1.3 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:

- 1.1 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.

Circuit Board Example	
1. Copy the template `thermal.dwg` file created in the first tutorial to the `\Tutorials\Thermal Desktop - legacy\board` 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.	
5.  or **Thermal** > **Thermophysical Properties** > **Edit Property Data**.	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.

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.
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.

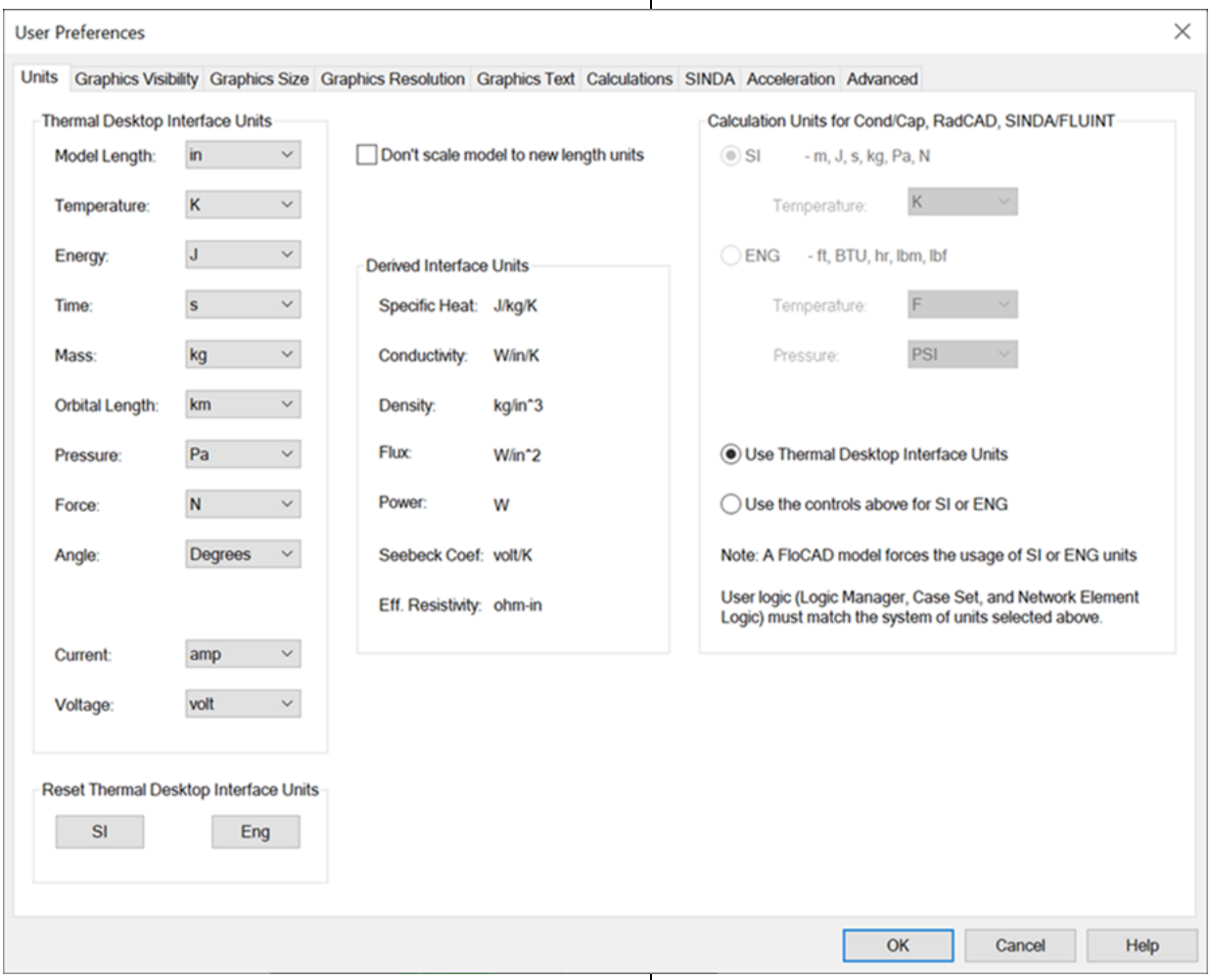
Circuit Board Example (Continued)

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: `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.	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.
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**	If OK is selected too soon, double-click the rectangle to re-enter the edit dialog.
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. 39. Highlight the current value in the **Thickness** field and type **0.05** 40. Select **OK** to close the **Thin Shell Data** dialog box.	Generate Nodes And Conductors must be selected to change the material and the thickness. Set the material as Aluminum with a thickness of 0.05 in.

Circuit Board Example (Continued)

41.  or **View** > **Zoom** > **Extents**.

At the end of these steps, the screen should look similar to the view below:

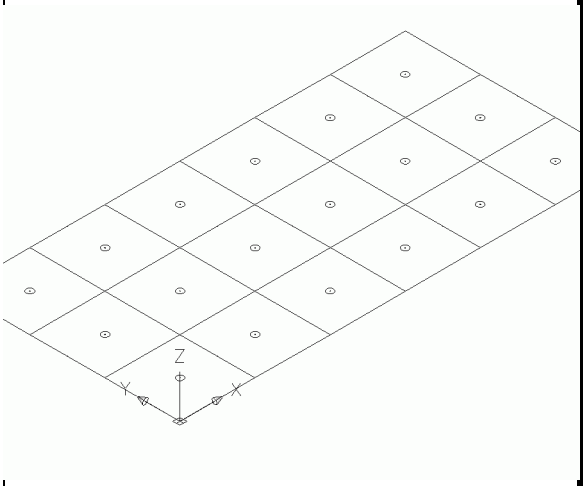


Figure 1-1: Aluminum Plate

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

The Command line area should show:

```
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>:
```

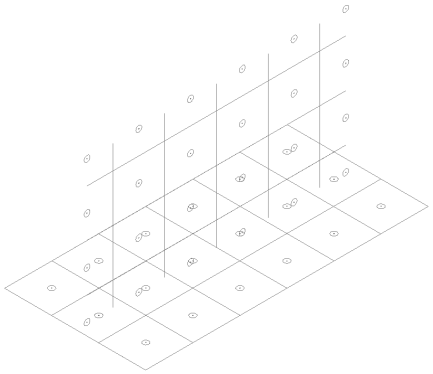
45. Type **@0,0,3**

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.

Circuit Board Example (Continued)	
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 sub-model, 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.
55. Click on the arrow next to the **Sub-model** 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 1-2: 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. **Thermal** > **Preferences**.

    The **User Preferences** dialog box appears.

61. Select the **Graphics Visibility** tab.

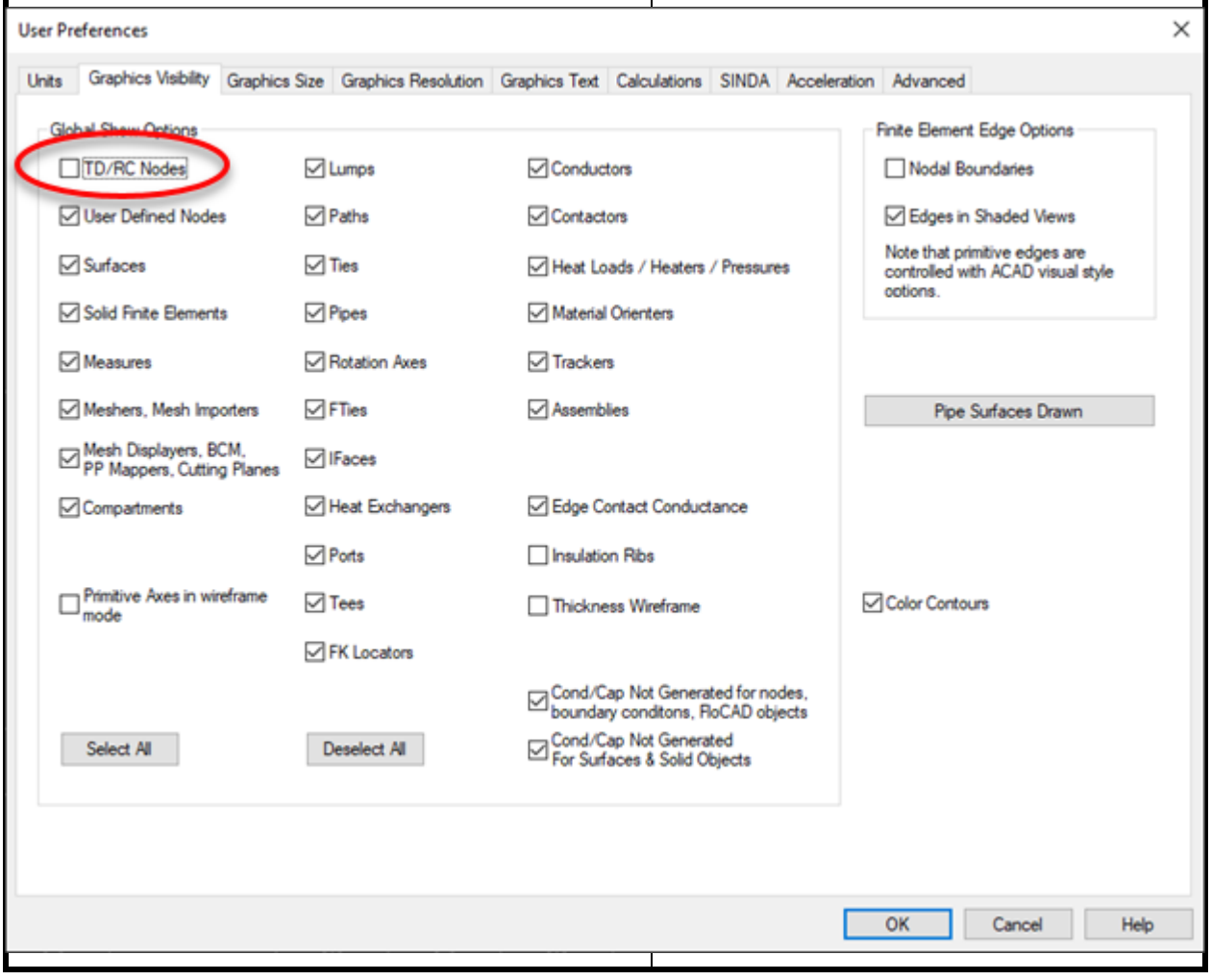
62. Clear **TD/RC Nodes**.

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

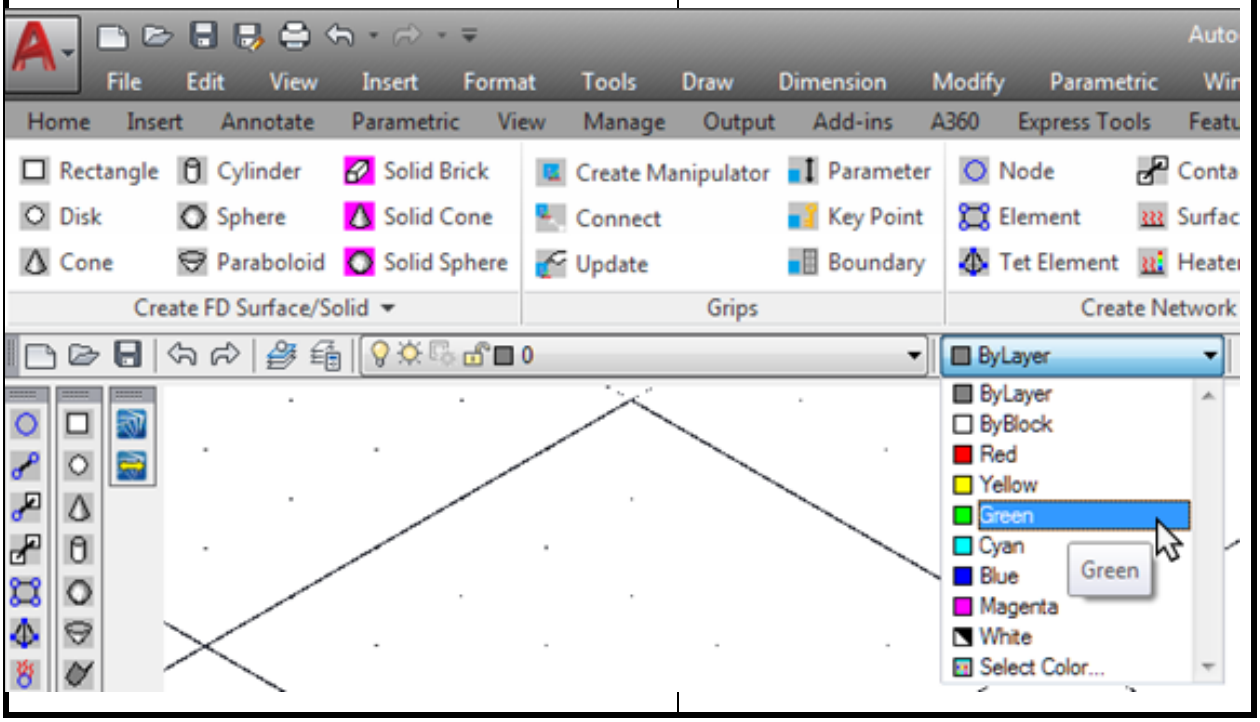
64. The nodes are no longer displayed.

The nodes (small circles at the center of each subdivision) are turned off to reduce visual clutter. They can be turned on when graphically selecting nodes is required.

The node visibility can be toggled on and off using the User Preferences window displayed below or by clicking this icon in the Ribbon or Toolbars.



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: `RcRectangle Origin point <0,0,0>:`	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 within the drawing area to access a menu for different types of snap points: select Node to snap to the center of a surface.
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.	
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.	If CHIP is not in the drop-down list, then type it in.
76. Highlight the current value in the **Thickness** field and type **0.1**	
77. Select **OK** to close the dialog box.	

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.

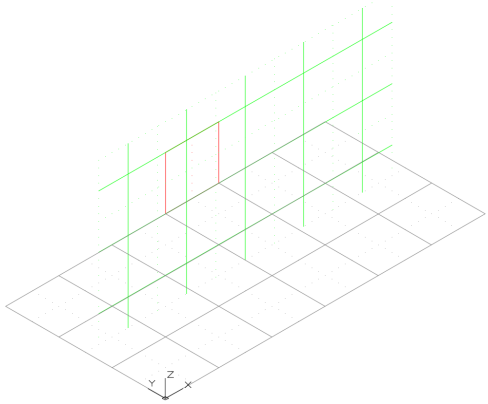


Figure 1-3: Chip on Circuit Board

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 or edge domains contacting from or [MB]:` 85. Select the green circuit board in the drawing area. The Command line should now read: `Select faces or edge domains contacting from or [MB]:` 86. Press <**Enter**>. The Command line should now read: `Select surfaces, solids, or nodes contacting to or [MB]:` 87. Select the aluminum plate (the horizontal surface in the drawing area). The Command line should now read: `Select surfaces, solids, or nodes 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]** in the `From (1):` list box.
93. In the **Select Edges** dialog, clear all checks except for **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 (circled in red) 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.

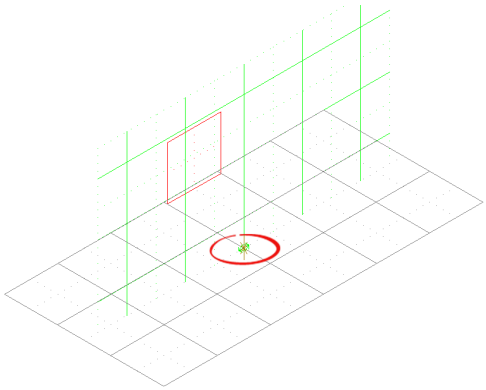
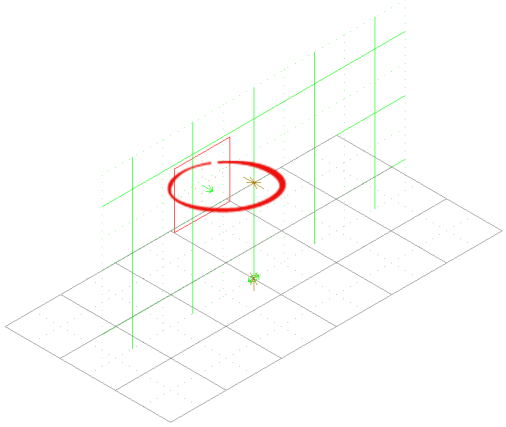


Figure 1-4: Edge Contactor

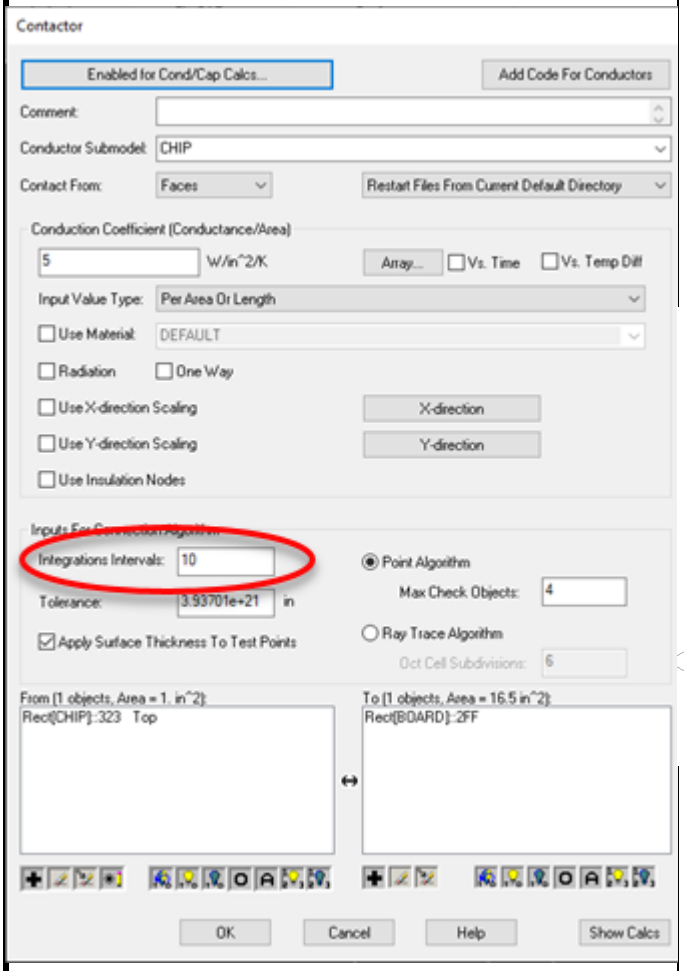
The edge for the contactor is determined by the creation of the BOARD surface: the X at Y=0 edge is the first edge specified after the origin.

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 (circled in red) 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 1-5: 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.

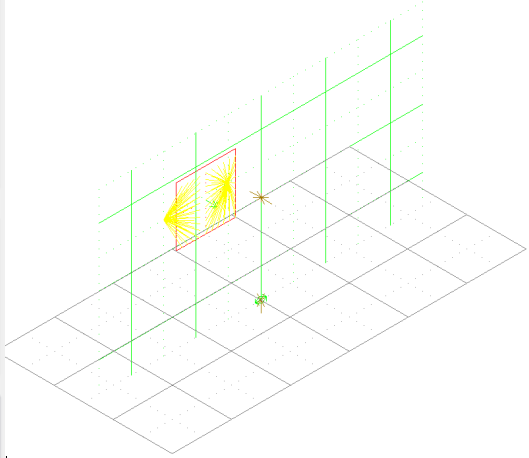
Circuit Board Example (Continued)

Yellow lines representing the contacts are displayed.



If the contactors dialog is closed without using Show Calcs, the contactor markers 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, used when creating the contactor, as the Integration Intervals. A value of 10 integration intervals means the 10*10, or 100, points will be tested for a face contactor. In this model, notice that the chip is connected to 2 nodes.

If a test point does not connect to a node, then it is displayed as a red +. To prevent test point from connecting, 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.