

---

---

# Symbolic Nuclear Analysis Package (SNAP)

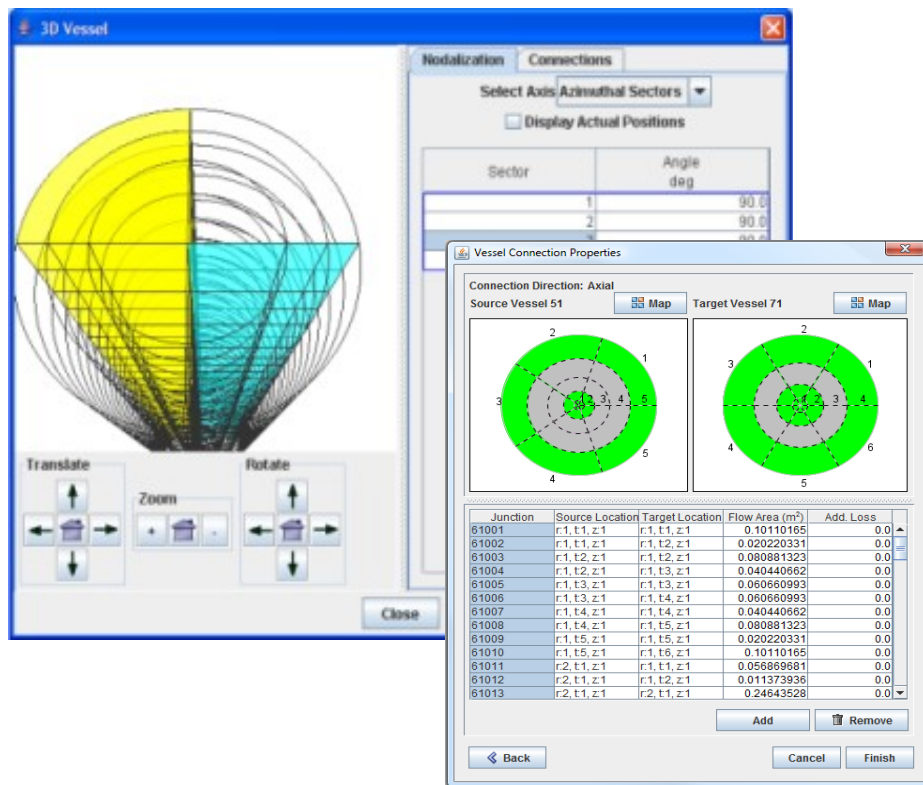
## TRACE/SNAP Users Workshop

### Introduction to SNAP and Job Streams

---

---

March 2018



## Table of Contents

Introduction.....	1
Exercise 1. Configuring SNAP.....	2
Exercise 2. Editing an Existing Model.....	5
Exercise 3. Working with Model Views.....	16
Exercise 4. Introduction to Job Streams.....	25
Exercise 5. Restart Editing.....	33
Exercise 6. Model/Component Diff Viewer.....	42
Exercise 7: 1D Hydraulic Renodalization.....	45
Exercise 8. Working with View Templates.....	52
Exercise 9. Resource Bundle Import/Export.....	53
Exercise 10. 2D Drawing Features.....	55
Exercise 11. Using AptPlot and the ACS Plug-in.....	61

## Introduction

This set of exercises is designed to provide basic instruction on how to configure SNAP and how to use many of the model editing capabilities in SNAP. The following topics are covered:

- Editing an existing model in the Model Editor.
- Creating new component in a model.
- Connecting components using the connect tool.
- Duplicating components using copy/paste.
- Manipulating drawn components in a view.

These exercises assumes you have properly installed the SNAP software along with the following applications:

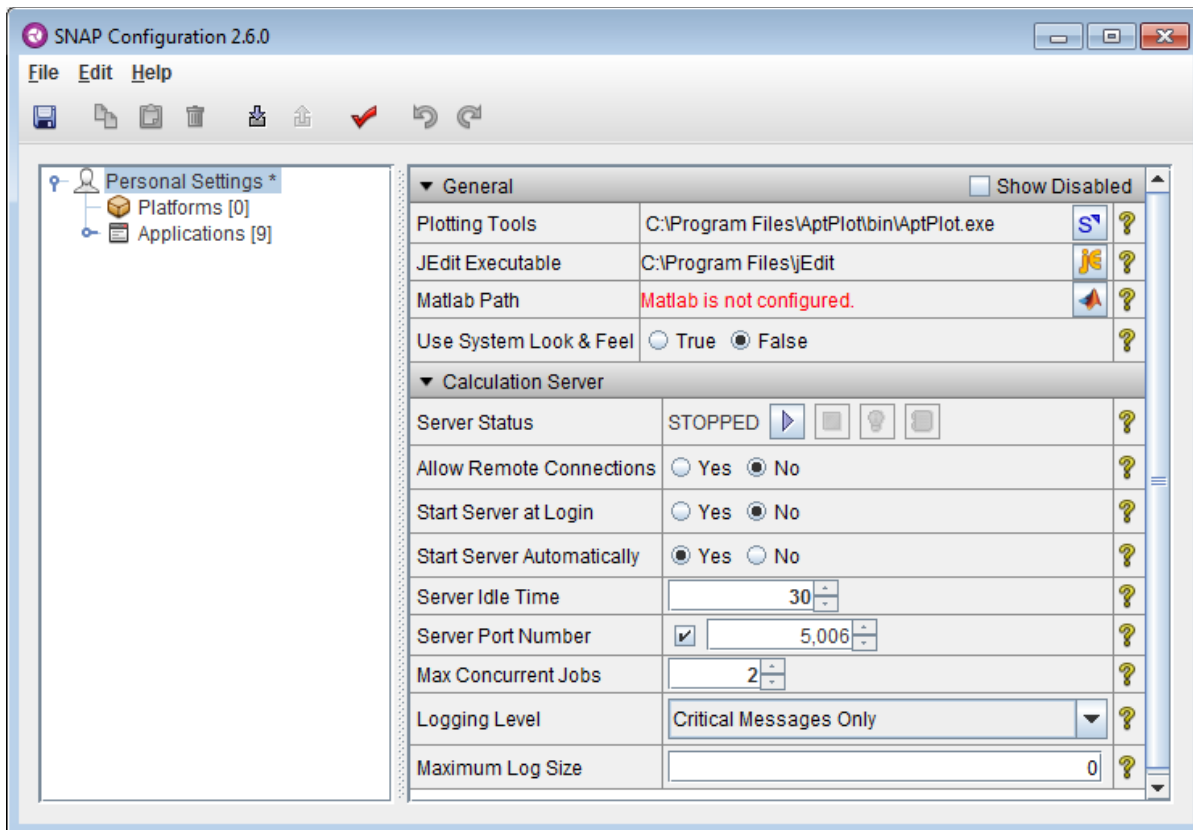
- The AptPlot plotting package. (<http://www.aptpplot.com>)
- The jEdit editor. (<http://www.jedit.org>)
- Adobe Acrobat Reader. (<http://www.adobe.com>)
- TRACE Executable
- The following software will be needed for later exercises:

Please refer to your system administrator or the installation instructions located on the SNAP website (<http://www.nrcsnap.com>) if you have any questions concerning installation.

## Exercise 1. Configuring SNAP

The following exercise provides a set of instructions for configuring a new SNAP installation for this workshop on a Windows system. The SNAP runtime was designed to work with a minimal amount of configuration, therefore most options can be left at their default values. The following steps assume that SNAP has not been previously configured on the system and that all properties are currently at their default values.

1. Open the Configuration Tool by selecting **All Programs** → **SNAP** → **Configuration Tool** from the Start Menu.



2. Select your AptPlot executable by pressing the **S** button located to the right of the **Plotting Tools** property.  
*This will open a file selection dialog.*
3. Navigate to the AptPlot installation directory and select the **AptPlot.exe** file located under the **bin** directory, then press the **Open** button.
4. Select your jEdit installation folder by pressing the **jEdit** button (**jE**) to the right of the **jEdit Executable** property.  
*This will open a directory selection dialog. If jEdit is not installed, skip this step.*
5. Navigate to and select your jEdit installation directory, then press the **Open** button.  
*A prompt will appear, indicating that the SNAP jEdit installation was successful.*

- Dismiss the prompt by pressing the **OK** button.

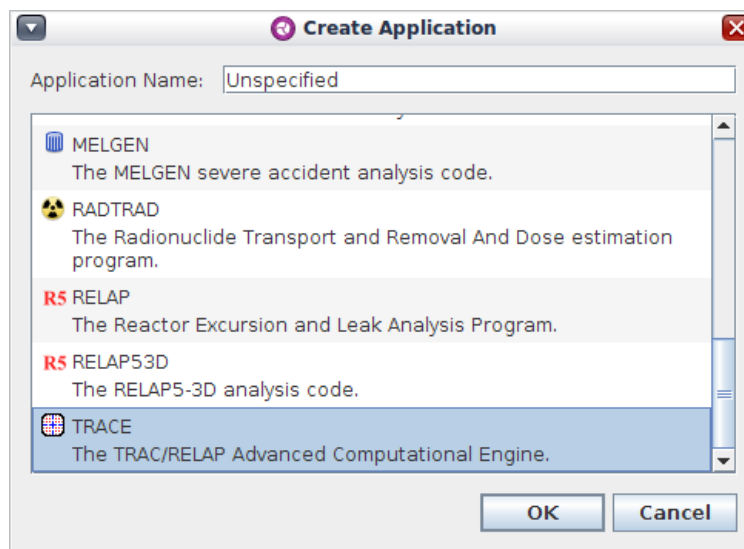
**Note:** Matlab can be configured for use as an external function calculation engine. This requires defining the location of the Matlab executable in the Configuration Tool. This is not required for this workshop.

- Expand the **Applications** node in the tree on the left.

*This will display the applications available for execution as part of a SNAP Job Stream. The applications currently displayed (if any) are those automatically defined as part of a SNAP installation. The next several steps will define the TRACE application used throughout these exercises.*

- Right-click on the **Applications** node and select **New** from the pop-up menu.

*The **Create Application** dialog is displayed, listing the available application types.*



- Select **TRACE** from the list of applications and press the **OK** button.

*A new, unspecified TRACE executable will be created for use in SNAP models. The new application should be selected with its properties displayed on the right. If not, follow the next step.*

- Select the **Unspecified** TRACE application in the **Applications** list.

*The properties to the right will change to reflect those of the selected application.*

- Set the **Name** property to “TRACE”.

*Typical application definition names for a TRACE executable might indicate the version or build number such as “V501147” or “V50P5”.*


- Enter a description for the TRACE executable by pressing the **E** button next to the **Description** property.

13. Press the **S** to the right of the **Local Location** property, located in the **Application Locations** group.

*A file browser will appear. This dialog is used to select the location of the executable on the local machine.*

14. Navigate to the location of the TRACE executable, select it in the file browser, then press the **Select** button.
15. Locate the **Application Version** property and make sure it is set to "TRACE:V 5.0 Patch 5".

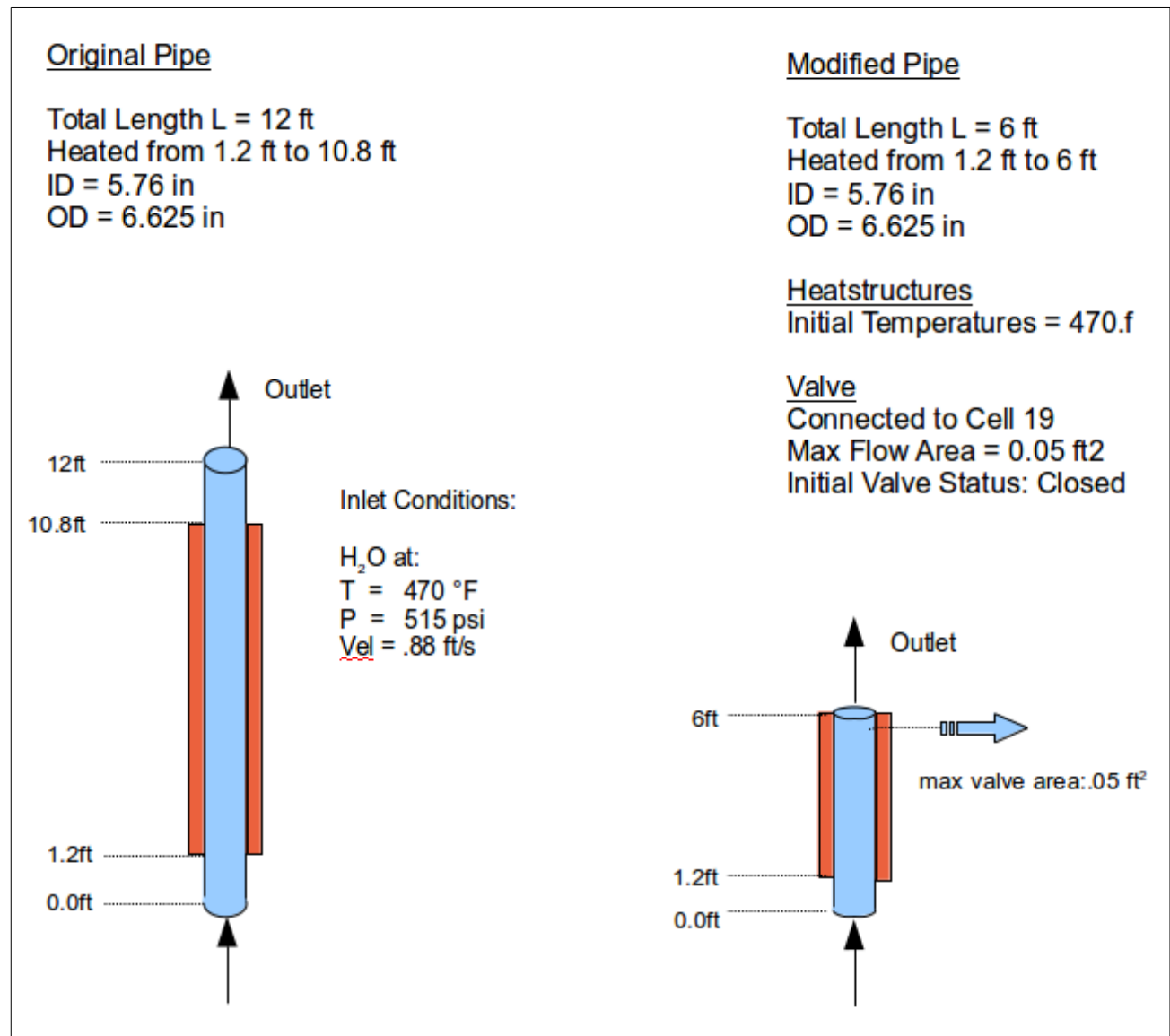
*The flavor and version of the application being defined. This serves as an additional indication to the job step of how this application should be handled.*

16. Save the configuration by either selecting **File** → **Save All** from the main menu or by pressing the **Save** button () in the tool-bar.
17. Exit the Configuration Tool.

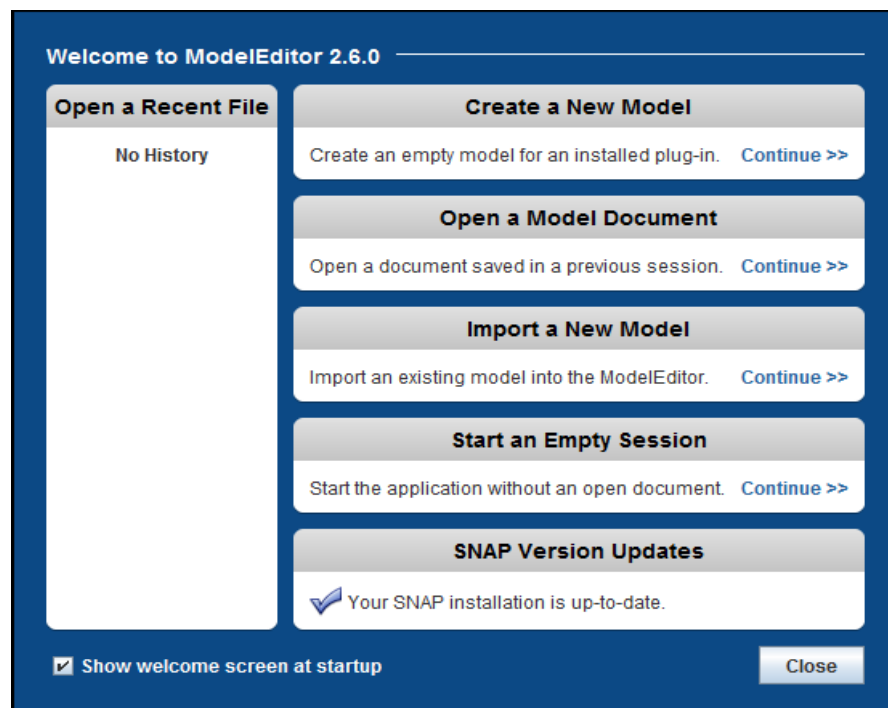
## Exercise 2. Editing an Existing Model

This exercise is designed to familiarize the analyst with the basic editing capabilities of the SNAP Model Editor using the TRACE plug-in. The following steps act as a guide through editing an existing TRACE model in SNAP. This exercise will cover a series of important topics including creating new components, editing component attributes, basic undo/redo functionality, and engineering units.

The exercise will take a simple stand pipe model, reduce the pipe length by half, and add a crossflow connected valve so that the resulting model appears as displayed below:



The first step is to open the ModelEditor application. Under Windows, select the “**All Programs** → **SNAP** → **Model Editor**” option from the Start menu. This will open the ModelEditor and display the welcome dialog shown below:



1. Select the **Open a Model Document** → **Continue >>** option.

*A file selection dialog will appear that is used to select a model file to open.*

2. Locate and select the standpipe model provided with this exercise and open the model. (**SNAP\_Exercises/StandPipe1.med**)
3. From within the navigator select and expand the **Hydraulic Components** → **Pipes** categories and select Pipe 1.

*The TRACE plug-in includes a property editor for each component attribute. Many of the attribute editors provide convenience functions such as copy/paste and plotting table data in addition to the basic data entry function. Note that the initial conditions property editor indicates that it contains invalid cell initial conditions.*

4. Select the **Show ASCII** item from the right-click pop-up menu item of the Pipe's navigator node.

*Each of the components in the TRACE plug-in provide a **Show ASCII** menu item which displays the input of the selected component. The ASCII view is automatically updated when any of the component's properties are modified. The ASCII view window highlights card numbers, comments, and unknown values in distinguishing colors.*

5. Specify the model units by opening the right-click pop-up menu from the model node in the Navigator and selecting **British** from the **Engineering Units** menu.



Notice that each of the attributes are automatically converted to the appropriate value in British units. The TRACE plug-in allows a user to specify the engineering units a model is defined in. For this exercise we will be describing each of the values in British units.

6. From within Pipe 1's property view, locate and open the property editor associated with the **Component Geometry** attribute.

The pipe geometry editor provides a series of columns and tabs for defining Pipe geometry data. The dialog is divided into two main segments, a component view and the property table. Each row in the table corresponds to a cell or edge. Selecting a cell/edge in the view will highlight the row in the table corresponding to the selected cell/junction. Selecting a row in the table highlights the cell/edge the table cell represents. Many of the table editors in the TRACE plug-in provide column header tool-tips which can further describe the values being edited.

7. Set the **Calculate** option to **Volume** and highlight all of the **Length** cells in the table.

Cell geometry in TRACE is defined by specifying any two of the Length, Area, or Volume properties. The third property is then calculated from the entered data.

8. Type in a length value of 0.3 ft and press Enter.

Cell Number	Volume (ft³)	Length (ft)	Vol. Avg. Flow Area (ft²)	DZ (ft)	2D Drawing Pivot
8	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
7	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
6	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
5	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
4	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
3	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
2	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
1	0.0543	0.3	0.181	0.3	<input type="checkbox"/>
<b>Total</b>	<b>1.086</b>	<b>6.0</b>	<b>3.62</b>	<b>6.0</b>	

Calculate  
☒ Volume ☐ Length ☐ Area

Note that the ASCII view updates when the value is modified. The last row in the geometry dialog displays the column totals for the volume and length data.

9. Undo the modifications made to Pipe 1's lengths by selecting the **Undo Cell Geometry Edit** option from the Model Editor's **Edit** menu.

Modifications performed in the Model editor can easily be undone and redone. This will update the editor and ASCII view to reflect the pipe a the length modifications were made.

10. Select the **Redo Cell Geometry Edit** option from the Model Editor's **Edit** menu.

11. Close the **Component Geometry** editor.
12. Scroll to the Gas Volume Fraction (alp) values in the ASCII Viewer and click on the third row of values.

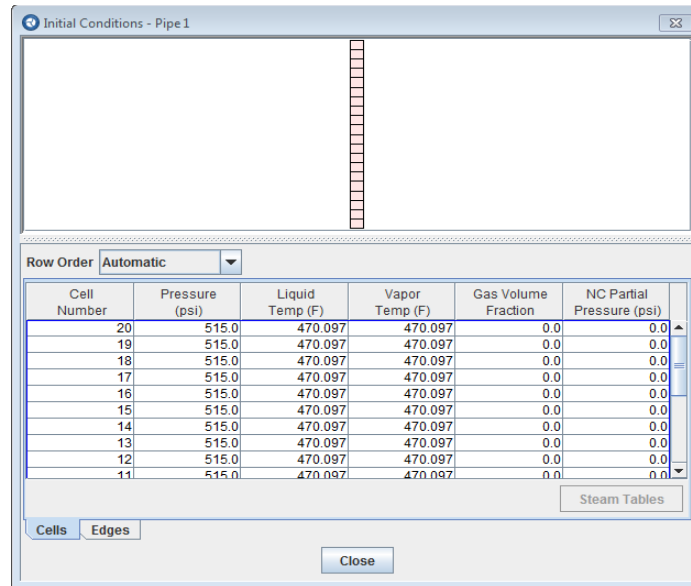
Label	Value 1	Value 2	Value 3	Value 4
* hd *	0.146304	0.146304	0.146304	0.146304s
* hd *	0.146304	0.146304	0.146304	0.146304s
* hd *	0.146304e			
* nff *	1	1	1	1s
* nff *	1	1	1	1s
* nff *	1	1	1	1s
* nff *	1	1	1	1s
* nff *	1	1	1	1s
* nff *	1e			
* alp *	Unknown	Unknown	Unknown	Unknowns
* alp *	Unknown	Unknown	Unknown	Unknowns
* alp *	Unknown	Unknown	Unknown	Unknowns
* alp *	Unknown	Unknown	Unknown	Unknowns
* alp *	Unknown	Unknown	Unknown	Unknowns
* v1 *	0.0	0.0	0.0	0.0s
* v1 *	0.0	0.0	0.0	0.0s
* v1 *	0.0	0.0	0.0	0.0s
* v1 *	0.0	0.0	0.0	0.0s
* v1 *	0.0	0.0	0.0	0.0s
* v1 *	0.0e			
* vv *	0.0	0.0	0.0	0.0s
* vv *	0.0	0.0	0.0	0.0s
* vv *	0.0	0.0	0.0	0.0s

The ASCII Viewer includes a toolbar with a brief description of the input at the current cursor location. Clicking on a supported portion of the input will update the cursor location and enable the **Edit** toolbar button. Pressing the **Edit** button will open the graphical editor corresponding to the current cursor location.

This same behavior is available for the entire model in the SNAP Validation Display which can be opened by pressing the Validation Display (🔍) button on the Model Editor's main toolbar.

13. Press the **Edit** button at the top of the ASCII Viewer to open the **Initial Conditions** editing dialog.

This will display the dialog shown below.



14. Click the **Gas Volume Fraction** entry for the first cell and enter 0.0.
15. Highlight the first cell in the **Gas Volume Fraction** column and choose the right-click pop-up menu **Copy** item to place the value on the clipboard.  
*Most tables in SNAP will allow you to copy and paste data values to and from spreadsheets, tab-delimited ASCII files, as well as between SNAP tables.*
16. Using the mouse, click and highlight the **Gas Volume Fraction** for cell 2 then drag the pointer to the last cell so that all of the remaining void fraction values are selected.
17. Right-click on the table to open the table pop-up menu and select the **Paste** item.  
*The remainder of the table should now be filled and the ASCII view will automatically update to reflect these changes.*
18. Select the first cell in the **Pressure** column and hold the Shift key.
19. While holding down the Shift key, scroll to and click the last cell in the table.
20. Type a pressure value of 515 psi and press Enter.  
*The editor will automatically calculate and set the temperature values of the liquid and gas using the built in steam tables.*  
*Notice that the attribute editor for initial conditions now indicates “[Valid Conditions]” have been entered.*
21. In the same dialog, select the **Edges** tab.
22. Set all **Liquid Velocity** values to 0.88 ft/s by selecting the liquid mass flow for junction 1 and dragging the cursor to the last junction to highlight all of the table cells and then entering the value 0.88 ft/s.

**Notes:**

The Shift and Control keys can be used to add contiguous ranges and individual cells to the current table selection, respectively.

When pasting a single value into a table, all selected values will be updated.


When pasting a set of multiple values from a spreadsheet, values are updated in a one-to-one manner until either the number of values contained on the clipboard or the end of the selection is reached.

23. Close the dialog and the ASCII view.

In the next few step we will add a discharge valve and some interactive control system components to the model.

24. Locate the **Hydraulic Components** → **Valves** category in the Navigator.
25. Create a new valve by selecting **New** from the **Valves** right-click pop-up menu.
26. Set the following values in the provided Valve completion dialog and then press the **OK** button:
  - Hydraulic Diameter: **0.48 ft**
  - Number of Cells: **0**
  - Orientation: **Horizontal**
27. Set the valve **Component Number** to 51 in the property view.
28. Locate the view labeled “Default View” from the **Views** category in the Navigator and select **Open** from the view's right-click pop-up menu.
29. Drag the valve off the Navigator and to the right hand side of pipe 1 cell 19.

The next steps will use the connect tool to connect the inlet of the valve to the cross-flow of pipe 1 cell 19.

30. Select the connection tool  from the view tool-bar.
31. Click on the circle on the left side of the valve.
32. Click on the diamond on the right side of the next-to-last cell of the pipe.

*Refer to the figure for the proper connecting point location.*

*The cursor will change to a target when the connect tool is selected. When the cursor hovers over the connection point located at the inlet of the valve a set of cross-hairs will appear in the target indicating that a connection can be made from this point.*

Clicking the left mouse button while the cursor is on a connection point will initiate a connection. A rubber band line is created from the connection point to the cursor. A filled blue circle will appear when the cursor hovers over a valid target connection point.

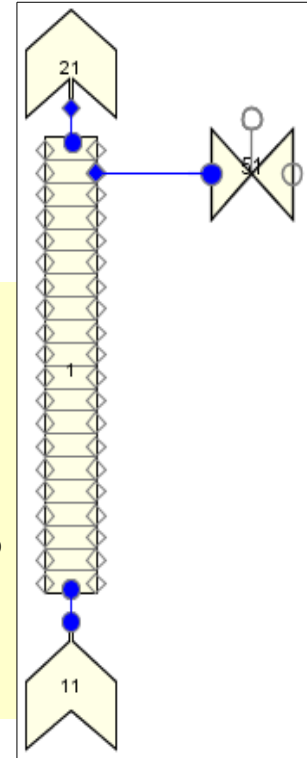
Move the cursor to the target connection point and click the left mouse button to create a connection.



#### Notes:

While a connection is being made, left clicking anywhere other than a valid connection point will create intermediate line segments for the connection.

Right clicking while creating a connection will remove the last intermediate line segment or will cancel the connection if there are no intermediate segments. The escape key can also be used to cancel a connection.

Connections have a right-click pop-up menu that can be used to add or remove points, to remove the connection from the view (cut) or to delete the connection from the model.



33. Choose the Select tool  from the view tool-bar.
34. Select the newly created drawn connection line in the view.
35. Set the target component theta **Angle** property to 0.0 deg.
36. Create a new Break from the Navigator.
37. Drag the newly created Break onto the 2D view to the right of the valve.
38. Connect the valve outlet junction to the Break inlet junction using the connect tool.
39. Choose the Select tool  from the view tool-bar.
40. Right click on the newly created Break drawing and set its **Drawn Orientation** to **Right**.
41. Select the valve in the navigator and locate the **Valve Type** attribute in the Property View.
42. Set the **Valve Type** to **[1] Flow Area Fraction Table**
43. Open the editor associated with the Valve's **Component Geometry** attribute.
44. Set the valve **Flow Area** to 0.05 ft<sup>2</sup> and close the editor.
45. Set the following values in the Valve property view:


Maximum Valve Rate: **1.0 (1/s)**  
Off Adjustment Rate: **0.0 (1/s)**  
Minimum Position: **0.0**  
Maximum Position: **1.0**

Valve Flow Area: **0.05 ft<sup>2</sup>**  
Valve Hydro Diameter: **0.48 ft**  
Initial Flow Area Fraction: **0.0**  
Valve Stem Position: **0.0**

46. Next, select the Break attached to the valve component and specify the following values in the property view:

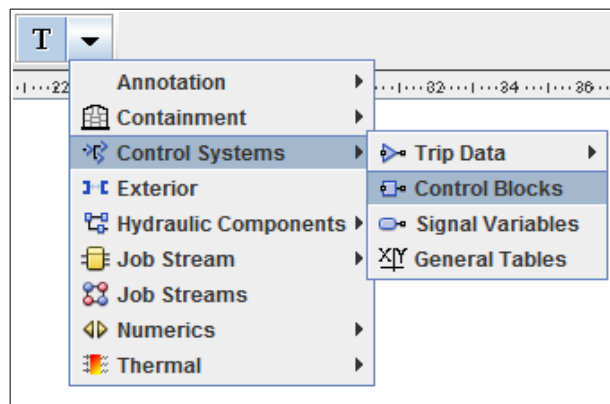
Component Number: **20**  
Length: **0.6 ft**  
Volume: **0.1086 ft<sup>3</sup>**  
Initial Gas Volume Fraction: **0.0**  
Initial Mixture Temperature: **470.0 F**  
Initial Pressure: **515.0 psi**

The next steps will use the insert tool to create new interactive controls for the model. These controls will be used to initialize and change the liquid flow rate, the valve flow area, and the surface heat flux of the outside surface of the heat structure.

47. In the 2D view, select insertion tool  and expand the drop down menu to select “Control Systems → Control Blocks”.

*This will define that the insert tool will create new control blocks. After clicking the mouse once, the mouse tool will automatically revert to the select tool.*

**Note:** The component insertion tool is located within the 2D view tool-bar as a drop-down menu. Selecting a component from the drop-down menu allows the user to create the specified component and place it in the 2D view. This method is an alternate to creating the component from the category New right-click menu items.



48. Click the mouse, using the insert tool to the left of Pipe 1, below the Interactive TRACE Demo text label.
49. Select type **[-9] Interactive Variable** in the provided completion dialog and press **OK**.
50. Verify that the Type of the Controller Block is **[-9] Interactive** and set the properties of the interactive variable as follows:

Control Block Name: **heatflux**  
Variable Name: **Heat Flux**

51. Construct another Interactive variable using the 2D insertion tool.

*This will be used as the valve stem position controller.*

52. Specify the following information for the newly constructed interactive variable:

Control Block Name: **valvepos**  
Minimum: **0.0**  
Maximum: **1.0**  
Constant 1: **0.0**  
Variable Name: **Valve Area**

53. Construct another Interactive variable using the 2D insertion tool.

*This will serve as the inlet flow velocity controller.*

54. Select the newly constructed interactive variable and specify the following information:

Control Block Name: **invel**  
Constant 1: **0.88**  
Variable Name: **Inlet Vel**

The next step is to connect the previously created interactive controllers to their associated hydraulic components.

55. Select Fill 11 and open its **Fill Table** property editor.

56. Set the **Independent Variable** to the **invel** interactive controller.

57. Press **OK** to close the dialog.

*The connection between the fill and the interactive controller in the 2D view will be indicated with a green dotted line.*

58. Next select the valve component and set its **Valve Table Indep. Var.** property to the **valvepos** interactive controller.

59. Press **OK** to close the dialog.

60. Open the editor for the **First Adjustment Table** property in the valve property view.

61. Press the **Add** button twice to add two rows to the table.

62. Set the cells in the first row to: 0.0, 0.0.

*This tells the valve to set the flow area fraction to 0.0 when the interactive variable provides a 0.0 value.*

63. Set the cells in the second row to: 1.0, 1.0

*This tells the valve to set the flow area fraction to 1.0 when the interactive variable provides a 1.0 value.*

64. Press the **OK** button to save the values and close the dialog.

65. Select “**Thermal → Heat Structures → Heat Structure 31**” in the Navigator.

66. Open the editor for the **Axial Nodes / Surface BCs** property.
67. Select Axial Cells 5 through 20 in the **Outer Surface Boundary Conditions** column.
68. Set the **Boundary Condition** property to **[10] Controlled Heat Flux**.
69. Set the **Control Signal** property to the heatflux interactive controller.

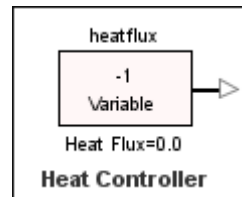
Next we will configure the heat structure to reflect changes made to the pipe length and initialize its temperature values.

70. Select all of the cells in the **Axial Cell** column.
71. Set the **Length** property of the nodes to 0.3 ft.  
*This reflects the modification made in an earlier step to Pipe 1 where we cut the length of the nodes from 0.6 ft from 0.3 ft.*
72. Press the **OK** button to accept the changes and close the dialog.
73. Open the **Initial Temperature** property editor.
74. Highlight all of the cells in the table, type in value 470.0, and press Enter.  
*This will set the initial temperature of the radial nodes of the heat structure to 470.0 F*
75. Verify that all of the values are set to 470.0.
76. Press the **OK** button to accept the changes and close the dialog.

In the next few steps we will add some basic annotations to the view.

77. Select the “**Annotation** → **Text Annotation**” item from the view insert tool drop-down menu.
78. Click on the view just below the heatflux interactive variable created earlier.
79. Enter “Heat Controller” as the **Text** property.
80. Set the **Font** to “Dialog, Bold, 12” by selecting the bold 'B' character in the font selection editor.
81. Select the “**Annotation** → **Rectangle**” item from the view tool-bar drop-down menu.
82. Use the insert tool to surround the new text annotation and the control component.

*Note that the rectangle is drawn over the top of these items and prevents them from being seen.*

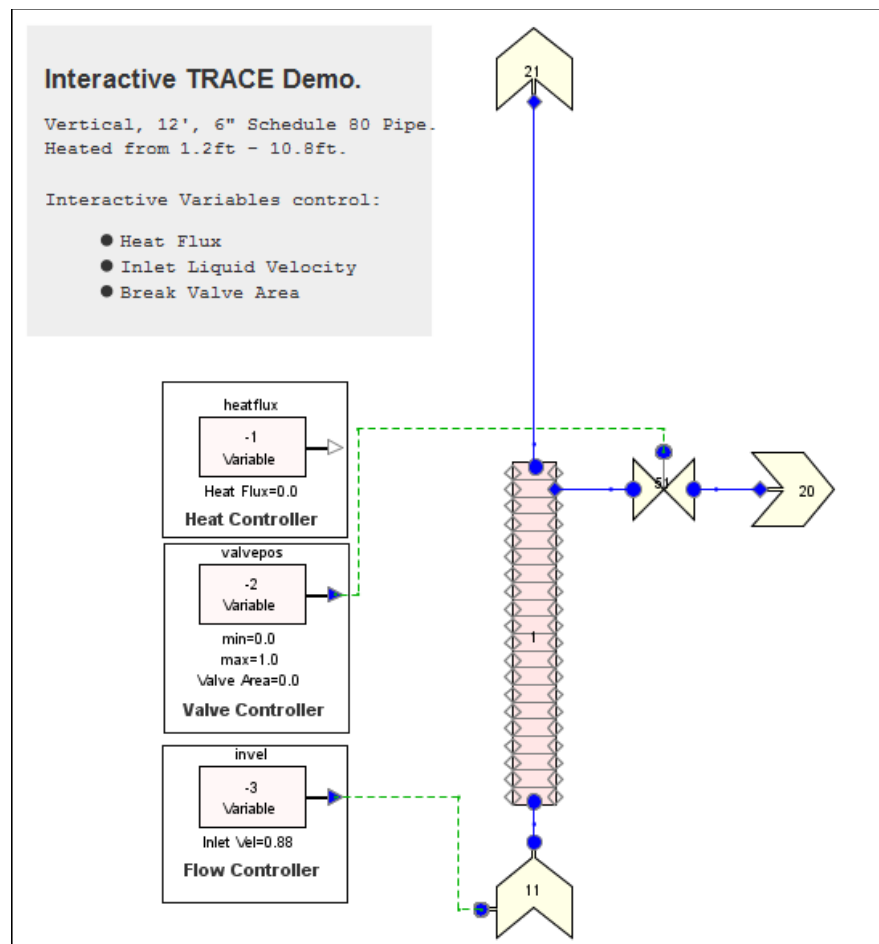


83. Open the right-click pop-up menu on the new Rectangle and select the “**To Back**” option.



84. Repeat this procedure to add labels and boxes around the Inlet Velocity Controller and Valve Controller control systems we created earlier.

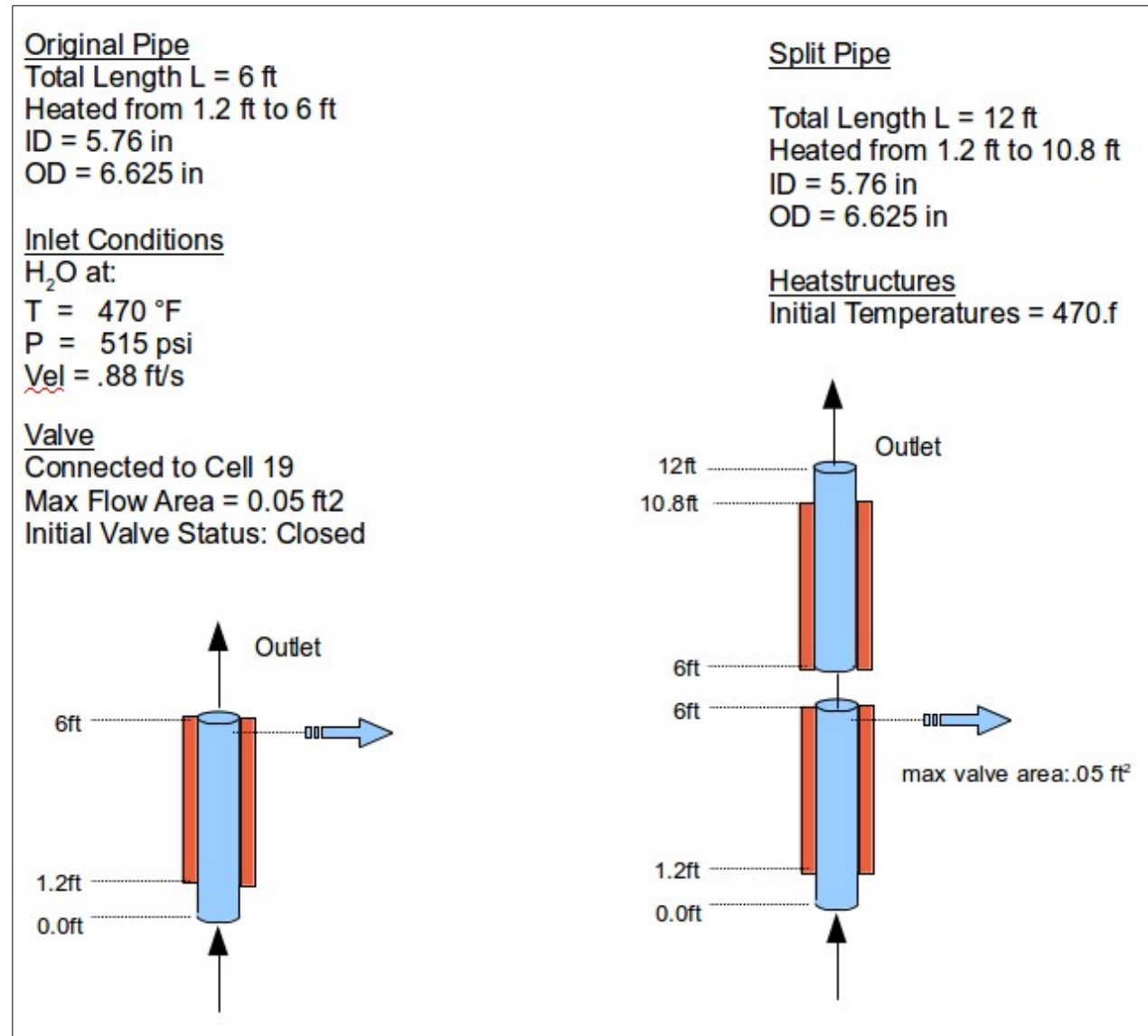
*The resulting view should resemble the one shown below:*



### Exercise 3. Working with Model Views

This exercise is designed to build upon the model created in Exercise 2 and continue introducing model building elements of SNAP. This exercise will cover topics including copying and pasting components, creating small control networks, basic undo/redo functionality, and engineering units.

The exercise will take a simple stand pipe model, duplicate the pipe, and insert the duplicated component between the existing pipe and the outlet. The resulting system will match the system displayed below:



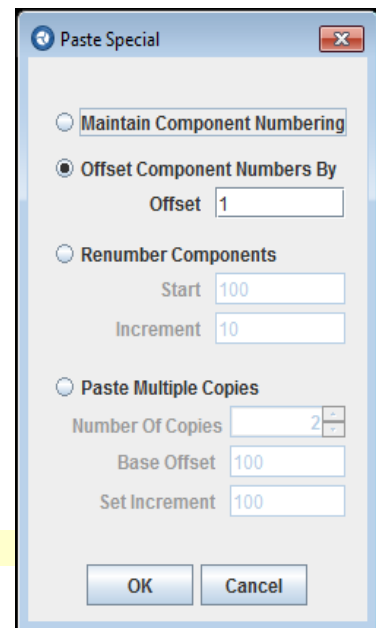
1. If you do not have the model from Exercise 2 open, open the file **SNAP\_Exercises/StandPipe2.med** included with these exercises.
2. Expand the **Views** Navigator node.

*Models can contain any number of 2D views. These views are loaded and saved with the model. Views contain 2D drawn representations of many types of model components and provide tools to help describe those components. The views can contain annotations, background colors, grid, etc.. Views allow component searching using the binocular icon provided in the view tool-bar. Zoom options are available by selecting the magnifying glass icon or using the right-click menu.*

In the next steps we will create a duplicate copy of the pipe component and a simple control system to generate a display value.

3. Select the **Hydraulic Components** → **Pipes** → **Pipe 1** node in the Navigator and select the **Copy** option from its right-click pop-up menu.
4. Choose the **Paste Special** option from the right-click pop-up menu of the **Pipes** category node.
5. Select the **Offset Component Numbers By** option and set the offset to 1.

*The paste special dialog allows a user to create multiple copies of the same component starting at a specific offset with a specific number increment. This also allows users to copy/paste model components within the same model or into a separate model.*



6. Press the **OK** button to paste the component.
  7. Locate the view labeled “Default View” and select **Open** from the view's right-click pop-up menu.
- Note:** The view's pop-up menu also includes a print option.
8. Expand the Hydraulic Components category in the Navigator and drag Pipe 2 onto the view just above Pipe 1.





*All components which have a 2D representation can be dragged onto a 2D view. Only one instance of each component can exist in a view (no duplicates of the same component are allowed in the same view). Components which have connections to other components in that view will automatically draw those connections when they are inserted into the view.*

9. Right-click on Pipe 2 in the view and select the **Scale Drawing** menu item.
10. Set the **Length Scale Factor** to 5.95, the **Width Scale Factor** to 10.0, and press the **OK** button.
11. Select the background of the 2D view to bring up the views properties.
12. Locate the **Canvas Size** property and re-dimension the views height to be 900 pixels.

*This will increase the amount of vertical space we have available so that there is adequate space for adding and moving components in the view.*

13. Select the connection from the outlet of Pipe 1 to Break 21.
14. Select the **Disconnect** option from the right-click pop-up menu.

**Note:** The undo/redo functionality can be used to resolve any issues where a component or connection has been mistakenly added or removed.

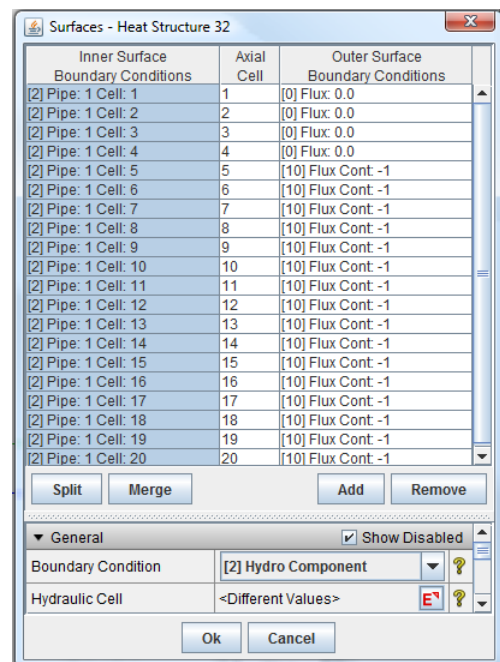
15. Select the connection tool  from the view tool-bar and create a connection between the inlet of Pipe 2 and the outlet of Pipe 1.
16. Use the connection tool  to create a connection between the outlet of Pipe 2 and the inlet of Break 21.
17. Choose the Select tool  from the view tool-bar.
18. Use the view's left-click rubber-band tool to select each of the display components which are arranged vertically. (This does not include the valve or its connected Break)
19. With the components selected, open the view right-click pop-up menu and select the **Align** → **Center** () item.  
*This will cause the components to center around a common vertical axis.*

In the next steps, the heat structure that was originally duplicated along with pipe 1 will be removed and replaced with a new heatstructure.

20. In the Navigator select Heat Structure 32 and select **Delete** from the right-click pop-up menu.

*Components which are deleted can be restored using the undo functionality.*

21. Select the **Copy** item from the right-click pop-up menu of Heat Structure 31.
22. Select the **Heat Structures** Navigator category node and select the **Paste Special** option.
23. Set the **Offset Component Number** property to 1 in the provided dialog and press the **OK** button.
24. Select the newly created Heat Structure in the Navigator.
25. Open the editor for the **Axial Nodes / Surface BCs** property.
26. Select all of the cells in the Inner Surface Boundary Conditions column and open the **Hydraulic Cell** property editor in the lower property view.



27. Change the referenced pipe value to the newly created pipe by opening the editor next to the **Hydraulic Component** property and selecting pipe 2 from the list.

*Once the Pipe has been selected as the **Hydraulic Component**, two spinners will appear to allow the selection of the range of hydraulic cells to map to the selected Heat cells.*

28. Press the **OK** button to save the value and close the dialog.
29. Verify the **Starting Cell** is set to 1 and the **Ending Cell** is set to 20.
30. Press the **OK** button at the bottom of the Heat Connections editor.

The dialog box 'Heat Connections' is shown with the following settings:

- Heatstructure: Heat Structure 32
- Selected Surface: Inside Surface
- Hydraulic Component: Pipe 2
- Starting Hydraulic Cell: 1 of 20
- Ending Hydraulic Cell: 20 of 20

The 'Surface Connections' table is as follows:

Heat Cell	Hydro Cell
1	Cell [1]
2	Cell [2]
3	Cell [3]
4	Cell [4]
5	Cell [5]
6	Cell [6]
7	Cell [7]
8	Cell [8]
9	Cell [9]
10	Cell [10]
11	Cell [11]
12	Cell [12]
13	Cell [13]
14	Cell [14]
15	Cell [15]
16	Cell [16]
17	Cell [17]
18	Cell [18]
19	Cell [19]
20	Cell [20]

Buttons: OK, Cancel

Next we will set the heatflux controller on the lower portion of the pipe.

31. Select the first four axial cells in the **Outer Surface Boundary Conditions** column.
32. Set the **Boundary Condition** property to **[10] Controlled Heat Flux**.
33. Set the **Control Signal** to the heatflux controller.
34. Select the last 4 axial cells (17-20).
35. Set the **Boundary Condition** property to **[0] Constant Heat Flux**.
36. Press the **OK** button to accept the changes and close the dialog.

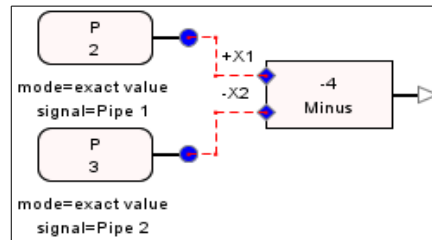
In the next steps a control block summer will be created to determine the pressure drop across the two pipes. Two signal variables will be created signifying the pressures at opposite ends of the pipes.

37. Using the insertion tool, select **Control Systems** → **Signal Variables**.
38. Insert the first signal variable on the view below the valve component.
39. Select type **[21] Pressure** in the provided completion dialog.
40. Insert the second signal variable directly below the last signal variable and again set the type to **[21] Pressure** in the provided completion dialog.
41. For the upper signal variable, open the editor for the **Signal** property.
42. Set the **Hydraulic** property to Pipe 1, the **Cell** to 1, and press the **OK** button.
43. For the lower signal variable, open the editor for the **Signal** property.
44. Set the **Hydraulic** property to Pipe 2, the **Cell** to 20, and press the **OK** button.

*The signal variables will be supplied to a subtract block in order to determine the pressure drop across the pipes.*

45. Using the insert tool drop-down menu, place a control block on the view to the right of the signal variables.

*The placement should be similar to the following figure.*

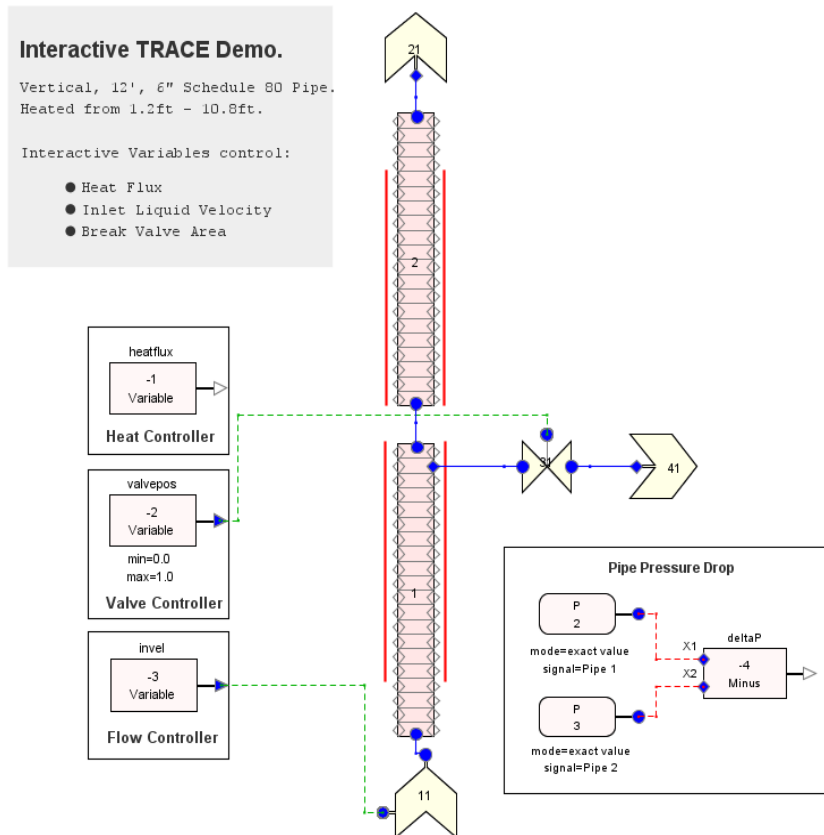


46. Select **[54] Subtract** when prompted from the completion dialog.
47. Set its **Control Block Name** property to **deltaP**.
48. Using the connect tool, connect the signal variables to the subtract block by connecting the upper signal variable to the upper connection point and the lower signal variable to the lower connection point.
49. Select the **Annotation** → **Text** item from the view insert tool drop-down menu and click on the view just above the new control network.
50. Enter “Pipe Pressure Drop” as the **Text** property.
51. Set the **Font** to “Dialog, Bold, 12” by selecting the bold 'B' character in the font selection editor.
52. Select the **Annotation** → **Rectangle** item from the view tool-bar drop-down menu and use it to surround the new text annotation and the control network.
53. Open the right-click pop-up menu on the new Rectangle and select the **"To Back"** option.

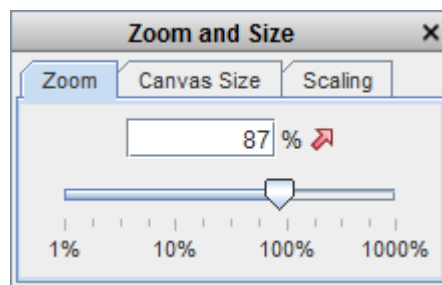
In the next section we will add some red lines adjacent to the pipe to signify the regions where power is being added to the heat structure.

54. Select the **Annotation** → **Line** annotation from the view tool-bar drop down menu and place a vertical line along the left side of Pipe 1, starting at the fourth vertical level from the bottom and extending to the topmost level of Pipe 1.
55. Set the **First Arrow Head** and **Second Arrow Head** properties to **None**.
56. Set the **Color** property to 255,0,0 and then close the dialog.
57. Use the right-click **Copy/Paste** menu items to copy the line to the right hand side of the Pipe and verify that it is placed at the same vertical position.

*These lines are a simple indicator of where the heatstructure powers the Pipe. The TRACE plug-in supports a series of annotation objects including text, rectangle, polygon, ellipse, image, and line.*



58. Copy one of the lines created in the previous step to left hand hand of Pipe 2.  
*Position the line such that the top of the line begins at the bottom of the fourth cell from the top of the pipe and stretches to the bottom of the pipe.*
59. Repeat these steps to place an annotation to the right-hand side of Pipe 2.
60. Select the background of the view and open the right-click pop-up menu. Locate and select the **Zoom** option.



*This opens the view zoom and scaling options pop-up dialog. This dialog allows controlling the zoom, the view size, and the view scaling factors in one convenient location.*

61. Click on the red arrow (↗) to bring up the zoom shortcut pop-up menu.
62. Click the **Fit to Window** option from the zoom shortcut menu.

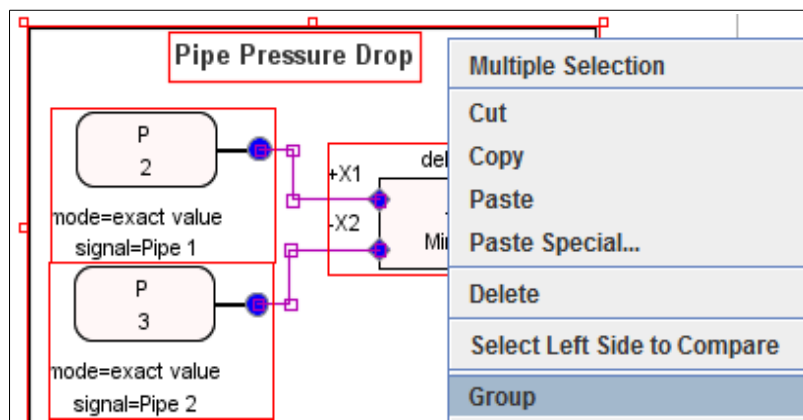
*This option adjusts the zoom in a view to display all of the drawn components and annotations inside that view.*

63. Select the background of the view and open the right-click pop-up menu, locate the **Tools** option, and select **Trim Excess Canvas**.

**Note:** The Trim Excess Canvas option can be used to easily reduce the canvas size to be the bounds of the components contained in the view.

In the next few steps we will use component grouping to group up each of the control systems and their associated annotations.

64. Locate the **Pipe Pressure Drop** control system block in the view and use the right-click rubber-band select tool to highlight the rectangular annotation, text annotation, and components.



65. With each of the pressure drop components selected, open the right-click pop-up menu and choose the **Group** option.
66. Select the **Pipe Pressure Drop** group and move the block until it is above the valve.  
*Notice that component grouping locks each component into a visual group. The elements in the group cannot be selected individually unless the group is first broken. As the group is moved, each of the items belonging to the group move with respect to their location within the group. Individual components can only be added to one group, although groups can be combined with other groups.*
67. Select the individual controller block components and group them using the procedure defined above.

The next few steps we will demonstrate additional view features.

68. In the property view of the Default View, locate the **Display Icon** attribute and select the camera icon to grab a snapshot of the current state of the view.
69. Press the **OK** button to accept the default image size.
70. Create a new view by selecting the Views category node in the Navigator and selecting the **New** option from the right-click pop-up menu.
71. Set the **Name** of the view to Embedded Views.



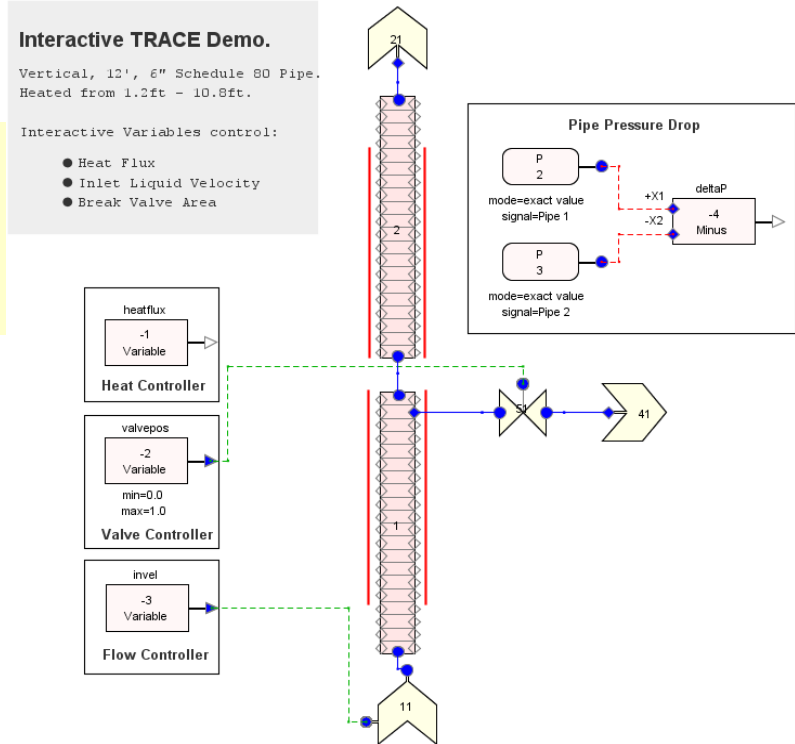
72. In the new view, select the **Annotation** → **Views** option from the view tool-bar drop-down and left-click inside the view to insert an embedded view.
73. Select the “Default View” in the dialog which identifies the view to embed in our current view and press the **OK** button.
74. Double-click on the embedded icon to switch to the Default View.  
*More than one embedded view can be placed within a view to provide convenient navigation between views.*
75. Once again bring up the right-click pop-up menu for the Default View and locate the **Export Image** option.
76. Select the **Entire View** option and choose **JPEG Image**.  
**Note:** Either the entire view or current perspective can be exported. The image exporter supports the following formats: JPEG, PDF, PNG, SVG, and TIFF. Additionally, the view can be exported to the system clipboard to paste the image into external applications such as MS Word.
77. Export the image as modelview.jpg using the default image settings.  
*Image size and scale can be modified by the dialog which is provided after a filename has been specified.*
78. Open the exported image and verify that it is an accurate representation of the exported view.

The next few steps will demonstrate reusing established views through the use of the SNAP view template feature. View Templates provide a means of saving a layout of a view that may then be applied to a similar model.

79. Create a new empty View by right-clicking on the **Views** node in the Navigator and selecting the **New** menu item.  
*This should create and open a new View with the name "View 1" and display its properties in the Main Property View.*
80. Open the view pop-up menu by right-clicking anywhere **inside** the empty view (the pop-up from the Navigator node does not contain the necessary menu item).
81. Select "**Tools** → **Import View Template**" in the pop-up.
82. At the file browser, select the file standpipe.mvt provided with these exercises (SNAP\_Exercises/standpipe.mvt) and press **Open**.

*This will apply the template to the view, which should now appear as shown below:*

**Note:** To export a View Template of an existing view, use the "Tools → Export View Template" item off the right-click pop-up menu in the View.



## Exercise 4. Introduction to Job Streams

This exercise introduces the basics of building Job Streams in the SNAP Model Editor. A stream consisting of a single TRACE step will be submitted to the local Calculation Server. Afterward, the stream will be monitored from the SNAP Job Status application and the task output examined.

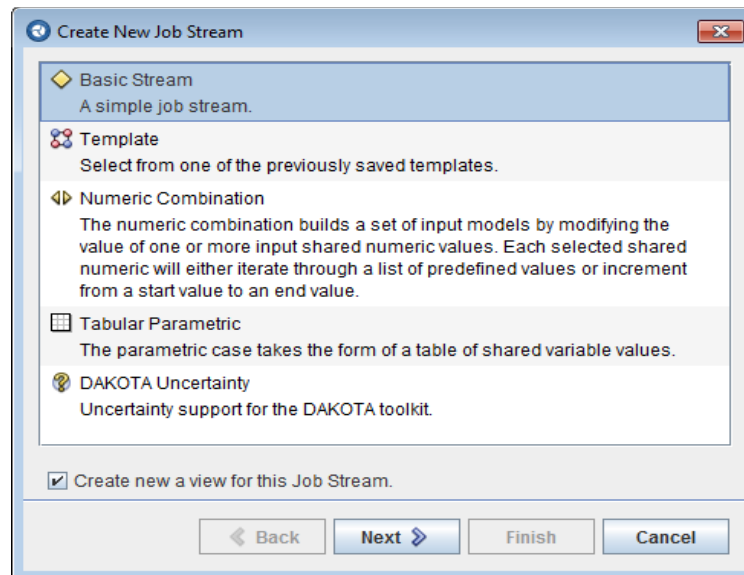
The following steps set up a **runs** folder on the Local Calculation Server. This is the location in which streams will be run through the exercises. If the SNAP configuration for the Local Calculation Server already contains a suitable submit location, skip the next six steps and use this folder whenever the exercise refers to the **runs** directory.

1. Open Job Status. On Windows, select from the Start menu: **All Programs** → **SNAP** → **Job Status**.
2. In the tree to the left, expand the **Local** node. If a prompt appears asking whether to start the Calculation Server, press **Yes**.  
*Expanding this node connects to the Local Calculation Server.*
3. At the **No Root Folders** prompt, press **Yes**.  
*A file browser will appear. This is used to select a location on the server where streams can execute.*
4. In the file browser, navigate to a suitable run directory or create one. Press the **Mount** button once the folder is selected.  
*A prompt will appear asking for the name of the new mount point.*
5. Enter the name “**runs**” for the folder and press the **OK** button.  
*The **runs** folder will appear under **Local**.*
6. Close Job Status.

The following steps build the simple stream and then submit it.

7. Open the Model Editor. On Windows, select from the Start menu: **All Programs** → **SNAP** → **Model Editor**.  
*The Model Editor will start up and display a welcome screen. Note that the welcome screen can be disabled and may not appear.*
8. On the welcome screen, select the **Open a Model Document** → **Continue >>** option.  
*A file selection window will appear prompting for the model to open. If the welcome screen was not displayed, select “**File** → **Open**” from the main menu.*
9. Open the standpipe sample model provided with this exercise. This file is located at “**SNAP\_Exercises/StandPipe3.med**”  
*After a moment, the model will be completely loaded and its primary view will be displayed. This is the model that will be submitted to the Calculation Server, which is covered in the next several steps.*

10. Right-click the **Job Streams** category and select **New** from the pop-up menu.  
*The **Select Stream Type** dialog is displayed, as shown below.*



11. De-select (un-check) the “**Create a new view for this Job Stream.**” check-box.

*A 2D View will be created to display the job stream in later steps.*

12. Select **Basic Stream** in the stream type dialog and press the **Next** button.

*The basic stream type is the simplest of the stream types and does not allow parametrics of any sort. The other stream types are used to create parametric streams, a type of stream where the model is executed multiple times with some type of iterative modifications between each task. Parametric stream types are explored in a later exercise.*

*The list of predefined job streams is displayed. Predefined job streams are used to speed the process of creating simple job streams for newly created or newly imported models. This exercise will start with an empty stream in order to examine the job stream creation process mode closely.*

13. Select **An Empty Stream** in the stream type dialog then press the **Finish** button.

*A new basic job stream is created and selected in the Navigator.*

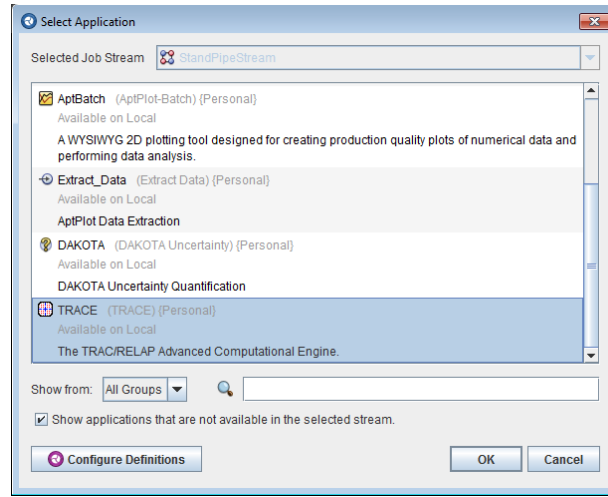
14. Set the following properties in the newly created stream:

Name: **StandPipeStream**  
Platform: **Local**  
Root Folder: **runs**  
Relative Location: **TRACE/**

15. In the Navigator, expand **StandPipeStream**.

*A set of child categories are shown, including **Stream Steps** and **Model Nodes**.*

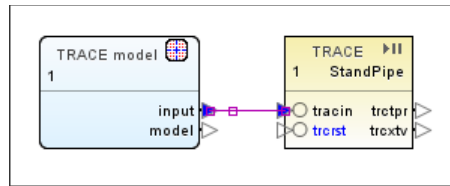
16. Right-click on the **Stream Steps** category and select **New** from the pop-up menu.  
*The **Select Application** dialog is displayed, as shown below.*



17. Select the appropriate **TRACE** application from the available list and press the **OK** button.  
*A new TRACE step is added to the stream. This is the step that will run the TRACE input represented by the model.*
18. Set the following properties on the new TRACE step:  
Name: **StandPipe**  
Interactive Step: **On**  
Start Paused: **On**  
***Interactive Step** and **Start Paused** will ensure that the TRACE job is run in interactive mode and, after a short initialization phase, waits before executing.*
19. Expand the **Model Nodes** category in **StandPipeStream**.  
*A single “model node”, **TRACE model 1**, will appear in the category. This node represents the StandPipe model.*
20. Right-click on **StandPipeStream** and select **Create View** from the pop-up menu.  
*A view will be created with the model node and the TRACE step.*
21. Re-position the elements of the new view so that the blue model node is to the left of the red TRACE node.

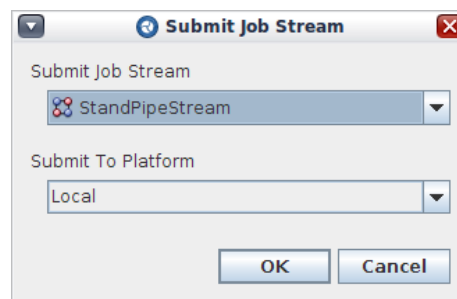
22. Using the Connection Tool in **StandPipeStream View**, connect the job stream model's **input** point to the **StandPipe** step's **tracin** point.

*The stream step will change color from red to yellow, indicating that its required inputs have been connected.*



*This simple process represents the most fundamental concept of job streams: connecting the outputs of a model, file, or step to the inputs of another step. When this stream is executed, the input created by the stream model will be run by the specified TRACE application as its **tracin** input.*

23. Select “**Tools** → **Submit Job**” from the main menu.



*The stream submission dialog is displayed, as shown.*

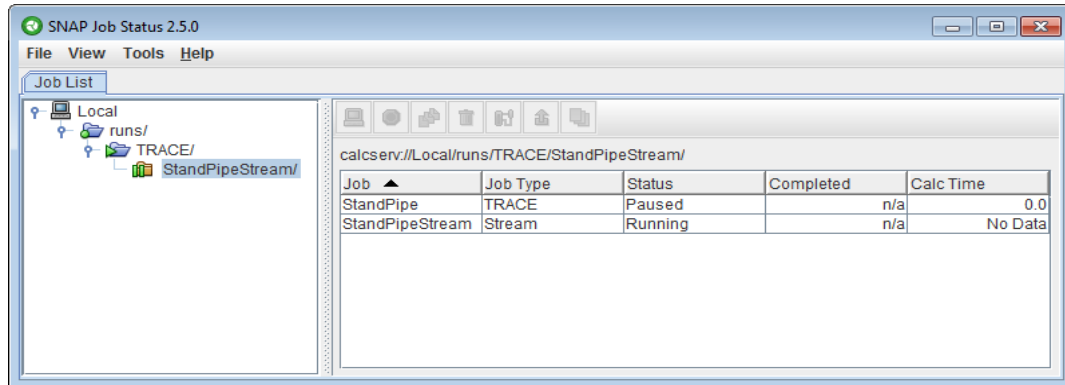
24. In the stream submission dialog, make sure that **StandPipeStream** and **Local** are selected, then press the **OK** button.
25. Press **OK** to confirm the stream submission.

*The job will be submitted to the Local Calculation Server. Job Status will appear shortly thereafter and is described below. Once the stream begins execution, it will perform its necessary initialization, then pause until told to proceed.*

The following steps introduce Job Status. This application provides functionality for accessing and monitoring streams submitted to a calculation server. In particular, the next several steps will examine the paused TRACE task created above and resume its calculation.

26. Wait for Job Status to appear after submitting the stream. If Job Status does not appear, select from the Start menu: **All Programs** → **SNAP** → **Job Status**.

*Job Status is shown in the image below. On the left is a list of known Calculation Servers and their mounted-folder hierarchies. The area on the right is a list of the jobs residing in the selected folder.*



27. Expand **Local**, then the **runs** folder, then the **TRACE** folder, and select the **StandPipeStream** folder.

*Note that this folder is a different color than those above it. This indicates that the directory was created specifically to house the contents of the stream. Selecting the stream shows the tasks running in that stream in the table to the right. Notice that **StandPipe** is listed and that its status is **Paused**.*

This final series of steps will issue an interactive command to the TRACE task, resuming its calculation. The interface for this process is the **Interactive Commands** dialog, which allows pausing, resuming, and completely halting jobs. The dialog also allows changing the values of interactive variables.

*Note: Terminating the run outside of the Interactive Commands dialog terminates the process, which may produce corrupted output files.*

28. Select **StandPipe** row in the table to the right.

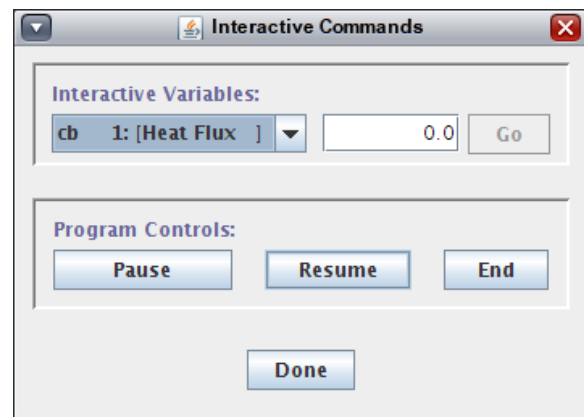
29. Press the **Interactive Commands** button (🔧) on the toolbar above the job list.

*The Interactive Commands dialog is displayed, as shown in the figure to the right.*

30. Press the **Resume** button.

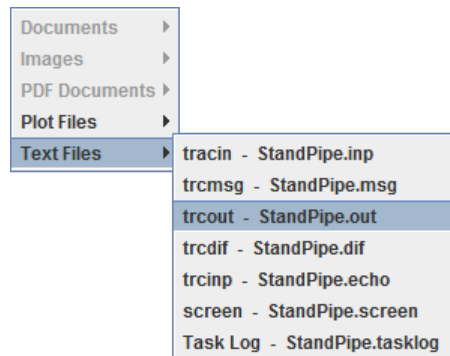
*The **Status** of **StandPipe** changes from **Paused** to **Interactive**, and the calculation resumes.*

31. Close the Interactive Commands dialog by pressing the **Done** button, then wait for the job to complete.



32. In the Job List, expand **Local**, and select the **/runs/TRACE/StandPipeStream** folder.
33. Press the **View Output** button (📄).

This will open a pop-up menu of available files related to the job. Files are broken down into several types, as shown in the image below. Selecting any items in the **Text Files** menu will open the Output Viewer, described below.



The TRACE **StandPipe** job provides several files which can be opened by the File Viewer. Some notable files include:

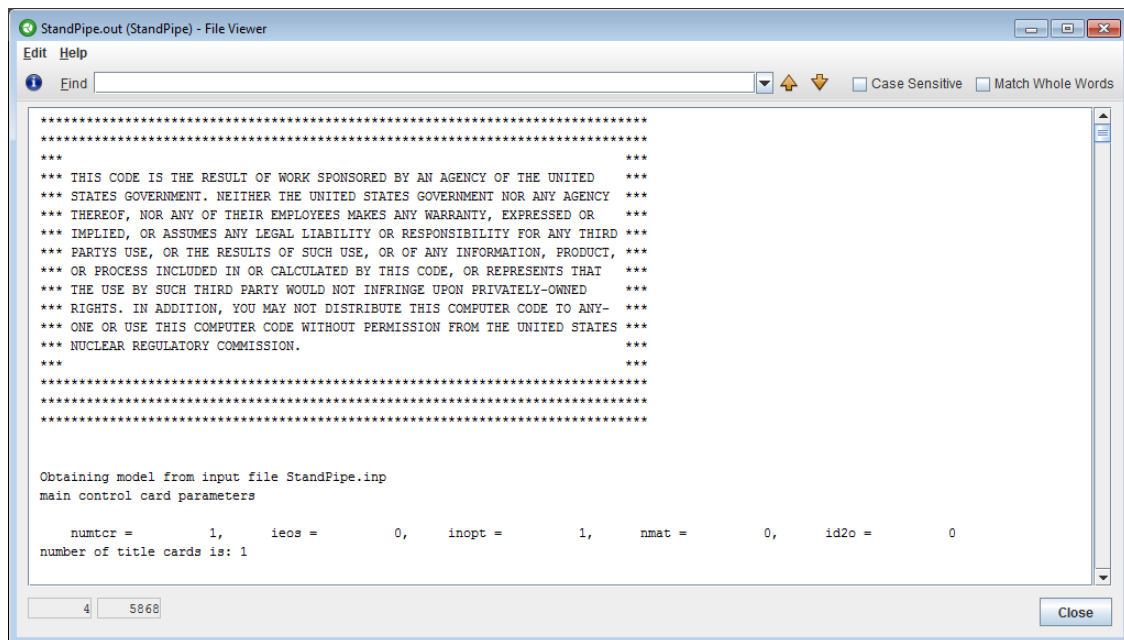
**tracin - StandPipe.inp.** The input file submitted to the TRACE executable.

**trcout - StandPipe.out.** The results of the calculation in ASCII form.

**trcmsg - StandPipe.msg.** Condensed output on the behavior of the numerical calculation and warning messages.

**screen - StandPipe.screen.** A collection of messages written to the console during the run.

34. From the pop-up menu, select the **trcout** entry.

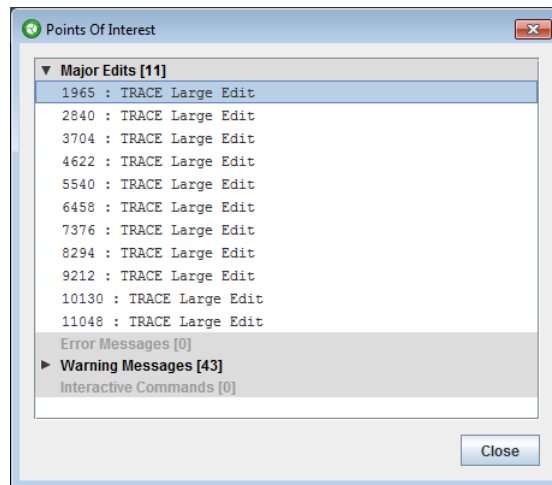




The File Viewer will appear and display the contents of the file. The File Viewer was designed to allow very large text files on a remote Calculation Server to be viewed and searched over a relatively slow connection.

35. Press the Points of Interest (i) button.

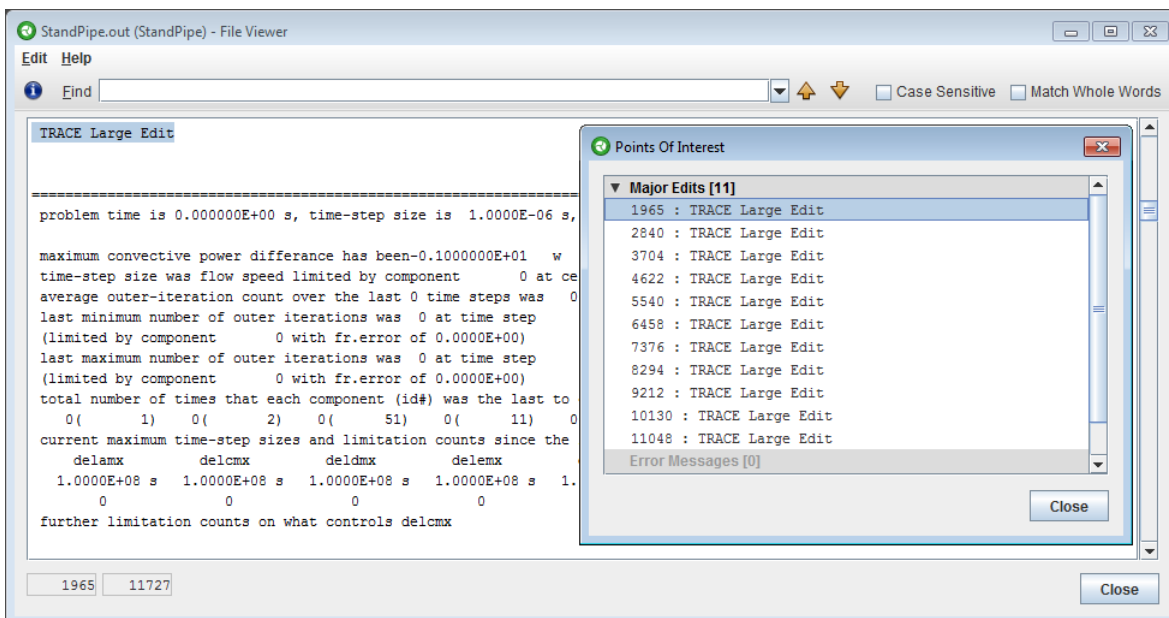
The **Points of Interest** search dialog will be displayed. The points of interest in the file are separated by type into expandable sections. Expanding each section will show the list of points of interest of that type in the file and the line number on which it occurs.



For TRACE output files the points of interest include major edits, error messages, warning messages, and interactive commands sent from SNAP.

36. Click on the **Major Edits** row to expand the list of major edits.

This will expand the Major Edits row to display the list of major edits in the file and will automatically select the first major edit in the list.



*Notice that the File Viewer is now positioned at the beginning of the first Major Edit. Each of the provided Points of Interest items will navigate to the point in the output where the item can be found.*

37. Close the **Points of Interest** window.

38. Close the File Viewer

## Exercise 5. Restart Editing

This exercise describes how to build restart cases in a Job Stream. These steps build on the stand pipe model built in previous exercises. The **StandPipe** job will be used as a source for initial conditions. A null transient restart case and a transient restart case will be created. The transient case will resupply the valve, with the valve opening at time 250. The initial conditions for the resupplied valve will be retrieved from the null transient case on the server.

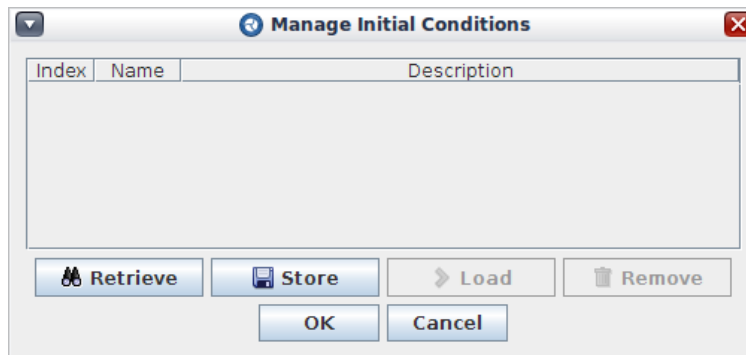
The following steps will guide you through the exercise.

1. Open **SNAP\_Exercises/StandPipe4.med**, included with these exercises, and make the following modifications:
  - Select **StandPipeStream** in the Navigator and make sure its **Platform** is set to **“Local”** and its **Root Folder** to **“runs”** (or the appropriate root folder used in place of the **runs** directory).
  - Select the **TRACE** step in **StandPipeStream** and make sure its **Application** is set to a valid TRACE application.

*The property view will display the current Application in red if the property is invalid for the local configuration.*

2. Right-click on the **StandPipe4.med** model node in the Navigator and select **Manage Initial Conditions** from the pop-up menu.

*The Manage Initial Conditions dialog displayed below is opened. This dialog is used to retrieve, store, load, and discard sets of initial conditions retrieved from completed jobs.*



3. Press the **Store** button.

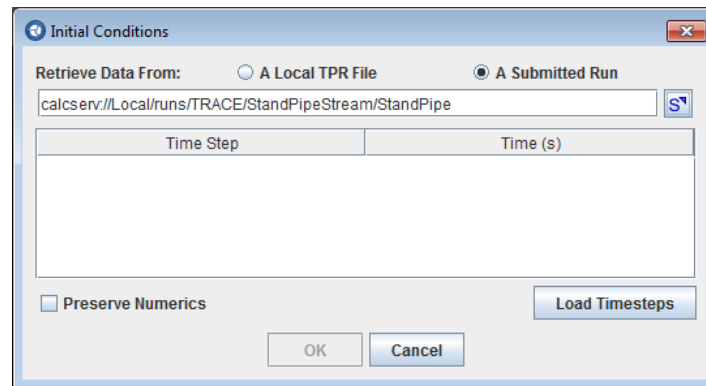
*A new row is created in the table, representing a set of initial conditions that have just been stored for later use. The contents of this initial conditions set were pulled from component values in the model. At any time in the future, the model's initial conditions can be reset to these values by selecting this row and then pressing the **Load** button.*

4. Double-click in the **Name** column of the newly created **unnamed** row and change the name to **“Original”**.

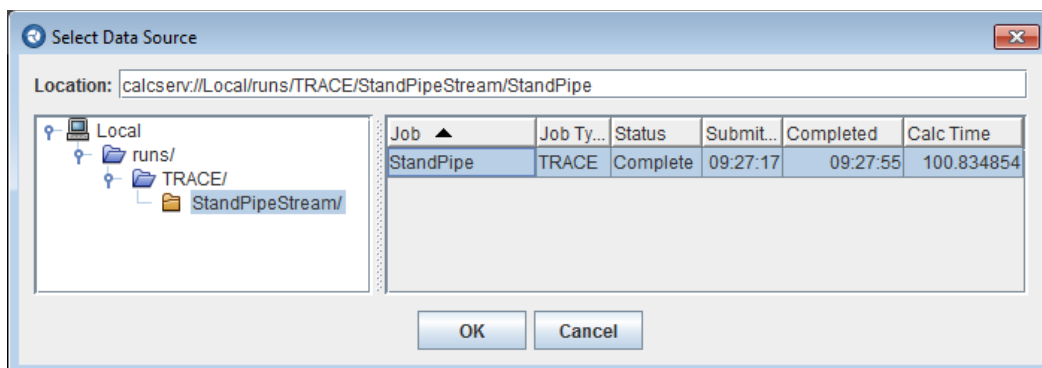
The **Name** and **Description** of the row are the only means of indicating the contents of the initial conditions, so it is important that they accurately describe the row.

5. Press the **Retrieve** button.

The **Initial Conditions** dialog will appear, as displayed below. This dialog is used to retrieve initial conditions from completed calculations.



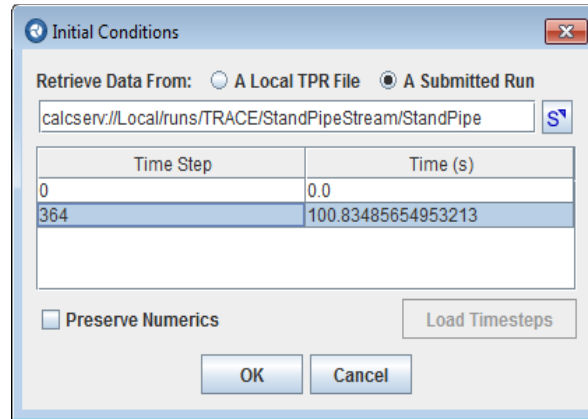
6. Select the **A Submitted Run** radio button.
7. Press the **S** to the right to select the job from which to retrieve initial conditions.  
This will open the job selection dialog shown below.



8. Expand **Local**, then the **runs** folder, then the **TRACE** folder, and select the **StandPipeStream** folder.
9. Select **StandPipe** in the table to the right and press **OK** to close the dialog.  
The previously submitted run is now the source of data from which initial conditions will be retrieved.
10. Press the **Load Timesteps** button.  
Two **Available Restart Times** will become available.
11. Select the last restart time in the list: approximately, Timestep **364** at **100.834** s.

12. Press the **OK** button.

*This will retrieve the initial conditions at the last timestep and assign them to each component in the model. Notice the initial conditions of **Pipe 1** displayed in the ASCII view have changed to reflect the data imported from the plot file.*



13. Press the **Store** button.

*As with storing the **Original** initial conditions, the initial conditions currently in the model will be represented by a new, **unnamed** row in the table.*

14. Double-click in the **Name** column of the newly created **unnamed** set and change the name to "**Steady State**".

15. Select the **Original** initial conditions row and press the **Load** button.

*The contents of the stored **Original** conditions will be loaded into the model. The **Steady State** contents will be used in a later step.*

16. Press the **OK** button in the Manage Initial Conditions dialog to close the dialog.

With the initial conditions set, the following steps edit the model as a restart.

17. Create a new restart case in the **Cases** category.

18. Set the Component Name to **Null Transient**.

19. Select the **Initial Conditions** by pressing the **S** button in the editor.

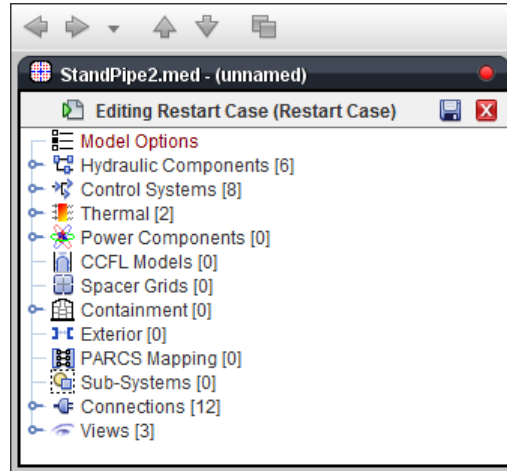
*A selection dialog will appear. This is used to select a set of initial conditions that the restart case will be based on. This has the effect that the selected initial conditions will be loaded automatically when editing a restart case in a virtual model (as shown in the next several steps).*

20. Select **Steady State** and press the **OK** button.

21. Set the **Editing Mode** property of the new case to "**Graphical**".

22. Begin editing the new Restart Case by pressing the **E** button for the **Restart Model** property.

A virtual model will be created, where the restart changes are made. Notice the red dot in the model node and the Restart Panel under it. The red dot is used to indicate models currently in Restart Editing mode and is used for all plug-ins that support restarts, while the panel is used to indicate that a restart case is being edited.



23. Select **Model Options** in the Navigator.

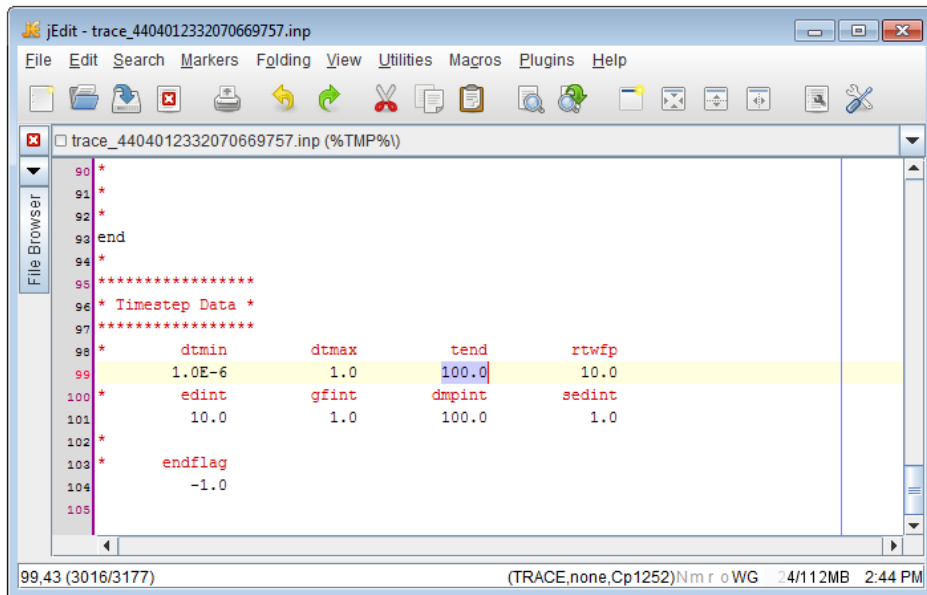
24. Change the **Transient Calculation** value to **[1] Transient**.

The following steps will illustrate how to perform manual ASCII edits to a restart case while performing a graphical edit.

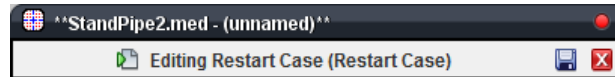
25. Select the **Tools** → **Export to jEdit** menu item.

26. Once jEdit opens, scroll to the bottom of the input model to the section with the Timestep Data heading.

27. Change the tend value to 200.0.



28. Save the input model to a local file location.
29. Right-click the **Model Node** in the Navigator and select the **Import** → **Restart** menu item.
30. Select the saved input model and press the **Open** button.
31. In the Restart Case panel, press the **Save Case** button (📁).



*The changes made in the virtual model are imported as a complete restart case. The ASCII view of a Restart Case displays the generated input for that case.*

Now that the null transient restart case is complete, the next several steps will build a transient calculation, where the valve is opened at 250.0 seconds automatically instead of relying on user interaction.

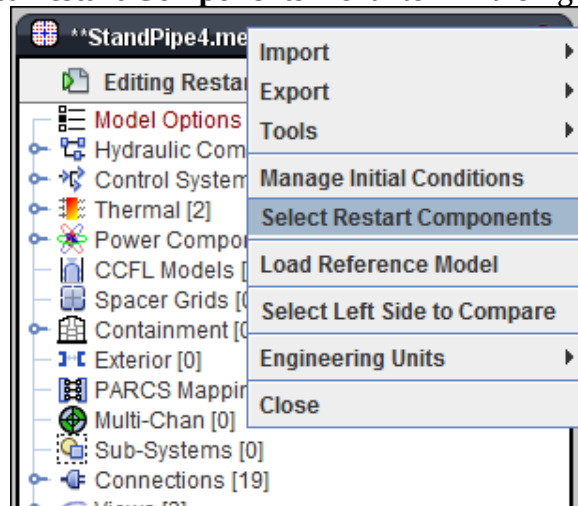
32. Create a second restart case
33. Change the **Component Name** to **Transient**.
34. Set the **Editing Mode** to **Graphical**.
35. Begin editing the new Restart Case by pressing the **E** button for the **Restart Model** property.
36. Select **Model Options** in the Navigator.
37. Change the **Transient Calculation** value to **[1] Transient**.
38. Press the **E** to the right of the **Timestep Data** property to open the timestep table.
39. Change the **End Time** from **100** seconds to **400** seconds, then press **OK**.
40. Expand the Hydraulic Components [6] → Valves [1] node in the Navigator.
41. Select Valve 51.
42. Set the **Valve Table Indep. Var.** to Signal Variable 1 Problem Time.
43. Open the **First Adjustment Table** property editing dialog.
44. Press the **Add** button.
45. Set the table values to match the following:

No Unit	Area Fraction
0.0	0.0
250.0	0.0
251.0	1.0

*This table begins opening the valve at 250.0 seconds and finishes opening the valve a second later.*

46. Right-click on the Model node in the Navigator.

47. Click on the **Select Restart Components** menu item in the right-click pop-up menu.



*This opens a dialog that allows selecting which components will be included in the restart case. Notice that Valve 51 is already included. When components are modified during a restart edit, they are automatically flagged for inclusion in a restart case.*

48. Set the **Available Categories** drop-down list to **Control Systems** [8].
49. Select **Problem Time 1** in the Available Components and press the Right Arrow button.

*This selects the Problem Time signal variable which must be included in the restart case because the Valve table is referencing problem time.*

50. Press the **OK** button.

51. In the Restart Case panel, press the **Save Case** button (💾).

Now that the restart cases are built, the next several steps will build a job stream that will execute.

52. Open the **StandPipeStream** view if it is not already open.
53. Expand **StandPipeStream** in the Navigator and add two new **Model Nodes**.
54. With **TRACE model 2** selected, select the check box in the **Restart Case** property editor.

*The property is now enabled, indicating that this model node represents a restart case.*

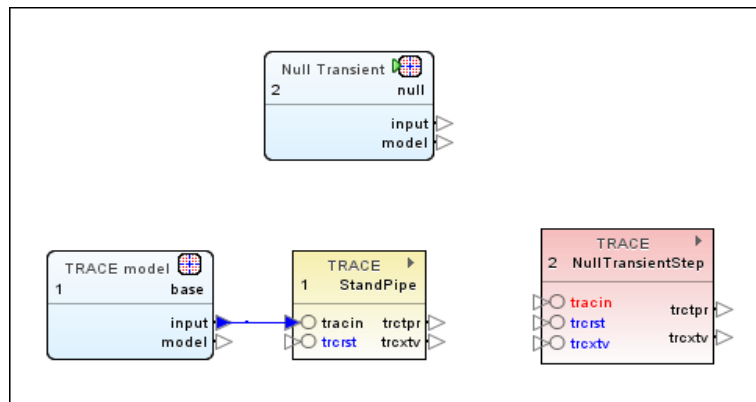
55. Press the **S** button in the **Restart Case** property editor and select **Null Transient** in the selection dialog.

*This model node now represents a specific restart, as indicated by its name changing to **Restart Case 2**. It will be used as the basis of a restart TRACE step in the job stream.*

56. Press the **OK** button to select the restart case.
57. Set the **Label** value to null.
58. Repeat the above steps to set **TRACE model 3** to use the **Transient** restart case, but set the **Label** to **trans**.
59. Create two new TRACE **Stream Steps**, as described in the previous exercise.



60. Name the first new step “**NullTransientStep**”.

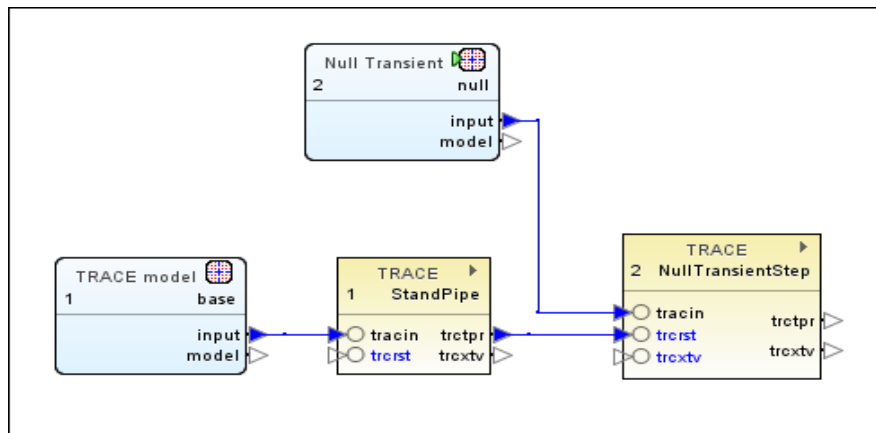


61. Drag the new **NullTransientStep** step from the Navigator into the “**StandPipeStream View**” to the right of the **StandPipe** step.

62. Drag the “Null Transient” model node from the Navigator into the “**StandPipeStream View**” just above the **StandPipe** step.

63. Using the Connection Tool, connect the **input** point of **Null Transient** to the **tracin** point of **NullTransientStep**.

64. Connect the **trctpr** output of **StandPipe** to the **trcrst** input of **NullTransientStep**.

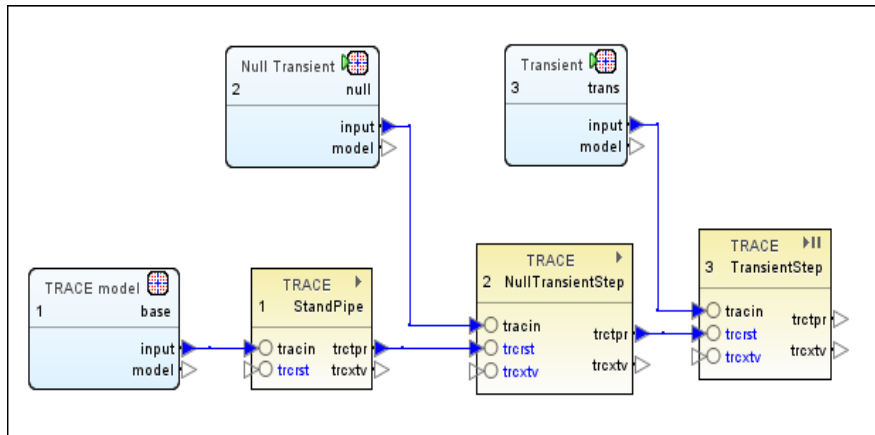


65. Set the third job step name to **TransientStep** and drag the **TransientStep** into the **SandPipeStream** view to the right of the **NullTransientStep**.

66. Drag the “Transient” model node from the Navigator into the “**StandPipeStream View**” just above the **NullTransientStep**.

67. Use the connect tool, as described above, to connect the **input** node of the **Transient** model node to the **tracin** input of the **TransientStep**.

68. Connect the **trctpr** output of the **NullTransientStep** to the **trcrst** input of the **TransientStep**.



The restart stream is now complete and should match the above image. When this stream is run, the **StandPipe** task will be run as before, using the base model. Once the **StandPipe** job is complete, the **NullTransientStep** task will run TRACE with the **NullTransient** restart case as its input and the TPR file created in the previous task as its restart input. Finally the **TransientStep** task will run TRACE using the **Transient** restart case as its input.

69. Select the **StandPipe** step and set its **Start Paused** property to **Off**.
70. Select the **NullTransientStep** step and set its **Interactive Step** to **On**.
71. Select the **TransientStep** step and set its **Interactive Step** and **Start Paused** properties to **On**.

**Start Paused** will not be available until **Interactive Step** is set to **On**.

The Transient restart case resupplies a hydraulic component but does include an Initial Condition set for defining the state conditions on the components. TRACE will report an error unless the state conditions of the resupplied component match the state conditions of the adjacent hydraulic components. The TRACE job steps supports retrieving state conditions for a restart job step from the preceding task.

72. Activate the **Retrieve Initial Conditions** property and select the **Use Last Timestep** radio button.

*Note that the Time field allows retrieving the conditions from a particular Time as well.*

73. Right-click **StandPipeStream** in the Navigator and select **Submit Stream to Local** from the pop-up menu.
74. Press **OK** to confirm the stream submission.

*The stream is submitted and Job Status will be displayed after the stream starts.*


75. In Job Status, expand **Local**, then the **runs** folder, then the **TRACE** folder, and select the **StandPipeStream** folder.

*The submitted stream steps will be displayed in the table to the right.*

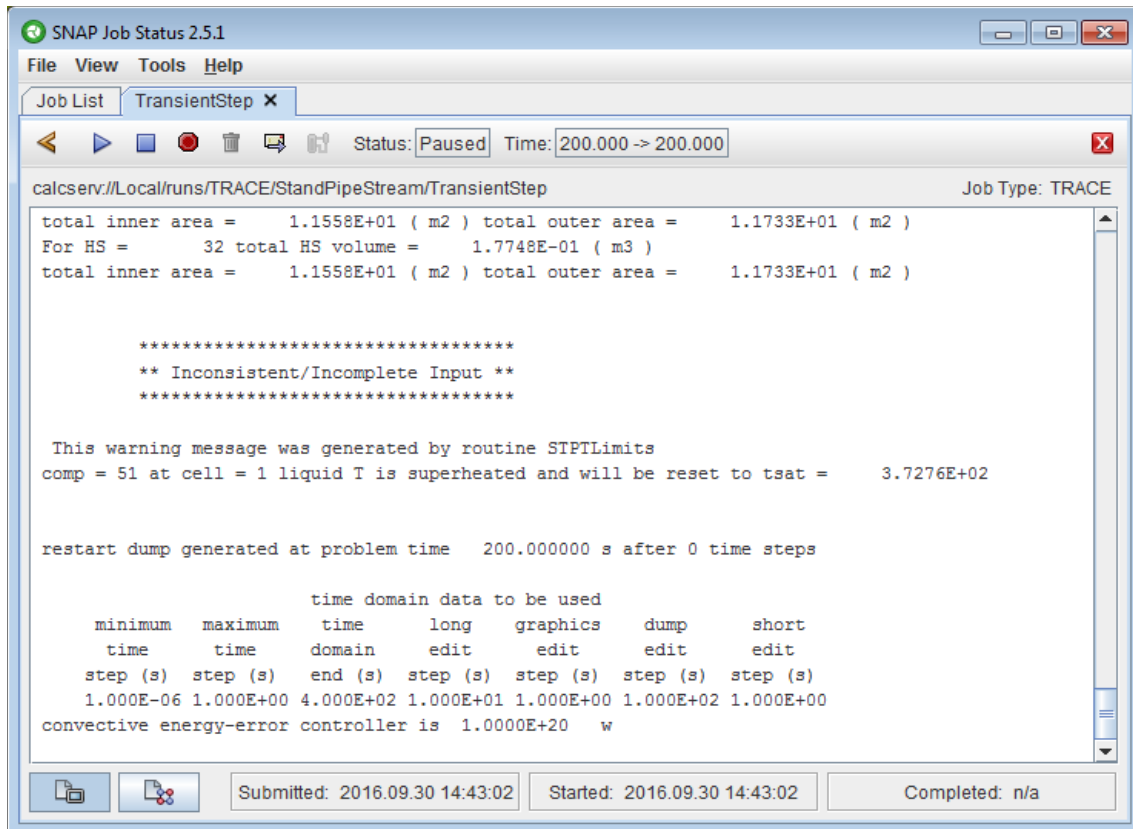
76. Wait for the **StandPipe** task to complete, then for the **TransientStep** job to initialize and switch to the **Paused** status.


*Note the distinction in terminology here. When creating a stream in the Model Editor, applications added to a stream are referred to as “steps”. When actually running the stream, these application runs in the stream are referred to as “jobs” or “tasks”.*

77. Select **TransientStep** in the table.

78. Press the **Job Console** button () in the Job List toolbar or double-click **TransientStep** in the table.

*The job console for **TransientStep** will open in a new tab, as shown below. A job console will display the console output of a task and can be used to issue commands to interactive jobs, terminate the task, open local files, etc..*



79. Press the **Play** button () in the job console toolbar and wait for the task to complete.

80. Close Job Status.

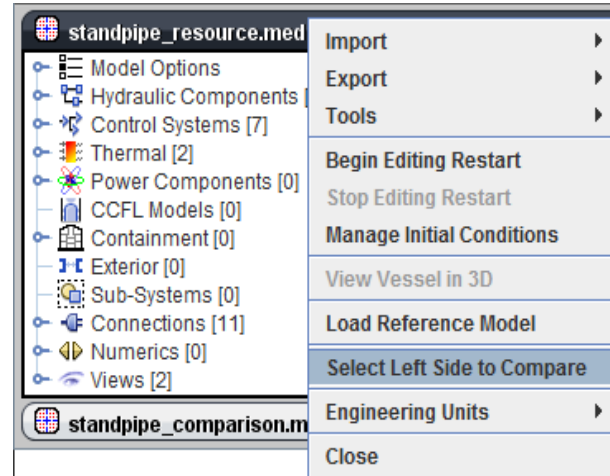
81. Select **File** → **Close All** from the main menu.

82. Press the **Discard All** button.

## Exercise 6. Model/Component Diff Viewer

This exercise will instruct the user on how to use the Model/Component differencing utility. The diff tool compares the ASCII output of model components. Comparisons can be made between components, subsystems, or models.

1. Open the sample TRACE model “**SNAP\_Exercises/StandPipe\_Resource.med**” provided with these exercises .
2. Open the sample TRACE model “**SNAP\_Exercises/StandPipe\_Comparison.med**” also provided with these exercises.
3. Right-click on the model node for **StandPipe\_Resource.med** and select the **Select Left Side to Compare** menu item.
4. Right-click on the model node for **StandPipe\_Comparison.med** and select the **Compare to unnamed** option.

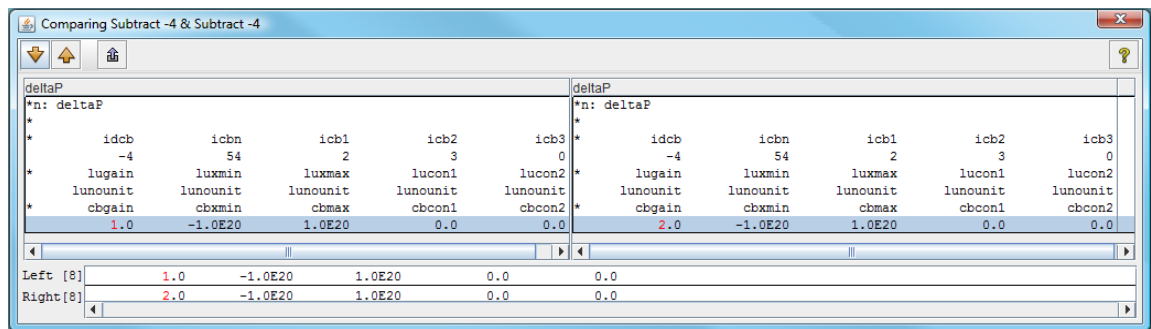


The left column includes each of the components which are contained in the selected Left model

(**StandPipe\_Resource.med**). The right column includes the components in the selected right model (**StandPipe\_Comparison.med**). The comparison column includes the status of each of the component comparisons. In the case where a model component does not exist in the other model, the comparison column will display a **Left/Right Side Only** tag in green font. Components which have no differences will be displayed as **No Differences**. The components that are different will display **Show Differences** in red. Clicking **Show Differences** will open the component comparison window.

Multiple Component Comparison		
Left	Comparison	Right
Model Options	Show Differences	Model Options
Break 21	No Differences	Break 21
Break 41	No Differences	Break 41
Fill 11	No Differences	Fill 11
Pipe 1	No Differences	Pipe 1
Pipe 2	No Differences	Pipe 2
Valve 51	Show Differences	Valve 51
	Right Side Only	Sum -5
Subtract -4 (deltaP)	Show Differences	Subtract -4 (deltaP)
Interactive Variable -3 (invel)	No Differences	Interactive Variable -3 (invel)
Interactive Variable -2 (valve...)	No Differences	Interactive Variable -2 (valv...
Interactive Variable -1 (heatf...	No Differences	Interactive Variable -1 (heat...
Problem Time 1	No Differences	Problem Time 1
Pressure 2	No Differences	Pressure 2
Pressure 3	No Differences	Pressure 3
Heat Structure 31	No Differences	Heat Structure 31
Heat Structure 32	No Differences	Heat Structure 32
Sample_Stream	No Differences	Sample_Stream

- Left-click the **Show Differences** table cell for control block **Subtract -4 (deltaP)**.  
The component comparison window is displayed, as shown below.



The **Next Difference** (⬇) and **Previous Difference** (⬆) buttons provide a way of navigating through a series of differences. As each difference is encountered, the table at the bottom of the screen updates displaying the differences encountered at that line. The data is organized such that the left model ASCII is displayed above the right model ASCII. The characters in the lines which are different are shown in a red font.

- Press the **Next Difference** button (⬇) at the top of the component comparison dialog.
- Visually identify the difference in the **cbgain** between the two components.

**Note:** Pressing the **Export Diff Output** button located at the top of the component diff viewer will export the difference to an ASCII file.



- Close the component diff viewer.
- Select the **Subtract -4 (deltaP)** control block component in the left model column of the Multiple Component Comparison dialog.
- Press the blue **Move Up** (⬆) button at the top of the dialog.

The Multiple Component Comparison window allows a user to modify the alignment of the compared components. The move up and move down buttons will take the selected component and swap it with the component in the selected direction. Once a component has been swapped to a new comparison position, a new diff is automatically generated for the compared components. This allows components of different types to be compared.

- Left-click the **Show Differences** table cell for **Subtract -4 (deltaP)** vs **Sum -5**.
- Visually identify the differences in the ASCII between the two components.
- Close the component diff viewer and exit the Multiple Component Comparison dialog.
- Switch to **StandPipe\_Resource.med** by clicking on the corresponding accordion node.
- Expand the **Hydraulic Components** category in the Navigator.
- Expand the **Pipes** category and select **Pipe 1**.
- Select **Left Side to Compare** from the pipe's right-click pop-up menu.

18. With the same pipe selected, select **Compare to Pipe 1** from the right-click pop-up menu.

*Components which are compared to themselves keep a copy of the original component ASCII in the left hand side of the component comparison dialog. When one of the components attributes change, the comparison dialog will update to display the exact effect the modification had on the component ASCII. This allows a user to easily view what changes a modification will make to the original component ASCII.*

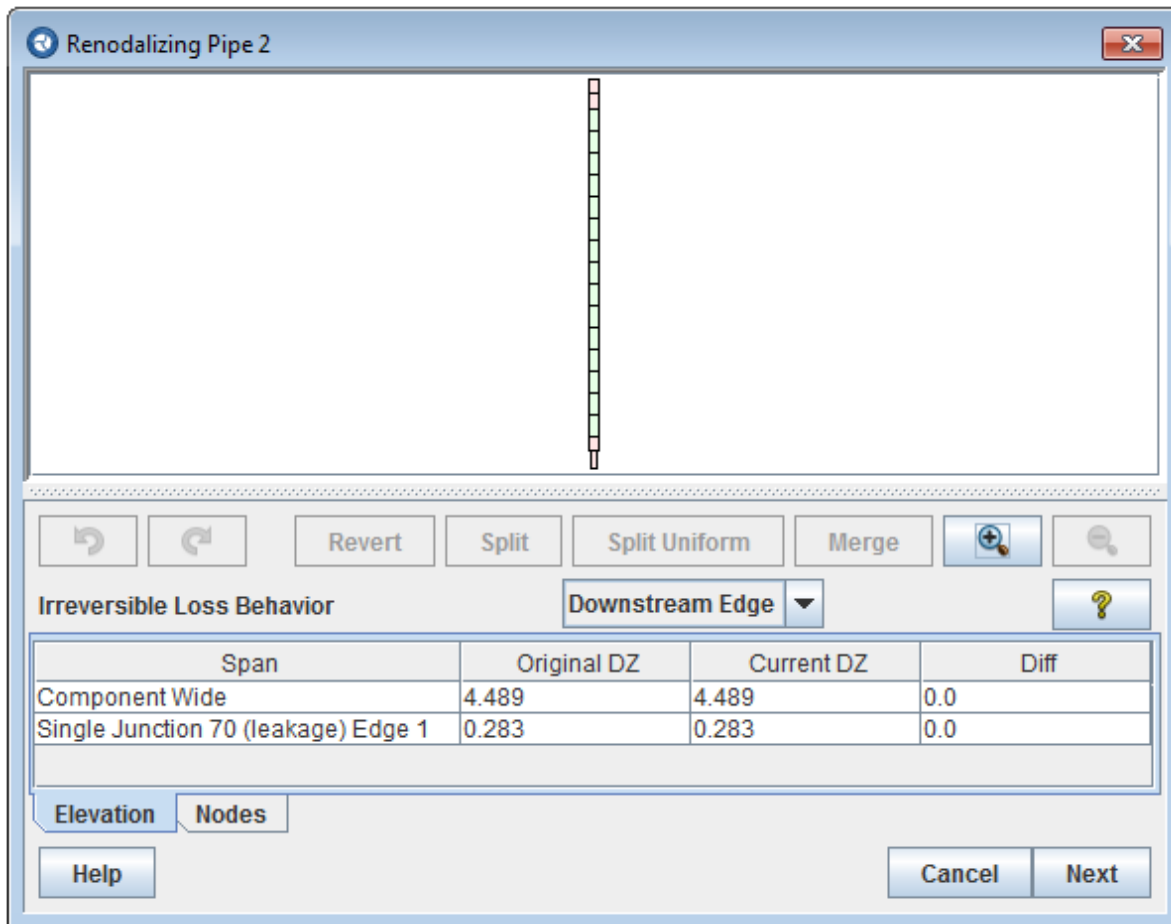
19. Select **Pipe 1** in the Navigator and open the editor associated with the **Component Geometry** attribute.
20. Change the **Length** of the first cell to “.2” ft and verify the component comparison dialog updates to reflect the modification.
21. Close the component comparison dialog and the geometry dialog.
22. Close the two models opened for this exercise.

## Exercise 7: 1D Hydraulic Renodalization

The purpose of this exercise is to introduce 1D hydraulic component renodalization in the TRACE plug-in. This will include observing the effects of nodalization changes on the rest of the simulation model.

1. Open the included **SNAP\_Exercises/SingleChannel.med** model.
2. Right-click on **Pipe 2** in the Default View and select the **Renodalization** → **Renodalize** menu item.

**Note:** This opens the 1D hydraulic renodalization wizard. This wizard allows the number and relative sizes of the volumes to be changed in a 1D component. The first panel defines the new nodalization by splitting and merging volumes. The undo-redo buttons allow stepping backward through the incremental changes as needed. It is important to note that nodalization changes are incremental. That is, if cells are merged together and then split again, values on the removed edges will be lost.



The next steps in this exercise will halve the number of nodes where the fuel heat structure is connected. The cells will be merged together into a single volume then split into 15 equally sized nodes.

3. Select the **Nodes** tab in the dialog.

4. In the table, select nodes 3 through 32.

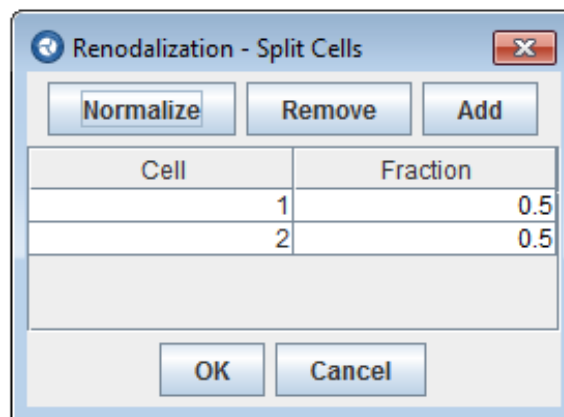
**Note:** The selected nodes are highlighted teal in the drawing at the top of the dialog. Nodes may be selected either in the table or the drawing. Multiple nodes may be selected in either the table or the view using shift-click or control-click.

5. Press the **Merge** button.

*The 8 selected cells have been merged together into a single cell. The data on the internal edges of the selected regions has been lost, with the exception of irreversible loss. The flow loss values are merged into the down-stream edge by default. How the loss coefficients are handled may be defined using the Irreversible Loss Behavior drop-down list.*

6. Select **Axial Node “3”** in the table and press the **Split** button

*This opens the Split Cells dialog. This dialog defines how the selected cell or cells should be split. The dialog shows the available fractions. Optionally, the **Split Uniform** button allows splitting the selected node into evenly spaced values.*



7. Press the **Add** button until there are 15 cells.

*Notice that the Fraction value is updated to preserve even spacing between the nodes. If the fractions do not sum to one when this dialog is closed, the values will be normalized before being applied to the target cell.*

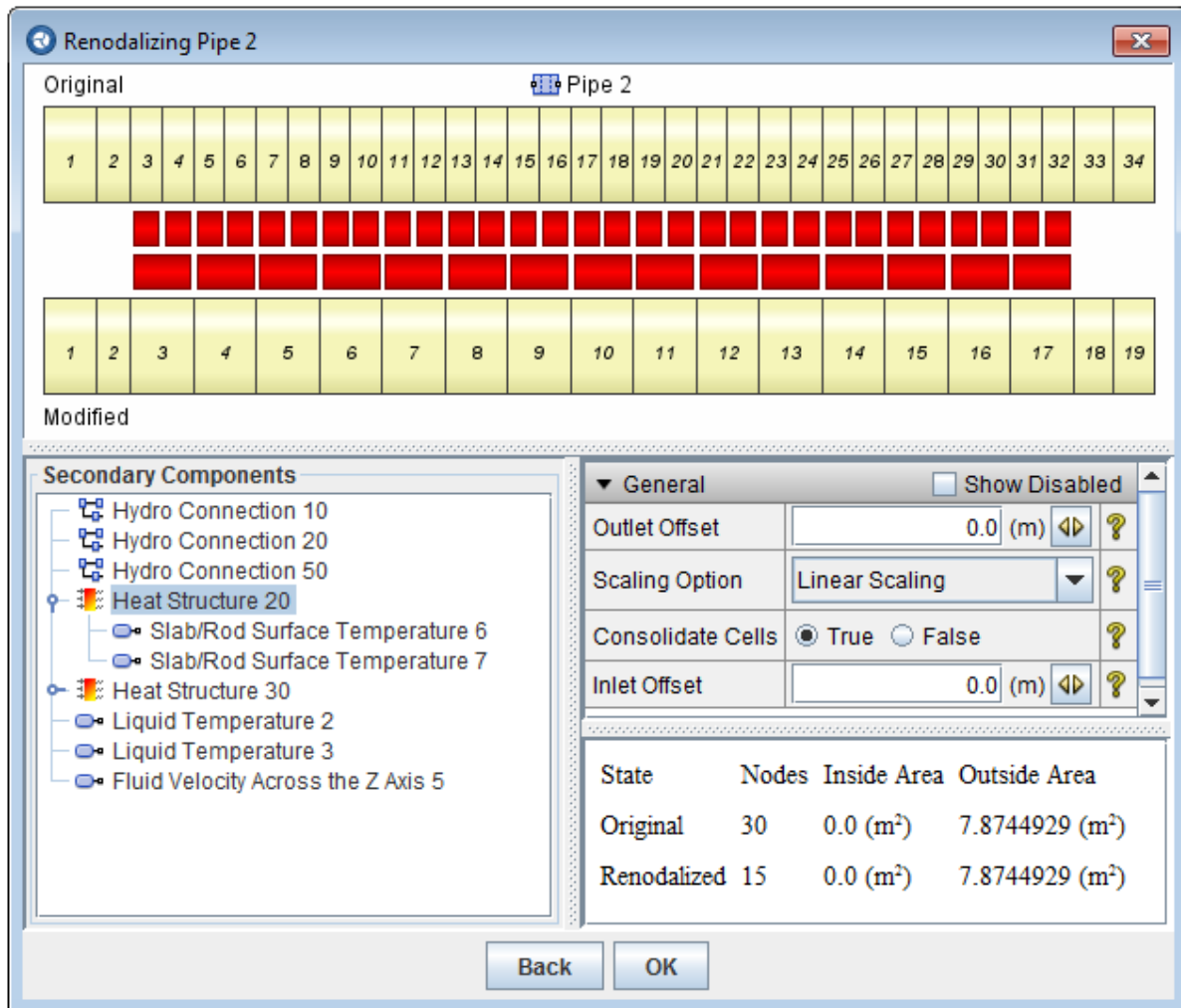
8. Press the **OK** button.

9. Press the **Next** button.

*This renodalization dialog now shows how the nodalization changes will affect other components in the model. The top of the dialog shows a before and after rendering of the 1D hydraulic component and the selected secondary component. The tree structure on the left side lists all of the components affected by the renodalization.*

10. Select **“Heat Structure 20”** in the **Secondary Components** tree.





Notice that the adjacent nodes have been merged together in modified version of the heat structure.

- Expand “Heat Structure 20” in the **Secondary Components** tree and select “**Slab/Rod Surface Temperature 6**”.

Notice that the change summary on the lower right indicates that the target locations for signal variable 6 are changed by the update. Signal variables on volumes or heat structure cells are located at the cell center. Renodalization can change where along the axial length of a component the signal is obtained.

- Select **Heat Structure 20** in the **Secondary Components** tree.
- Set the **Consolidate Cells** option to False.

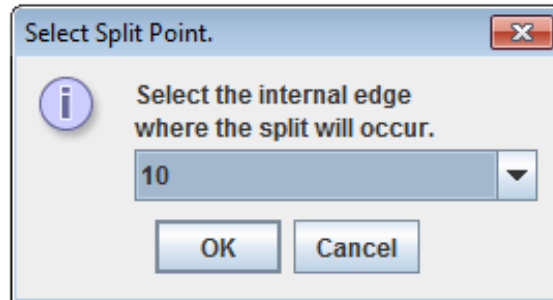
**Note:** The consolidate cells option tells the system to merge together adjacent heat structure cells with identical left and right boundary condition data.

- Press the **OK** button.
- Press the **Notes** toggle button to hide the informational messages.

*The only message displayed is a warning message indicating that a signal variable source location has been modified. Notice the location values are displayed in red.*

16. Press the **Close** button.

17. Right-click Pipe 10 in the view and select the **Renodalization** → **Split** menu item



18. Select internal edge **11** where the split should occur and press the **OK** button.

*Pipe 10 has been split into two 10 cell pipes, with the outlet half being renumbered to pipe 80. Notice that heat structure 30 connects to both pipe 10 and pipe 80. An error would be reported if a signal variable has volume or edge location references that are on both sides of the pipe split location.*

19. Select the **Standard Stream** view and unlock the view.

20. Select the restart model node.

21. Set the **Restart Case** property to “Split”.

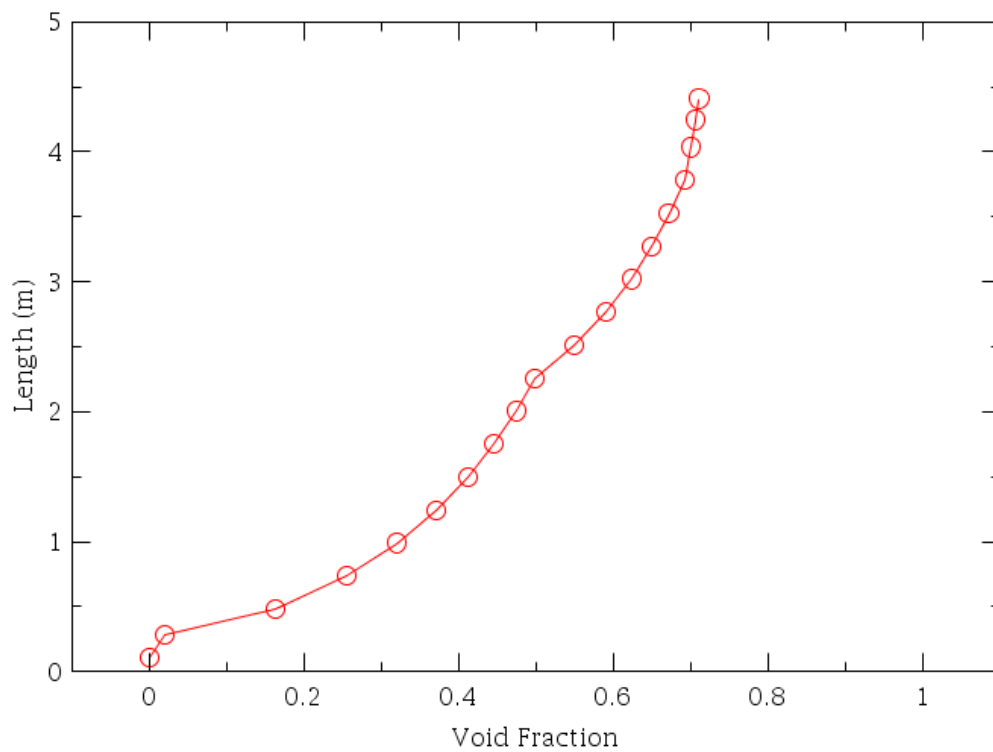
**Note:** This restart case is a transient case that includes Pipe 80 in the iorder array. This is required to restart the renodalized model. The Short restart case would suffice if the only change was the Pipe 2 renodalization. Restart cases must be taken into consideration when performing renodalization, as existing restart cases will not be automatically updated.

22. Lock the view, press the **Execute** button, then wait for the calculation to complete.

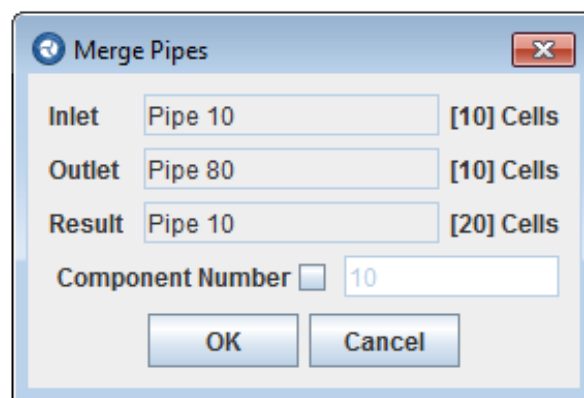
23. Right-click the **Plots** step and view the “void\_png” Image output file.

The next steps will work backwards to revert the nodalization to the previous state.

## Void Fraction



24. Right-click Pipe 10 in the view and select the **Renodalization** → **Select for Merge** menu item.
25. Right-click Pipe 80 in the view and select the **Renodalization** → **Merge with Pipe 10** menu item.

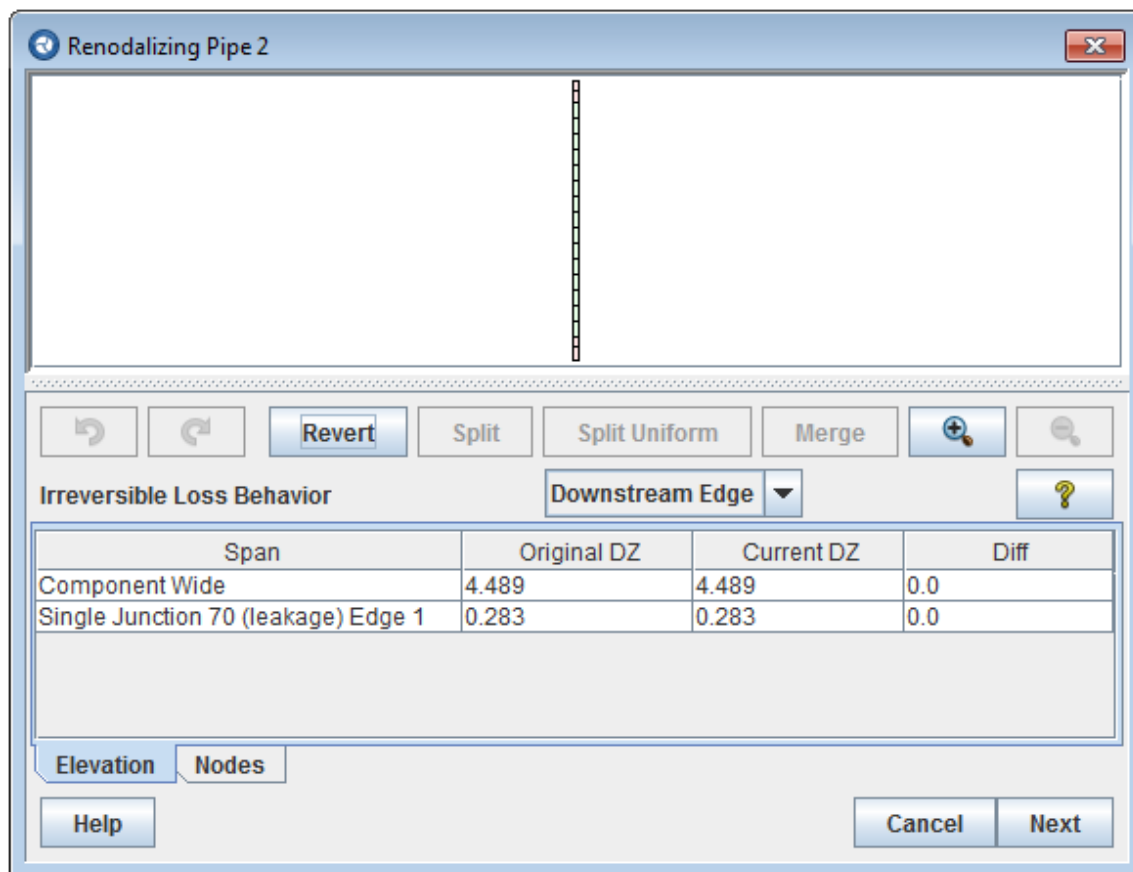


26. Press the **OK** button in the Merge Pipes dialog.

The next steps will use the renodalization wizard to revert the nodalization of a 1D hydraulic component to match the values entered in a different model. This is done by selecting a model as a “reference model” and then selecting the source of the geometry from the renodalization dialog.

27. Select the **File** → **Save As** menu item and save the file to a new file location.
28. Right-click the **Model Node** in the Navigator and select the **Load Reference Model** menu item.
29. Select the **SNAP\_Exercises/SingleChannel.med** model included with these exercises and press the **Open** button.
30. Right-click Pipe 2 in the view and select the **Renodalization** → **Renodalize** menu item.

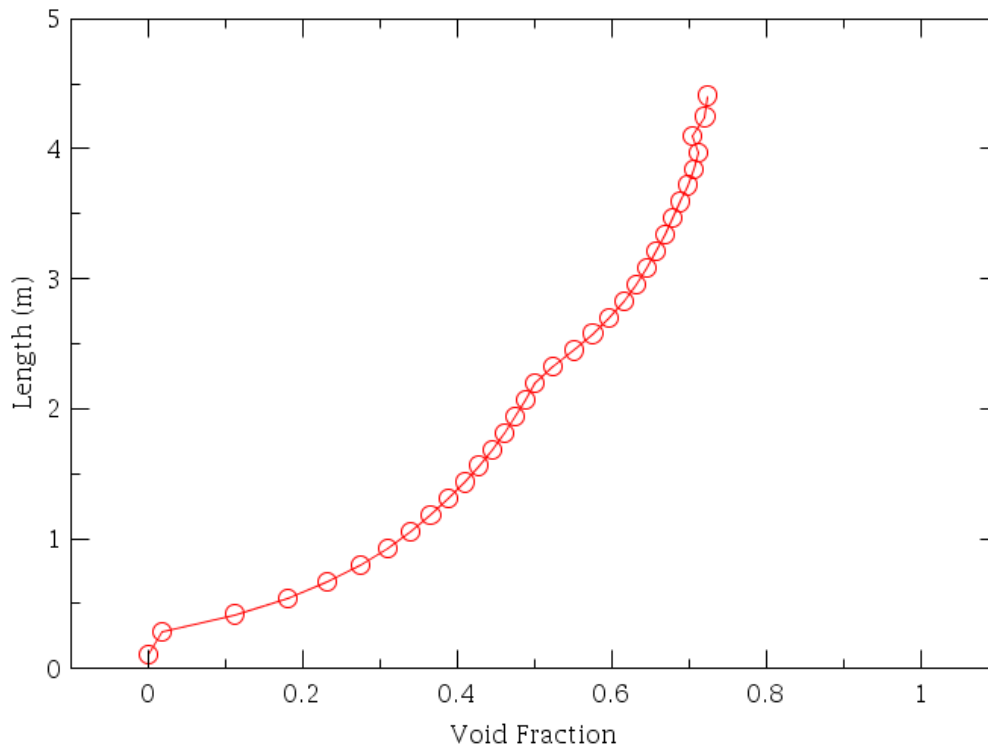
*Notice that the revert button is enabled.*



31. Press the **Revert** button.
32. Select **Pipe 2** in the selection dialog and press the **OK** button.
33. Select “**all cell and junction data**” from the confirmation dialog and press the **OK** button.
34. Press the **Next** button.
35. Press the **OK** button.

36. Change the restart model node in the Standard Stream back to the “Short” restart case.
37. Submit the Standard Stream model again and wait for the jobs to complete.
38. View the “void\_png” Image output from the plot exercise.

## Void Fraction



This completes the 1D hydraulic renodalization exercise.

## Exercise 8. Working with View Templates

This exercise is designed to familiarize the user with importing existing models and View Templates. View Templates provide a means of saving a layout of a view that may then be applied to a similar model.

In this exercise the user will use a sample View Template to lay out an imported model.

1. Select the **"File → Import → TRACE"** main menu item.
2. Set the **File Type** option on the right to **"ASCII"**.
3. Select **SNAP\_Exercises/standpipe\_viewtemplate.inp**, included with these exercises.
4. At the **Create Views** prompt, uncheck all check-boxes and press the **OK** button.

*The model will be imported without views.*

5. Create a new empty View by right-clicking on the **Views** node in the Navigator and selecting the **New** menu item.

*This should create and open a new View with the name "View 1" and display its properties in the Main Property View.*

6. Right-click anywhere **inside** the newly created View.

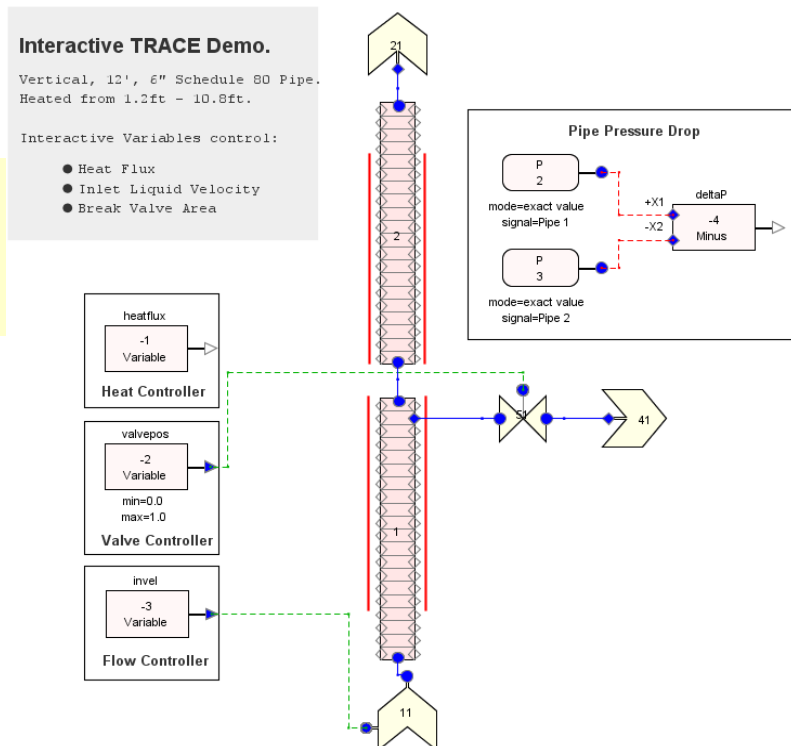
*This will open the 2D View pop-up menu with items such as Organize and Zoom.*

7. Select **"Tools → Import View Template"** in the pop-up.

8. Select **SNAP\_Exercises/standpipe.mvt**, included with these exercises, and press **Open**.

*This will apply the template to the view, which should now appear as shown here.*

**Note:** To export a View Template of an existing view, use the "Tools → Export View Template" item off the right-click pop-up menu in the View.

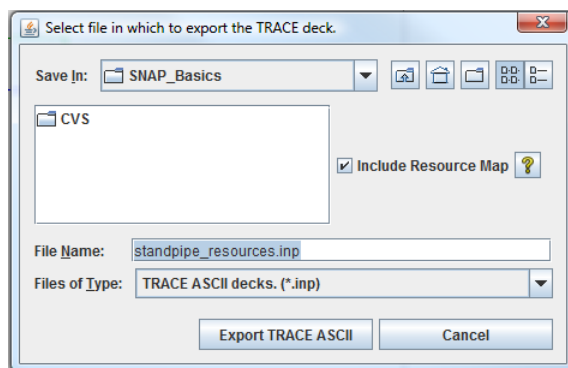


## Exercise 9. Resource Bundle Import/Export

The TRACE plug-in provides a series of features which facilitate the editing of input models. This exercise will familiarize the user with using the Resource Bundle feature to quickly modify a set of components outside of the Model Editor as ASCII and re-import the model without losing model data such as views, documentation, or SNAP variables.

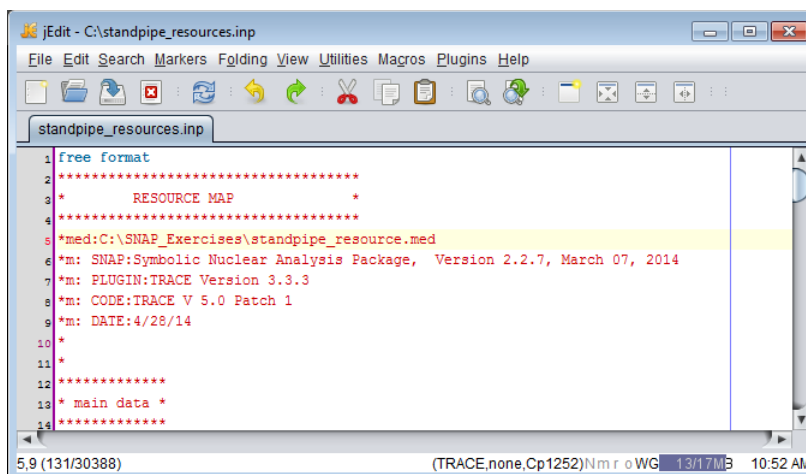
1. Select **File** → **Open** from the Model Editor's main menu.
2. Select the TRACE model **SNAP\_Exercises\StandPipe\_Resource.med**, included with these exercises.
3. Export the sample model as a resource bundle by selecting **File** → **Export** → **ASCII Full** from the main menu.
4. Select the **Include Resource Map** check-box in the Export File selector.

*Note that the TRACE plug-in will prompt the user to save the current model before the resource export completes if the model has not yet been saved.*

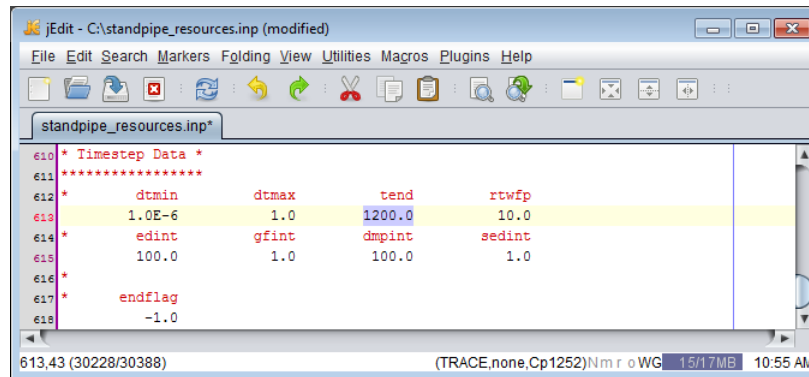


5. Export the resource model as: **“standpipe\_resources.inp”**
6. Using jEdit, locate and open the exported resource model.

*Notice that the file includes a section at the top for a numerics map. This allows SNAP variables to be reconstituted from ASCII resource import.*



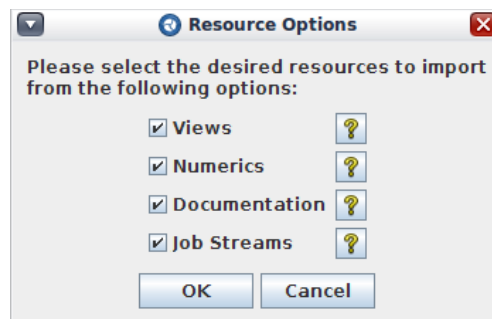
7. Locate the Timestep Data section located at the end of the input deck.



8. Change the end time (**tend**) to “1200.0” and save the model.
9. Close jEdit.
10. In the model editor select **File** → **Import** → **TRACE** from the main menu and choose the file “**standpipe\_resources.inp**” exported previously.
11. Press the **Yes** option when the plug-in asks whether to import the resource file.

**Note:** The resource import allows the user to specify which resources to include from the resource file. This includes views, SNAP variables, and model documentation.

12. Verify that each of the available options are selected in the dialog and press the **OK** button.

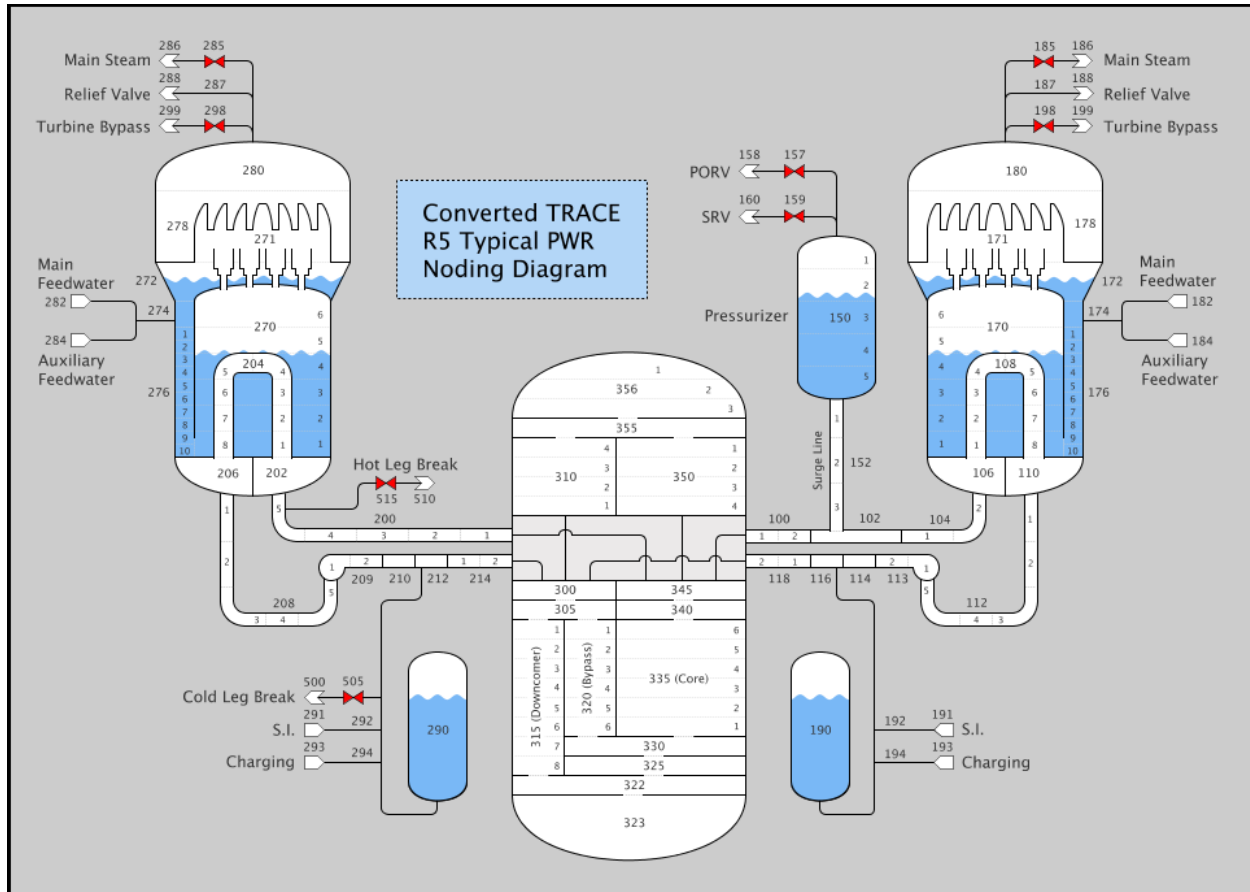


13. Right-click the **Views** category and select **Open Views** from the pop-up menu.  
*To speed the resource import process, imported 2D Views are not automatically opened.*
14. Select the **Model Options** node in the Navigator.
15. Locate the **Timestep Data** property and open the associated editor.
16. Verify the **End Time** was updated to **1200.0** seconds and close the dialog.
17. Select **File** → **Close All** from the main menu.
18. Press the **Discard All** button.




## Exercise 10. 2D Drawing Features

The goal of this exercise is to demonstrate some of the features used to create larger and more complex nodalization views with the Model Editor. This process begins with Layers, to illustrate some of the ways layers can be used to organize complex views. Afterward, the drawing capabilities of the Polygon bean are used to complete the illustration of the broken loop.



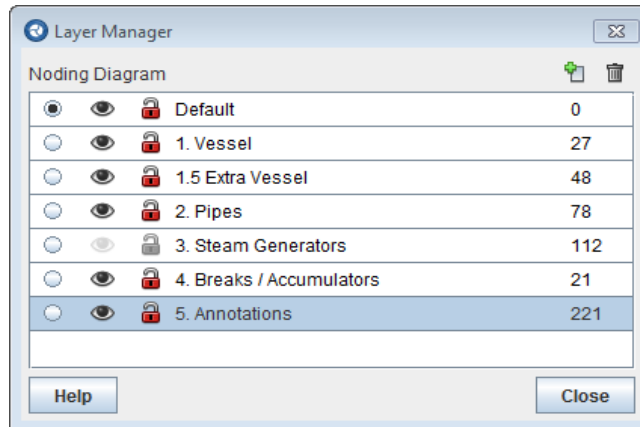
1. Open the file "**SNAP\_Exercises/TraceTypPWR.med**" included with this exercise.
2. Open the **Noding Diagram** view by right-clicking the view's node in the Navigator and selecting the **Open** item.

*This model opens with a single 2D View visible: a free form "nodalization view" of the full plant model. In appearance, it is similar to the full plant view in the Typical PWR animation mask. There are, however, a few significant differences that will be described later.*

3. Open the Layer Manager by pressing the **Layer Manager** button () on the view toolbar.

**Note:** Layers provide a means of organizing the elements in a view. They can be locked and hidden independently. Locking a layer prevents elements from being clicked on or selected while the view is unlocked. Hiding a layer hides the elements in the layer from view.

The Layer Manager is used to create, hide, and lock layers as well as to set the current layer. All elements added to a View are added to the current layer.



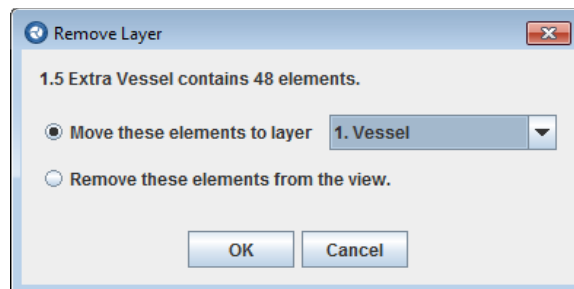
Layers are listed in alphabetical order following the **Default** layer, which appears first. The Default layer is always available and contains all of the elements in the view that have not been moved to another layer.

- Click the **Hide/Show** button (👁) to the left of Steam Generators layer.

Notice that the secondary side components appear in the view when this layer is made visible. Hiding components in a layer is conceptual (primary vs. secondary side) and practical. Working with stacks of elements (polygons in this case) is easier when the elements below or above can be hidden. In this case, the secondary side of the steam generators can be hidden to make working with the u-tubes easier, or vice-versa.

Layers may also be locked, which prevents elements in that layer from being selected. Layers may be locked or unlocked by pressing the Lock (🔒) button or the Unlock button (🔓).

- Select the **1.5 Extra Vessel** layer by left-clicking on the layer name in the Layer Manager.
- Remove the layer by pressing the **Remove** button (🗑) at the top of the dialog.



*In this case a window is displayed asking what to do with the contents of the layer being removed. The options are to either move the contents into another layer or to remove the elements. A drop-down to the right has the list of layers the elements can be moved into. The initial choice is to move the elements into the **Default** layer.*

7. Expand the Drop-down list and select **1. Vessel** from the list of available target layers.
8. Press **OK** to move the contents of the **Extra Vessel** layer into the **Vessel** layer.

*To the right of each layer name in the Layer Manager is the number of elements in that layer. Notice that the **Vessel** layer had 27 elements and now has 75.*

9. Create a new layer by pressing the **New Layer** button (  ) at the top of the dialog.

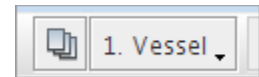
*This will be the layer for the new parts created for this exercise, but first, it will need an appropriate name.*

10. Rename the new layer to "**New Parts**" by double-clicking on the name **Layer 1** and entering the new name.

*That gives it an appropriate name, but unless it is the Current layer the new parts won't be automatically added to it.*

11. Set the **Vessel** layer as the current layer by pressing the radio button to the left of its **Hide/Show** button.

*Notice that **Vessel** now appears as the current layer displayed in the Layer Selector on the view toolbar to the right of the **Layer Manager** button. The Layer Selector can be used to set the Current layer directly, without opening the Layer Manager.*



12. Set the newly created **New Parts** as the current layer by left-clicking on the Layer Selector and selecting "**New Parts**" from the list.

*Also note that the list of layers also included a **New** entry. This is a quick way to create a new layer and set it as the Current layer.*

**Note:** Additional information about Layers and the Layer Manager is available in the SNAP User's Manual by pressing the **Help** button at the bottom of the Layer Manager window.

That completes the Layers section. Next, we'll look at the Associated Component feature. Then, we'll create a few polygon beans to replace those that are missing in the view.

13. Zoom in on the broken loop (left) steam generator using either Ctrl + Mouse Wheel or the Zoom Tool.


- This will select the polygon that was added to represent pipe 204 in the view.*

- 

- Nothing in the view will change to indicate where it was clicked, however, the main Properties View will show the properties of **Pipe 204 (bsgtbs)** and the Navigator will scroll to and select **Pipe 204**.

- Now that we see how drawing elements can be associated with components in the model, lets take a look at how to create some of the shapes required to build a detailed nodalization view.

- Notice that the broken loop accumulator (290) does not appear to have any water in it while the intact loop accumulator (190) does. This is a purely aesthetic difference, as the nodalization view does not directly reflect the conditions of the accumulator. However, it does make the drawing more distinct.*

23. Activate the **Polygon** insertion tool () by selecting it from the **Insert Menu** to the right of the Connection Tool on the view toolbar.

- Follow the points in the given diagram. If you click in the wrong place, right-click to remove the last point.

- Notice that each point will try to form a straight line from the previous point.*

-

*You'll notice that the existing accumulator's boundaries are slightly thicker than the new polygon. We'll fix that first so that the lines will be easier to match up.*

27. In the Properties View, set the **Outline Thickness** property to "2".
28. Set the Polygon's **Fill Color** property to a light blue, "121,184,246".

*Now, it's time to round off the bottom to match the accumulator.*

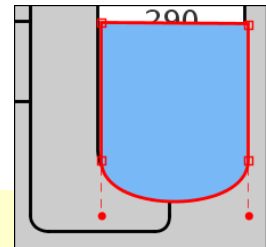
29. Right-click the bottom edge of the newly created polygon.

*This will display the Polygon pop-up menu. At the bottom of this menu are several items specific to the Polygon.*

30. Select the **Curve Segment** item from the pop-up menu.

*This will change the straight segment into a curved segment. Curved segments have a start and end point (like straight segments) but they also have two control points. These control points are used to control the curve of the segment.*

**Note:** The Curve Segment item will only be enabled when right-clicking exactly on the line segment. If you miss, just click away to close the pop-up menu and try again.



31. Drag the left control point straight down from the bottom left point to the location shown.

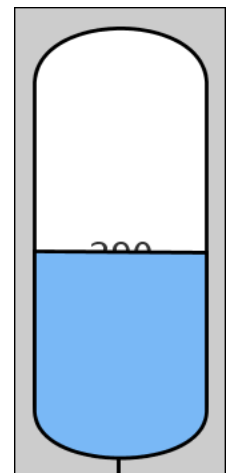
*The control point locations don't have to be perfect. They can always be corrected later if the shape doesn't look quite right.*

32. Drag the right control point straight down even with the left.

*This will add a nice rounded shape to the bottom of the accumulator.*

33. Use the arrow keys to move the new polygon directly on top of the original.

*The new polygon gives a sufficient hint that there is liquid in the accumulator, but it could be better. It more resembles a pill than an accumulator. It would look better without the black line between the new and original polygons and maybe a wavy top to the liquid.*



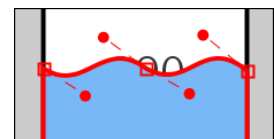
34. Right-click on the center of the top segment and select the **Add Point** item to add a point to the top of the new polygon.

*To get a good wavy look the line will need a center point for the two pairs of curves. But first, the top segment lines need to be hidden.*

35. Right-click on each of the two top segments and select **Curve Segment** to curve the new segments.

36. Drag the left control point for each segment down and to the right.

37. Drag the right control point for each segment up and to the left.



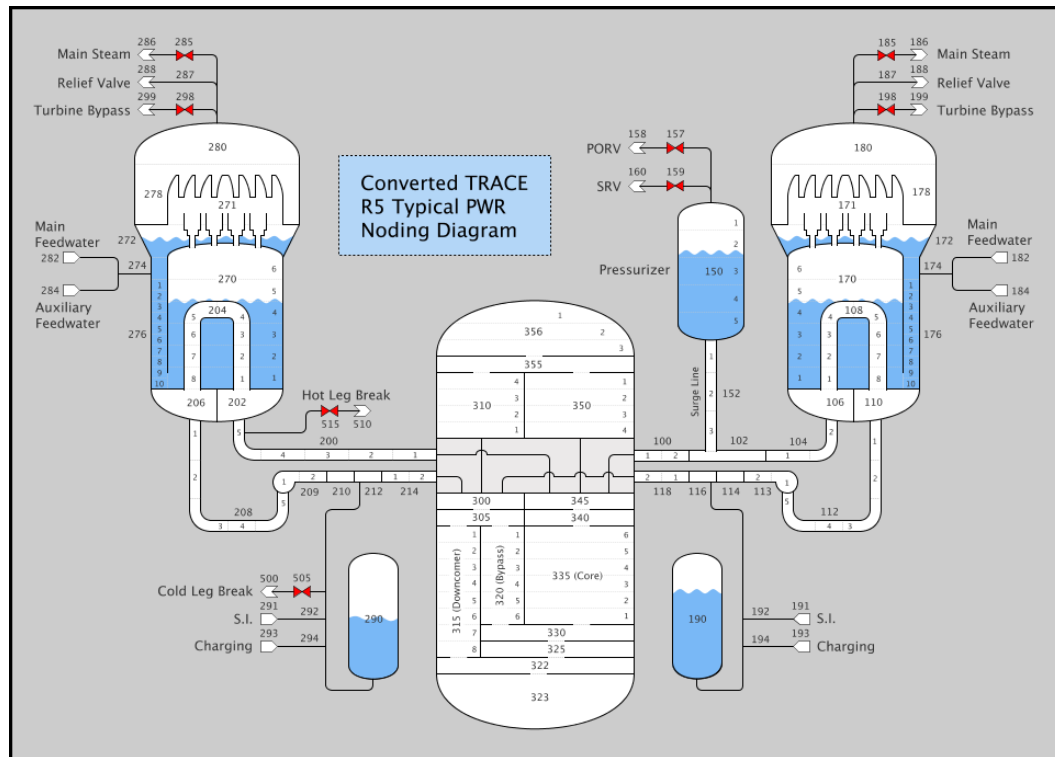
*That takes care of the wavy appearance of the top of the liquid. More ripples can be added by adding more points.*

38. Right-click on each of the top segments and select **Hide Segment** to remove the outline from the new segments.

*In this particular example the accumulator number (290) has been hidden by the water. As a final step, we'll pull that to the foreground.*

39. Right-click the accumulator **290** text annotation and select the **To Front** item from the pop-up menu.

That completes the Intact Loop accumulator.



## Exercise 11. Using AptPlot and the ACS Plug-in

This exercise is designed to familiarize the analyst with the basic functionality of AptPlot and the ACS plug-in. A TRACE plot-file will serve as a source of data for several graphs on a plot. In addition, some introductory graph and data set formatting will be performed.

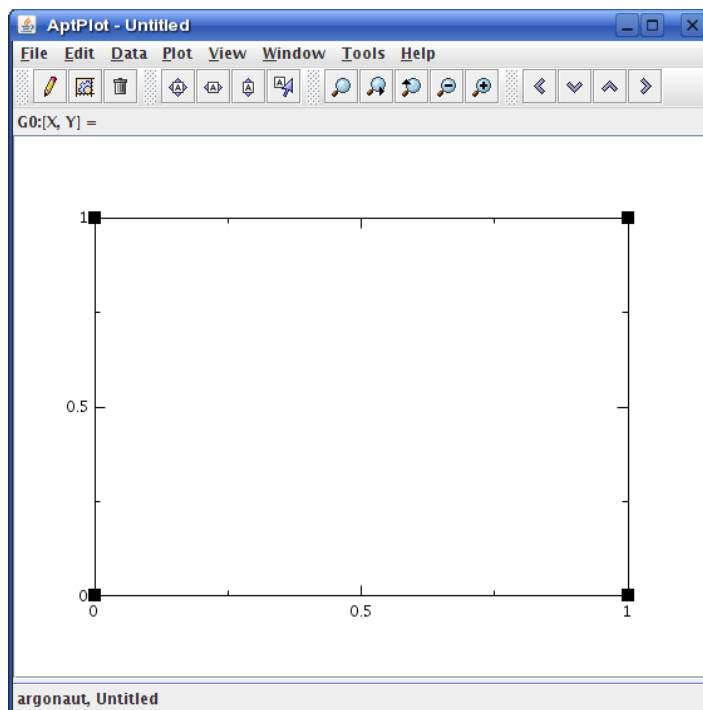
The following steps will guide the user through the exercise.

1. Open AptPlot. From the Windows Start menu, select **All Programs** → **Plotting Tools** → **AptPlot**.

*This will open AptPlot with a new plot, as shown below. From here, data will be added to the plot from a TRACE plot file.*

*First, a quick rundown of the window. The white area is the plot. Anything appearing within this “canvas” is part of the plot and will appear in print-outs and exported images. The rectangle within the plot is a graph. The data itself is represented by Data Sets: linked columns of data contained within a graph. From the bottom up: data sets form lines on the graph, graphs create a window into their data, and plots are the root container for everything.*

*The new plot in the window contains a single graph without any data sets. The first several steps of this exercise will add a data set to the graph by reading values from a TRACE plot file.*



2. From the AptPlot main menu, select **File** → **Read** → **TRACE data....**

*A file selection window is displayed.*

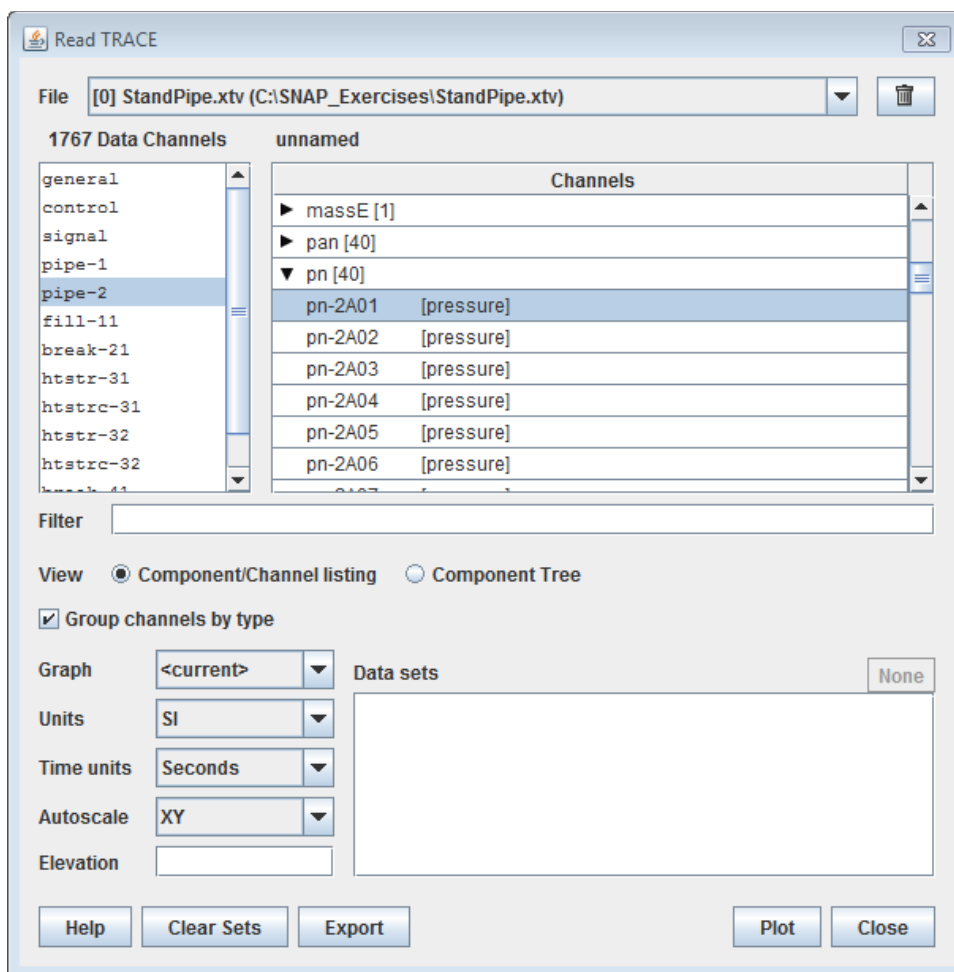
3. In the file dialog, select **Next Available** for the **File Index** and **XTV** for the **File Type**.

*The file type tells AptPlot how to attempt to read the selected file. Most TRACE plot files will use XTV, even if the file has been demultiplexed. A demultiplexed file is a version of the plot file reorganized for more efficient plotting. Demultiplexing is explained in detail in the AptPlot documentation.*

*As for the file index: each code type supported by the ACS plug-in allows opening up to 30 plot files at any given time. **File index** specifies which of those slots is assigned to the selected file. With **Next**, the file is assigned to the next unused plot index.*

4. In the file dialog, navigate to the **SNAP\_Exercises** folder provided with this exercise.
5. Select the file **StandPipe.xtv**.
6. Press the **Open** button.

*A channel selection window will appear, similar to the image below, which will be used to read the channel data into the plot.*



7. Select **pipe-2** in the list of categories of the left.



The list of channels will change to those associated with the pipe-2 component.

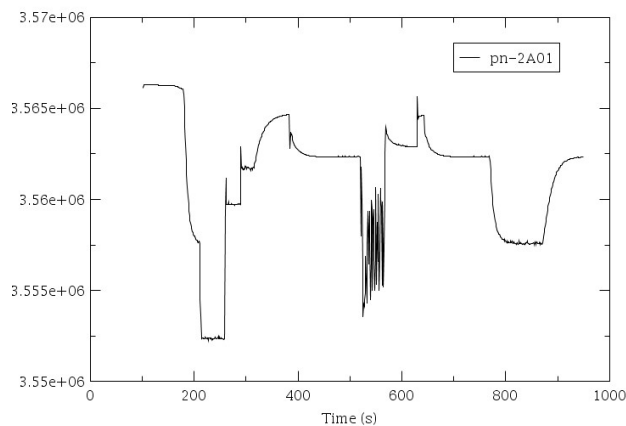
8. Type “**pn-\***” (sans quotes) into the **Filter** field and press the **Enter** key.

The list of channels will be filtered to only those channels whose names begin with “**pn-**”. The \* character will match any string of letters (including none). “**pn-2\*1\***” would match any channel name that starts with “**pn-2**” and ends with “**1**”.

9. Select **pn-2A01** from the list of data channels and press the **Plot** button.

The channel data will be plotted in the graph, as shown below. Notice that the graph's displayed axis values change to fit the data. Each graph has a set of World Coordinates that define which the range of values it displays. To illustrate, zooming out on a graph widens the world coordinate window.

Unfortunately, this graph's numeric axis labels reach to the edge of the plot with the default formatting. While it is not currently visible, a vertical unit label “**Pressure (Pa)**” has been added to the Y axis: the long axis labels have pushed it off the plot. The next several steps will re-size the graph until this label is visible.



10. Close the channel dialog.
11. Double-click and hold on the upper-left corner of the graph, where the small black square is placed.

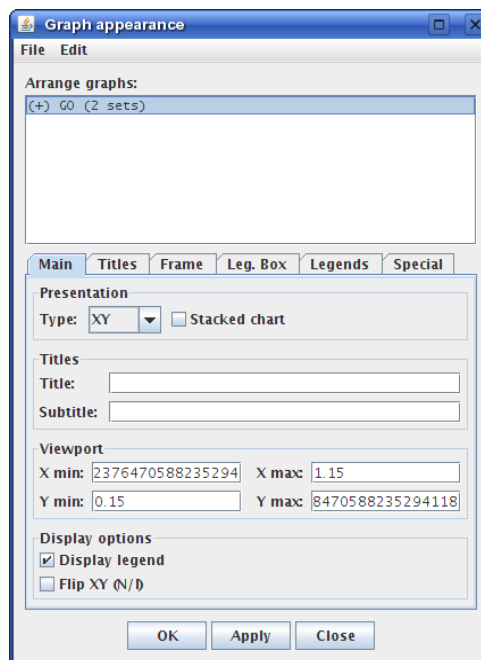
This will initiate a graph re-size operation, indicated by the red “rubber-bands” displayed on the plot. By moving the mouse around, the rubber-bands indicate the new bounds of the plot. Clicking once will place the corner in its new location and re-size the graph.

12. Make the graph a little smaller by placing the corner slightly to the right of its original location.
13. Repeat the previous two steps until the “**Pressure (Pa)**” label is completely visible.

The current graph displays large values in a relatively narrow range. Further into the exercise, a channel will be plotted with very small values. If the new data is plotted alongside the current data, the graph will appear as two straight lines with a wide margin between them. To avoid this, the new data will be placed on a second graph.

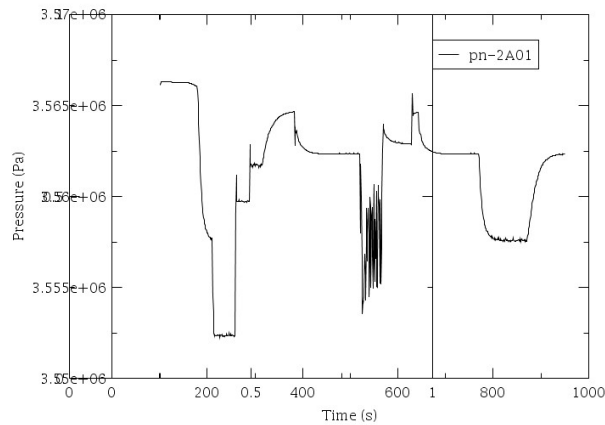
14. From the main-menu, select **Plot** → **Graph appearance....**

The **Graph Appearance** window is displayed. The top of the Graph Appearance dialog is a graph list. Selecting a graph in the list determines which graph's appearance is being edited below. This type of graph list is used frequently throughout AptPlot, and each one provides identical functionality. This particular dialog was opened specifically to create a new graph from the graph list. In practice, any AptPlot dialog with a graph list could have been used.



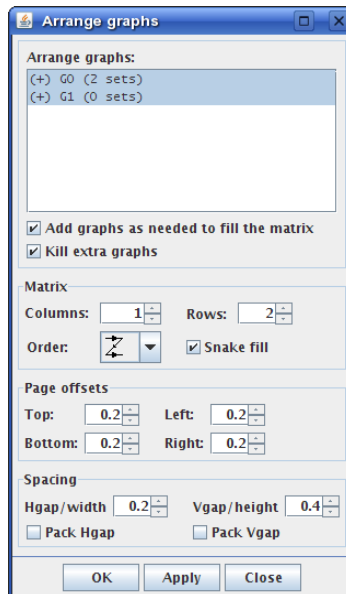
15. Right-click anywhere within the graph-list and select **Create new** from the resulting pop-up menu.

A new graph is created. The plot should look similar to the one below. The new graph overlaps a significant portion of the original graph. This could be fixed by moving the graph corners around, but this would ignore several options AptPlot makes available for quickly laying out plots.



16. From the main-menu, select **Edit** → **Arrange graphs....**

*The **Arrange Graphs** window will be shown. With this dialog, graphs can be arranged in a grid of any dimensions, automatically assigning each the same size, vertical and horizontal spacing, and overall margin from the edges of the plot. Additionally, the dialog can create and remove graphs as needed to fill the grid.*



17. Select both graphs in the list. To select both graphs, select one, press and hold the Shift key, then select the other.

*Both graphs must be selected to indicate they are the graphs being arranged. If only one graph is selected, AptPlot would create a third graph, as the parameters entered in the next step tell AptPlot to create a graph grid with two rows in one column.*

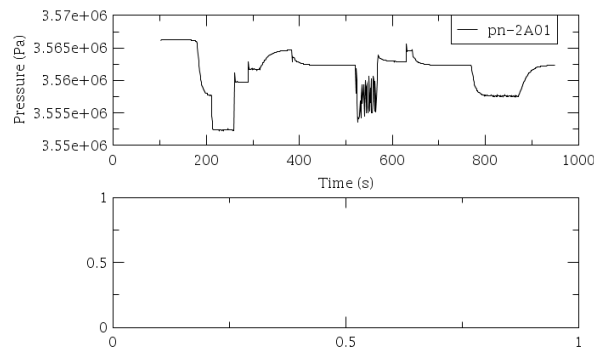
18. Set the following properties in the Arrange Graph window to the indicated values:

Matrix	Columns:	1
	Rows:	2
Page offsets	Top:	0.2
	Bottom:	0.2
	Left:	0.2
	Right:	0.2
Spacing	Hgap/width:	0.2
	Vgap/height:	0.4

**Note:** The offsets and spacing above are specified in View Coordinates. A quick explanation: AptPlot creates a coordinate space for graphs that is based on ratios. The shortest side of the plot is 1 viewpoint long; the longer side's length is a ratio to the shorter side. This coordinate space is used to retain relative dimensions regardless of how the plot dimensions are changed. For example, if a user doubles the width of a plot, a square graph remains a square instead of transforming into a rectangle. If both dimensions of that same plot are doubled, the graph expands to correctly fill the space, retaining its original relative dimensions within the plot. The viewpoint coordinate system is the backbone of AptPlot's ability to create plots of any resolution that still appear identical to what's displayed on screen.

19. Press the **OK** button.

*The **Arrange Graphs** window disappears and the graphs are rearranged. The next several steps will add data to the second graph.*



20. From the main menu, select **Edit** → **Edit plug-in data** → **TRACE data....**

*The channel dialog will reappear. Once a plot file has been opened, its data can always be accessed from the **Edit** menu (unless another file is opened in its file index).*

21. Select **rovn-2A01** in the channel list.

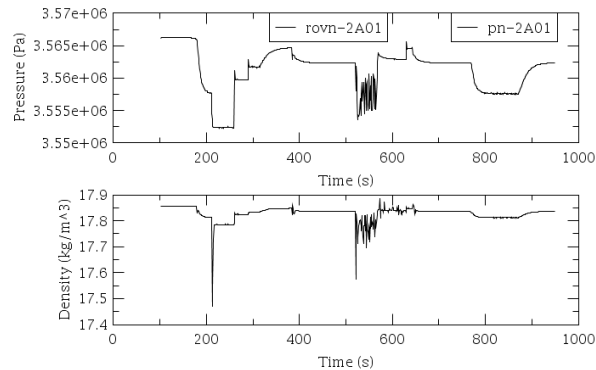
*Remember that a channel filter can prune the list.*

22. Select “**(+) G1**” from the **Graph** drop-down at the bottom of the window.

*This selects the graph into which the data will be read.*

23. Press **Plot** to display the channel data, then close the dialog.

*At this point, the plot should have two graphs with data, axis labels, and legends. Notice that the second graph's legend is placed on the first graph. The next several steps will move the legend to a better location.*



24. Click once anywhere on the second graph to select it.

*Many operations in AptPlot depend on the graph selection to determine which graph is affected. Moving the legend in the next step is one of them.*

25. Press **Ctrl+L**.

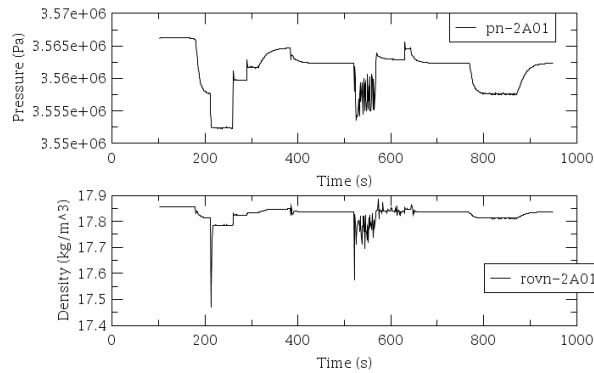
*AptPlot will enter “Move Legend” mode in the plot. This is visually indicated by a change in the mouse cursor.*

26. Click and hold on the legend containing the entry **rovn-2A01**.

*This will begin an operation to move the legend, indicated by a shaded preview of the new location. Moving the mouse will move the preview indicator. Clicking once will set the legend in its new location. The move can be canceled by pressing the escape key or right-clicking.*

27. Reposition the preview indicator to a location on the right-side of the second graph, then click once.

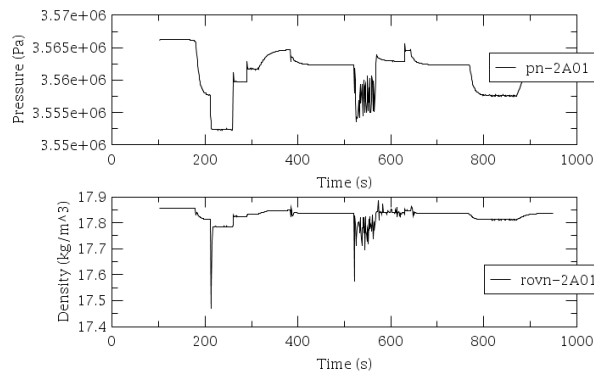
*The legend will be placed in the new location. The plot should look similar to the image below.*



28. Press the Escape key or right-click.

*While this exercise moved the legend with the mouse, its coordinates can also be explicitly specified by editing the graph's appearance.*

29. Repeat the last five steps to reposition the first graph's legend. When complete, the plot should resemble the following:

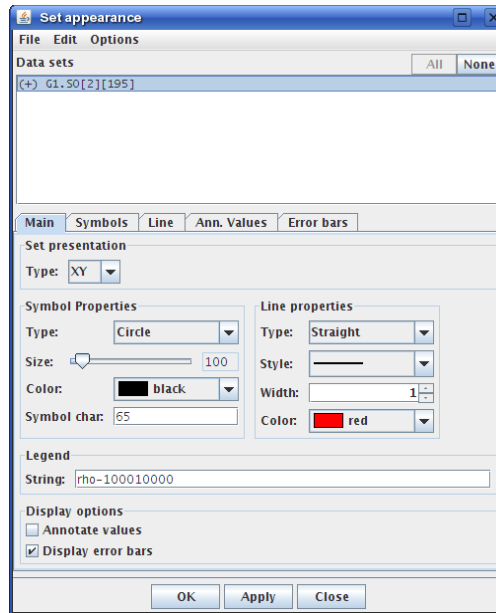


*The final steps of this exercise will modify the appearance of the data set in the second graph.*

30. Double-click anywhere within the second graph.

*The **Set Appearance** window shown below is displayed. Double-clicking within a graph opens the **Set Appearance** dialog for that graph's data sets. This dialog could have also been displayed by selecting "Plot → Set appearance..." from the main menu.*

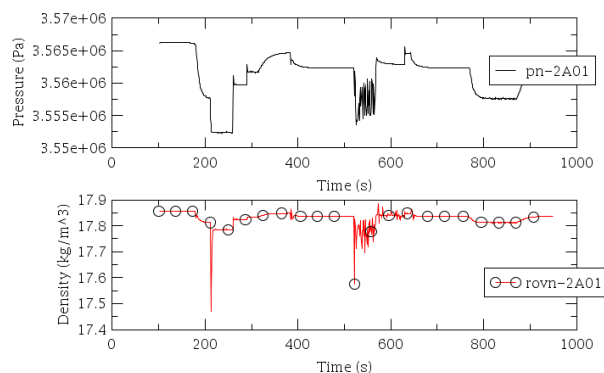
31. Make sure the data set is selected in the set-list the top of the dialog.



*Like graph lists, set lists are used throughout AptPlot to select data sets. Also like graph lists, right-clicking on a set list will open a pop-up menu of set-related operations. Unlike how a graph list displays all graphs in the plot, the contents of a set list will always be the sets of the currently selected graph or those of the graph selected in a nearby list.*

32. In the **Main** tab, set **Line properties Color** to **Red**.
33. In the **Main** tab, set the **Symbol Properties Type** to **Circle**.
34. In the **Symbols** tab, set the **Extra** property **Symbol Skip** to 35.
35. Press the **OK** button.

*The data set will be adjusted with the new formatting. Notice how the set is now painted as a red line with a circle around every 35<sup>th</sup> point. The plot should look similar to the image below.*



36. From the main menu, select **File** → **Save**.

*A file dialog is displayed requesting where to save the plot.*

37. Navigate to the **SNAP\_Exercises** folder in the directory provided with this exercise and save the file as “**UsingAptPlot.apf**”.

38. Close AptPlot.