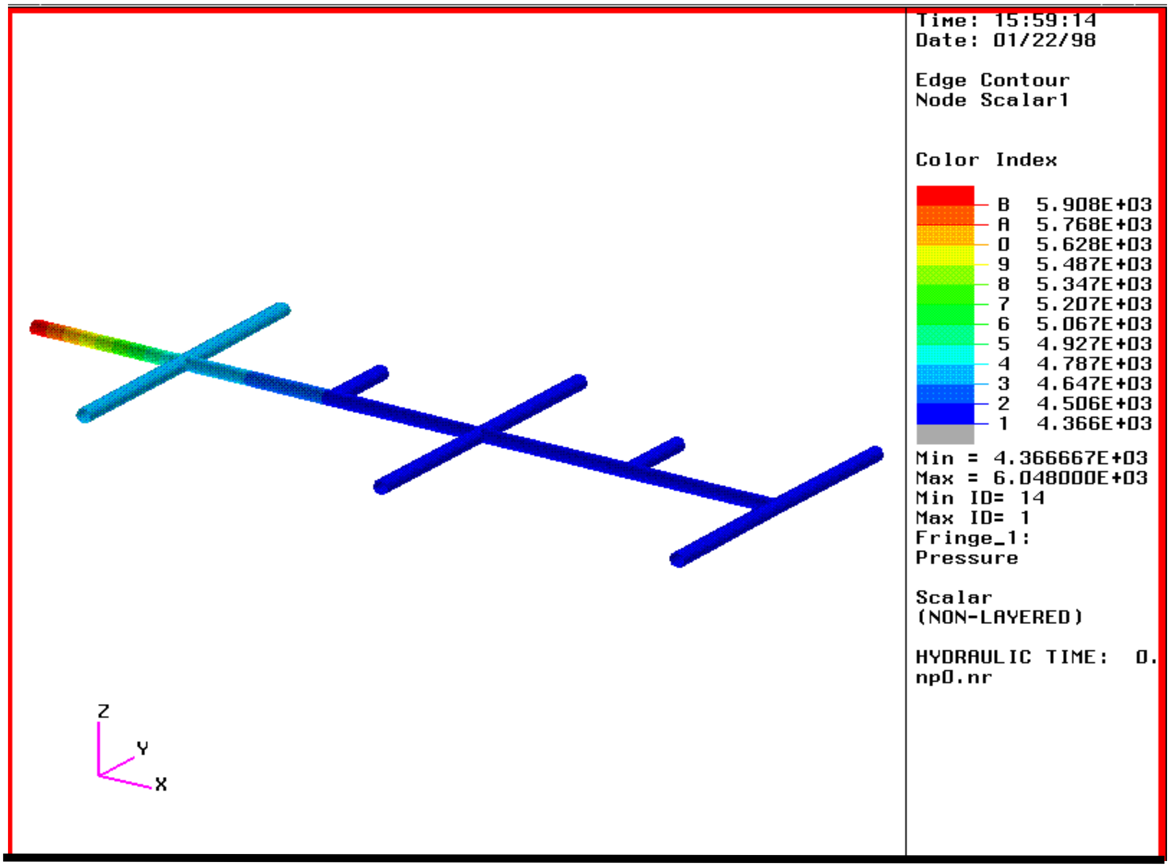
Exercise 13

A Sprinkler System Hydraulic Analysis



Objective:

- Model a schematic of a home sprinkler system.
- Use microfunctions to apply pressure varying mass flow functions at the sprinkler heads.
- Run a hydraulic analysis to evaluate the pressure drop and total mass flow through the system.

Model Description:

In this exercise you will create schematic geometry which models a sprinkler system for a medium size lawn. The model is a schematic since the actual pipe lengths in the circuit will be defined via the Element Properties form. All fitting losses have been included as additions to pipe lengths.

The home for which this sprinkler is designed can comfortably deliver 12 gallons per minute (GPM) of water at 42 psi through the existing main. Since both the volumetric flow through and the coverage from each sprinkler head are a function of pressure at the head, this analysis will determine whether the pressure at each head is above 30 psi and whether the entry volumetric flow demand is less than 12 GPM.

All data provided yield an analysis in English Engineering units, lbf, lbm, s, feet. However here are some useful conversion factors for evaluating the results: 1 cu. ft. = 7.481 gal. and 1 sq. ft. = 144 sq. in.

Exercise Overview:

- Create a new database named **exercise_13.db.** Set *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**.
- Create flow network schematic geometry using the **Geometry** form **Create** and **Transform** Actions.
- Create FEM entities. Create nodes matching geometric points and mesh curve with **BAR2** elements.
- Equivalence nodes.
- Define element properties for **1D Flow network bar** elements using *IOPT*=**2** for automatic friction factor calculation.
- Use **Utilities/Thermal/Hydraulic Icon** to check flow direction.
- Create three **NonSpatial/General** fields which define the sprinkler head volumetric flow as a function of pressure.
- Define inlet pressure and sprinkler head mass flow conditions.
- Complete boundary condition in **template.dat.apnd** file.
- Select **Analysis** to prepare and to submit the model for analysis and to **Read Results**.
- Run the analysis and read the results into the database.
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_13.db**.

Within your window environment change directories to a convenient working directory. Run MSC/PATRAN by typing **p3** in your xterm window.

Next, select **File** from the *Menu Bar* and select **New…** from the drop-down menu. Assign the name exercise_13.db to the new database by clicking in the *New Database Name* box and entering **exercise_13**.

Select **OK** to create the new database.

File

New

New Database Name **exercise_13**

OK

MSC/PATRAN will open a Viewport and change various *Main Form* selections from a ghosted appearance to a bold format. When the New Model Preferences form appears on your screen, set the *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences form.

Tolerance ◆ **Default**

Analysis Code **MSC/THERMAL**

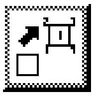
OK

Create network schematic geometry

2. Create flow network schematic geometry using the Geometry form **Create** and **Transform** Actions.

Select the **Geometry** *Applications radio button*. Create points using the following *Action*, *Object*, and *Method*. Click in the appropriate list boxes to edit the default values and change them to values listed below.

First, turn on the labels with the *Show labels* icon.



◆ **Geometry**

Create/Point/XYZ

Point List **[0 6 0]**

Apply

Point List	**[6 0 0]**

Apply

Point List	**[6 12 0]**

Apply

Now translate points using the following *Action*, *Object*, and *Method*. Click in the appropriate list boxes to edit the default values and change them to values listed below.

Create curves using the following *Action*, *Object*, and *Method*. Click in the

☐ **Auto Execute**

Transform/Point/Translate

Translation Vector	**<6 0 0>**
Repeat Count	**5**
Point List	**Point 1**

Apply

Translation Vector	**<12 0 0>**
Repeat Count	**2**
Point List	**Point 2, 3**

Apply

Translation Vector	**<0 3 0>**
Repeat Count	**1**
Point List	**Point 5, 7**

Apply

appropriate list boxes to edit the default values and change them to values listed below.

Create/Curve/Point

Starting Point LIst	**Point 1**
Ending Point LIst	**Point 4**

Apply

Repeat this create curves step for the following starting/ending point list.

Table 1:

Starting Point	Ending Point	Resulting Curve
4	5	2
5	6	3
6	7	4
7	8	5
4	2	6
4	3	7
5	13	8
6	9	9
6	10	10
7	14	11
8	11	12
8	12	13

Create nodes and elements

3. Create FEM entities. Create nodes matching geometric points and mesh curve with **BAR2** elements.

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Method* to **Create/Node/Edit**. Select all of the geometry for inclusion in the *Node Location List* to duplicate the node number over the corresponding geometric point number.

◆ Finite Elements

Create/Node/Edit

Node Location LIst

<drag a rectangle around all geometry in the viewport>

Set the *Action*, *Object*, and *Type* to **Create/Mesh/Curve**. Change the *Global Edge Length* to **12.0** and select all curves for inclusion in the *Curve List*.

Create/Mesh/Curve

Global Edge Length	**12.0**
Curve List	**<select all curves in the viewport>**

Apply

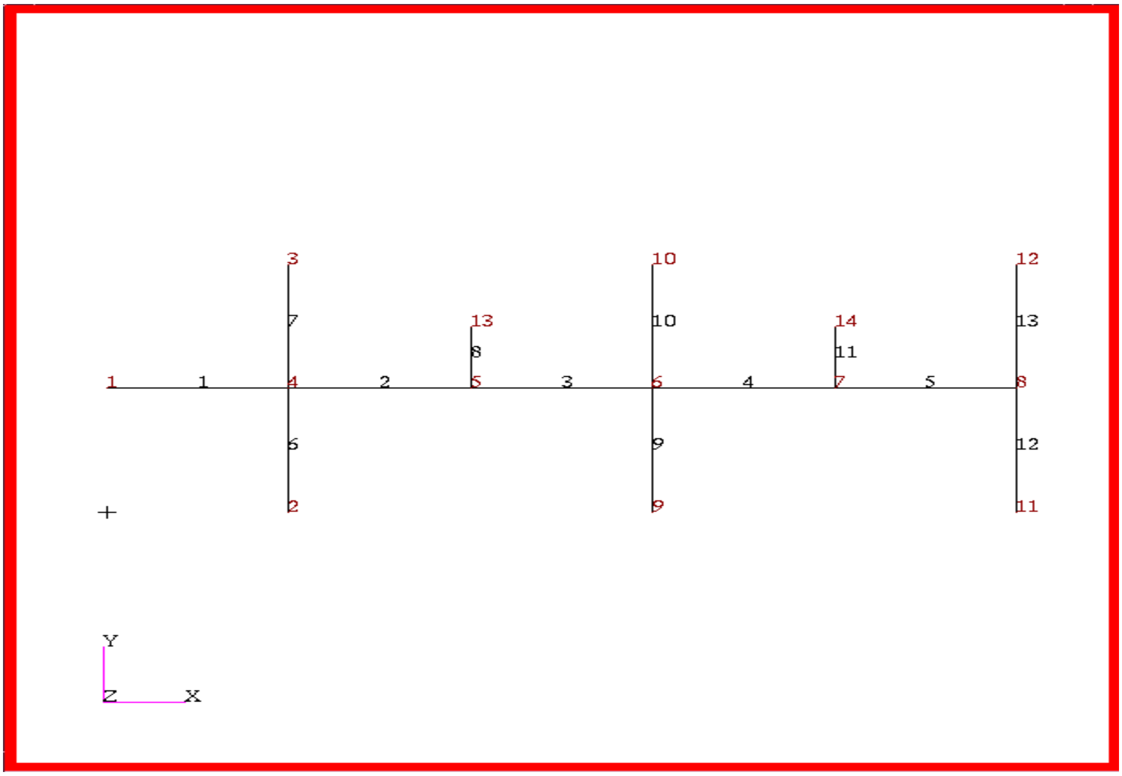
4. Equivalence nodes.

Equivalence Nodes

Select the **Finite Elements** *Applications radio button* if not already selected. Set the *Action*, *Object*, and *Method* to **Equivalence/All/Tolerance Cube**. Select apply to complete the function.

The nodes bounding the interior cracks will be circled in the display and the *Command Line* will indicate that a number of nodes are deleted.

The model should now appear as shown below.



Create micro-functions

5. Define element properties for 1D Flow network bar elements using *IOPT* **2** for automatic friction factor calculation.

Select the **Properties** *Applications radio button*. Set the *Action*, *Dimension*, and *Type* to **Create/1D/Flow network bar**. Enter *Property Set Name* **Entry**. Select the *Input Properties...* box. In the Input Properties chart, follow the steps below and enter the values that correspond to the property name. Select **OK** to close the form.Click in the *Select Members* box and select Curve 1 in the viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties	
Create/1D/Flow network bar	
Property Set Name	**Entry**
Input Properties...	
TID	**1**
IOPT	**2**
Pipe diameter	**0.0625**
Pipe length	**50.00**
Pipe roughness	**5.0e-6**
Fluid density	**62.4**
Fluid viscosity	**1.5e-3**
OK	
Select Members	**<select Curve 1>**
Add	
Apply	

Repeat the above steps with the following property set names and change the property values that are listed, the rest should stay untouched. To select more than one curve in the *Select Members* box, hold down the *<Shift> key* and select the curves.

Set Name	Pipe length	Application Region
L_manifold	13.0	Curve 2, 5
L_riser	26.0	Curve 6, 7, 9, 10, 12, 13
S_manifold	7.0	Curve 3, 4
S_riser	8.0	Curve 8, 11

Scroll through the *Existing Property Sets* box to make sure there are five property sets.

Verify Flow Directions

6. Use **Utilities Thermal Hydraulic Icons** to verify flow directions.

Utilities
Thermal
Hydraulic Icons...

If a disclaimer message appears, click: **OK**

Apply
Clear
Close

Create Fields

7. Create 3 **NonSpatial/General** fields which define the sprinkler head volumetric flow as a function of pressure.

In the Fields form use the *Action/Object/Method* **Create/Non Spatial/ General**. Enter **Full-GPM** in the *Field Name* box. After selecting **Input Data** the General Field Input Data form will show the complete list of microfunctions in the *Select Function Term* list box.

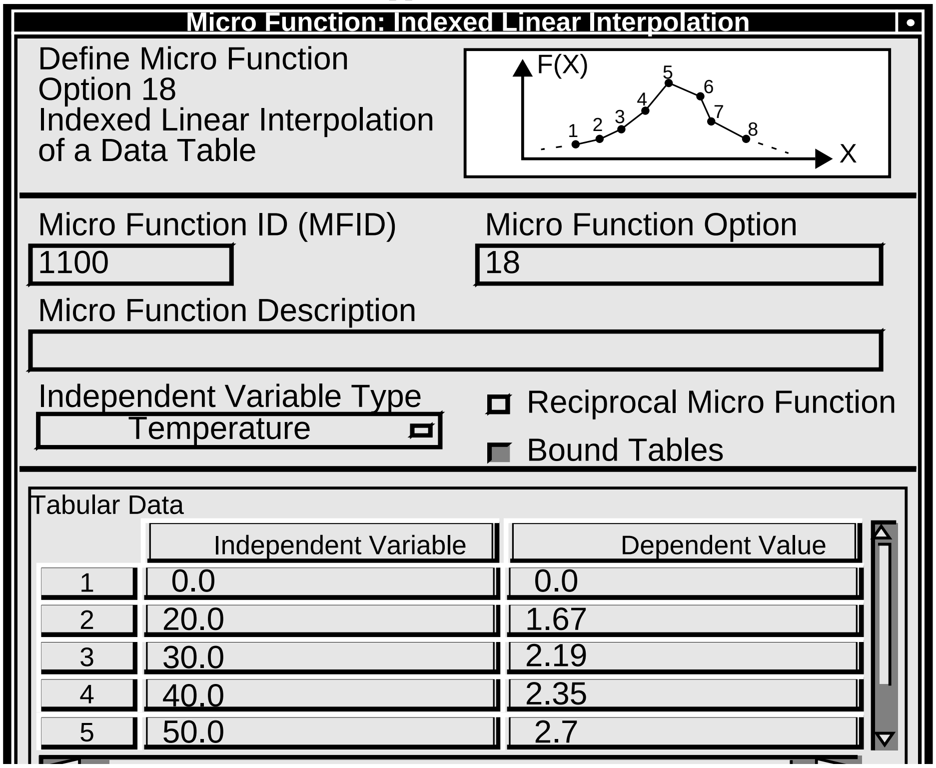
The entries and selection below will guide you through the process of creating the microfunctions. An image of each completed microfunction form is included to facilitate microfunction entry.

When modeling hydraulic networks the independent variable names remain the same on the General Field Input Data form; however, Temperature now refers to Pressure and any other independent variable choices other than Time or Temperature should be ignored.

◆ Field	
Create/Non Spactial/General	
Field Name	**Full_GPM**
Input Data...	
Select Function Term	**mfid_indx_linr_tabl**
Micro Function ID	**1100**
Independent Variable Type	**Temperature**
Independent Variable, (X)	**0.0**

Value, Function(X)	**0.0**
Enter	
Independent Variable, (X)	**20.0**
Value, Function(X)	**1.67**
Enter	
Independent Variable, (X)	**30.0**
Value, Function(X)	**2.19**
Enter	
Independent Variable, (X)	**40.0**
Value, Function(X)	**2.35**
Enter	
Independent Variable, (X)	**50.0**
Value, Function(X)	**2.7**
Enter	
OK	
OK	
Apply	

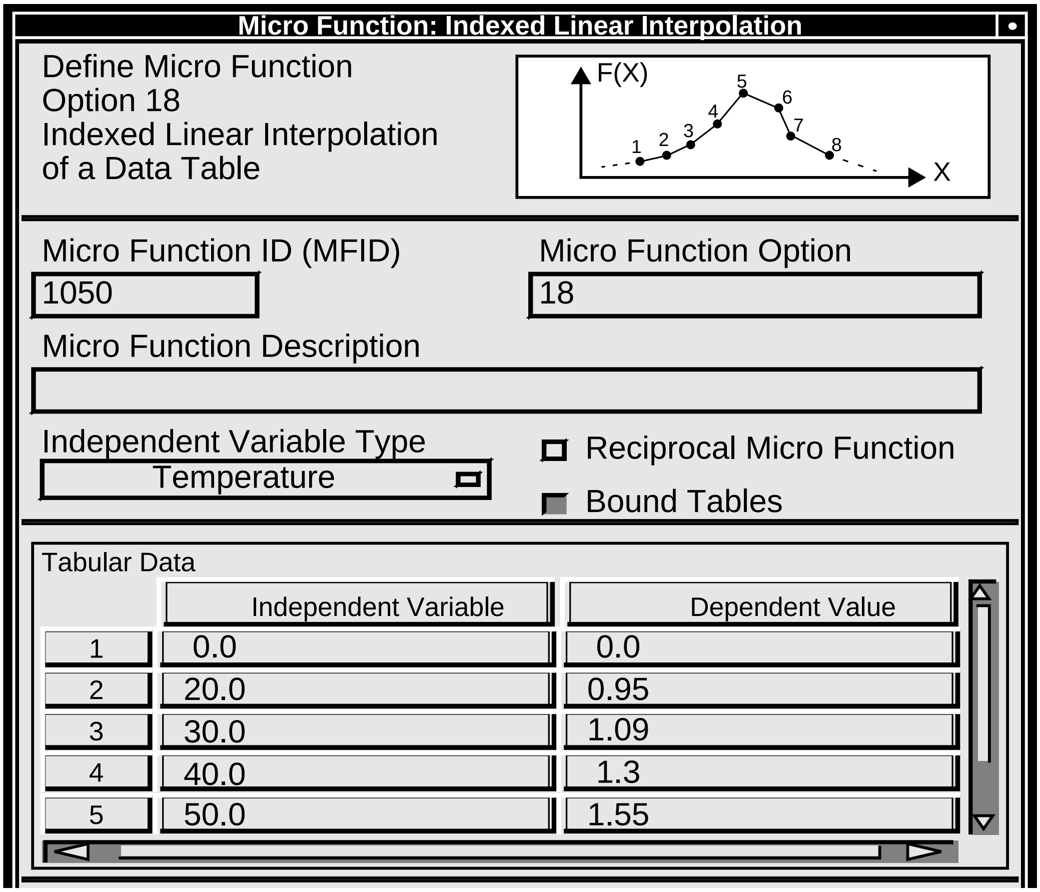
The Micro Function form should appear as shown below.



Field Name	**Half_GPM**
Input Data...	
Select Function Term	**mfid_indx_linr_tabl**
Defaults	**(This clears the data form)**
Micro Function ID	**1050**
Independent Variable Type	**Temperature**
Independent Variable, (X)	**0.0**
Value, Function(X)	**0.0**
Enter	
Independent Variable, (X)	**20.0**
Value, Function(X)	**0.95**
Enter	
Independent Variable, (X)	**30.0**

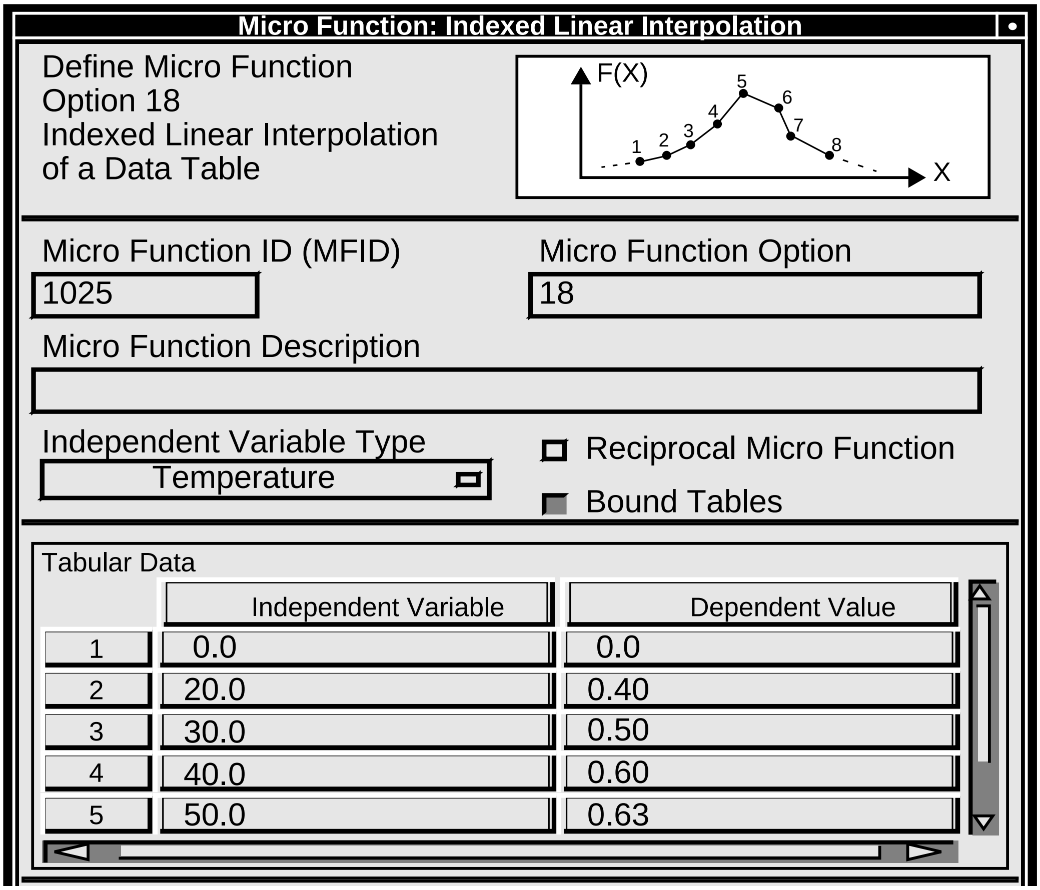
Value, Function(X)	**1.09**
Enter	
Independent Variable, (X)	**40.0**
Value, Function(X)	**1.3**
Enter	
Independent Variable, (X)	**50.0**
Value, Function(X)	**1.55**
Enter	
OK	
OK	
Apply	

The Micro Function form should appear as shown below



Field Name	**Quarter_GPM**
Input Data...	
Select Function Term	**mfid_indx_linr_tabl**
Default	
Micro Function ID	**1025**
Independent Variable Type	**Temperature**
Independent Variable, (X)	**0.0**
Value, Function(X)	**0.0**
Enter	
Independent Variable, (X)	**20.0**
Value, Function(X)	**0.40**
Enter	
Independent Variable, (X)	**30.0**
Value, Function(X)	**0.50**
Enter	
Independent Variable, (X)	**40.0**
Value, Function(X)	**0.60**
Enter	
Independent Variable, (X)	**50.0**
Value, Function(X)	**0.63**
Enter	
OK	
OK	
Apply	

The Micro Function form should appear as shown below



Apply boundary conditions

8. Define inlet pressure and sprinkler head mass flow conditions.

Begin applying boundary conditions. Select the **Load/BCs** *Applications radio button*. Create a nodal pressure named **Entry**. In the Input Data form define the fixed pressure. In the Select Application Region form pick **Point 1** located at the left most section of the geometry.

◆ **Load/BCs**

Create/Pressure/Nodal

Option: **Fixed**

New Set Name **Entry**

Input Data...

Fixed Pressure **6048**

OK

Select Application Region...

Select Point	**<select Point 1>**
Add	
OK	
Apply	

Create a **Variable, Mass Flow Rate** named **Full** with *Template Id* **100** in the Input Data form. In the Select Application Region form pick **Point 13, 14**.

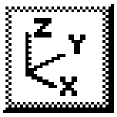
Create/MassFlow Rate/Nodal	
Option:	**Variable**
New Set Name	**Full**
Input Data...	
Template ID	**100**
OK	
Select Application Region...	
Select Points (hold down *<Shift> key*)	**<select Point 13, 14>**
Add	
OK	
Apply	

Repeat these steps for *New Set Name* **Half** with *Template ID* **50** on **Point 9, 10** and **Quarter** with *Template ID* **25** on **Point 2, 3, 11, 12**.

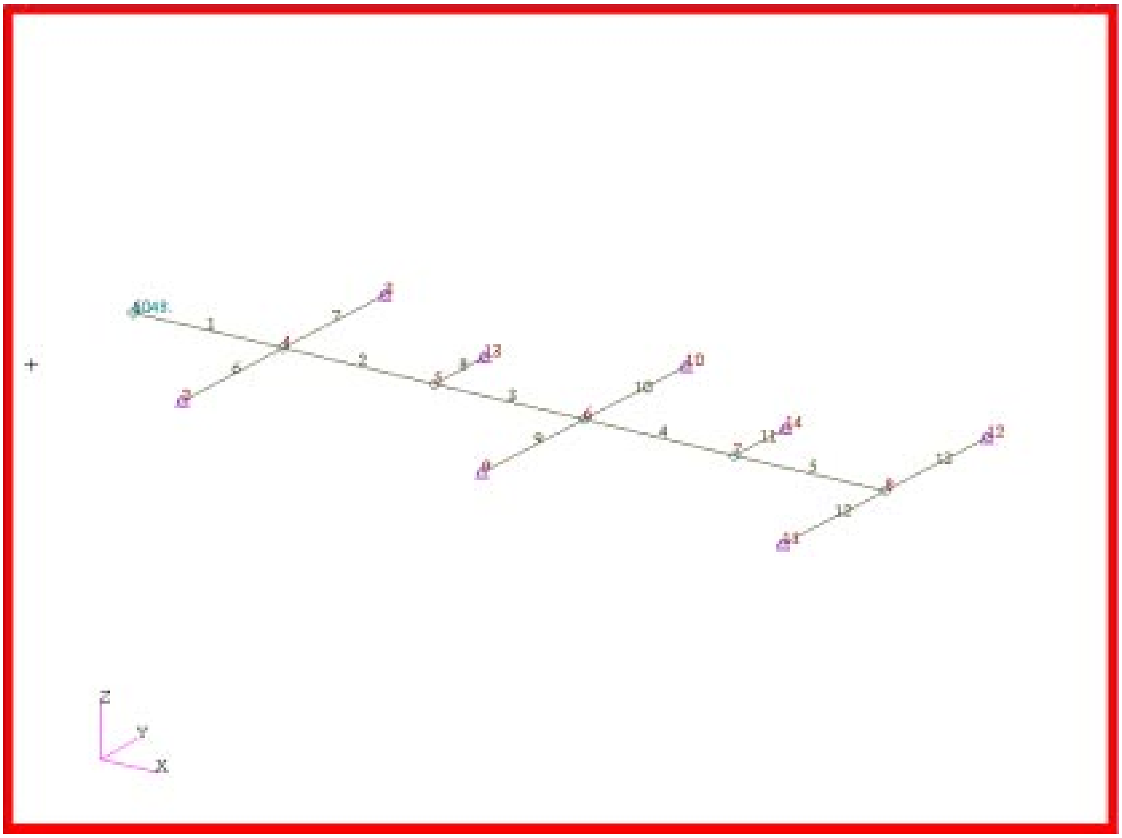
New Set Name	**Half**
Input Data...	
Template ID	**50**
OK	
Select Application Region...	
Select Points (hold down *<Shift> key*)	**<select Point 9, 10>**
Add	
OK	
Apply	

New Set Name	**Quarter**
Input Data...	
Template ID	**25**
OK	
Select Application Region...	
Select Points (hold down *<Shift> key*)	**<select Point 2, 3, 11, 12>**
Add	
OK	
Apply	

Change the view to an isometric view using the *Iso 3 View* icon.



With boundary conditions applied the model should appear as shown below



9. Complete boundary condition definition **template.dat.apnd** file.

In unix create template.dat. apnd file

Using the system editor, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC/PATRAN is running. (You may need to open a new window.) The format for the MACRO functions are as follows:

If a template.dat.apnd already exists in this directory rename it to associate it with that previous analysis. For instance, in Exercise 12 you created a tempate.dat.apnd file. Use the following unix command to move it to a new name associated with that analysis:

> mv template.dat.apnd 12_template.dat.apnd

Using the system editor, typically vi, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC/PATRAN is running.

Use the following chart to help you define the MACRO functions for the pressure boundary conditions assigned to all points.

Table 2:

TID#	Micro_function_count	Node 1	Node 2	scale_factor	mfid#
25	1	0	0	-0.139	1025
50	1	0	0	-0.139	1050
100	1	0	0	-0.139	1100

Note: Nodes 1 and 2 are set to zero since the argument is time.

Shown below is the final form of the **template.dat.apnd** file created for this exercise. Note that any comment lines must be started with an * in column 1 and make sure that there are no blank lines especially at the end of the file.

```
*================
MACRO 25 1 0 0 -0.139
1025
MACRO 50 1 0 0 -0.139
1050
MACRO 100 1 0 0 -0.139
1100
*================
```

10. Prepare and submit the model for analysis.

Prepare and run analysis

Select the **Analysis** *Applications radio button* to prepare the analysis. Select the parameter forms reviewing and changing the settings as shown below. The analysis is submitted by selecting **Apply** in the Analysis form.

◆ Analysis	
Analyze/Full Model/Full Run	
Solution Type...	**◆ Perform Hydraulic Analysis**
OK	
Solution Parameters...	
Calculation Temperature Scale	**◆ Fahrenheit**
OK	
Output Requests...	
Units Scale for Output Temperatures	**◆ Fahrenheit**
Nodal Results File Format ..	**<Select all Hydraulic Node and Element Entries to Output>**
OK	
OK	
Apply	

11. **Read Result** via the Analysis Form.

Read and plot results

From within MCS/PATRAN the only indication that the analysis has successfully finished is the existence of an nrX.nrf.01 results file in a subdirectory one level below your working directory.

P3 was initiated from a working directory which contained the exercise_013.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*; **exercise_013/**. By using **Read Result** in the Analysis form and Selecting **Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of a results file.

◆ Analysis	
Read Results/Result Entities	
Select Results File...	
Directories	**<path>/exercise_13**
Filter	

Available Files	**np0.nrf.01**
OK	
Select Rslt Template File...	
Files	**pthermal_1_pnodal.res_tmpl**
OK	
Apply	

Once he hydraulic results are read in, use Insight to post process the data. Insight provides a mechanism for increasing the diameter of the **Bar 2** elements to view the results.

◆ Insight	
Create/Fringe	
Results Selection...	
Current Load Case(s)	**2.1-Hydraulic Time: 0.0000**
Update Results	
Fringe Result	**1.1-Pressure,**
OK	
Fringe Attributes...	
Edge Width	**5 <use slider bar>**
Style	**Cylinder**
OK	
Target:	**All Edges**
Apply	

The result should now appear as shown on the front page of the exercise.

Use the **vi** editor in UNIX to open the **qout.dat.01** file in the **exercise_13** *Job Name* subdirectory and determine whether the design requirements are met: an entry volumetric flow rate not to exceed 12 GPM and a sprinkler head pressure above 30 psi.

12. **Quit** MSC/PATRAN

Quit MSC/Patran

Do not delete the database when you finish this exercise it will be used in a future exercise. In that exercise we will improve execution time and reduce viewfactor file size. To stop MSC/PATRAN select **File** on the *Menu Bar* and select **Quit** from the drop-down menu.