

---

---

## 22 FloCAD® Tutorials

---

---

This set of tutorials is focused on the FloCad application. Five example tutorials follow which show how to model air flow, heat pipes, and flow through a manifold.

- Section 22.1: Air Flow Through an Enclosure on page 22-3
- Section 22.2: Heat Pipe Model on page 22-23
- Section 22.3: Manifolded Coldplate on page 22-37
- Section 22.4: Drawn Shape Heat Pipe on page 22-85
- Section 22.5: FEM Walled Pipe on page 22-99

At this point, the first three tutorials from Chapter 20: "Setting Up a Template Drawing" should have been completed. These tutorials give an overview of Thermal Desktop and how to create geometry, as well as define properties. The typographical conventions are defined there as well.



## 22.1 Air Flow Through an Enclosure

What will be learned:

- How to create a fluid model
- How to create all types of fluid elements

In this example, the components necessary to model airflow through an enclosure with electronic components will be added. The exercise will start using a model of an enclosure for which the geometry has already been created. The enclosure has a flat base 30x30 cm. Three boards extend up from the base, with each containing a chip producing 25 watts of heat. There is a set of conductors connected to the base plate to simulate convection to an ambient temperature. Two flow paths through the enclosure are created. The temperature distribution in the enclosure will be computed for a given flow rate.

*Want to Learn More? Refer to Chapter 5: "Fluid Models" in the Thermal Desktop User's Manual for detailed information on fluid models.*

*Want to Learn More?*

### Air Flow Example

1. Double click on the file **fluid.dwg** located in the **Tutorials\Thermal Desktop\air-flow** folder.

Thermal Desktop opens with the fluid drawing on the screen.

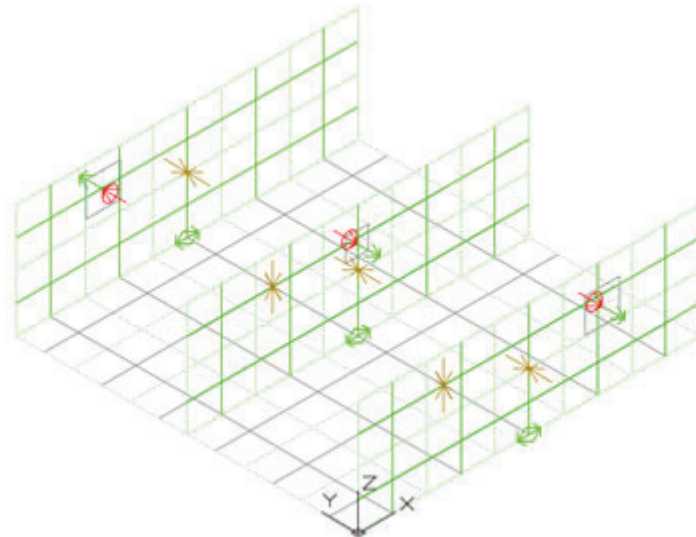



Figure 22-1 Fluid Drawing Initial View

2. Type **Zoom** and then **0.2**

## Air Flow Example (Continued)

3.  or **Thermal > Fluid Modeling > Lump**.

The Command line now reads:

Enter location of lump:

4. Type **-20,15,5** in the Command line. (*<ENTER> is implied when something is typed into the Command line*)

The Command line now reads:

Type a Command:

5. Press **<Enter>**.

*Note: <Enter> is used to repeat the last command and create a second lump for the outlet of the fan.*

The Command line now reads:

Enter location of lump:

6. Type **-10,15,5** in the Command line.

The Command line now reads:

Command:

Create two lumps on one side of the circuit boards. One lump will be changed to a Plenum later in the tutorial.

[--] [SW Isometric] [2D Wireframe]

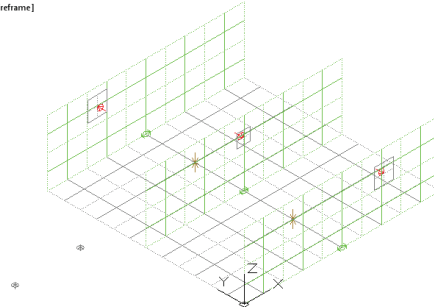



Figure 22-2 After Step 6

7.  or **Thermal > Fluid Modeling > SetFlow**.

The Command line now reads:

Select from lump:

8. Click on the **1st lump**.

The Command line now reads:

Select to lump:

9. Click on the **2nd lump**.

Create a fan between the two lumps that were just created.

The default direction of positive flow in all paths will be in the direction in which the lumps are selected.

The fan will be modified shortly to set the flow rate.

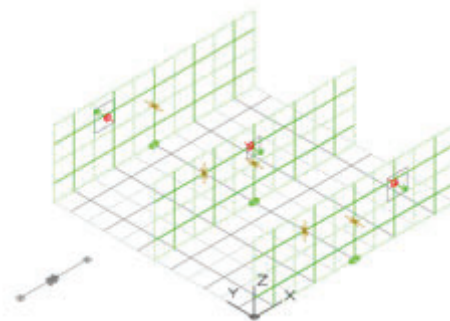
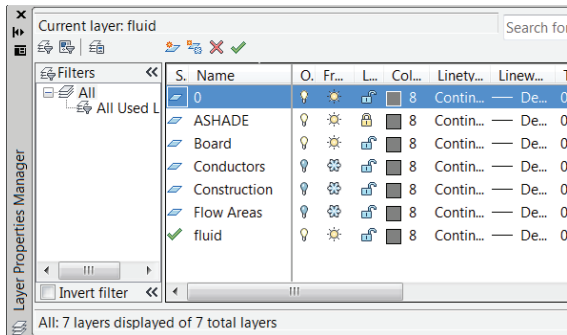


Figure 22-3 After Step 9

## Air Flow Example (Continued)

10.  or type **Layer** in the Command line or select **Format > Layer**.

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



11. Click on the **On (lightbulb)** and the **Freeze (snowflake)** icons of the **Construction** layer to turn the layer **on** (lighten the lightbulb and change the snowflake to a sun).
12. Click on the **On (lightbulb)** and the **Freeze (snowflake)** icons of the **Flow Areas** layer to turn the layer **on** (lighten the lightbulb and change the snowflake to a sun).
13. Click on the **Freeze (sun)** icon of the **Board** layer to turn the layer **off** (snowflake).
14. Close the Layer Property Manager.

Turn on and off some layers to make it easier to select the points to generate the lumps and paths within the enclosure.

Notice that Fluid is already the current layer (green check mark), which is where all of the fluid submodel components will be placed.

The geometry was already created for convenience, but can easily be created with AutoCAD Draw commands.

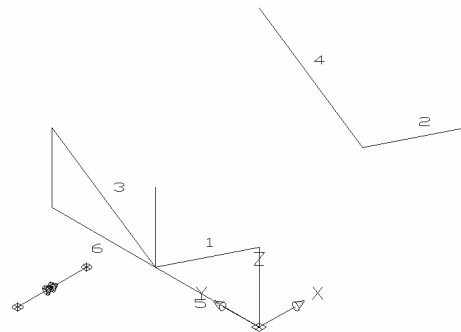
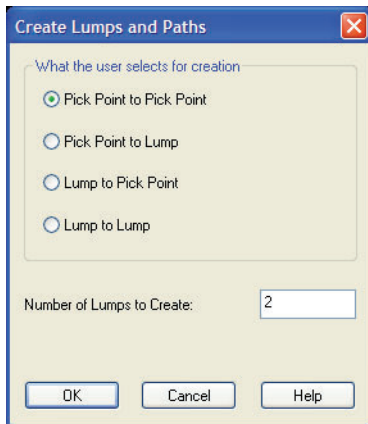


Figure 22-4 After Visibility Changes

## Air Flow Example (Continued)

15.  or **Thermal > Fluid Modeling > Lumps and Paths.**

The **Create Lumps and Paths** dialog box appears.



16. Click on the **Pick Point to Pick Point** radio button if not already selected (place dot in the circle).
17. Highlight the current value in the **Number of Lumps to Create** field and type **5**.
18. Select **OK**.

The Command line now reads:

Select start point:

19. Click at the **midpoint of line 1**.

*Note: A midpoint snap point (a triangle) should appear as the cursor moves midway on line 1.*

The Command line now reads:

Select end point:

20. Click at the **midpoint of line 2**.

The Command line now reads:

Select Upstream Entity for  
Area Calculation (Enter for  
User Specified Area):

21. Click on a **point on line 5**.

Create all of the lumps and paths within each side of the enclosure with one command.

Notice that the shape of line 5 follows along the edge of the boards for this flow channel. This provides the wetted perimeter for the flow area. The shape is closed to get the area by assuming there is a line between the two endpoints. The code then computes the hydraulic diameter and flow area for all four paths from this shape.

This shape was created with a polyline.

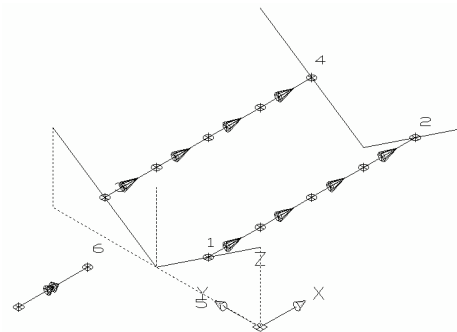


Figure 22-5 New Lumps and Paths after Step 23

## Air Flow Example (Continued)

The Command line now reads:

Select Downstream Entity for  
Area Calculation (Enter for  
Same Entity):

22. Press **<Enter>** to reuse the same .

23. **Repeat the command** picking the  
midpoints of lines 3 and 4, and line 6  
for the shape.

- Press **<Enter>** to recall the  
command.

The **Create Lumps and Paths**  
dialog box appears.

- Leave *Pick Point* to *Pick Point*  
selected and 5 as the number of  
lumps to create.
- Select **OK** to close the dialog  
box.

The Command line now reads:

Select start point:

- Click at the **midpoint of line**  
**3.**

The Command line now reads:

Select end point:

- Click at the **midpoint of line**  
**4.**

The Command line now reads:

Select Upstream Entity for  
Area Calculation (Enter for  
User Specified Area):

- Click on a **point of line 6.**

The Command line now reads:

Select Downstream Entity for  
Area Calculation (Enter for  
Same Entity):

- Press **<Enter>**.

Enter or right-click will repeat the  
previous command.

## Air Flow Example (Continued)

24.  or type **Layer** in the Command line or select **Format > Layer**.

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

25. Click on the **Freeze (sun)** icon of the **Construction** layer to turn the layer off (display a snowflake).
26. Click on the **Freeze (sun)** icon of the **Flow Areas** layer to turn the layer off (display a snowflake).
27. Close the **Layer Properties Manager**.

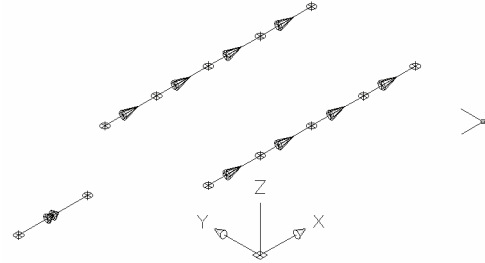



Figure 22-6 After Step 27

28.  or **Thermal > Fluid Modeling > Lump**.

The Command line now reads:

Enter location of lump:

29. Type **50,15,5** in the Command line.

Create the lump for the outlet plenum.

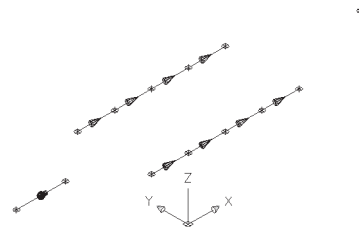


Figure 22-7 After Step 29



## Air Flow Example (Continued)

30.  or **Thermal > Modeling Tools > Toggle Selection Filter.**

In the Command area you should see:

```
RcFilter Thermal Desktop  
filter turned on
```

31.  or **Thermal > Modeling Tools > Turn Numbers On.**

The Command line now reads:

```
Select entity(s) to display  
ids or [MB GRP]:
```

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

The Command line now reads:

```
Select entity(s) to display  
ids or [MB GRP]:
```

33. Press **<Enter>**.

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

34. Highlight **Lumps[13]** in the **Select type to filter** field if not already selected.

35. Select **OK**.

To view the lump numbers for selecting them in the following steps, set the selection filter “on”. Make sure that the Command line shows “on” after selecting the toggle command.

Then use the selection filter to turn on the IDs for the lumps.

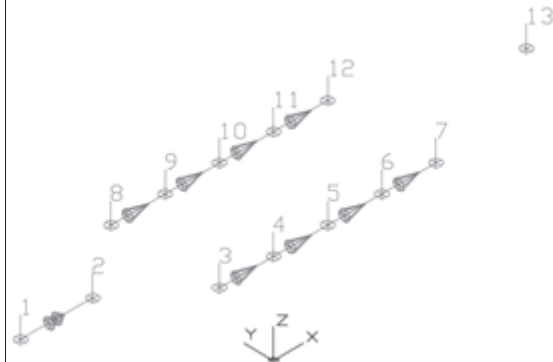
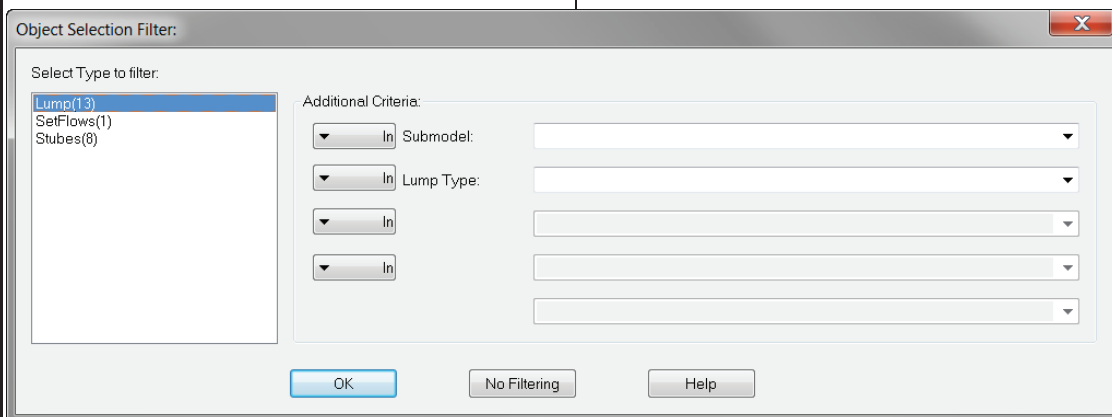



Figure 22-8 Lumps with IDs

*Note: If you had to delete and recreate some lumps during this tutorial, then your numbering will be slightly different. The remaining tutorial will refer to the numbers as previously shown.*



## Air Flow Example (Continued)

36.  or **Thermal > Fluid Modeling > Loss.**

The Command line now reads:

Select from lump:

37. Click on **lump 2**.

*Note: Click on the lump numbers rather than on the lump symbols for easier selection.*

The Command line now reads:

Select to lump:

38. Click on **lump 3**.

The Command line now reads:

Select Entity for Area Calculation (Enter for User Specified Area):

39. Press **<Enter>**.

40. Use the same command to create losses from each of: Lump 2 to Lump 8; Lump 7 to Lump 13; and, Lump 12 to Lump 13.

- Press **<Enter>**.

The Command line now reads:

Select from lump:

- Click on **lump 2**.

The Command line now reads:

Select to lump:

- Click on **lump 8**.

The Command line now reads:

Select Entity for Area Calculation (Enter for User Specified Area):

- Press **<Enter>**.

Lumps 2 and 8 are connected.

Now generate all of the entrance and exit paths. Use the loss coefficient to model the entrance and exit effects.

The losses will be edited in a later step.

As with the Pump/Fan command, **the order the lumps are selected determines the direction for positive flow.**

Note that picking on the lump numbers with the mouse is probably the easiest way to select each lump.

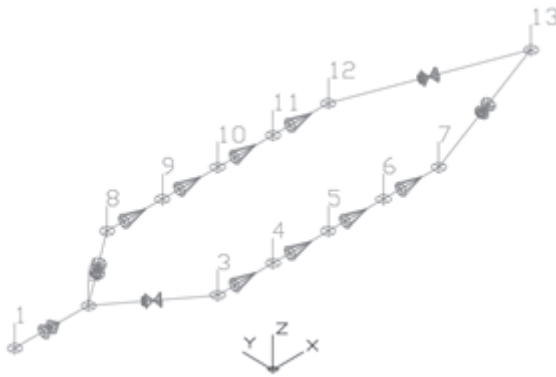


Figure 22-9 View after Step 40

### Air Flow Example (Continued)

- *Press <Enter>.*

The Command line now reads:

Select from lump:

- *Click on **lump 7**.*

The Command line now reads:

Select to lump:

- *Click on **lump 13**.*

The Command line now reads:

Select Entity for Area Calculation (Enter for User Specified Area):

- *Press <Enter>.*

Lumps 7 and 13 are connected.

- *Press <Enter>.*

The Command line now reads:

Select from lump:

- *Click on **lump 12**.*

The Command line now reads:

Select to lump:

- *Click on **lump 13**.*

The Command line now reads:

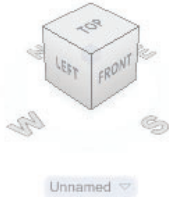
Select Entity for Area Calculation (Enter for User Specified Area):

- *Press <Enter>.*

Lumps 12 and 13 are connected.

## Air Flow Example (Continued)

41. Select **View > 3D Views> Top** or click **TOP** on the View Cube



View Cube

Simplify the view for selecting.

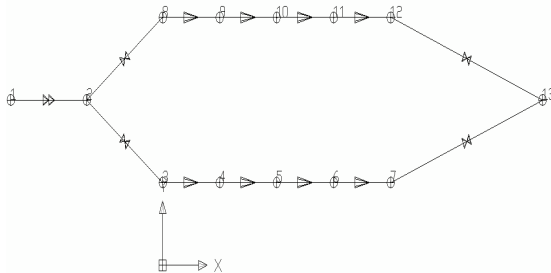


Figure 22-10 Top View

42. Select the **SetFlow** (the fan created in Step 6 from Lump 1 to Lump 2) with the mouse.



43. or **Thermal > Edit**.

The **SetFlow** edit form dialog box appears.

44. Make sure the **SetFlow Data** tab is selected. Highlight the current value in the **Mass Flow Rate** field and type **0.1**.

45. Select **OK**.

The SetFlow looks like this:



Here the object (the SetFlow) is selected before the command is issued. In this order, the command acts on the item(s) that have already been selected.

Assume the flow rate for this fan is known and select a mass flow rate device.

## Air Flow Example (Continued)

46.  or **Thermal > Edit**.

The Command line now reads:

Select objects or [Indiv MB  
GRP] :

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

The Command line now reads:

Select objects or [Indiv MB  
GRP] :

48. Press **<Enter>**.

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

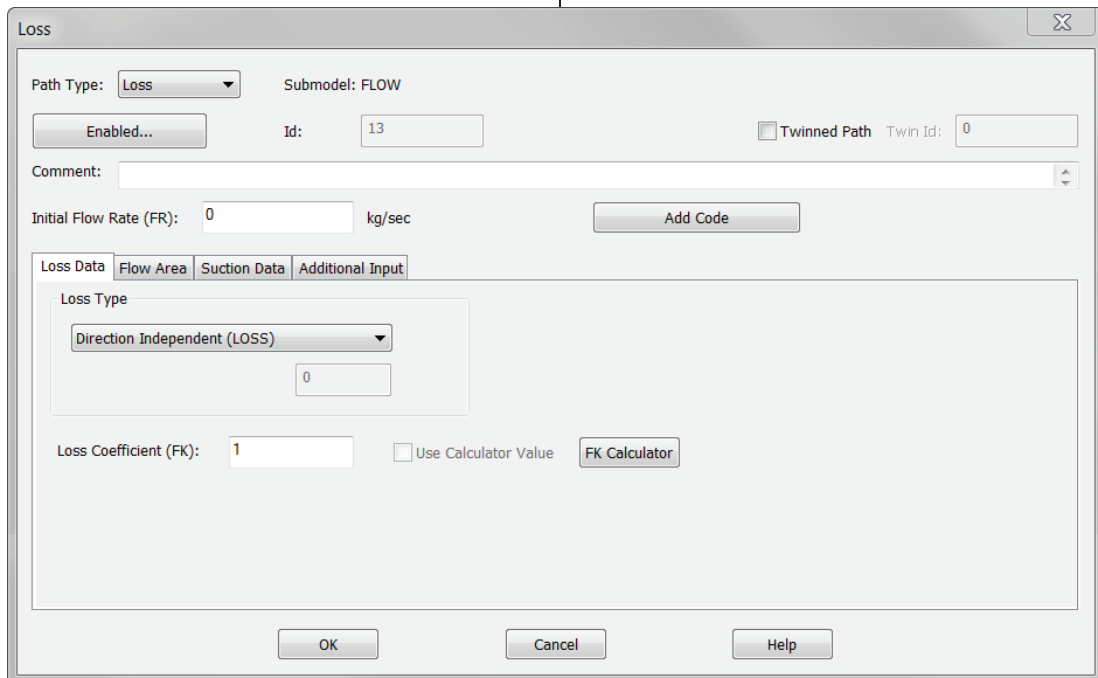
49. Select **Losses[4]** in the **Select type to filter** field.

50. Select **OK**.

The **Loss** edit form dialog box appears.

Here the command (Edit) is issued before anything is selected. In this order, the command requests that objects be selected.

Use 100 cm<sup>2</sup> for all of the inlet and outlet flow areas. Because they are all the same type of path, they can be selected using the Object Selection Filter. Also leave the FK value at 1.0 for an entrance and exit loss.



Loss

Path Type: **Loss** Submodel: FLOW

Enabled... Id: 13 ☐ Twinned Path Twin Id: 0

Comment:

Initial Flow Rate (FR): 0 kg/sec Add Code



Loss Data Flow Area Suction Data Additional Input

Loss Type


Direction Independent (LOSS) 0

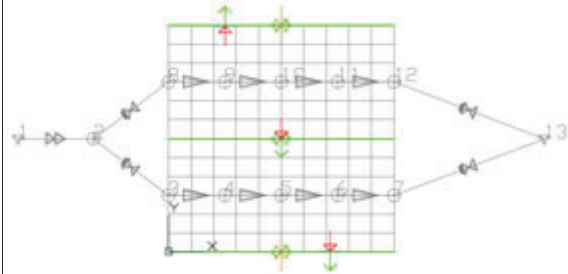
Loss Coefficient (FK): 1 ☐ Use Calculator Value FK Calculator

OK Cancel Help


Air Flow Example (Continued)	
<p>51. Select the <b>Flow Area Tab</b>. In the <b>Flow Area (AF)</b> field, type <b>100.0</b> in the box.</p> <p>52. Select <b>OK</b>.</p> <p>A <b>Multi Edit Dialog</b> box appears confirming the change.</p> <p>53. Select <b>Apply Changes</b>.</p>	
<p>54.  or <b>Thermal &gt; Modeling Tools &gt; Toggle Selection Filter</b>.</p> <p>In the Command area you should see:</p> <pre>_RcFilter Thermal Desktop filter turned off</pre>	<p>Turn off the selection filter.</p> <p>Make sure the Command line output states the filter is ‘turned off’.</p>
<p>55. Select lumps <b>1</b> and <b>13</b>.</p> <p><i>Note: Click on the lump numbers to insure only the lumps are selected.</i></p> <p>56.  or <b>Thermal &gt; Edit</b>.</p> <p>The <b>Lump Edit Form</b> dialog box appears.</p> <p>57. Click the <b>Plenum (Boundary - Infinite Volume)</b> radio button to select it (place a dot in the circle).</p> <p>58. Select <b>OK</b> to close the dialog box.</p> <p>A <b>Multi Edit Dialog</b> box appears confirming the change.</p> <p>59. Select <b>Apply Changes</b>.</p>	<p>The lumps created by default are junctions. For the model to run, the source and sink lumps must be plena. Lumps 1 and 13 can be changed together by selecting them together and changing their type in the <b>Lump</b> edit form dialog box.</p> <p>Notice that the shape of the lumps changes from a circle to a triangle. Thermal components share a shape with the fluid submodel components, but the fluid shapes also have interior lines and a vertical line normal to the plane.</p>

## Air Flow Example (Continued)

60.  or type **Layer** in the Command line.  
The **Layer Properties Manager** dialog box appears.
61. Click on the **Freeze (sun)** icon of the **Board** layer to turn the layer on (display a sun).
62. Close the **Layer Properties Manager**.



## Air Flow Example (Continued)

63.  *or* **Thermal > Fluid Modeling > Tie to Surface.**

The Command line now reads:

Select the lumps to be tied to or [MB GRP]:

64. **Select lumps 3 to 7** by using a left-to-right selection box as shown to the right with the heavy line.

The Command line now reads:

Select the lumps to be tied to or [MB GRP]:

65. Press **<Enter>**.

The Command line now reads:

Select the surfaces for the tie or [MB GRP]:

66. Type **GRP** in the Command line.

The **Select Groups** dialog opens

67. **Select RIGHT** from the Select Groups list and click **OK**.

In the Command area you should see:

Added 4 members 4 found  
Select the surfaces for the tie or [MB GRP]:

68. Press **<Enter>**.

The Command line now reads:

Select the paths to be tied to (None for user specified HTC) or [MB GRP]:

69. **Select paths between lumps 3 and 7** as done previously for selecting the lumps.

The Command line now reads:

Select the paths to be tied to (None for user specified HTC) or [MB GRP]:

70. Press **<Enter>**.

Create the thermal connection between the fluid submodel and the thermal submodel using a “tie”. The lumps and path sets are both selected using a left to right box (thick line) shown below.

Note that the heat transfer coefficient will be computed by SINDA/FLUINT since there is a selected a path set.

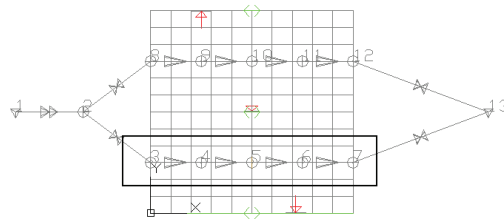


Figure 22-11 Selecting lumps or paths

As an aid in selection, the AutoCAD groups named ‘left’ and ‘right’ have been pre-created in the DWG file by the authors. Groups can be created by using the ‘groups’ command in AutoCAD.

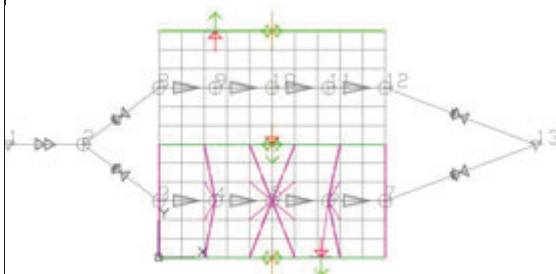



Figure 22-12 View after Step 70



## Air Flow Example (Continued)

71.  *or* **Thermal > Fluid Modeling > Tie to Surface.**

The Command line now reads:

Select the lumps to be tied to or [MB GRP]:

72. *Select lumps 8 to 12 by using a left-to-right selection box.*

The Command line now reads:

Select the lumps to be tied to or [MB GRP]:

73. *Press <Enter>.*

The Command line now reads:

Select the surfaces for the tie or [MB GRP]:

74. *Type **GRP** in the Command line.*

The **Select Groups** window opens.

75. *Select **LEFT** in the Select Groups list and click **OK**.*

In the Command area you should see:

Added 5 members 5 found  
Select the surfaces for the tie or [MB GRP]:

76. *Press <Enter>.*

The Command line now reads:

Select the paths to be tied to (None for user specified HTC) or [MB GRP]:

77. *Select **paths between lumps 8 and 12** by using a left-to-right selection box.*

The Command line now reads:

Select the paths to be tied to (None for user specified HTC) or [MB GRP]:

78. *Press <Enter>.*

The steps from the previous page are repeated for the left group

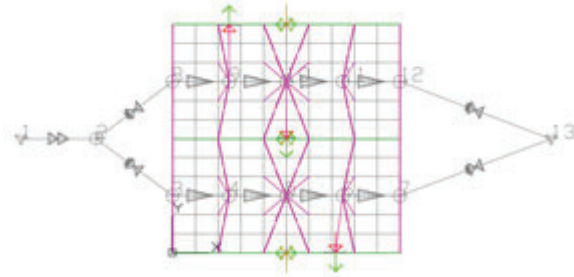
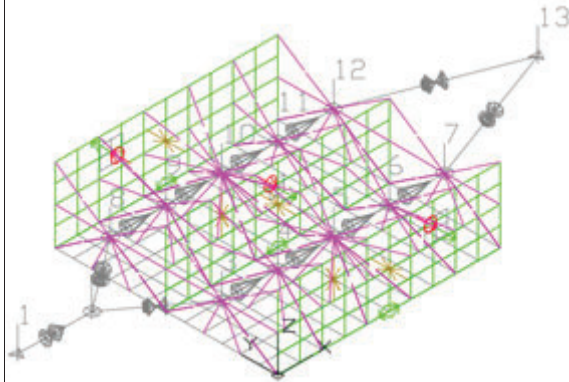


Figure 22-13 View after Step 78

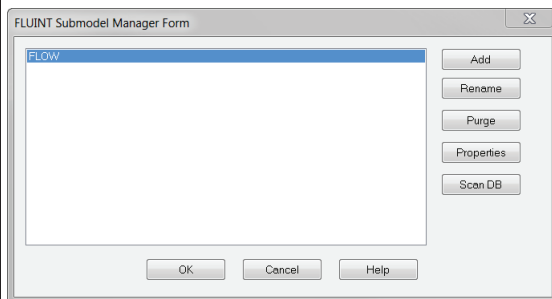
## Air Flow Example (Continued)

79. Select **View > 3D Views> SW Iso-metric**.



80. Select **Thermal > Fluid Modeling > Submodel Manager**.

The **FLUINT Submodel Manager Form** dialog box appears.

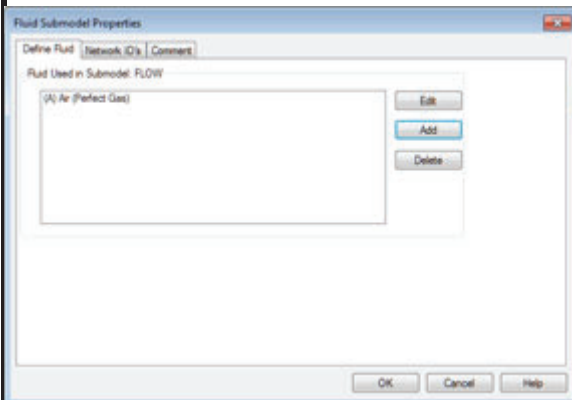


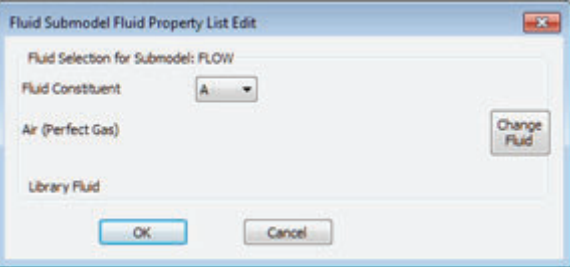
The FLUINT Submodel Manager is where the fluid is set. The default is Air, so there is nothing to do for this model.

In addition to the library fluids, the user could define their own fluid.

81. With **FLOW** selected, select the **Properties** button.

The **Fluid Submodel Properties** dialog box appears.



Air Flow Example (Continued)	
<p>82. Select <b>(A) Air (Perfect Gas)</b> (if not already selected).</p> <p>83. Select the <b>Edit</b> button.</p> <p>The <b>Fluid Submodel Fluid Property List Edit</b> dialog box appears.</p> 	
<p>84. Select <b>OK</b> to close the <b>Fluid Submodel Fluid Property List Edit</b> dialog box.</p> <p>85. Select <b>OK</b> to close the <b>Fluid Submodel Properties</b> dialog box.</p> <p>86. Select <b>OK</b> to close the <b>FLUINT Submodel Manager Form</b> dialog box.</p>	
<p>87. Select <b>File &gt; Save</b>.</p>	

90. Click **OK** to close the dialog box.

After the solve is completed, the final temperatures are displayed on the model in the post-processing state. You may need to select SW Isometric view.

## Air Flow Example (Continued)

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

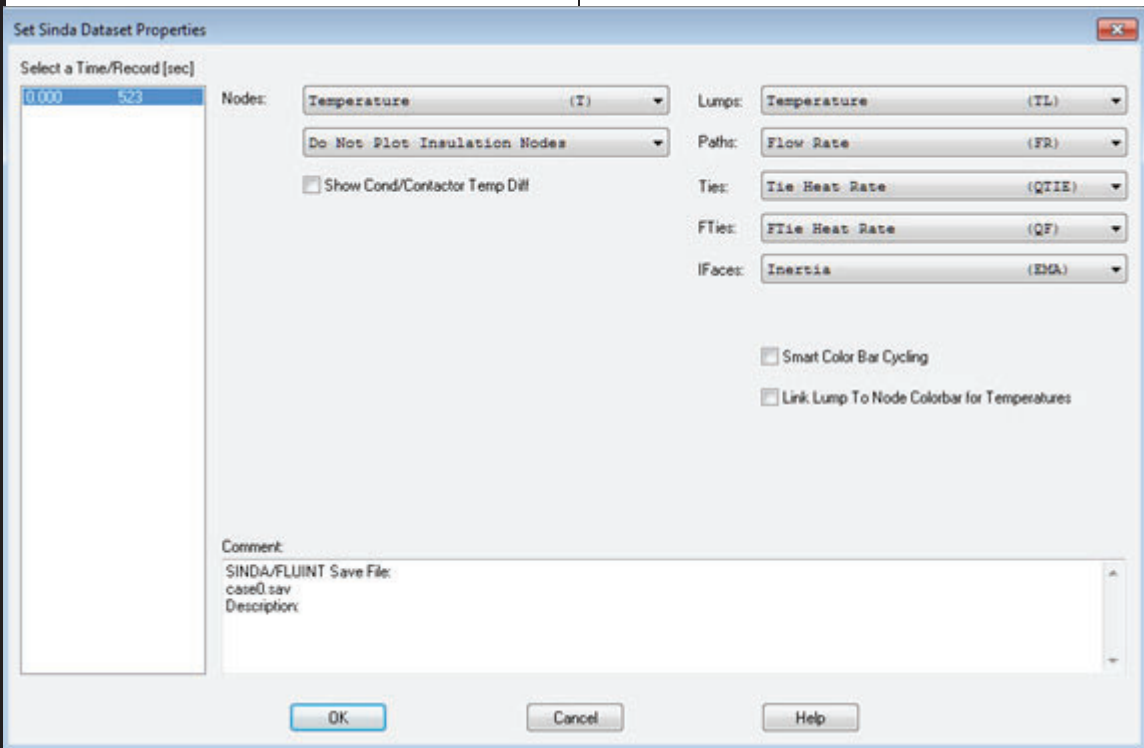
The **Set Sinda Dataset Properties** dialog box appears.


92. *Select the drop-down menu beside each item to see the options available.*

93. Click **OK** to close the dialog box.

The **Set Sinda Dataset Properties** window allows the user to choose the Time or Record to postprocess and which data to view for different objects.

Since this run was steady state and the output was written only after the solution was reached, only one record is available.





94.  or **Thermal > Post Processing > Cycle Color Bars.**

95. *Repeat Cycle Color Bars several times noting the change to the color bar and its labels.*

The Cycle Color Bars command can be used to cycle between the four types of color bars (Node, Lump, Path, and Tie). The color scale for each as well as the current variable being displayed can be found on the each of the individual color bars.

To make viewing easier, objects not associated with the current color bar are drawn as gray. Smart Color Bar Cycling, applied next, hides all other objects.

Air Flow Example (Continued)	
<p>96.  <i>or</i> <b>Thermal &gt; Post Processing &gt; Edit Current Dataset.</b></p> <p>The <b>Set Sinda Dataset Properties</b> dialog box appears.</p> <p>97. <i>Select the <b>Smart Color Bar Cycling</b> checkbox (so a check mark is displayed).</i></p> <p>98. <i>Click <b>OK</b> to close the dialog box.</i></p>	<p>Smart Color Bar Cycling turns off the visibility of all objects except those associated with the active color bar.</p>
<p>99.  <i>or</i> <b>Thermal &gt; Post Processing &gt; Cycle Color Bars.</b></p> <p>100. <i>Repeat <b>Cycle Color Bars</b> several times noting the change to the display.</i></p>	<p>With Smart Color Bar Cycling selected, the visibility of objects changes along with the color bar such that nodes and surfaces are shown with the Node color bar, lumps are shown with the Lump color bar, etc.</p>
<p>101. <i>Select <b>File &gt; Exit.</b></i></p> <p>A <b>Thermal Desktop/AutoCAD</b> dialog box appears asking to save the drawing changes.</p> <p>102. <i>Select <b>Yes.</b></i></p>	<p>Exit Thermal Desktop and save as prompted.</p>

As time permits, advanced users may consider making the following changes to the model:

- Change the inlet plenum or the SetFlow as a function of time (sine wave function or array)
- Run the model as a transient solution.