total`
    `Select nodes for linear element or [MB]:`

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

    The Command line should now read:

    `Select nodes for linear element or [MB]:1 found, 3 total`
    `Select nodes for linear element or [MB]:`

24. 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]:1 found, 4 total`
    `Select nodes for linear element or [MB]:`

25. 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.
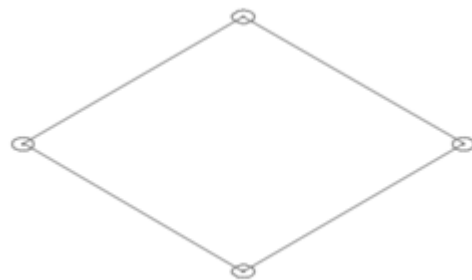
The order in which the nodes are picked is extremely important. Refer to the drawing below to select the nodes.

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)

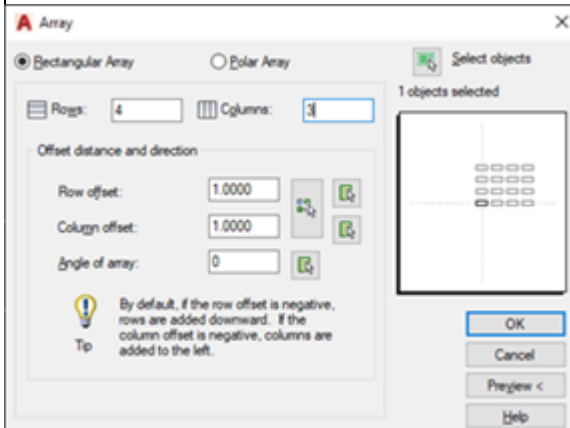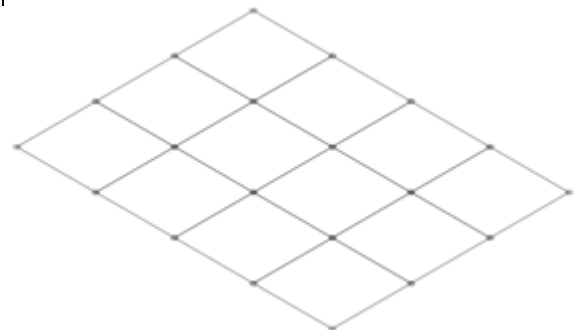|	
26. Double-click the new quad element to edit it. The **Thin Shell Data** dialog box appears. 27. Select the **Cond/Cap** tab. 28. Click on the arrow next to the **Material** field and select **Aluminum** from the drop-down list. 29. Select **OK** to close the dialog box.	The element is being edited to apply the material property created earlier. The thickness does not matter since this element is used for an extrusion.
30. Select the new quad element. 31. Type **ARRAYCLASSIC**. The **Array** dialog box appears. 32. Select **Rectangular Array** if not already selected. 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 calcu-`
`lations or [MB]:`
 	
40. Type **all** in the Command line.
 	
The Command line should now read:
 	
`Select the elements for free edge calcu-`
`lations 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 **0.01** 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.
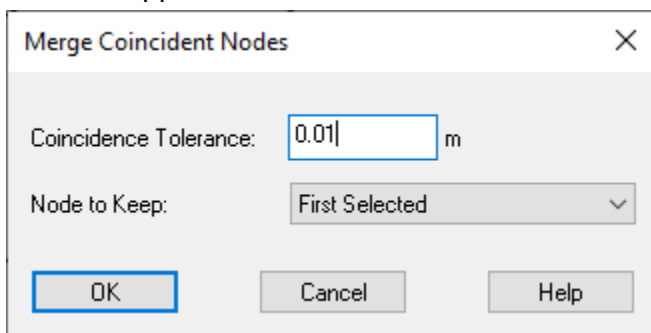
Finite Element Example (Continued)

48. or type **REGEN** in the Command line. The free edges are turned off. 49. Select **Thermal** > **Model Checks** > **Show Free Edges**. The Command line should now read: `Select the elements for free edge calcu-lations or [MB]:` 50. Type **all** in the Command line. The Command line should now read: `Select the elements for free edge calcu-lations or [MB]:` 51. Press **<Enter>**. The outside edge of the array turns red. 52. or type **REGEN** in the Command line. The free edges are turned off.	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.
 	
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.
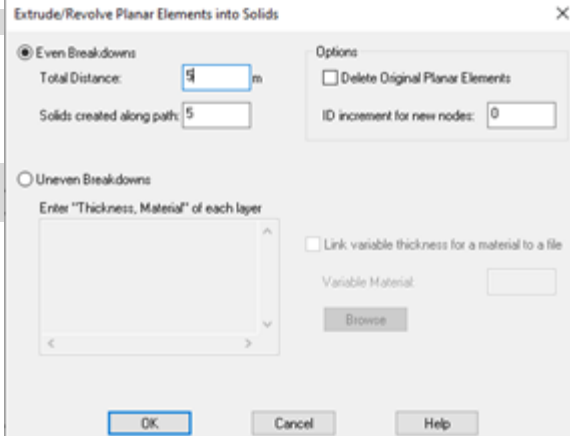
Finite Element Example (Continued)

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.
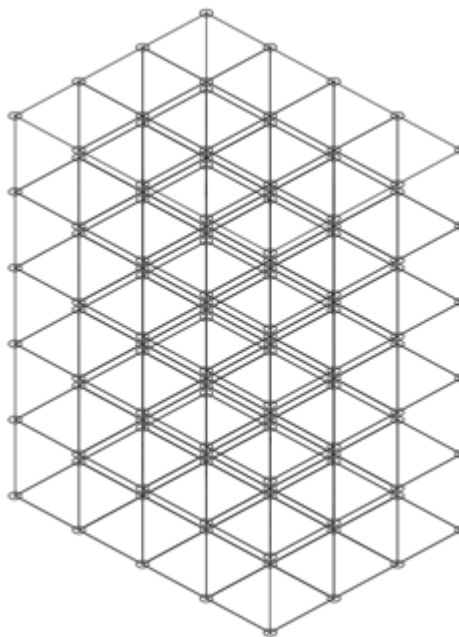
The Command line should now read:

`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.
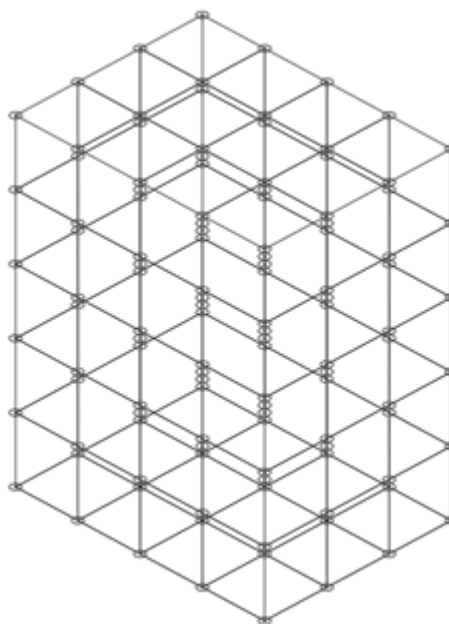
Finite Element Example (Continued)

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.
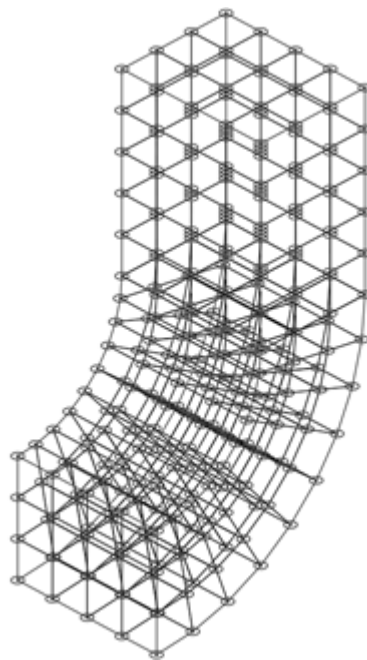
Finite Element Example (Continued)

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]:
```

68.   Press **<Enter>**.

      The Command line should now read:

```
Select base point to revolve from:
```

69.   Type **–3,0**

      The Command line should now read:

```
Select point to define revolve axis:
```

70.   Type **–3,3**

      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.

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.

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.
76. Turn on the Selection Filter by selecting **Thermal > Modeling Tools > Toggle Selection Filter**. The Command line should now read: `Thermal Desktop filter turned on`	The model now consists of planar element and solids. The planar elements must be deleted because their function is complete. The Object Selection Filter is used to delete only the planar elements. 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)

77. ✏️ , select **Modify > Erase**, or type **ERASE**. The Command line should now read: `Select objects:` 78. Type **all** in the Command line. The Command line should now read: `Select objects:` 79. Press **<Enter>**. The **Object Selection Filter** dialog box appears.	These steps will delete the planar elements. If the Object Selection Filter does not open and everything is deleted, press **Ctrl–Z** and repeat Step 76.

Object Selection Filter:

Select Type to filter:
- Surface/Thin Shell Element(12)
- Nodes(300)
- Solid Element(168)

Additional Criteria:
- In Submodel:
- In Material:
- In Optics:
- In Analysis Group:
- Other:

[OK]   [No Filtering]   [Help]

80. Leave the current selections as they appear: **Surfaces/Thin Shell Elements(12)**.
 	
81. Select **OK** to close the dialog box.
 	
The command line should read:
`Filtering from 480 objects`
`12 objects meet criteria`	

82. Turn off the Selection Filter by selecting **Thermal > Modeling Tools > Toggle Selection Filter**. The Command line should now read: `Thermal Desktop filter turned off`	

Finite Element Example (Continued)

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

    The Command line should now read:

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

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

    The Command line should now read:

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

85. 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.

86. Review the **Radiation** tab.

87. Select the **Cond/Cap** tab.

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

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

The **Multi Edit Dialog** window appears asking to confirm the change.

90. Select **Apply Changes** to close the dialog box. A text file indicating the type and number of surfaces created will appear. You may close that as well.

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.

Finite Element Example (Continued)

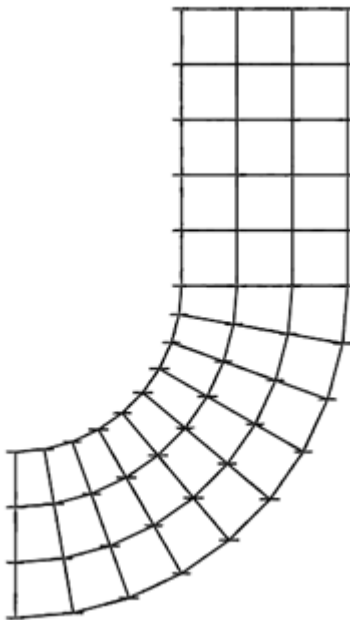91. Select **View > 3D Views > Front**.

92. 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>:
```

93. 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.
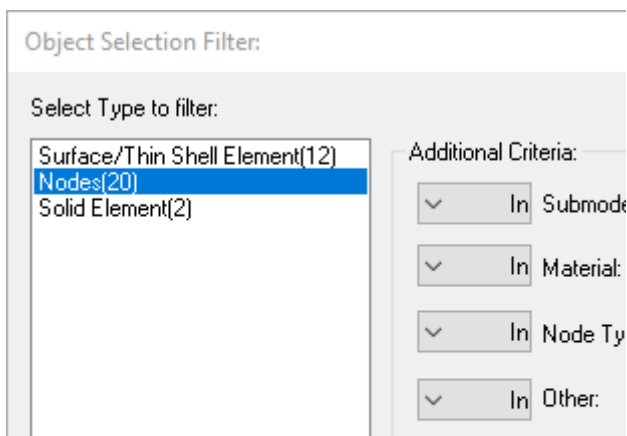
Finite Element Example (Continued)

94. Select from points 1 to 2 as shown in the figure to the right.

95. [icon] or **Thermal > Edit**.

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

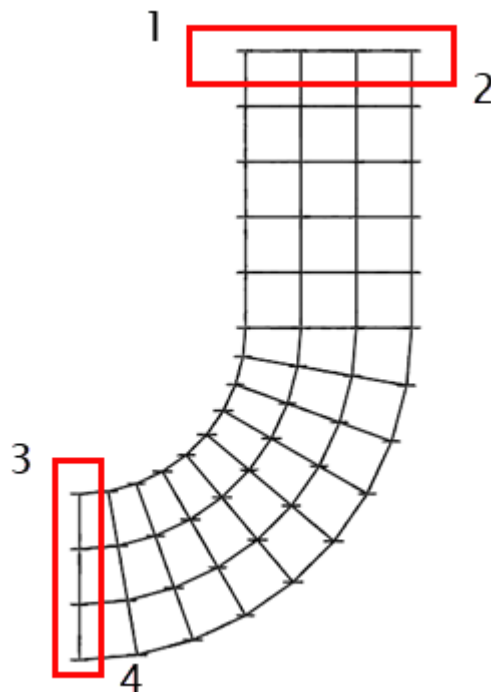96. Select **Nodes(20)** in the `Select Type to filter` field.

97. 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 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 so the **Object Selection Filter** dialog box is displayed regardless of the Toggle Filter setting.

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.

98. Select **Override calculation by elements/surfaces**.

The **Type** frame activates.

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.

99. Click on **Boundary**.

100. Highlight the current value in the **Initial temp** field and type **373.15**

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

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

102. Confirm the changes to close the dialog box.

Finite Element Example (Continued)

103. Select from points 3 to 4 as shown in the figure to the right.

104. or **Thermal > Edit**.

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

105. Select **Nodes(20)** in the **Select Type to filter** field.

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

    The **Node – Multi Edit Mode** dialog box appears.

107. Select **Override calculation by ele-ments/surfaces**.

    The **Type** field activates.

108. Click on **Boundary**.

109. Highlight the current value in the **Initial temp** field and type **273.15**

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

111. 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.
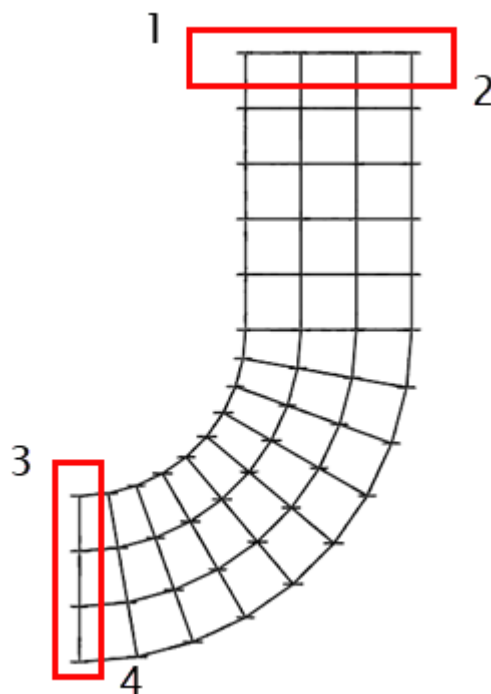
Finite Element Example (Continued)

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

The Command line should now read:

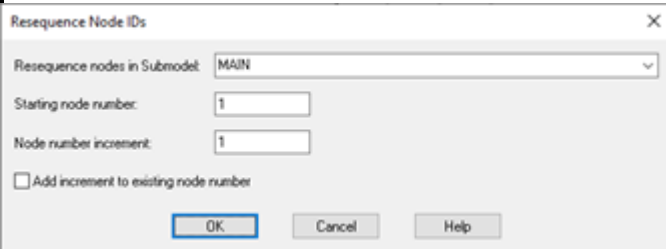`Select entity(s) for Node ID Resequenc-ing or [MB GRP]:`

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

The Command line should now read:

`Select entity(s) for Node ID Resequenc-ing or [MB GRP]:`

114. Press **<Enter>**.

The **Resequencing Node IDs** dialog box appears.

115. 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)

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

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

117. 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 con- ductors 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 tem- perature 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.	

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

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

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

120. 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.

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

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

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.

Finite Element Example (Continued)

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

Node

>373.1

373.1

363.1

353.1

343.1

333.1

323.1

313.1

303.1

293.1

283.1

273.1

<273.1

Temperature [K], Time = 0 s, Steady State, Loopct=2/1000
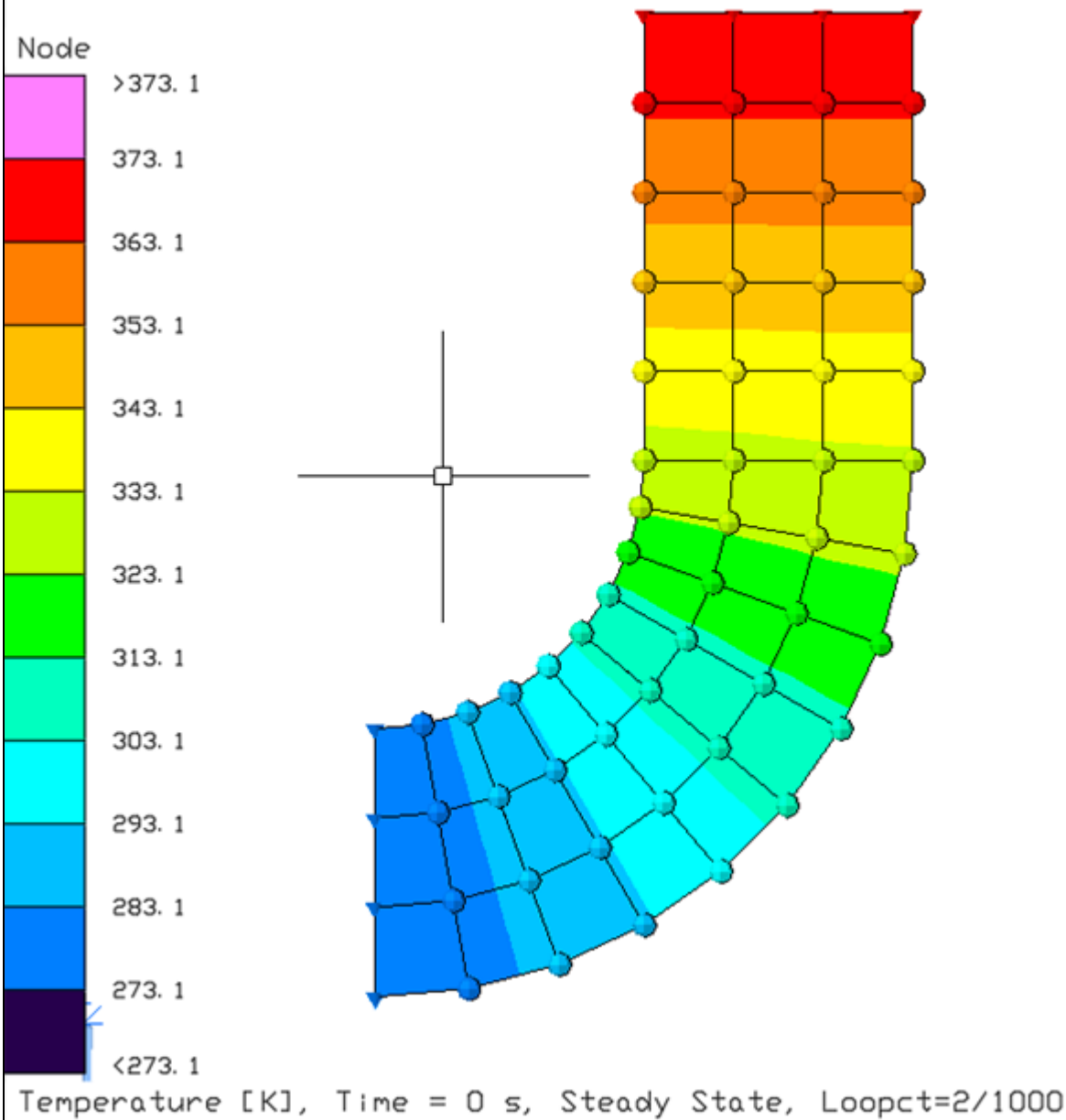
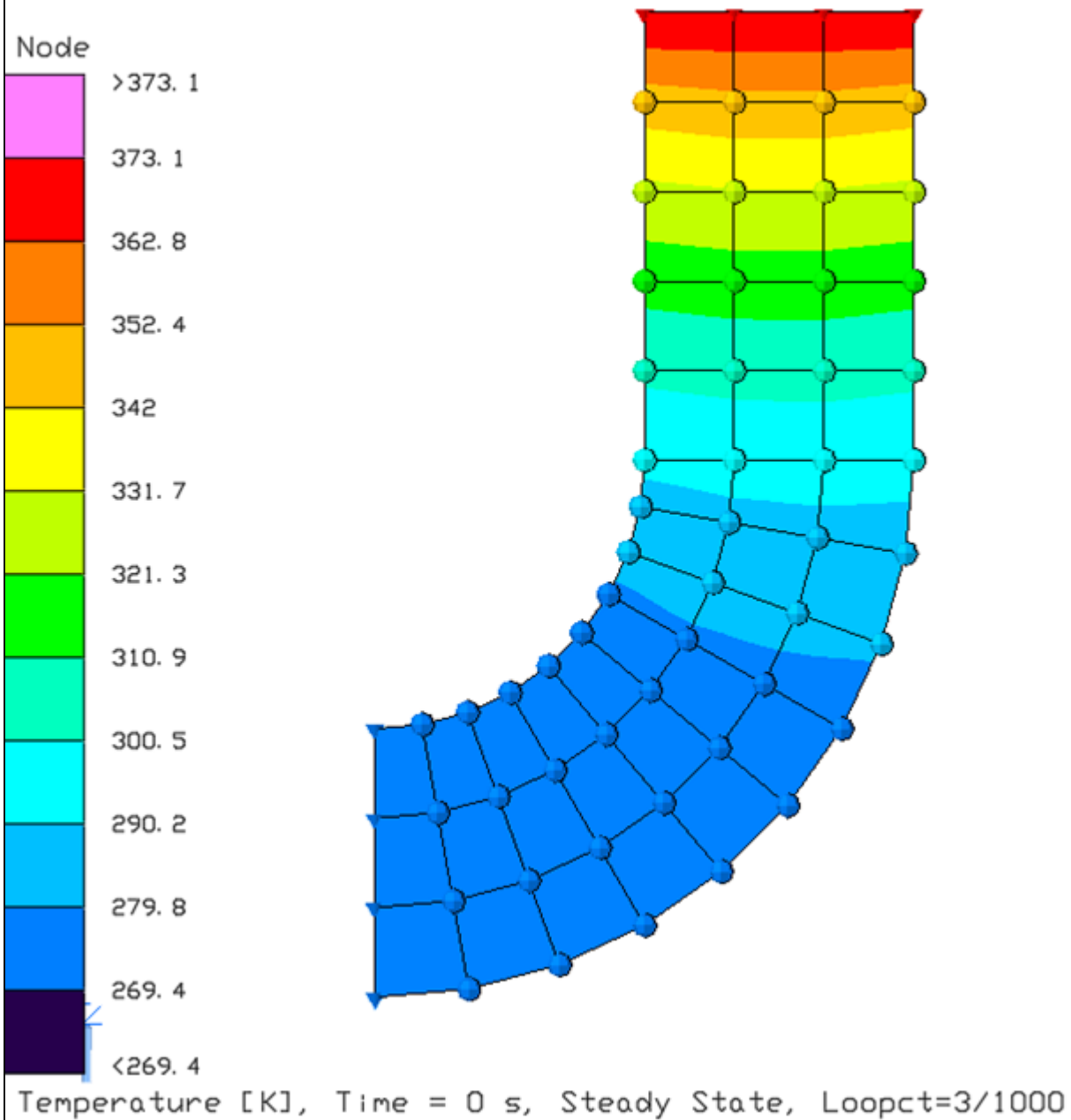| 123. Select **File > Save**	Save the model before adding radiation.
If you have access to a RadCAD license, you may continue; otherwise, skip ahead to Step 138.	

Finite Element Example (Continued)

124. ⬤ or **Thermal > Case Set Manager**. The **Case Set Manager** dialog box appears. 125. Click on **Copy**. The **Copy Case Set** dialog box appears. 126. Type **Radiation** for the **New Case Set Name** 127. Select **OK** to close the dialog box. The **Case Set Manager** dialog box reappears. 128. Double-click **Radiation** Case Set. The **Case Set Information - Radiation** dialog box appears.	
129. Select the **Radiation Tasks** tab, if it is not already selected. 130. Select the **Add** button. The **Radiation Analysis Data** dialog box appears. 131. Confirm **Radks** is selected in the **Calculation Type** region. 132. Confirm **BASE** is in the **Analysis Group** field. 133. Confirm **Monte Carlo** is selected in the **Calculation Method** region. 134. Select **OK** to close the dialog box. The **Case Set Information - Radiation** dialog box reappears. 135. Select **OK** to close the dialog box. The **Case Set Manager** dialog box reappears. 136. 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. 137. 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.

Finite Element Example (Continued)

The radiation calculation includes a sink temperature of 2.73 K by default, explaining the cooler temperatures.

Node

	>373. 1
	373. 1
	362. 8
	352. 4
	342
	331. 7
	321. 3
	310. 9
	300. 5
	290. 2
	279. 8
	269. 4
	<269. 4

Temperature [K], Time = 0 s, Steady State, Loopct=3/1000

Finite Element Example (Continued)

138. Select **File > Exit**. A **Thermal Desktop/AutoCAD** dialog box appears asking if the user wants to save the changes made to the drawing. 139. 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.