1.5 Simple Satellite

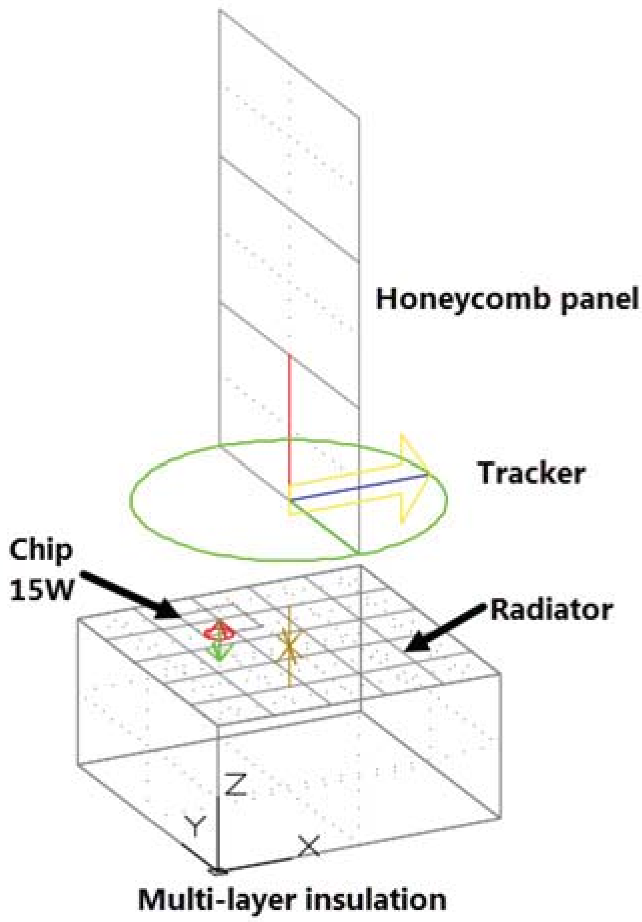
What will be learned:

- Using trackers
- Using insulation
- Creating double sided Thermal Desktop surfaces

Before Starting:

- Create a template

In this example the simple satellite shown below will be built. This satellite will have two radiation analysis groups, one for internal of the box and one for the external. The five lower surfaces of the box will be coated with insulation. Conduction within the box is going to be ignored.



Simple Satellite Demo Example

Start by opening the folder titled simpleSatellite. Copy the template DWG file into the demo-Satellite folder.

1. Open the folder named simpleSatellite. (create it if it does not exist).
2. Copy the template thermal.dwg file created in the first tutorial to the new `..\Tutorials\RadCAD\simpleSatellite` 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.

3. Rename the copied template file to simpleSatellite.
4. Start Thermal Desktop by double clicking on the simpleSatellite drawing file icon in the simpleSatellite directory.

5.  or **Thermal** > **Thermophysical Properties** > **Edit Property Data.**

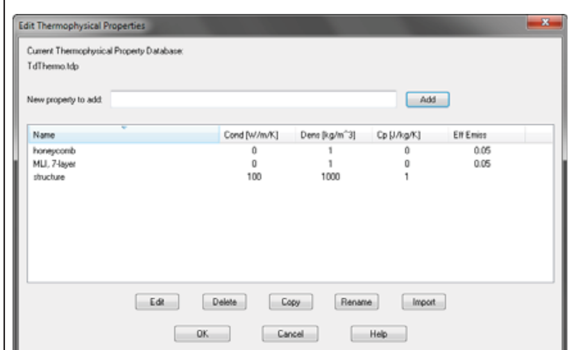
   The Edit Thermophysical Properties dialog box appears. Your properties list may or may not be empty.
6. Type structure in the New property to add field.
7. Select Add.

   The Thermophysical Properties dialog box appears.
8. Highlight the value in the Conductivity field and enter 100.
9. Highlight the value in the Density field and enter 1000.
10. Select OK to close the dialog box.

    The Edit Thermophysical Properties dialog box reappears with 'structure' displayed in the list field. Notice that the conductivity is 100, specific heat is 1 and density is 1000.

Add three new materials:

- structure: k = 100; Cp = 1; ρ = 1000
- MLI, 7-layer: k=Cp=ρ=0; ε* = 0.05
- honeycomb: k=20; Cp=ρ=0; ε*=0.05



Simple Satellite Demo Example	
11. Type MLI, 7-layer in the New property to add field. 12. Select Add. The Thermophysical Properties dialog box appears. 13. Highlight the current value in the Conductivity field and type 0. 14. Highlight the current value in the Specific Heat field and type 0. 15. Highlight the current value in the Effective Emmissivity e-star field and type 0.05. 16. Select OK. The Edit Thermophysical Properties dialog box reappears displaying MLI.	The MLI material has a conductivity of zero and an effective emissivity of 0.05. This means that when used as insulation, or as a core material for surfaces with different node numbers on each side, the conductor through the material will be radiation. If the conductivity were greater than zero and the effective emissivity were zero, the conductors through the material would all be linear. If both values were greater than zero, both radiation and linear conductors would be generated through the material. The internal properties of the insulation are given in the thermophysical properties and the surface properties are given in the optical properties.
17. Select MLI, 7-layer in the property list and select the Copy button. 18. Type honeycomb in the Copy Material Property form that appears. 19. Select OK. 20. Double-click Honeycomb. The Thermophysical Properties dialog box appears. 21. Highlight the current value in the Conductivity field and type 20. 22. Select OK. The Edit Thermophysical Properties dialog box reappears displaying MLI. 23. Select OK.	The honeycomb material has a conductivity of 20 and an effective emissivity of 0.05. This means that when used as insulation, or as a core material for surfaces with different node numbers on each side, two conductors will be created through the material: one linear and one radiation.

Simple Satellite Demo Example

24.  or **Thermal** > **Optical Properties** > **Edit Property Data.**

    The Edit Optical Properties dialog box appears. Your properties list may or may not be empty.

25. Type MLI surface in the New property to add field.

26. Select Add.

    The Thermophysical Properties dialog box appears.

27. Set Solar Absorptivity to 0.15

28. Set Infrared Emissivity to 0.05

29. Select OK.

Define optical properties:

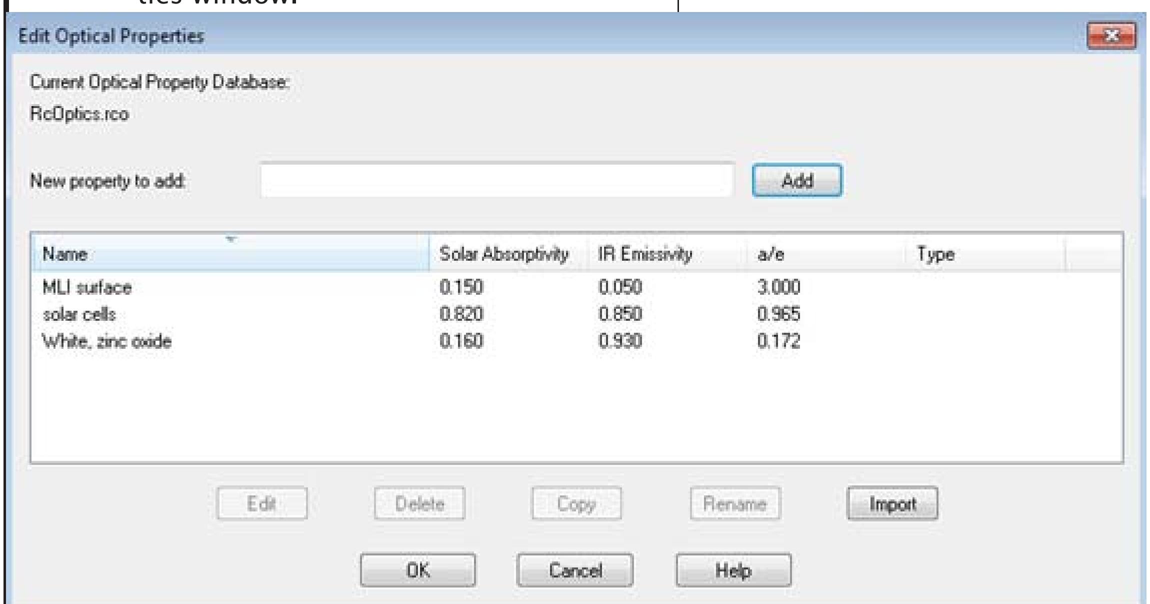
- MLI surface: α = 0.15; ε = 0.05
- White, zinc oxide: α = 0.16; ε = 0.93
- solar cells: α = 0.82; ε = 0.85

Any surfaces without an assigned property will have the DEFAULT property which is α = 1 and ε = 1

The surface properties are given in the optical properties and the internal properties, including internal effective emissivity, are given in the thermophysical properties.

30. Repeat for 'white, zinc oxide' and 'solar cells' properties.

31. Select OK. to close Edit Optical Properties window.



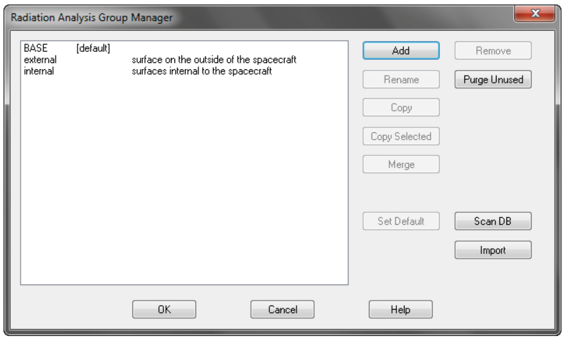
32. Select **View** > **3D Views** > **SE Isometric.**

    The UCS icon reflects the change.

Simple Satellite Demo Example

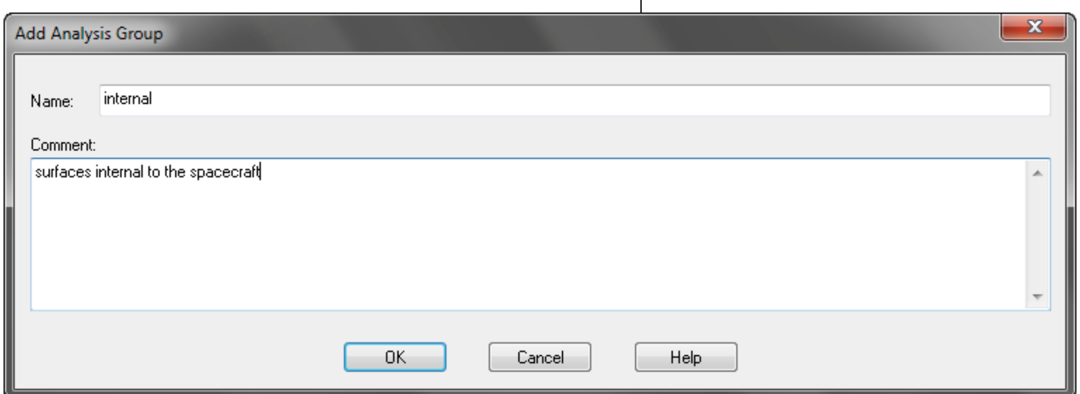
33. Select **Thermal** > **Radiation Analysis Groups.**

    The Radiation Analysis Group Manager dialog box appears.



34. Select Add.

    The Add Analysis Group dialog box appears.



35. Type Internal in the New radiation group name field.

36. Select OK to close the Add Analysis Group dialog box.

    The Radiation Analysis Group Manager dialog box reappears with Internal displayed in the Analysis Group list field.

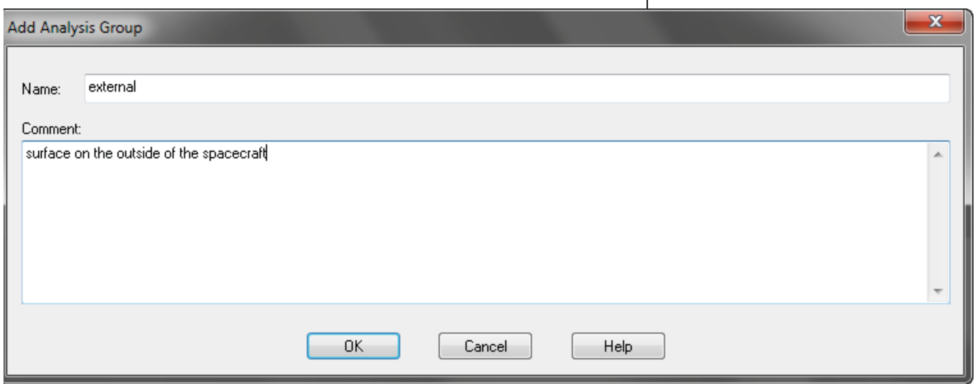
Radiation analysis groups designate isolated regions of the model for radiation calculation purposes. Two analysis groups are created for this model:

- Internal
- External

The default analysis group Base can remain in the model and will not affect the calculations if it is not used for any calculations.

Simple Satellite Demo Example

37. Select Add.



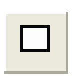
The Add Analysis Group dialog box appears.

38. Type External in the New radiation group name field.

39. Select OK to close the Add Analysis Group dialog box.

The Radiation Analysis Group Manager dialog box reappears with the 2 new groups displayed in the Analysis Group list field.

40. Select OK to the close the dialog box.

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

The Command line should now read:

`Origin point <0,0,0>`

42. Press <Enter>.

The Command line should now read:

```
Point for +X axis and X-size
<@1,0,0>: 2,0,0
```

43. Type 2,0,0 in the Command line.

The Command line should now read:

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

44. Type 0,0,1 in the Command line.

The Thin Shell Data dialog box appears.

45. Click on the Radiation tab.

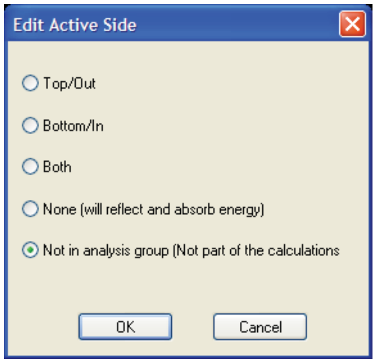
46. Double click on External in the Analysis Group Name, Active Side field.

Create the box shown in the inital image of this tutorial by creating a series of six rectangles.

Set the top side to be in the External Analysis Group and the bottom side to be in the internal group. Make sure to generate nodes and conductors and put insulation on the top side.

Simple Satellite Demo Example

The Edit Active Side dialog box appears.



47. Select Top/Out.
48. Select OK.

    The Thin Shell Data dialog box reappears with top/out displayed next to External.
49. Double click on Internal in the Analysis Group Name, Active Side field.

    The Edit Active Side dialog box appears.
50. Select Bottom/In.
51. Select OK.

    The Thin Shell Data dialog box reappears with top/out displayed next to External.
52. Using the drop-down list for Top/Out under Optical Properties for Radiation Calculation, select MLI surface.

The surface was created such that the top (the +Z of the surface) is on the outside of the spacecraft.

Simple Satellite Demo Example

53. Click on the Cond/Cap tab.

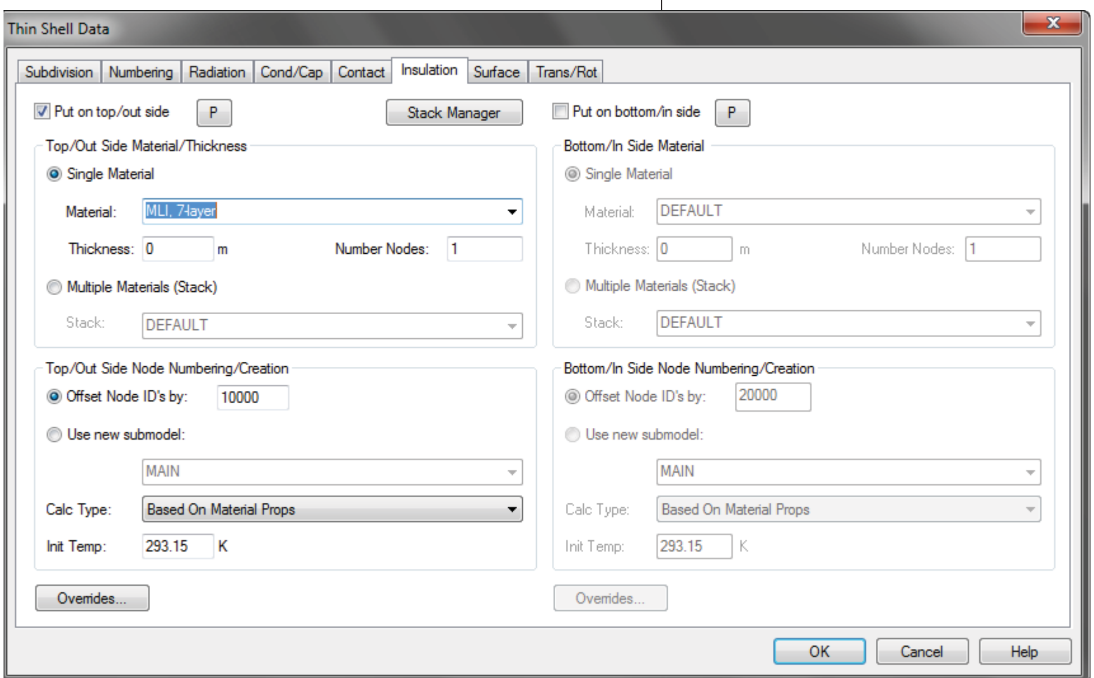
    Generate Nodes and Conductors is already set by default.

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

55. Select the Insulation tab.

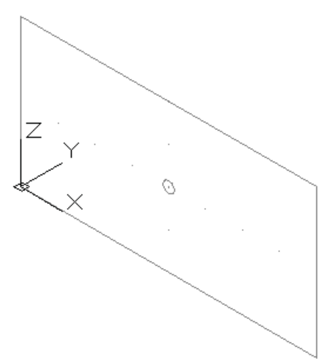
56. Click in the check box next to Put on top/out side to select it.

    The Top/Out field activates.



57. Click on the arrow next to the Material field and select MLI, 7-layer from the drop-down list.

58. Select OK to close the dialog box.

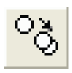


Simple Satellite Demo Example

Create the top and the other sides of the box by copying the first rectangle. Do this so that the top side is always out.

One option is to begin by copying the first rectangle and rotating it 90 degrees to create a new side. The rotation can be completed using the grip point Aim X Rotating About Y.

A second, possibly easier, option is to use the `ARRAY` or `ARRAYCLASSIC` command to create a polar array.

59. Select the newly created rectangle.
60.  (Copy).

    The Command line should now read:

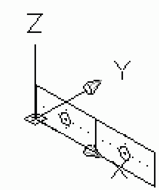
    Specify base point or [Displacement/mOde] <Displacement>:
61. Click at the origin to set the first point.

    The Command line should now read:

    Specify base point or [Displacement/mOde] <Displacement>: Specify second point or <use first point as displacement>:
62. Click on bottom right corner of the existing rectangle (as currently oriented in the drawing area), on the X axis.

    A second rectangle appears adjacent to the first.
63. Click on the second rectangle to select it.

The steps to create the first of the 5 remaining sides is shown to the left.



Simple Satellite Demo Example

64. Type ROTATE3D.

    The Command line should now read:

    `Specify first point on axis or define axis by [Object/Last/View/Xaxis/Yaxis/Zaxis/2points]:`

65. Click on the point at the base of the line separating the two rectangles.

    The Command line should now read:

    `[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: Specify second point on axis:`

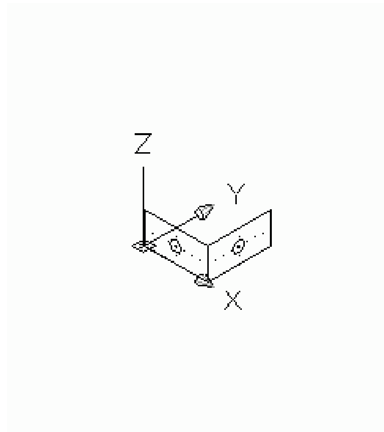
66. Click on the point at the top of the line separating the two rectangles.

    The Command line should now read:

    `Specify rotation angle or [Reference]:`

67. Type 90 in the Command line.

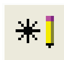
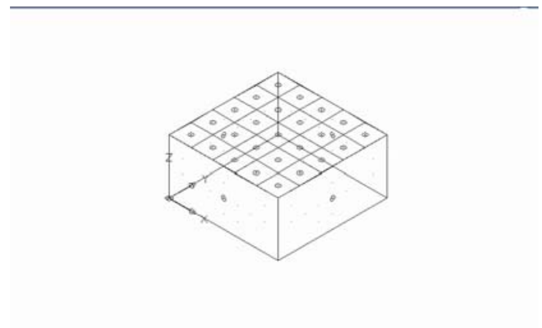
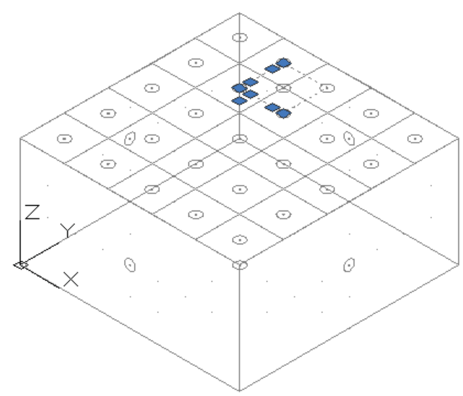
    The second rectangle now displays as a side of the box.



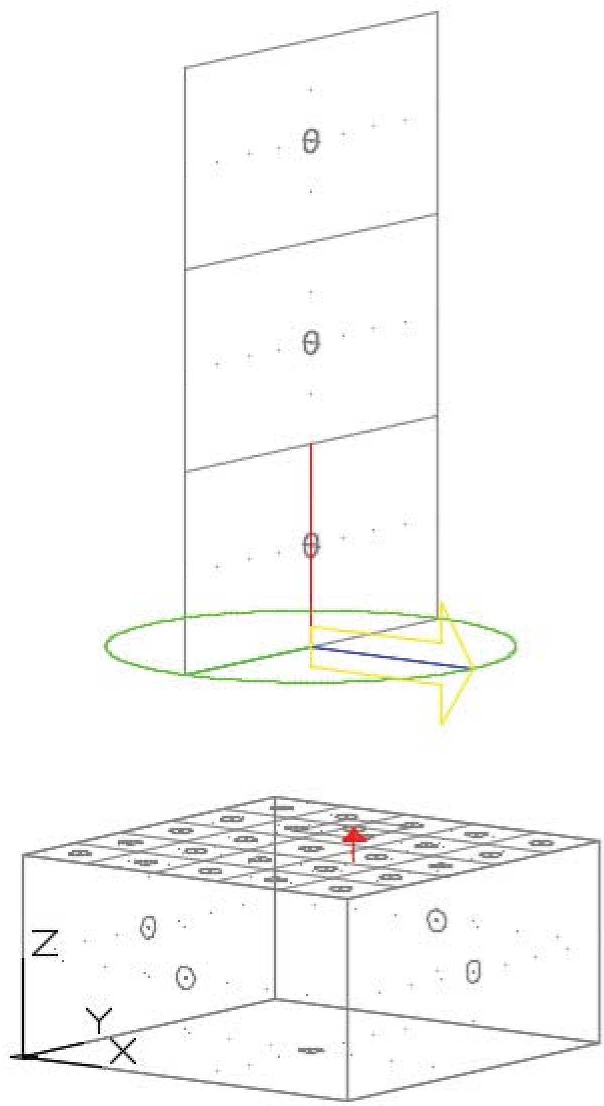
68. Repeat the process to create the other 2 sides of the box and the top and the bottom of the box.

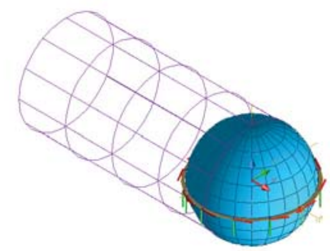
    The top and bottom of the box use the same concepts of copying and rotating. Pay attention to the angles and the axes for the top and bottom. The sides of the box are smaller than the top and the bottom so they will need to be stretched to fit—use the grip points. It is also possible to use the copy and move commands, but remember to keep the top sides out.

    It is also fine to zoom in for a closer view and rotate the view.

Simple Satellite Demo Example	
69. Click on the new top side of the box to select it. 70.  or **Thermal** > **Edit.** The Thin Shell Data dialog box appears with the Insulation tab displayed. 71. Clear Put on top/out side. 72. Select the Subdivision tab. 73. Highlight the current value in the X-direction Equal field and type 5. 74. Highlight the current value in the Y-direction Equal field and type 5. 75. Leave Centered Nodes selected. 76. Select the Radiation tab. 77. Select white, zinc oxide from the Top/Out drop-down list of the Optical Properties for Radiation Calculations. 78. Select OK to close the dialog box.	The top side of the box is the radiator. Change the top side of the box to have a 5x5 breakdown and also to take the insulation off of it. 
79. Create a rectangle to represent the chip using the snap points on the radiator to place the chip as shown in figure to the right. Orient the chip such that the +Z (top) of the chip is facing the inside of the spacecraft. The Thin Shell Data dialog box appears as a part of the creation process. 80. Place the Node ID in the submodel **CHIP.** 81. Set the radiation for the top side in the Internal analysis group. 82. Place the conductors in the CHIP submodel. 83. Set the material to structure. 84. Close the Thin Shell Data form	Create the CHIP. Use the snap points to put it directly in the same plane as the radiator.. 
85. Select the chip. 86. Select **Modify** > **Move.** 87. Follow the prompts and move the chip down in the Z direction 0.01.	Offset the chip from the radiator for radiation calculations. Move the chip into the box. **Note**: Hint – Select a corner of the chip as a base and use @0,0,-.01 as the “to” point.

Simple Satellite Demo Example	
88. Create a contactor from the chip to the radiator with a value of 5 W/m^2K and place in the CHIP submodel.	
89. Create a heat load with power of 15 W on the chip surface and place in the CHIP submodel.	
90. Create a double sided solar panel. **Note**: Refer to this image on the next page for a visual reference. Pan and zoom as needed.	Create a solar panel and a tracker. For the solar panel: ■ Place the surface origin at 1,0,2 ■ Make the rectangle 2 m x 3 m ■ Make it a double sided surface and set initial node ID for bottom to 11 **Note**: Hint – Go to Numbering tab and uncheck **Use Same ID's on both sides** ■ Place nodes and conductors in the SOLAR_PANEL submodel ■ Set the separation distance to be 0.01. ■ Use the structure material for the faces and honeycomb for the separation. ■ Set the radiation analysis External group to "Both". ■ Use the solar cells optical property for the Top and white, zinc oxide for the bottom. ■ Subdivide 3 in the y and 1 in the x.

Simple Satellite Demo Example	
91. Create the tracker.	Create the tracker and modify it to be oriented as shown in the below graphic.
	
92. Attach the solar panel to the tracker.	Attaching geometry to a tracker can be accomplished by either using the **Thermal** > **Articulators** > **Attach Geometry** command or by using drag-and-drop in the Tracker list of the Model Browser.

Simple Satellite Demo Example	
93. Create a basic orbit with default properties	
94. Display the vehicle in the orbit to ensure the solar panel orientation makes sense. 95. Edit the orbit to change the orientation (rotation about X)	You may need to open the radiation analysis group manager to set BASE or EXTERNAL to the default group
96. Use the display active sides and the Model Browser to make sure the model is correct.	
97. Display the Case Set Manager. 98. Set up and run the case.	■ Edit the radiation jobs to calculate ■ radks for the internal analysis group ■ articulating radks for the external analysis group (the geometry is changing over the orbit) ■ heating rates for the external group. ■ Set for a steady state solution followed by a transient run of 15000 seconds. ■ Set the output increment to 100 seconds. ■ Run 1 Selected Case (allow node IDs to be automatically resequenced)
99. Select **File** > **Exit**. A Thermal Desktop/AutoCAD dialog box appears asking to save the drawing changes. 100. Select Yes.	Exit Thermal Desktop and save as prompted.