AFT Fathom™

Examples - English Units

AFT Fathom Version 13

Incompressible Pipe Flow Modeling

Published: September 13, 2023

Dynamic solutions for a fluid world ™

Contents

Walk-Through Examples

The AFT Fathom Walk-Through Examples are designed to get new users trained to use AFT Fathom correctly in the shortest amount of time. If you are using AFT Fathom for the first time, start with the Beginner - Three Reservoir Example because it offers an in-depth explanation of how to use AFT Fathom. Otherwise, choose an example from the table below that you wish to learn more about. For examples using the AFT Fathom Add-on Modules, see the AFT Fathom Modules Overview.

Completed versions of the example model files are included with the default installation: C:\AFT Products\AFT Fathom 13\Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Three Reservoir	Beginner	Water	3	-	A new AFT Fathom user should start here. Get in-depth instructions of model building by solving a classic three reservoir flow problem. Learn how to define a fluid in Analysis Setup, place pipes and junctions on the Workspace, define pipes and junctions, use the Model Data window, run a model, use the Output window, generate graphs in the Graph Results window, and view results in the Visual Report window.
Sizing a Pump	Beginner	Water	2	1	Learn how to size a pump to get the desired flowrate and head through an open system. Explore the differences between the Sizing and Pump Curve features in the Pump junction. See how to interpret results, select a pump curve, and generate a Pump vs. System curve graph.
Control Valve	Beginner	Water	2	-	Learn about the Control Valve junction and how to use the Valve Summary tab in the Output window. See how to use the results

Example	Complexity	Fluid	Pipes	Pumps	Description
					to adjust the pipe size until reaching a desired minimum pressure drop across the control valve.
Heat Transfer in a Pipe	Beginner	Water	1	-	Learn how to enable heat transfer calculations in Analysis Setup and how to use the Convective Heat Transfer model on the Heat Transfer tab of the Pipe Properties window. See how to use the Output Control window to display relevant heat transfer output parameters.
Heat Exchanger System	Beginner	Water	7	1	Learn how to define a Resistance Curve in a Heat Exchanger junction and how to size a pump in a closed system. See how to use the Pump Summary and Heat Exchanger Summary tabs in the Output window and how to generate a Pump vs. System Curve in the Graph Results window.
Spray Discharge System	Beginner	Water	11	-	Learn how to define the Spray Discharge junction using the Cd Spray Loss Model. See how to use the Output Control window to display the discharged flow rate parameter. Finally, learn how to enable the Isometric Grid on the Workspace, route pipes on the grid, and rotate junctions.
Pump Sizing and Selection with Flow Control Valves	Beginner	Water	9	1	Learn how to resolve the Reference Pressure Needed window that may appear when modeling

Example	Complexity	Fluid	Pipes	Pumps	Description
					pumps and flow control valves in series. See how to use the results to select a pump curve for the system.
Freon Delivery System with Heat Transfer	Beginner	R11 and R123	7	1	Learn how to enable heat transfer, use a NIST REFPROP fluid, and use the Convective Heat Transfer model in a Heat Exchanger junction. See how to view and interpret temperature results in the Pipes section of the Output window.
Cooling System	Beginner	Water	20	2	Learn how to import pump curve data to evaluate the performance of different pump curves. See how to use the Pump Summary tab in the Output window to view the power consumed by the pumps.
Housing Project	Beginner	Water	25	-	Learn how to use Scenario Manager and Global Pipe Edit to rapidly change pipes sizes to analyze different system configurations. Also see how Assigned Flow junctions and Special Conditions work.
Beginner - Cost Calculation	Beginner	Water	10	1	Learn how to add, connect, and use both engineering and cost libraries. See how to define the Cost Settings panel in Analysis Setup, include objects in the cost calculations, and view the Cost Report in the Output window.
Non-Newtonian Phosphates Pump-	Beginner	Phosphate	2	1	Learn how to use and compare the Homogeneous

Example	Complexity	Fluid	Pipes	Pumps	Description
ing					Scale-up, Power Law, and Bingham Plastic non-Newtonian viscosity models in a simple system.
NFPA Firewater System	Beginner	Water	71	-	Learn how to enable NFPA calculations in the Miscellaneous group in Analysis Setup. Explore how to use Groups to create an NFPA Report. See where to find the NFPA Report in the Output window.
Crude Oil Pipeline	Intermediate	Viscous Fluid	3	1	Learn how to specify a User Defined Fluid. See how results change when using the Viscosity Correction on the Optional tab of the Pump Properties window. Explore the sensitivity of junction elevation of pump speed as it relates to pressure in the system.
Hot Water System	Intermediate	Water	24	3	Learn how to add, connect, and use a library to define a pump curve. See how to use Global Junction Edit to rapidly apply changes. Explore how to use the Pump Configurations feature and Scenario Manager to analyze the performance effects of impeller size and pump speed.
Plant Cooling Cost Calculation	Intermediate	Water	34	4	Explore how to create cost libraries for pipe materials, pipe fittings and losses, and junctions. Learn how to use the Library Manager window to connect libraries. See how to interpret the Cost Report in the Output window.

Example	Complexity	Fluid	Pipes	Pumps	Description
Non-Newtonian Mill Discharge Slurry	Intermediate	Mill Discharge Slurry	1	-	Learn how to use raw rheological data to define the Bingham Plastic, Power Law, and Homogeneous Scale-up viscosity models. See how the various viscosity models produce different results using the Multi-Scenario Output feature.

Beginner - Three Reservoir

Summary

This example provides an overview of AFT Fathom's layout and structure. The basic features will be used to build a three-pipe, four-junction model to solve the classic three-reservoir problem.

Topics Covered

- Drawing the system with pipes and junctions
- Entering pipe and junction properties
- Completing Analysis Setup

Required Knowledge

No previous experience with AFT Fathom is required to complete this example. It begins with the most basic elements of laying out the pipes and junctions and solving the system hydraulics with the Newton-Raphson methodology.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Beginner - Three Reservoir.fth***

Step 1. Start AFT Fathom

➢**To start AFT Fathom**, click Start on the Windows taskbar, choose AFT Fathom 13, and launch the program. (This refers to the standard menu items created by setup. You may have chosen to specify a different menu item during installation).

As AFT Fathom starts, the start-up screen, as shown in Figure 1, appears with several options before to start building a model. Some of the actions available are:

- Open a recent model, browse to a model file, or browse to an Example
- Activate an Add-on Module
- Select ASME Water or a recently used fluid to be the Working Fluid
- Review or modify Modeling Preferences
    - Select a Unit System
    - Filter units to include Common Only or Common Plus Selected Industries
    - Choose a Grid Style
    - Select a default Pipe Material
- Access other Resources, such as Quick Start Guides, Help Files, and Video Tutorials

Note: If you are working through the Metric Units version of the Examples, be sure to specify either Metric Only or Both with Metric Defaults as the unit set by modifying the Modeling Preferences.
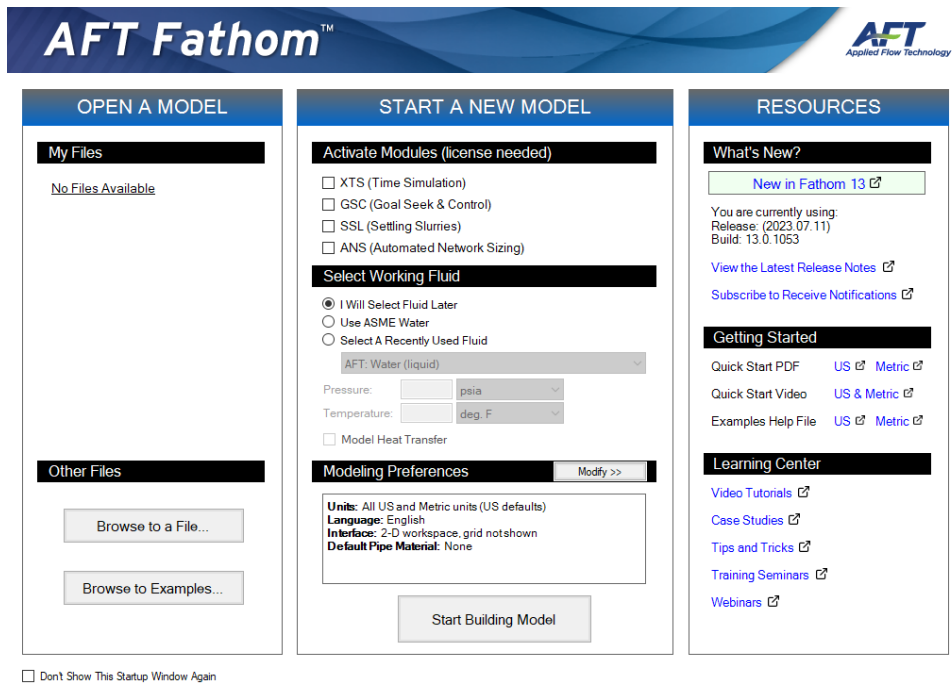

Figure 1: AFT Fathom 13 Start-Up Window

After clicking the Start Building Model button on the start-up window, the Workspace window is initially always the active (large) window, as seen in Figure 2. The tabs such as "Workspace" in the AFT Fathom window represent the Primary Windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

AFT Fathom supports dual monitor usage. You can click and drag any of the primary window tabs off of the main AFT Fathom window. Once you drag one of the primary windows off of the AFT Fathom window, you can move it anywhere you like on your screen, including onto a second monitor in a dual monitor configuration. To add the primary window back to the main Fathom primary tab window bar, simply click the X button in the upper right of the primary window.

To ensure that your results are the same as those presented in this documentation, this example should be ran using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

The Workspace window

The Workspace window is the primary vehicle for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox is the bundle of tools on the far left.

The Quick Access Panel is on the right. It is possible to minimize the Quick Access Panel by clicking on the thumbtack pin in the upper right of the Quick Access Panel in order to allow for greater Workspace area.

The Workspace takes up the rest of the window.

You will build your pipe flow model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Pipe Drawing Tool and Annotation Tool. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. The Annotation tool allows you to create annotations and auxiliary graphics.

Below the two drawing tools are twenty-three icons that represent the different types of junctions available in AFT Fathom. Junctions are objects that connect pipes and also influence the pressure or flow behavior of the pipe system. The twenty-three junction icons can be dragged from the Toolbox and dropped onto the Workspace.

When you pass your mouse pointer over any of the Toolbox tools, a tooltip identifies the tool's function.
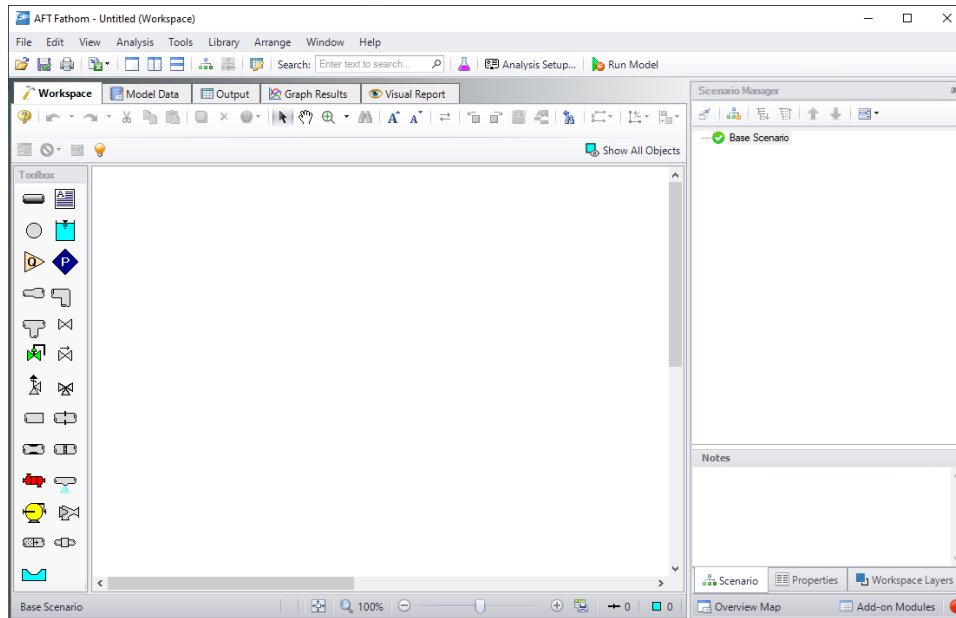
Figure 2: The Workspace window is where the model is built

Step 2. Complete the Analysis Setup

➢**Next, click Analysis Setup on the Toolbar** that runs across the top of the AFT Fathom window. This opens Analysis Setup (see Figure 3). Analysis Setup contains seven groups (additional groups will be displayed if the GSC, XTS or ANS modules are active). Each group needs to be completed (indicated with a green checkmark next to the group name) before AFT Fathom allows you to run the Solver.

Analysis Setup can also be accessed by clicking on the Status Light on the Quick Access Panel. Once the Analysis Setup is complete, the Model Status light turns from red to green.
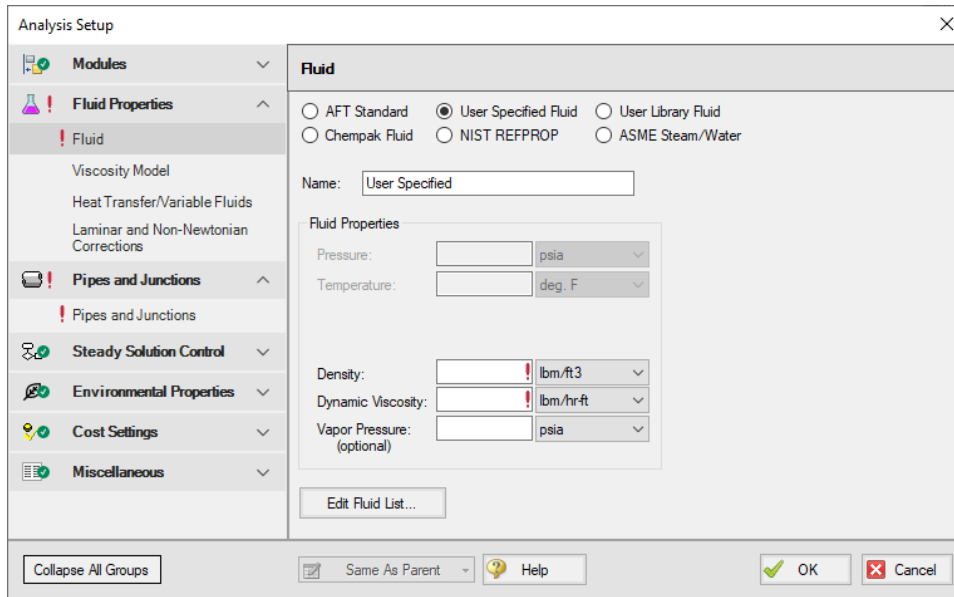
Figure 3: The Analysis Setup tracks the model's status and allows users to specify required parameters

A. Define the Modules Group

Although Analysis Setup initially opens to the Fluid panel of the Fluid Properties group, the first group, Modules, always has a green check when you start AFT Fathom because there are no modules activated by default. No further input is required here for this example.

B. Define the Fluid Properties Group

Next is the Fluid Properties group. This group allows you to specify your fluid properties (density, dynamic viscosity, and optional vapor pressure), viscosity model, heat transfer, and fluid corrections. Start by clicking on Fluid to open the **Fluid panel** (see Figure 4).

You can model the fluid properties using the following data sources:

1. **AFT Standard:** This fluid model uses fluid data from the AFT Standard library. These fluid properties are either temperature dependent or dependent on the solids concentration. Type in the desired condition (e.g., temperature), click the Calculate Properties button and the required properties are calculated. Users can add their own fluids to this option. This model supports heat transfer analysis if specific heat and thermal conductivity data is included. Custom fluids are created by opening the Library Manager window from the Library menu or by clicking the Edit Fluid List button on the Fluid panel.
2. **User Specified Fluid:** This fluid model allows you to directly type in the density, viscosity and vapor pressure. You cannot perform heat transfer analysis with a User Specified Fluid fluid.
3. **User Library Fluid:** This fluid model allows you to select user created fluids that are stored in the user's Fluid Library. You can perform heat transfer analysis with a User Library Fluid fluid if

specific heat, thermal conductivity, enthalpy data has been included in the fluid properties.

4. **ASME Steam/Water:** This fluid model obtains water data from the ASME Steam tables built into AFT Fathom. It also supports steam data, which can be used in AFT Fathom if incompressible.

5. **NIST REFPROP:** This fluid model allows you to select a single fluid or create a mixture of fluids from the REFPROP list. These fluid properties are pressure and temperature dependent, although some are temperature dependent only. This fluid model supports heat transfer analysis.

6. **Chempak Fluid:** This fluid model allows you to select a single fluid or create a mixture of fluids from the Chempak list. These fluid properties are pressure and temperature dependent, although some are temperature dependent only. This fluid model supports heat transfer analysis. Chempak is an optional add-on.

After selecting ASME Steam/Water, NIST REFPROP, or Chempak Fluid, you must then select a Fluid Phase: Liquid or Gas. Then, upon entering a pressure, AFT Fathom will display a temperature range applicable to liquid or gas, depending upon which was selected.

Note: Because heat transfer has a significant impact on compressible flow fluid properties and AFT Fathom calculates flow on an incompressible basis, heat transfer is disabled when Gas is selected as the Fluid Phase.

➤**Select AFT Standard**, by clicking the radio button next to the name. Then choose Water (liquid) from the list and click Add to Model. The properties for AFT Standard water are given only as a function of temperature. Enter 70 deg. F in the temperature box and click the Calculate Properties button.

➤**Check the groups on the left side of Analysis Setup** and you should now see the second group is checked complete with green checkmarks.
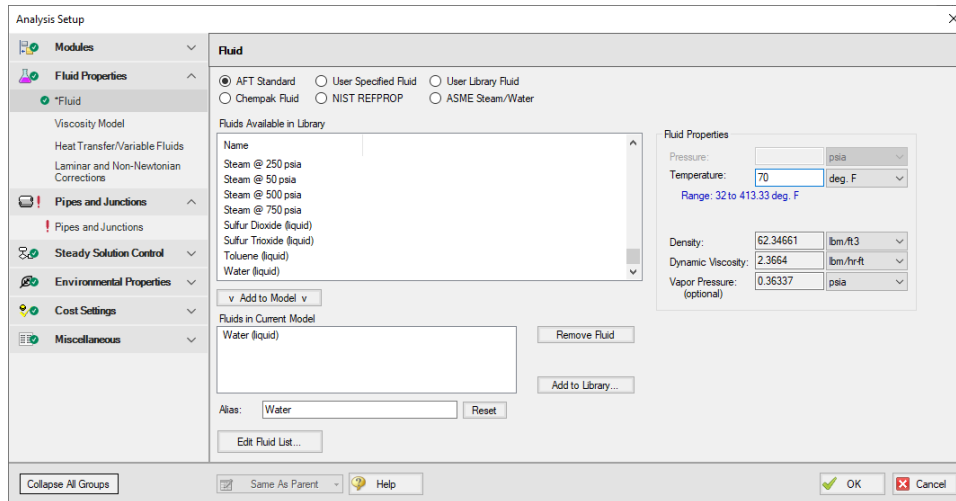

Figure 4: The Fluid panel lets you enter physical properties

On the **Viscosity Model panel**, users can specify how AFT Fathom will treat viscosity. By default, Newtonian is the viscosity model selected. AFT Fathom offers a variety of non-Newtonian models. However, for this example, the default selection of Newtonian will suffice.

On the **Heat Transfer/Variable Fluids panel**, users can specify variable fluid properties or enable heat transfer. For models with variable fluid properties, the values for density and viscosity are default fluid

properties. You can then enter different property values, if desired, for any pipe in the Pipe Properties window. Fluids that allow heat transfer can be modeled by choosing the Heat Transfer with Energy Balance option. In this case, you will be required to also enter heat transfer information for the pipes and temperature for some of the junctions. By default, Constant Fluid Properties is selected and will suffice for this example.

On the **Laminar and Non-Newtonian Corrections panel**, users can specify the type of corrections that the solver will apply when flow is laminar or if a non-Newtonian viscosity model is selected. The default selections are applicable for most cases, so no input is required for this example.

C. Define the Pipes and Junctions Group

In order to fully define this group, there needs to be pipes and junctions on the Workspace. To lay out the classic three-reservoir model, you will place the three reservoir junctions and a branch junction on the Workspace. Then you will connect the junctions with pipes.

To go back to the Workspace and save the inputs made in Analysis Setup, click OK.

I. Place the first reservoir

➤**To start, click on and drag a reservoir junction from the Toolbox** and drop it on the Workspace. Figure 5 shows the Workspace with one reservoir.

Objects and ID numbers

Items placed on the Workspace are called *objects*. All objects are derived directly or indirectly from the Toolbox. AFT Fathom uses three types of objects: *pipes*, *junctions*, and *annotations*.

All pipe and junction objects on the Workspace have an associated ID number. For junctions, this number is, by default, placed directly above the junction and prefixed with the letter J. Pipe ID numbers are prefixed with the letter P. You can optionally choose to display either or both the ID number and the name of a pipe or junction. You also can drag the ID number/name text to a different location to improve visibility.

The reservoir you placed on the Workspace will take on the default ID number of 1. You can change this to any desired number greater than zero and up to 99,999.

Editing on the Workspace

Once on the Workspace, junction objects can be moved to new locations and edited with the features on the Edit menu. Cutting, copying, and pasting are all supported.
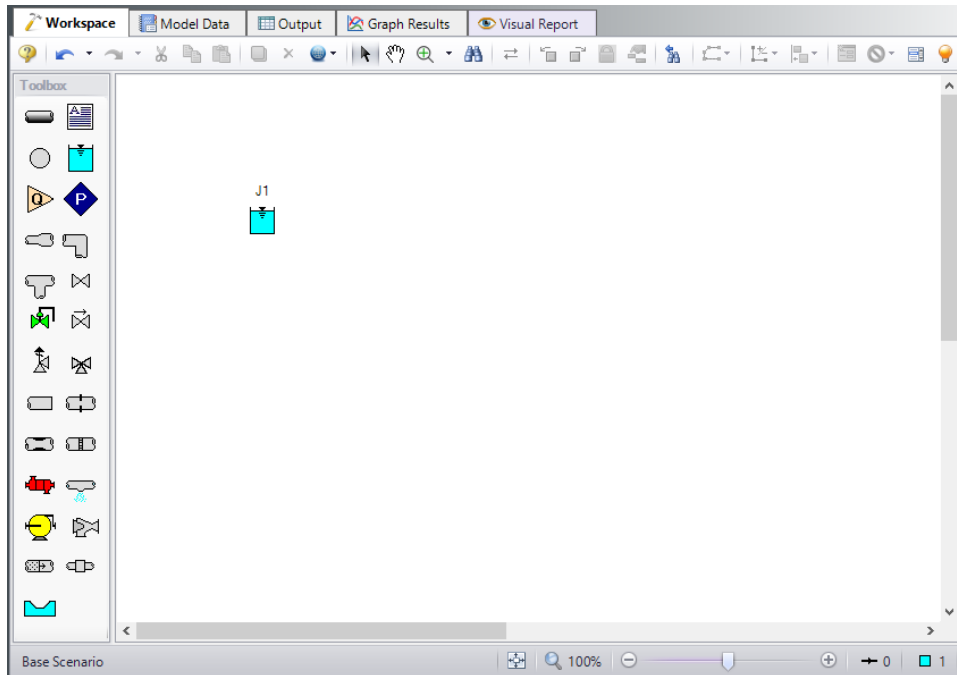
Figure 5: Three Reservoir Problem with one reservoir placed

II. Place the second and third reservoirs

The remaining two reservoirs can be created the same way as the first one or they can be derived from the existing reservoir.

➢**To create a second reservoir from the existing one**, select junction J1 by clicking it with the mouse. A red outline will surround the junction. Choose Duplicate from the Edit menu and move the J2 junction to the right of J1. Your Workspace should appear similar to that shown in Figure 6.

If you like, you can Undo the Duplicate operation and then Redo it to see how these editing features work. To undo an operation, click on the undo button on the Toolbar or choose Undo from the Edit menu. To redo an operation, choose Redo from the Edit menu.

➢**To create the third junction**, select one of the two reservoirs on the Workspace and once again choose Duplicate from the Edit menu. Arrange the three junctions, numbered J1, J2, and J3, as shown in Figure 7.
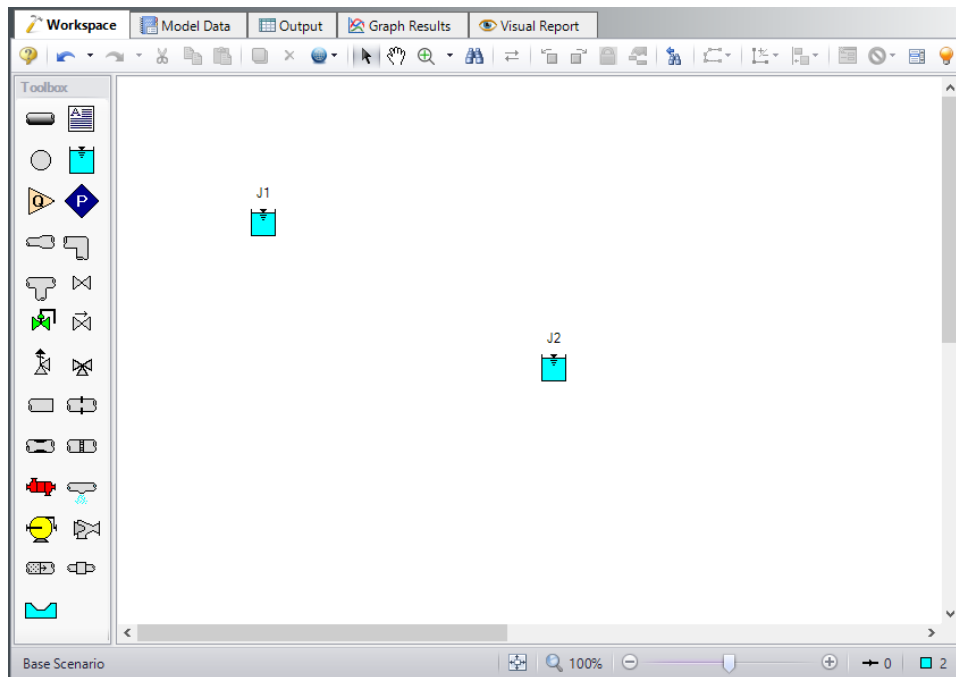
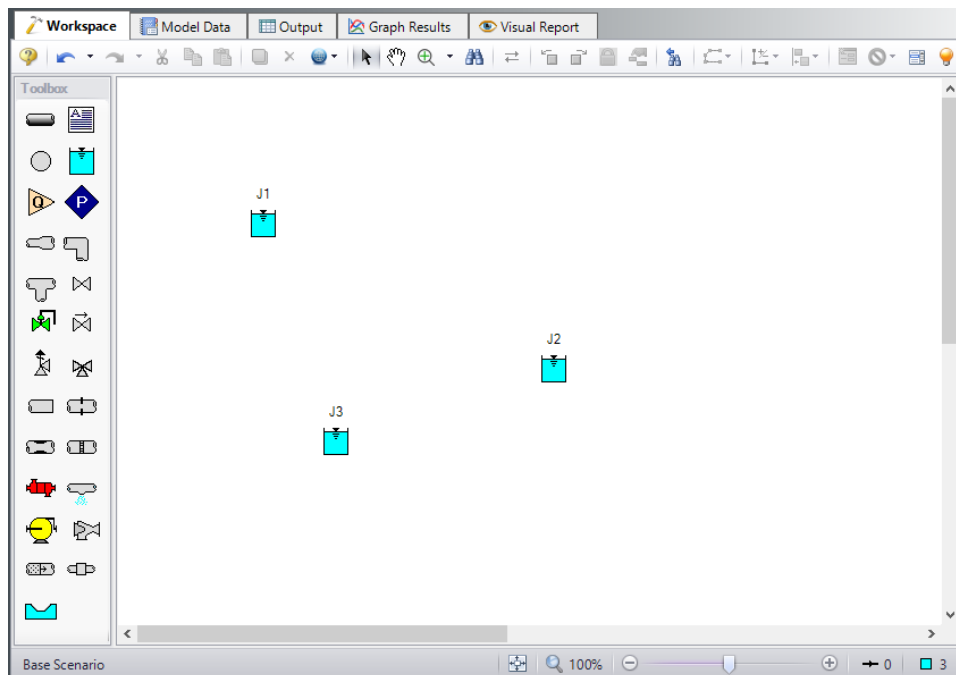*Figure 6: Three Reservoir Problem with two reservoirs placed*

Figure 7: Three Reservoir Problem with three reservoirs placed

III. Place a branch junction

➢**To add a Branch junction**, select a Branch from the Toolbox and place it on the Workspace as shown in Figure 8. The Branch will be assigned the default number J4.

Note: The relative location of objects in AFT Fathom is not important. Distances and heights are defined through dialog boxes. The relative locations on the Workspace establish the *connectivity* of the objects, but have **no bearing on the actual length or elevation relationships.**

➢**Before continuing, save the work you have done so far**. Choose Save As from the File menu and enter a file name (Three Reservoir Problem, perhaps) and AFT Fathom will append the *.fth* extension to the file name.
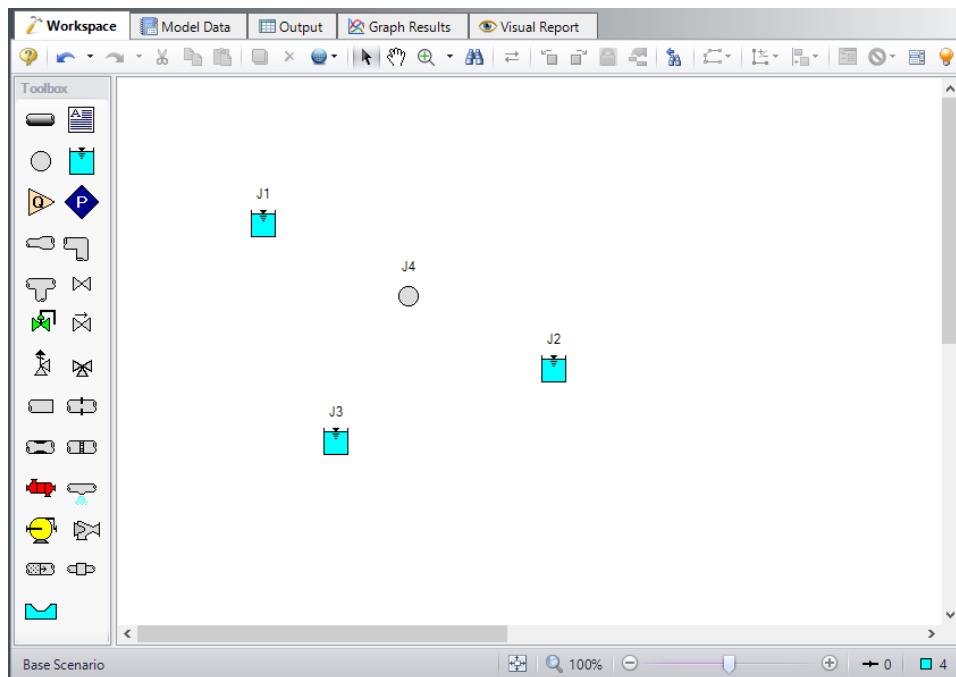

Figure 8: Three Reservoir Problem with all junctions placed

IV. Draw a pipe between J1 and J4

Now that you have four junctions, you need to connect them with pipes.

➢**To create a pipe**, click the Pipe Drawing tool icon. The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe above the junctions, similar to that shown in Figure 9.

The pipe object on the Workspace has an ID number (P1) shown near the center of the pipe.

➢**To place the pipe between J1 and J4**, use the mouse to grab the pipe in the center, drag it so that its left endpoint falls within the J1 Reservoir icon, then drop it there (see Figure 10). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J4 Branch icon (see Figure 11).
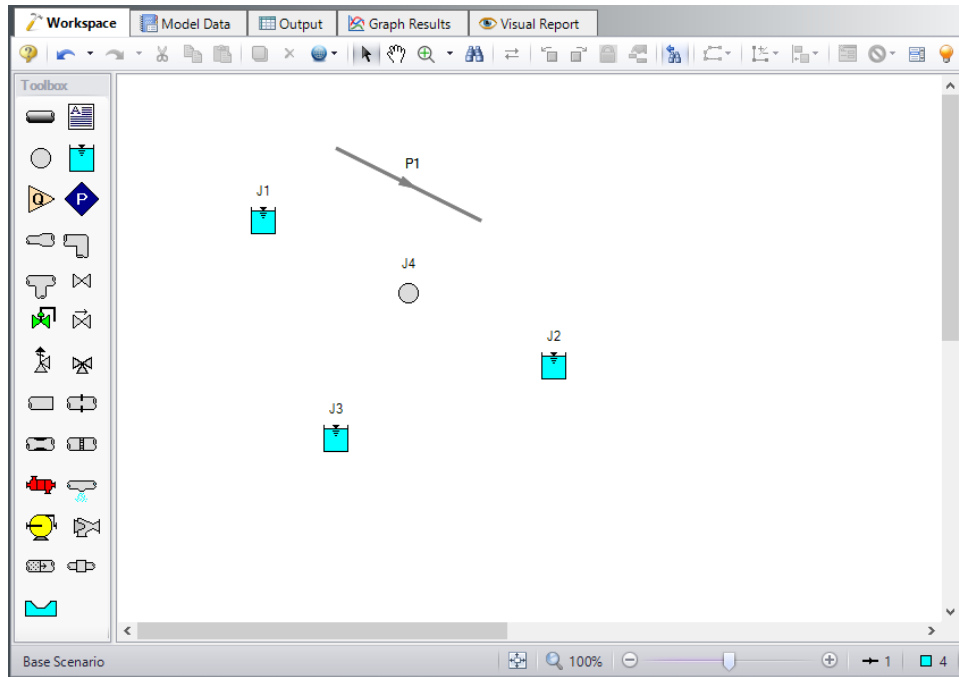
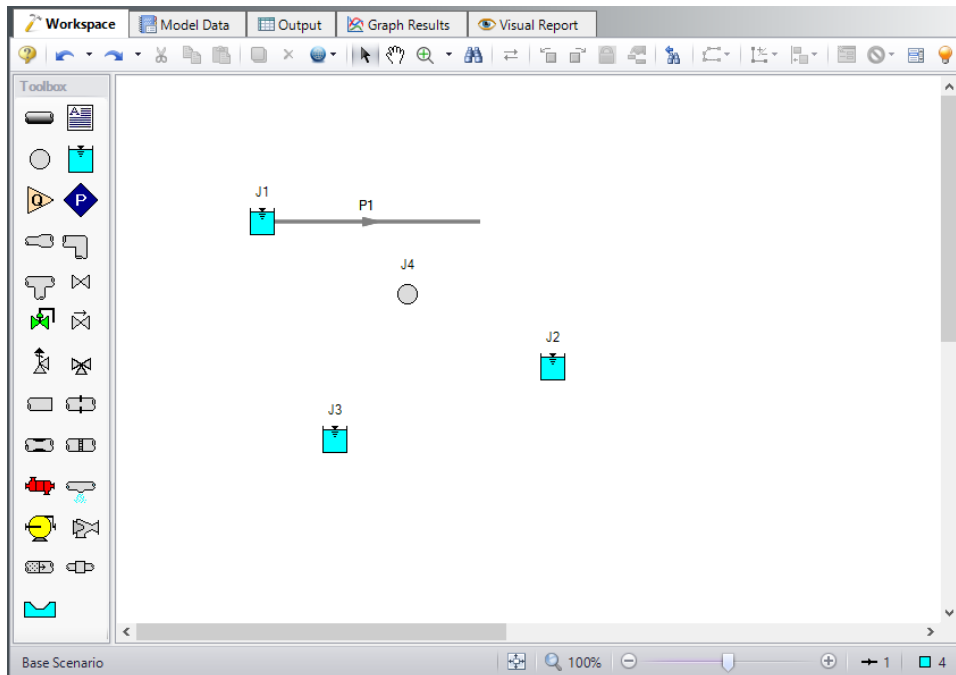*Figure 9: Three Reservoir Problem with first pipe drawn*

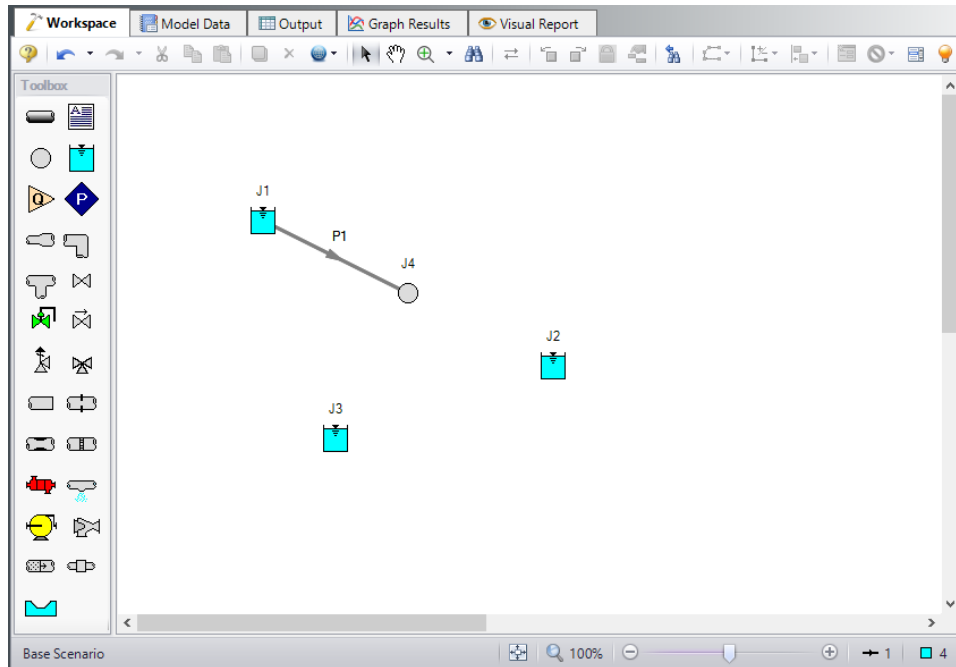*Figure 10: Three Reservoir Problem with first pipe inlet end connected*

Figure 11: Three Reservoir Problem with first pipe connected

Reference positive flow direction

Located on the pipe is an arrow that indicates the reference positive flow direction for the pipe. AFT Fathom assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the Arrange menu or selecting the reverse direction button on the Workspace Toolbar.

In general, the reference positive flow direction indicates which direction is considered positive. However, when used with pumps and certain other junction types the pipes must be in the correct flow direction because that is how AFT Fathom determines which side is suction and which is discharge. If the reference positive direction is the opposite of that obtained by the Solver, the output will show the flow rate as a **negative number**.

V. Add the remaining pipes

A faster way to add a pipe is to draw it directly between the desired junctions.

➢**Activate the pipe drawing tool again** (double-click to allow pipe drawing tool to stay active). Position the cursor on the J4 Branch. Press and hold the left mouse button. Stretch the pipe to the J2 Reservoir then release the mouse button. Then draw a third pipe from the J4 Branch to the J3 Reservoir. Your model should now look similar to Figure 12. In Figure 12, the J3 label has been moved slightly to the left (using drag-and-drop) so it is more visible.

At this point all the objects in the model are graphically connected. Save the model by selecting Save from the File menu or Toolbar.
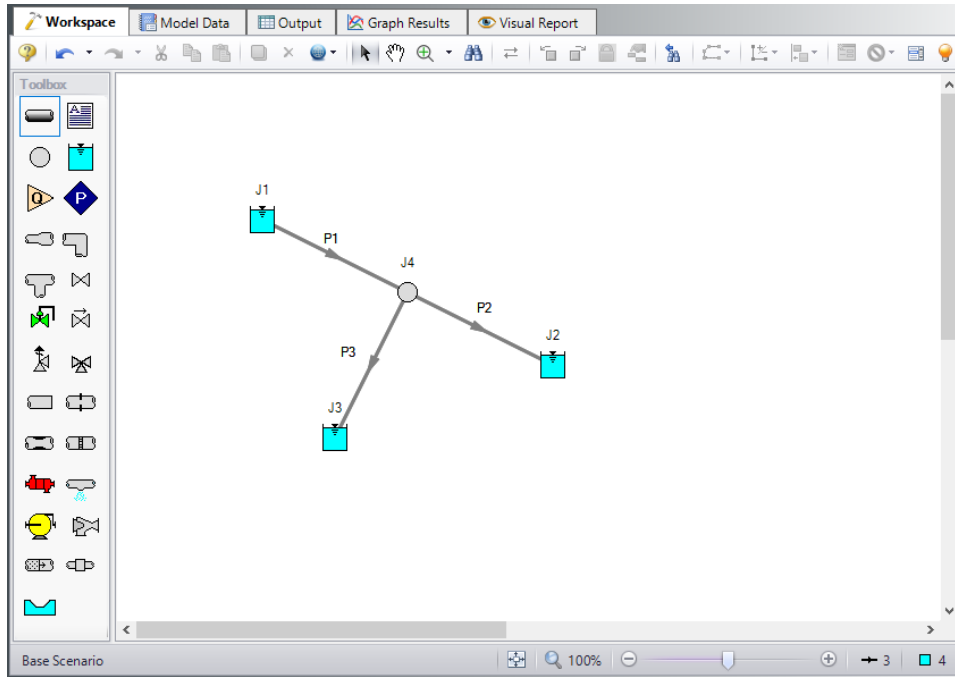
Figure 12: Three Reservoir Problem with all objects connected

VI. Define the pipes and junctions

To fully define the Pipes and Junctions group in Analysis Setup, all pipes and junctions must be connected and have the proper input data.

Object status

Every pipe and junction has an object status. The object status tells you whether the object is defined according to AFT Fathom's requirements. To see the status of the objects in your model, click the floodlight icon on the Workspace Toolbar (alternatively, you could choose Show Object Status from the View menu). Each time you click the floodlight, Show Object Status is toggled on or off.

When Show Object Status is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. (These colors are configurable through User Options from the Tools menu.)

Because you have not yet defined the pipes and junctions in this sample problem, all the objects' ID numbers will change to red when you turn on Show Object Status.

Undefined Objects window

The Undefined Objects window lists all undefined pipes and junctions and further displays the items that are not yet defined.

VII. Define Reservoir J1

➤**To define the first reservoir**, open the J1 Reservoir Properties window (see Figure 13) by double-clicking the J1 icon. Enter a reservoir Liquid Surface Elevation of 300 feet. You can assign any unit of length found in the adjacent dropdown list of units.

Note: You can also open an object's Properties window by selecting the object (clicking on it) and then either pressing the Enter key or clicking the Open Pipe/Jct Window icon on the Toolbar.

➤**Enter a Liquid Surface Pressure of 0 psig** and a Pipe Depth of 0 feet.

Note: You can specify preferred units for many parameters (such as meters for length) in the User Options window.

You can give the object a name, if desired, by entering it in the Name field at the top of the window. In Figure 13, the name of this reservoir is Supply Tank A. By default the junction's name is the junction type. The name can be displayed on the Workspace, Visual Report or in the Output.

Most junction types can be entered into a custom library allowing the junction to be used multiple times or shared between users. To select a junction from the custom library, choose the desired junction from the Library list. The current junction will get the properties from the library junction.

The Copy Data From Jct list will show all the junctions of the same type in the model. This will copy the selected parameters from an existing junction in the model to the current junction.

The pipe table on the **Pipe Depth & Loss Coefficients** tab allows you to specify entrance and exit loss factors for each pipe connected to the reservoir (in this case there is one). You can enter standard losses by selecting the option buttons at the right. The default selection is the Custom option with loss factors specified as zero. To later change the loss factors, click within the pipe table and enter the loss. You can also specify a depth for the pipe.

The **Optional** tab allows you to enter different types of optional data. You can select whether the junction number, name, or both are displayed on the Workspace. Some junction types also allow you to specify an initial pressure as well as other junction specific-data. The junction icon graphic can be changed, as can the size of the icon.

Design Alerts can be entered for most junctions, which are applied to the pressure loss calculations for the junction in order to give additional safety margin to the model.

Each junction has a tab for **Notes**, allowing you to enter text describing the junction or documenting any assumptions.

The **Status** tab will list undefined parameters of the junction.

The highlight feature displays all the required information in the Properties window in light blue as shown in Figure 13. The highlight is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window or by pressing the F2 key. The highlight feature can also be turned on or off by selecting it on the User Options menu. For the remainder of the examples, we will keep the highlight feature turned off.

➢**Click OK**. If Show Object Status is turned on, you should see the J1 ID number turn black again, telling you that J1 is now completely defined.
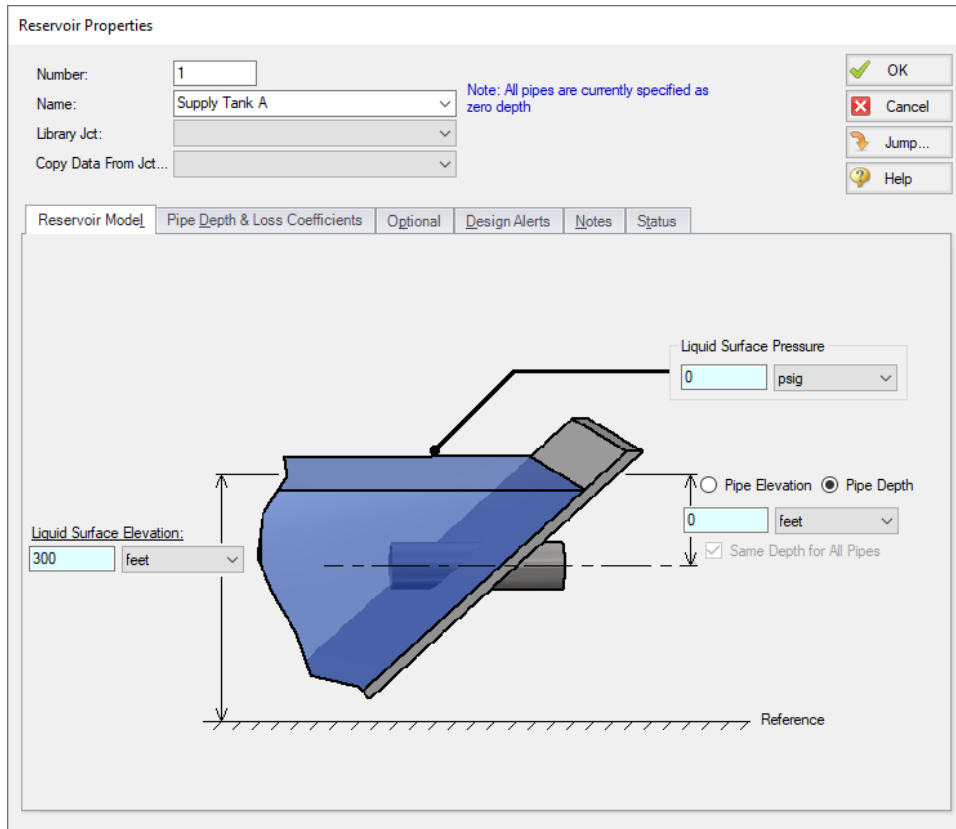

Figure 13: Properties window for Reservoir J1

The Inspection feature

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspection feature. Position the mouse pointer on J1 and hold down the right mouse button. An information box appears, as shown in Figure 14.

Inspecting is a faster way of examining the input in an object than opening the Properties window.
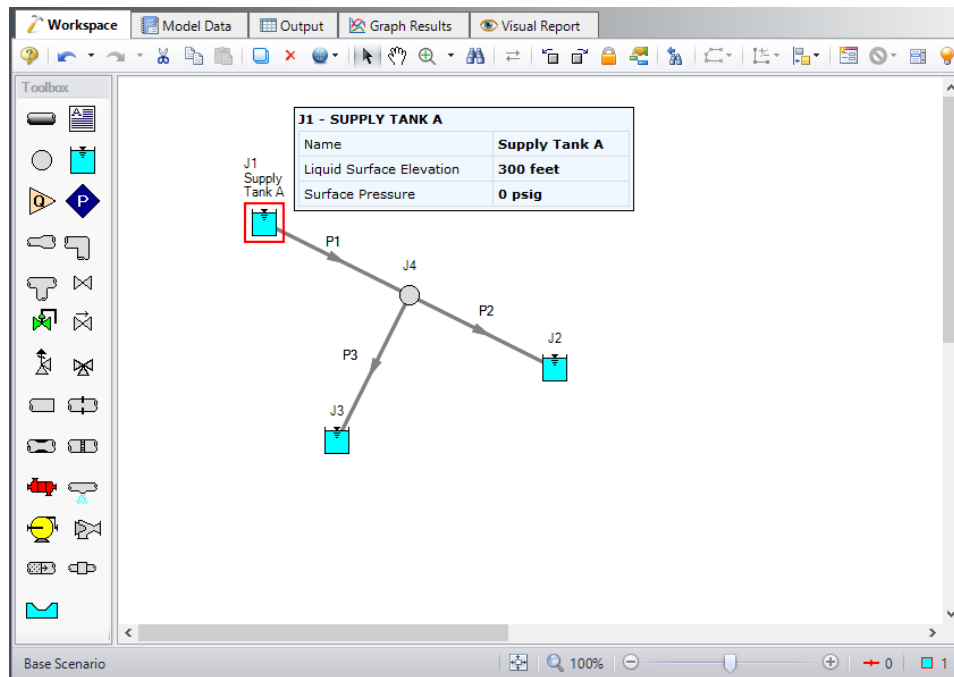
Figure 14: Inspecting from the Workspace with right mouse button

VIII. Define junctions J2, J3, and J4

- J2 Reservoir
    - Liquid Surface Elevation = 200 feet
    - Liquid Surface Pressure = 0 psig
    - Pipe Depth = 0 feet (default)
- J3 Reservoir
    - Liquid Surface Elevation = 100 feet
    - Liquid Surface Pressure = 0 psig
    - Pipe Depth = 0 feet (default)

➢**Open the J4 Branch Properties window** (see Figure 15). In this window, all three connecting pipes should be displayed in the pipe table area. You could associate loss factors with each pipe by clicking within the pipe table and entering the data.

➢**Enter an elevation of zero feet** for the J4 Branch junction (an elevation must be defined for all junctions). You could also click the Optional tab then specify a flow source or sink at the junction; for now leave this as zero. Click OK. All the junctions have now been defined.
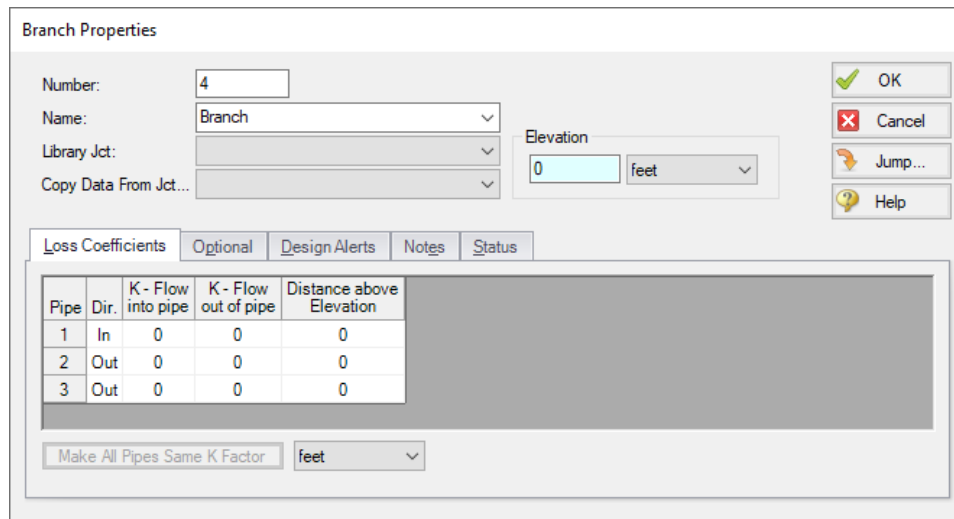
➢**Save the model** again before proceeding.

Figure 15: Properties window for Branch J4

IX. Define Pipe P1

Data for pipes and junctions can be entered in any order. In this example, we did the junctions first. The next step is to define all the pipes. To open the Pipe Properties window, double-click the pipe object on the Workspace.

➤**First open the Pipe Properties window for Pipe P1** (see Figure 16). For Pipe P1, enter a length of 1000 feet, select the Pipe Material as Steel - ANSI, Size of 8 inch , and Type as STD (schedule 40). Click OK. Pipe P1 is now defined.

The Pipe Properties window

The Pipe Properties window offers control over all important flow system parameters that are related to pipes.

The Inspect feature also works within the Pipe Properties window. To Inspect a connected junction, position the mouse pointer on the connected junction's ID number and hold down the right mouse button. This is helpful when you want to quickly check the properties of connecting objects. (You can also use this feature in junction Properties windows for checking connected pipe properties.)

By double-clicking the connected junction number, you can jump directly to the junction's Properties window. Or you can click the Jump button to jump to any other part of your model.

Figure 16: Properties window for Pipe P1

X. Define Pipes P2 and P3

Both pipes have a Pipe Material of Steel - ANSI, Size of 8 inch, and Type of STD (Schedule 40).

1. P2
   a. Length = 2000 feet
2. P3
   a. Length = 3000 feet

The Pipes and Junctions group should now be complete. Everything is ready to submit to the Solver.

➢**Before running the model, save it one more time.** It is also a good idea to review the input using the Model Data window.

D. Define the remaining groups in Analysis Setup

Open Analysis Setup once more to confirm that all groups are defined. No inputs or alterations are required to complete the last three groups: Steady Solution Control, Environmental Properties, and Miscellaneous. All seven groups should have a green checkmark next to its name. Click OK.

D. Review input in the Model Data window

The Model Data window is shown in Figure 17. To change to this window, you can select it from the Primary Window tabs, Window menu, or pressing CTRL+M. The Model Data window gives you a text-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to an Adobe PDF, or printed out for review.

Data is displayed in three general areas. The top is called the general area, the middle the pipe area and the bottom the junction area.

The Model Data window allows access to all Properties windows by double-clicking the appropriate ID number in the far left column of the table. You may want to try this right now.
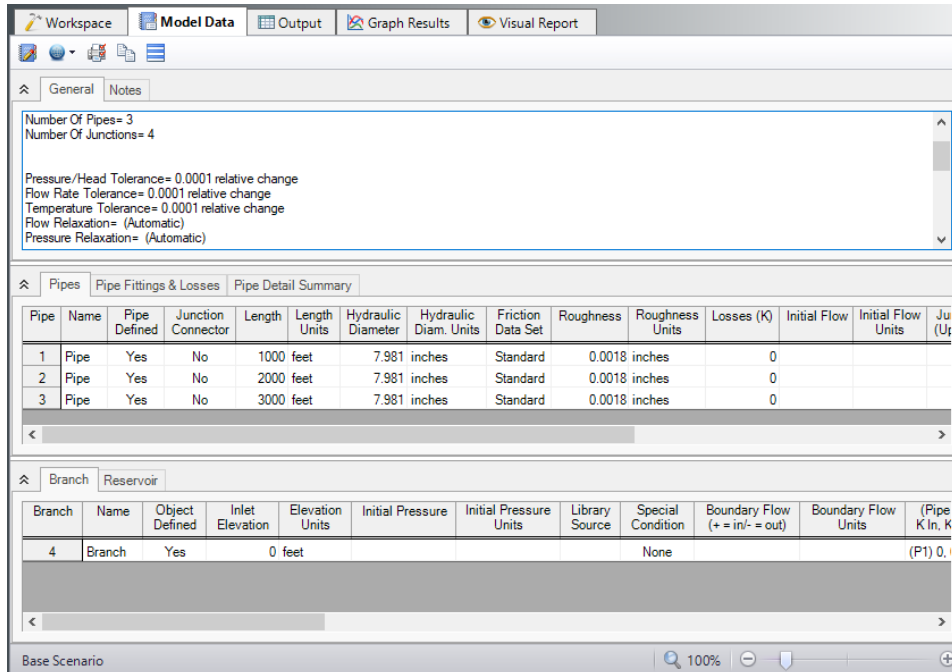

Figure 17: The Model Data window shows all input in text form

Step 3. Run the Solver

➢**Click Run Model from the Toolbar** or from the Analysis Menu. During execution, the Solution Progress window is displayed (see Figure 18). You can use this window to pause or cancel the Solver's activity. When the solution is complete, the Output, Visual Report, and Graphs Results windows can be accessed directly from the Solution Progress window. Click the Output button and the text-based Output window will appear (see Figure 19). The information in the Output window can be reviewed visually, saved to file, exported to a spreadsheet-ready format, copied to the clipboard, printed to an Adobe PDF file, and printed out on the printer.

Figure 18: The Solution Progress window displays the convergence progress
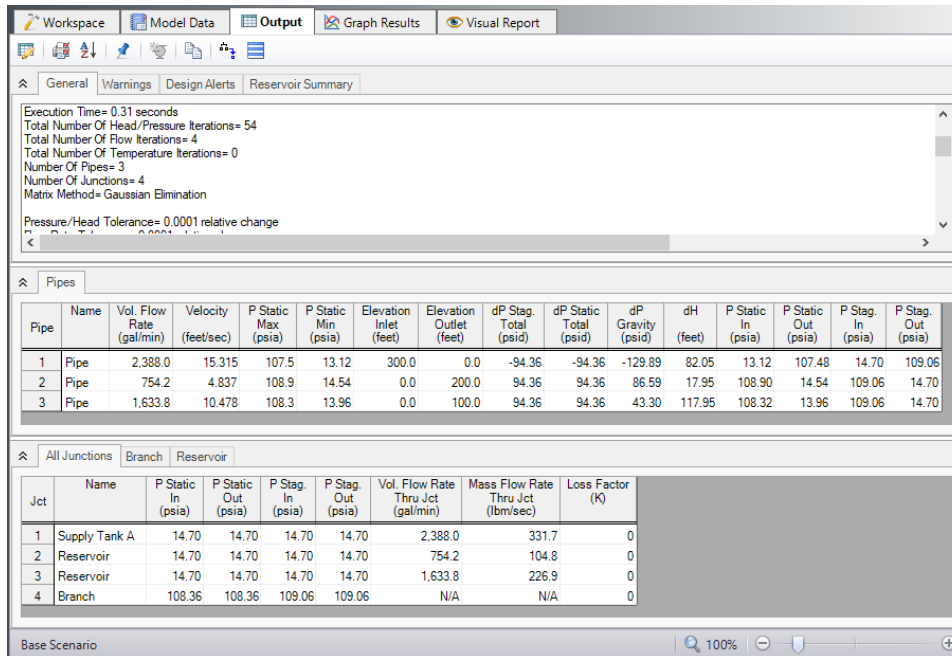
Figure 19: The Output window displays output in text form

Step 4. Review the output

The Output window is similar in structure to the Model Data window. Three areas are shown, and you can minimize or enlarge each area by clicking the arrow next to the General, Pipes, and Junctions tabs or from the View menu.

A. Specify output control

By default, AFT Fathom has a predefined set of Output Control parameters that are specified and the default units used depend on which unit set was selected when first running AFT Fathom (see Figure 1). A default title is also assigned in the Output Control.

➢**Open Output Control** from the Tools menu (Figure 20). Click the General tab, enter a new title (if you like you can title this Classic Three-Reservoir Problem), then click OK to accept the title and other default data.

In addition, this window allows you to select the specific output parameters you want in your output. You also can choose the units for the output.
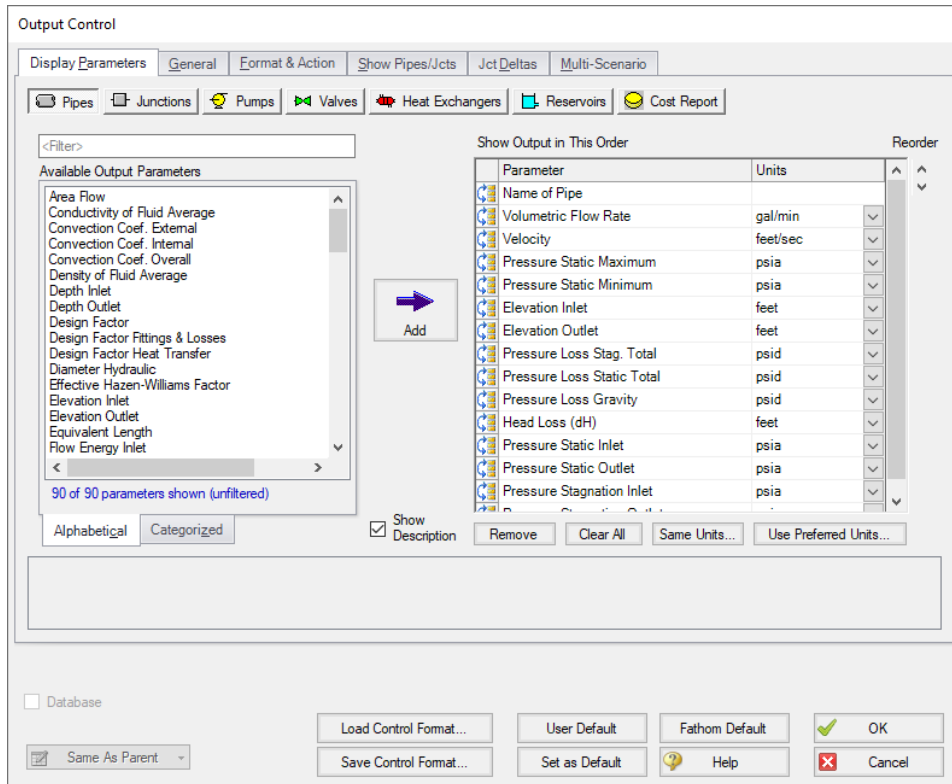
Figure 20: The Output Control window lets you customize the output

B. Modify the output format

If you selected the default AFT Fathom Output Control, the Pipe Results table will show volumetric flow rate in the second column (Figure 19) with units of gal/min (gallons per minute).

- **Select Output Control from the Tools menu** or Toolbar. On the right side of the Pipe tab is the list of selected output parameters. Click Volumetric Flow Rate and change the units by selecting gal/hr (gallons per hour) from the list at the bottom.

- **Click OK to display changes to the current results.** You should see the volumetric flow rate results, still in the second column, in units of gal/hr. Notice the Velocity results in the third column.

- **Select Output Control from the Tools menu one more time.** The reorder icons on the left side of the Output List allows you to reorder parameters in the list.

- **Select the Velocity parameter** and use the Reorder scroll bar to move it up to the top of the parameter list.

- **Click OK to display the changes to the current results.** You will see in the Pipe Results table that the first column now contains Velocity and the third column contains the volumetric flow rate. The Output Control window allows you to obtain the parameters, units and order you prefer in your output. This flexibility will help you work with AFT Fathom in the way that is most meaningful to you, reducing the possibility of errors.

- **Lastly, double-click the column header Velocity in the Output window Pipe Results Table.** This will open a window in which you can change the units again if you prefer. These changes are extended to the Output Control parameter data you have previously set.

C. View the Output on the Workspace with Workspace Layers

- **Return to the Workspace window** by choosing it from the Window menu, clicking on the Workspace tab on the tool bar, or by pressing CTRL+W. Workspace Layers allows you to integrate your text results with the graphic layout of your pipe network.

- **Select the Workspace Layers tab** at the bottom right of the screen within the Quick Access Panel. Select the layer named All Objects Layer and then select the single gear icon above to edit the Layer Settings. Alternatively, while hovering over the layer name, the Layer Settings icon will be revealed to the right. Select the Pipe Parameters tab and search for Volumetric Flow Rate with the Filter or within the Pipe Parameters list. Double click Volumetric Flow Rate to add it to the Workspace Layer. Select the Junction Parameters tab and search for both Junction Name and the Pressure Static Inlet. After adding these parameters to the display list, press Close. Rearrange the labels as desired for readability.
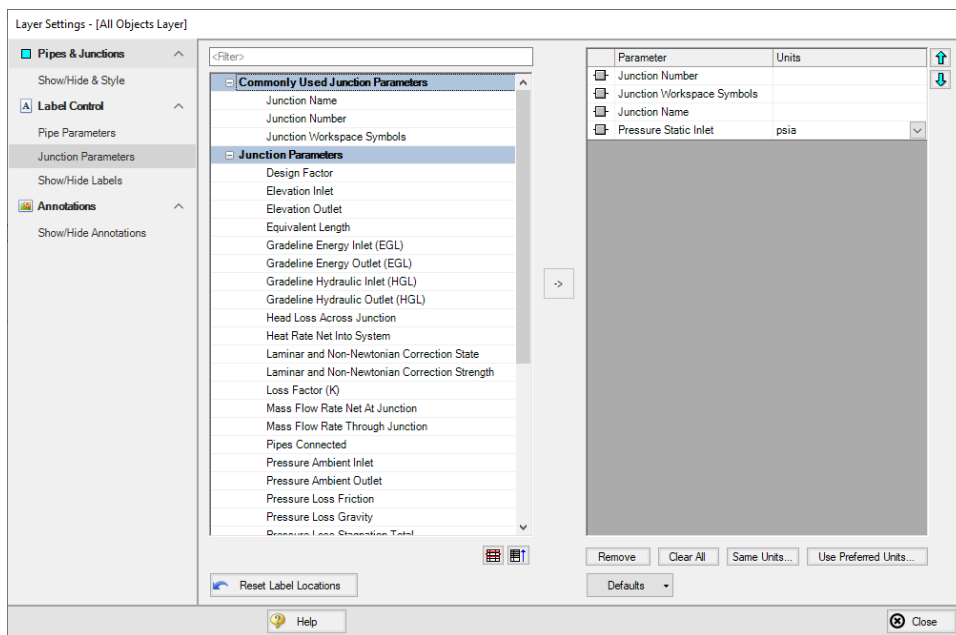

Figure 21: The Layer Settings window selects visible parameters for the Workspace Layers
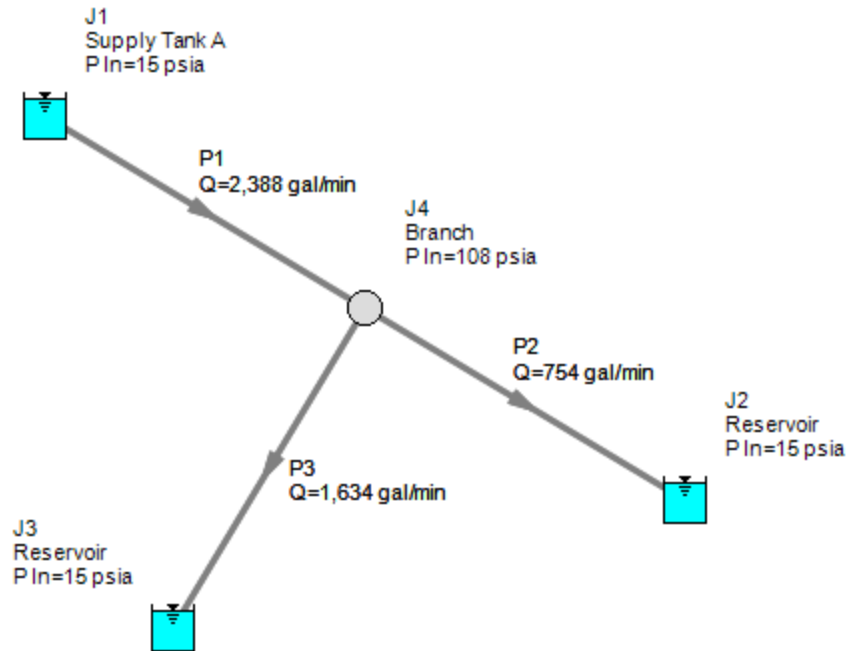
Figure 22: The Workspace window displays input and output data utilizing Workspace Layers

D. Graph the results

➢**Change to the Graph Results window** by clicking the Graph Results tab. The Graph Results window offers full-featured Windows plot preparation.

The Graph Parameters menu will automatically be displayed in the Quick Access Panel on the far right of the Graph Results Window and is where you are to specify which graphs to generate.

The Profile graph tab in the Graph Parameters menu will also be selected by default on Quick Access Panel. Make sure the Plot Single Path option is selected from the Pipes drop-down menu. Select pipes 1 and 2 from the pipes list. Choose Pressure Static from the Parameter drop-down menu. Make sure feet is selected as the Length Units and psia as the Pressure Static units (see Figure 23). Then click Generate. The Show Junction Locations feature is located on the Graph Results toolbar and can be turned on by clicking the icon.

A graph appears showing the static pressure along the flow-path defined by pipes 1 and 2 in Figure 24.

You can use the other buttons in the Graph Results window to change the graph appearance and to save and import data for cross-plotting. The Graph Results window can be printed, saved to file, copied to the clipboard, or printed to an Adobe PDF file. The graph's x-y data can be exported to file or copied to the clipboard.

Note that the graph guide, located at the top right of the Graph Results window and represented with the What Would You Like to Do? icon, can guide you through the development of your graph. This feature can be hidden by clicking on the icon.

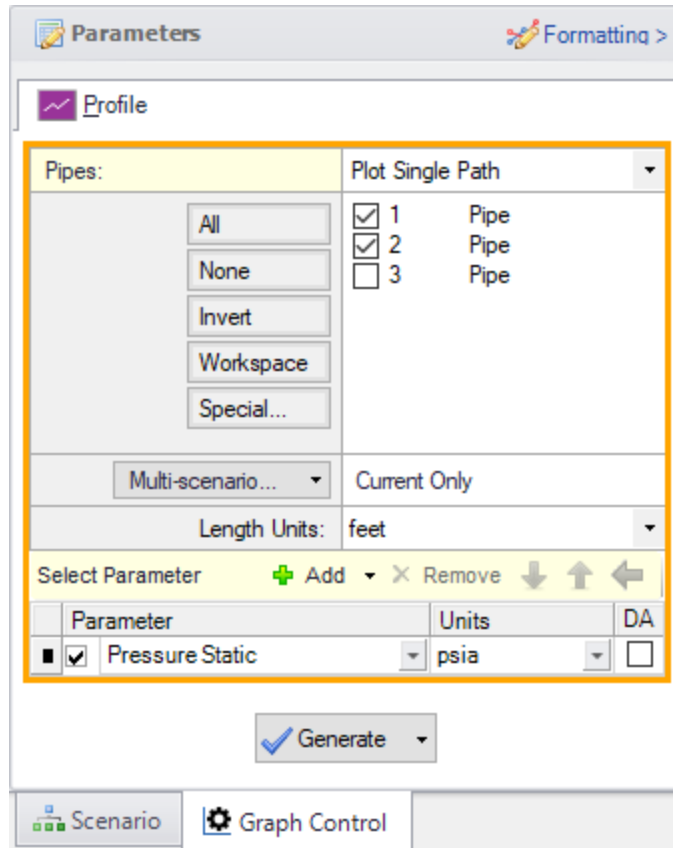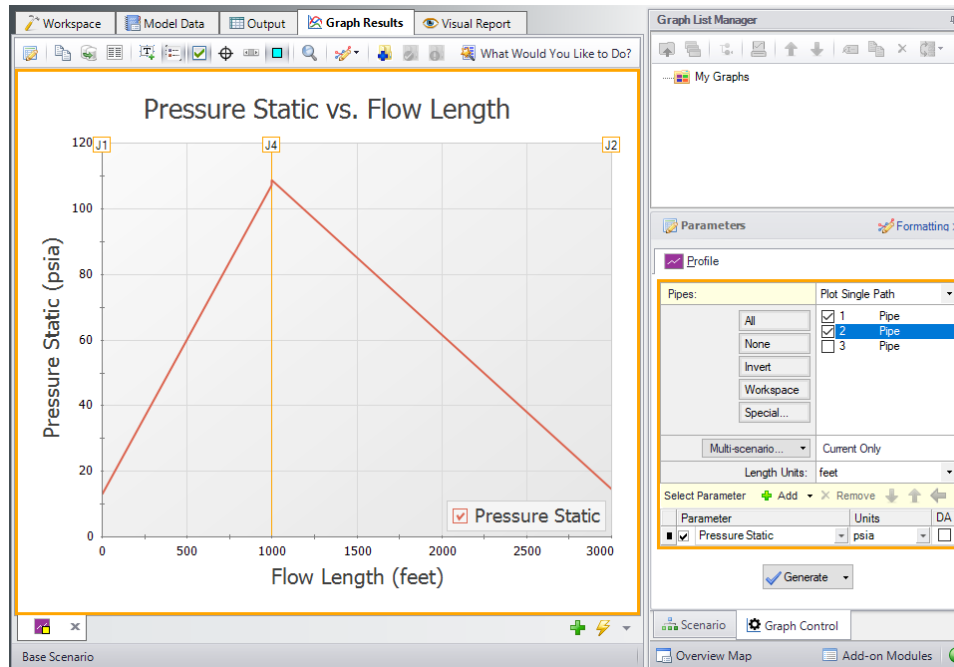*Figure 23: The Graph Control window controls the Graph Results content*

Figure 24: The Graph Results window offers full-featured plot generation

Conclusion

You have now used AFT Fathom's Primary Windows and functions to build a simple model. Review the rest of the Fathom Help File contents for more detailed information on each of the windows and functions.

Sizing a Pump

Summary

This example will walk you through a simple calculation to size a pump.

A pump is to be used to transfer water from a supply reservoir to a holding reservoir at the top of a hill. The system consists of a supply reservoir, a pump, a discharge reservoir, and two pipes.

The first pipe from the supply reservoir to the pump is 10 feet long, and the second pipe leading from the pump to the discharge reservoir is 990 feet long. Both pipes are 4 inch, Steel - ANSI, STD (Schedule 40) pipes. Constant fluid properties are assumed.

The supply reservoir has a liquid surface pressure of 0 psig, and the liquid surface is at an elevation of 10 feet. The discharge reservoir has a liquid surface pressure of 0 psig, and the liquid surface is at an elevation of 200 feet. The pipes for both reservoirs connect to the reservoirs at a depth of 10 feet.

What is the head requirement for a pump to supply flow in this system at 500 gal/min?

Topics Covered

- Specifying a pump in the Pump Properties Window
- Defining pipe fittings and losses lumped into a pipe
- Sizing a pump
- Entering pump curve data

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Sizing a Pump.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
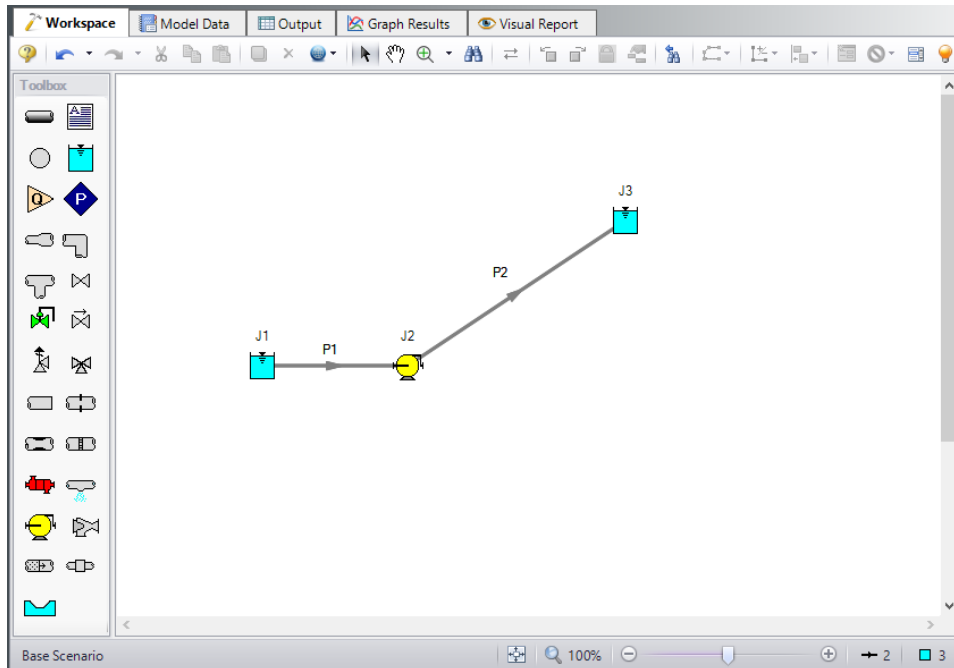

Figure 1: Workspace for Sizing a Pump Example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 4 inch
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Lengths =

Pipe	Length (feet)
1	10
2	990

The pipe model also allows for fittings and losses like valves or elbows. Select the Fittings & Losses tab for P2, and type in 25 for the Total K Factor. Now click OK to close the Pipe Properties window and accept your changes.

> **Note:** A red ampersand (&) will now appear next to the pipe number for pipe P2. This indicates that Fittings & Losses are entered in the Pipe Properties window. This symbol can be edited or hidden in User Options.

Junction Properties

1. J1 Reservoir
    a. Name = Lower Reservoir
    b. Liquid Surface Elevation = 10 feet
    c. Liquid Surface Pressure = 0 psig
    d. Pipe Depth = 10 feet
2. J2 Pump
    a. Inlet Elevation = 0 feet
    b. Pump Model = Centrifugal (Rotodynamic)
    c. Analysis Type = Sizing
    d. Parameter = Volumetric Flow Rate
    e. Fixed Flow Rate = 500 gal/min
3. J3 Reservoir
    a. Name = Upper Reservoir
    b. Liquid Surface Elevation = 200 feet
    c. Liquid Surface Pressure = 0 psig
    d. Pipe Depth = 10 feet

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will

have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The Output window (Figure 2) contains all the data that was specified in the output control window. Because we were interested in the pump requirements for this system, select the Pump Summary tab in the General (top) window. The pump data can also be found in the Junction (bottom) window. We are primarily interested in the pump head (or pressure) requirement. This information can be found in the Pump Summary window.

Select Output Control from the Common Toolbar or Tools menu to open the Output Control window. Click the General tab, enter the title Water Transfer System, then click OK to save the title.

For this system, the head requirement of the pump is 380.7 feet. Because the difference in elevation of the reservoirs was 190 feet this is the minimum head rise regardless of flow rate or pipe size. Coincidentally, the friction loss was also approximately 190 feet. This can be seen in the *dH* column of the Pipe output window.

Figure 2: Output window for Pump Sizing Example

Step 6. Add a Pump Curve

Now that the pump head requirement has been identified as 380.7 feet at the design flow rate of 500 gal/min, a pump of the correct size can be purchased to meet this requirement. Once the actual pump has been identified, the pump characteristics can be added to the pump by adding the pump curve data to the pump junction.

Open the Pump Properties window and enter the following data:

1. Pump Model = Centrifugal (Rotodynamic)
2. Analysis Type = Pump Curve
3. Enter Curve Data
    a. Flow Q Parameter = Volumetric
    b. Head/Pressure dH Parameter = Head

Volumetric	Head
gal/min	feet
0	400
500	381
1000	250

    c. Curve Fit Order = 2
    d. Click Generate Curve Fit Now
    e. Click OK

Note: This is a situation where a user could create a new scenario using the Scenario Manager to examine a what-if situation, without disturbing the basic model. For an example that illustrates how to use the Scenario Manager, see the Water to Housing Project example.

Figure 3: The Pump Configuration Window is used to enter pump curve data

Step 7. Re-evaluate the model

Re-run this model. Examine the Pump Summary in the General Output Section. The output (Figure 4) shows the pump operating at a flow rate of 500.2 gal/min, and a head rise of 381.0 feet, which is acceptably close to the previous sizing calculation.

Figure 4: Output when running with a pump curve

Change to the Graph Results window (e.g., using the Windows menu), and click on the Pump vs. System tab (Figure 5) in the Graph Parameters menu on the Quick Access Panel. Click Generate to graph a pump vs. system curve, as shown in Figure 6.

Figure 5: Select Graph Data for creating a pump vs. system curve
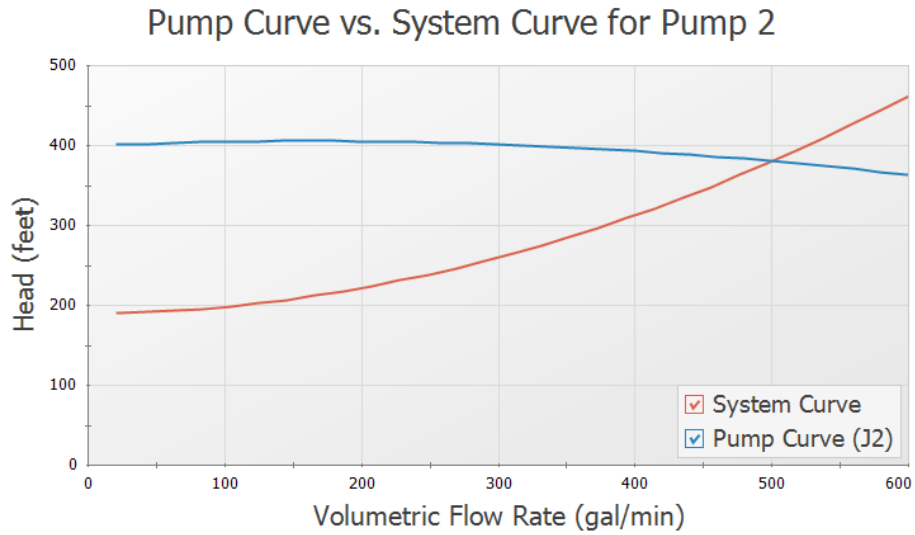
Figure 6: Pump vs. system curve using selected pump

Conclusion

AFT Fathom can be used to size pumps. Once the actual pump has been selected, the model can be modified to include the actual pump curve data. Then the actual system properties and behavior can be calculated.

Control Valve

Summary

This will walk you through a simple calculation to calculate a pipe size for a control valve system. Water will be flowing from a reservoir on a hill down to another reservoir. The pipe must be sized correctly to allow 250 gal/min to flow from one reservoir to the other. A control valve is used to control this flow, but the valve must have a minimum pressure drop of 10 psid to operate correctly. Only STD (schedule 40) pipes will be considered.

Topics Covered

- Using flow control valves
- Entering and changing pipe size data

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Control Valve.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

    a.   Fluid Library = AFT Standard

    b.   Fluid = Water (liquid)

            i.   After selecting, click Add to Model

    c.   Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for Control Valve Problem

The system is in place but now we need to enter the properties of the objects. For this example, we need to find the correct pipe size but we are limited to STD (schedule 40) pipe sizes. We need to use our engineering judgment to take an initial guess at the pipe size for 250 gal/min. Let's try 3 inch pipe. Specify the pipes as the following:

Pipe Properties

1.   All Pipes
    a.   Pipe Material = Steel - ANSI
    b.   Pipe Geometry = Cylindrical Pipe
    c.   Size = 3 inch

    d.   Type = STD (schedule 40)

    e.   Friction Model Data Set = Standard

    f.   Length = 100 feet

Junction Properties

1. J1 Reservoir
   a. Name = Upper Reservoir
   b. Liquid Surface Elevation = 40 feet
   c. Liquid Surface Pressure = 0 psig
   d. Pipe Depth = 0 feet
2. J2 Control Valve
   a. Inlet Elevation = 10 feet
   b. Valve Type = Flow Control (FCV)
   c. Control Setpoint = Volumetric Flow Rate
   d. Flow Setpoint = 250 gal/min
3. J3 Reservoir
   a. Name = Lower Reservoir
   b. Liquid Surface Elevation = 10 feet
   c. Liquid Surface Pressure = 0 psig
   d. Pipe Depth = 0 feet

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the Output toolbar or Tools menu. This window allows the user to specify the output from the Fathom simulation. The window starts with a default set of output parameters which you are free to change.

In this example, we want to review the effects of changing the nominal pipe size. To simplify this, lets add this parameter to the output.

1. From the Pipes output list, select the parameter Nominal Size in the list on the left.
2. Click Add (or double-click the parameter), which adds the parameter to the list on the right.

3. Use the Reorder buttons (in the upper right) to move the Nominal Size up so that it becomes the second parameter in the list (see Figure 2).
4. Click OK.
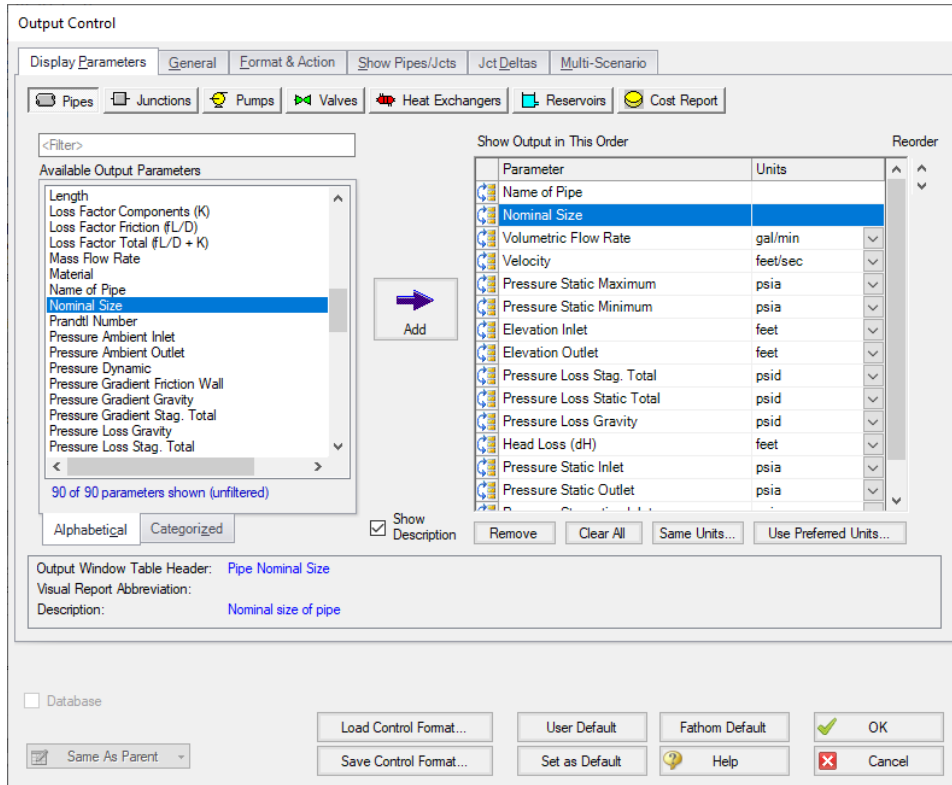
Figure 2: Output Control Window

The Output window contains all the data that was specified in the output control window. The output is shown in Figure 3. In the summary window, there is a Valve Summary tab. Select this tab. You can see that the flow control valve achieved the flow rate of 250 gal/min but the pressure drop (*dP*) is only 1.316 psid (see Figure 3). The pressure drop requirement is 10 psid minimum. This pressure drop is too low. The pipe size needs to be increased.

Figure 3: Output Window for Control Valve Example

Step 6. Change Pipe Sizes

Go back to the Workspace and change the pipe properties to a new pipe size. By trial and error, you can determine that by using 4 inch for both pipes, a pressure drop of 10.04 psid can be obtained (see Figure 4).

Figure 4: Output Window for modified Control Valve Example

Conclusion

To meet the system requirements, use 4-inch pipe.

Heat Transfer in a Pipe

Summary

The objective of this example is to demonstrate how to calculate heat transfer in a pipe. For this example we will specify a mass flow rate and an inlet temperature and calculate the outlet temperature and total heat loss for a 100 foot section of pipe.

Topics Covered

- Modeling heat transfer in a single pipe

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Heat Transfer in a Pipe.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)

> i. After selecting, click Add to Model
>
> c. Temperature = 70 deg. F
>
> d. This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.

3. Open the Heat Transfer/Variable Fluids panel:

   a. Heat Transfer = Heat Transfer With Energy Balance (Single Fluid)

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Although one pipe is all that we are modeling, we still need junctions at each end to define the pressure and flow conditions for this pipe.
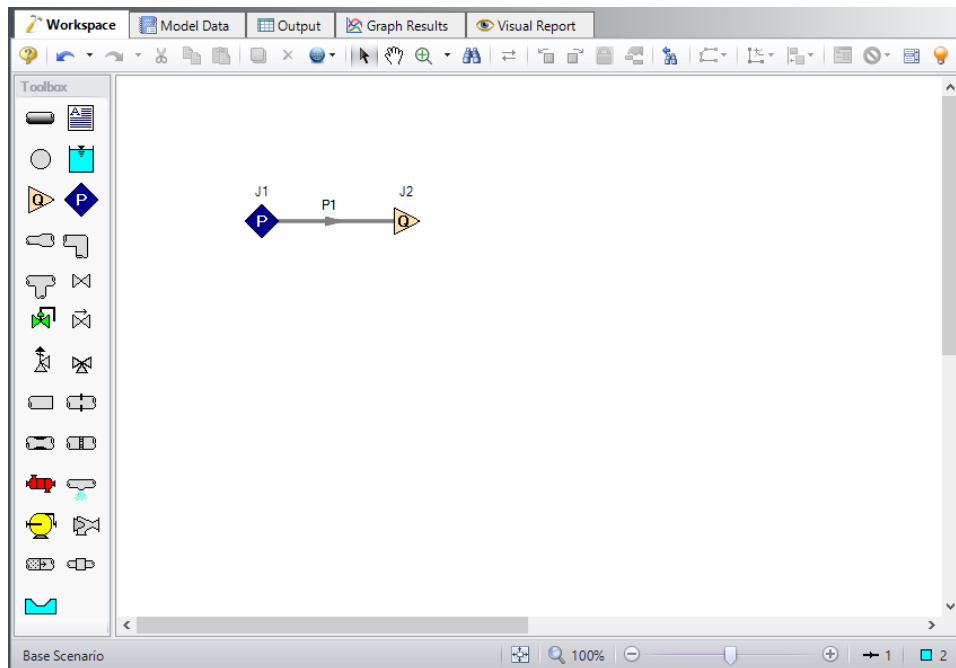
Figure 1: Workspace for Heat Transfer Problem

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

Because heat transfer was selected on the Heat Transfer/Variable Fluids panel in Analysis Setup, the Pipe Properties window has an additional tab to define: Heat Transfer. We will use Convective Heat

Transfer for this example. Fathom allows you to perform a layered heat transfer analysis with internal and external convection coefficients and conduction through multiple insulation layers. For this example, we will consider the internal convection, conduction through the pipe wall, and external convection as affecting heat transfer in our system.

Pipe P1

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 6 inch
    b. Type = STD (schedule 40)
    c. Friction Model Data Set = Standard
    d. Length = 300 feet
2. Heat Transfer tab
    a. Heat Transfer Model = Convective Heat Transfer
    b. Ambient Conditions
        i. Temperature = 32 deg. F
        ii. Fluid Velocity = 20 miles/hr
        iii. Fluid = Air
    c. Fluid Internal
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = Correlation
        iii. HT Area Ratio = 1
        iv. Convection Coefficient = Gnielinski
    d. Pipe Wall
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = Material Library
        iii. Conductivity = From Library
        iv. Thickness = 0.28 inches
        v. HT Area Ratio = 1
    e. External
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = Correlation
        iii. HT Area Ratio = 1
        iv. Convection Coefficient = Forced (Churchill-Bernstein)

The heat transfer window looks like Figure 2 below.

Figure 2: Pipe properties window for Heat Transfer Problem

Junction Properties

Now that the pipe is defined, specify the junction properties:

1. J1 Assigned Pressure
   a. Elevation = 0 feet
   b. Pressure = 50 psia
   c. Temperature = 150 deg. F
   d. Pressure Specification = Stagnation
2. J2 Assigned Flow
   a. Elevation = 0 feet
   b. Type = Outflow
   c. Flow Specification = Mass Flow Rate
   d. Flow Rate = 10 lbm/sec

Note: The temperature at J2 will not be used by Fathom because the outlet temperature of the pipe is defined by the inlet temperature pipe due to the Assigned Pressure junction and the heat loss through the pipe.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the Output toolbar or Tools menu. From the Display Parameters tab, Pipes selection, double-click the parameters Heat Flux, Heat Rate, Temperature Inlet, and Temperature Outlet in the left list to add these output parameters to this model. Click OK to return to the Output window.

The Output window contains all the data that was specified in the output control window. Because we added Inlet and Outlet temperatures to the output, we can see that the outlet temperature is 140.9 deg. F. The output is shown in Figure 3 below.

Note: If the default Output Control parameters are being displayed, a warning will be shown stating the *HGL, EGL, and head loss results may not be meaningful for variable density systems* as can be seen in Figure 3. If all head output parameters are hidden using Output Control, this warning will no longer appear.
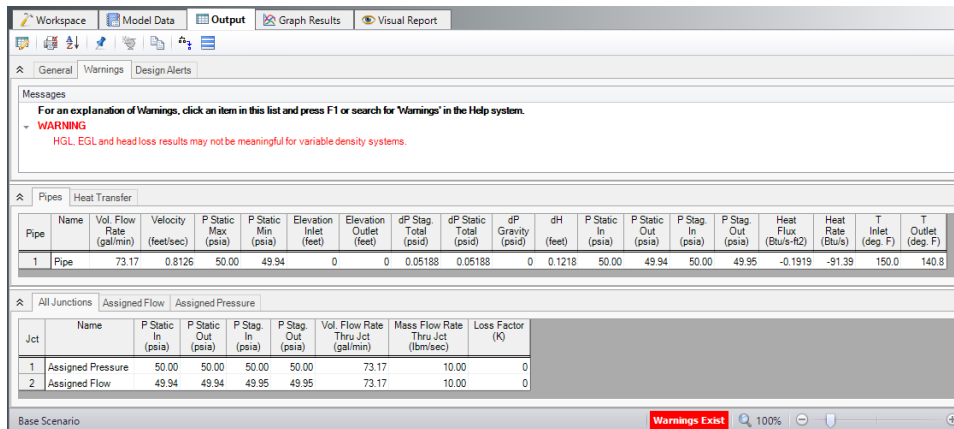
Figure 3: Output window for Heat Transfer Problem

Heat Exchanger System

Summary

This example will walk you through a simple calculation to size a pump for a closed loop heat exchanger system. First you will build the system with a fixed flow pump and then enter a pump curve that meets these requirements.

Topics Covered

- Sizing a pump for a specific system
- Entering a loss curve for a Heat exchanger
- Graphing system curves versus pump curves

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Heat Exchanger System.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard

b. Fluid = Water (liquid)

    i. After selecting, click Add to Model

c. Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Layout of Heat Exchanger Problem

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab

    a. Pipe Material = Steel - ANSI

    b. Pipe Geometry = Cylindrical Pipe

    c. Size = 4 inch

    d. Type = STD (schedule 40)

e. Friction Model Data Set = Standard

f. Lengths =

Pipe	Length (feet)
1	200
2	15
3	100
4	100
5	200
6	15
7	200

Junction Properties

Specify each junction by entering the following properties in each corresponding Junction Properties window:

1. J1 Heat Exchanger
   a. Inlet Elevation = 15 feet
   b. Loss Model = Resistance Curve
   c. Enter Curve Data =

Volumetric	Pressure
gal/min	psid
0	0
100	5
200	20

   d. Curve Fit Order = 2
   e. Click Generate Curve Fit Now
1. J2 & J7 Bend
   a. Inlet Elevation = 15 feet
   b. Type = Standard Elbow (knee, threaded)
   c. Angle = 90 Degrees
2. J3 & J6 Bend
   a. Inlet Elevation = 0 feet
   b. Type = Standard Elbow (knee, threaded)
   c. Angle = 90 Degrees

3. J4 Assigned Pressure
    a. Elevation = 0 feet
    b. Pressure = 100 psia
    c. Pressure Specification = Stagnation
4. J5 Pump
    a. Inlet Elevation = 0 feet
    b. Pump Model = Centrifugal (Rotodynamic)
    c. Analysis Type = Sizing
    d. Parameter = Volumetric Flow Rate
    e. Fixed Flow Rate = 100 gal/min

Figure 2: Heat Exchanger Loss Curve Fit Specification

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the

results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The Output window contains all the data that was specified in the output control window. Because we are interested in the pump requirements for this system, select the Pump Summary tab in the General (top) window. The pump data can also be found in the Junction window (bottom). If we are primarily interested in the head (or pressure) requirement, this is found in the summary window.

For this system, the head requirement of the pump is 7.281 psid (16.82 feet). At the assigned flow rate of 100.0 gal/min, the pressure drop across the heat exchanger was 5.000 psid (11.55 feet) so the total pipe and elbow loss was approximately 2.281 psi (5.27 feet). This output is shown in Figure 3a and Figure 3b.

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)
5	Show ...	Pump	100.0	13.89	7.281	16.82	100.0	N/A	0.4247	N/A	N/A	229.5	N/A

General | Warnings | Design Alerts | Pump Summary | Heat Exchanger Summary

Figure 3a: Pump Summary for Heat Exchanger Example

Jct	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP Stag. (psid)	dH (feet)	dT Loss (deg. F)	Heat Rate In (Btu/s)	T Inlet (deg. F)	T Outlet (deg. F)	T 2nd Inlet (deg. F)	T 2nd Outlet (deg. F)
1	Heat Exchanger	100.0	13.89	5.000	11.55	N/A	N/A	70.00	70.00	N/A	N/A

General | Warnings | Design Alerts | Pump Summary | Heat Exchanger Summary

Figure 3b: Heat Exchanger Summary for Heat Exchanger Example

Step 6. Add Pump Curve

Now that we know the general requirements of the pump, we can select an appropriate pump and enter this data. Open the Pump Properties window and enter the following data:

1. Pump Model = Centrifugal (Rotodynamic)
2. Analysis Type = Pump Curve
3. Enter Curve Data
   a. Flow Q Parameter = Volumetric
   b. Head/Pressure dH Parameter = Head

Volumetric	Head
gal/min	feet
0	20

Volumetric	Head
gal/min	feet
100	17
200	12

c. Curve Fit Order = 2
d. Click Generate Curve Fit Now

Step 7. Re-evaluate the Model

Select Run Model from the toolbar or Analysis menu. Because the pump head at 100 gal/min was slightly higher than required, the system will adjust with a slightly higher flow rate. The output of this run is shown in Figure 4.
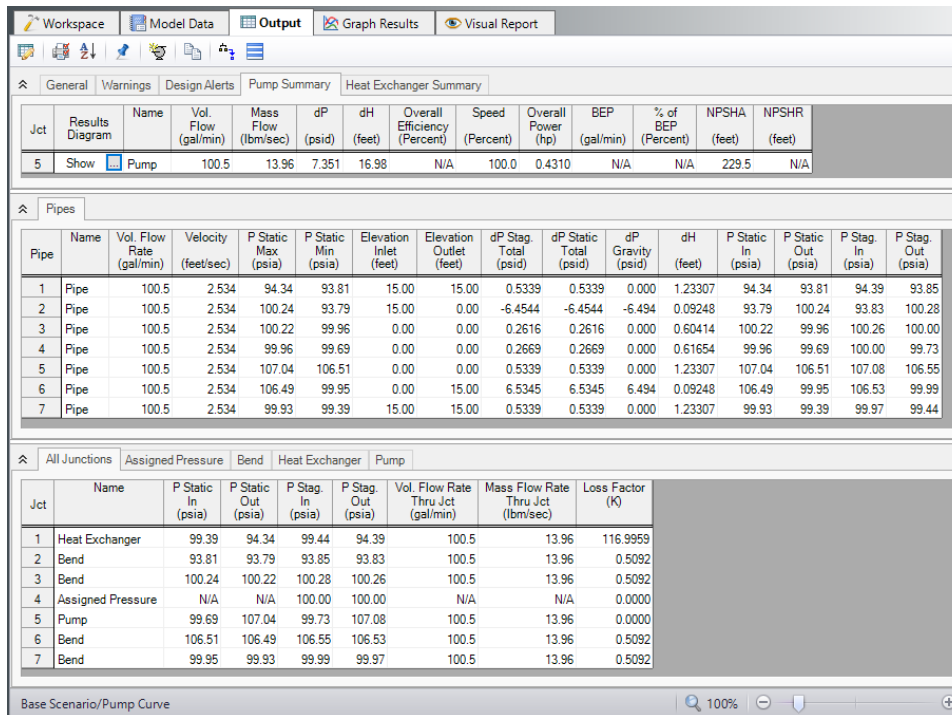
Figure 4: Output window for Heat Exchanger Example with real pump data

This system is a good example to demonstrate how to use the Graph Results Tab. Change to the Graph Results window by choosing it from the Window drop-down on the menu bar, clicking the Graph Results tab on the toolbar, or by pressing CTRL+G. Now the Graph Control window will be displayed on the Quick Access Panel as shown in Figure 5. Select Pump Curve vs. System Curve and Fathom automatically selects the data you want to graph.

Figure 5: Select Graph Data window for Heat Exchanger System

Click Generate to view this graph. This graph is shown in Figure 6. The red line shows the pump curve that we entered. This starts out with 20 feet of head available at 0 gal/min and decreases slowly with increasing flow rate. The blue line is the overall system curve. The head requirement increases significantly with flow rate. Most of the pressure loss (head loss) is through the heat exchanger but this graph also includes the pipe and elbow loss. The lines cross right at 100.5 gal/min which is the operating point shown in the output in Figure 4.
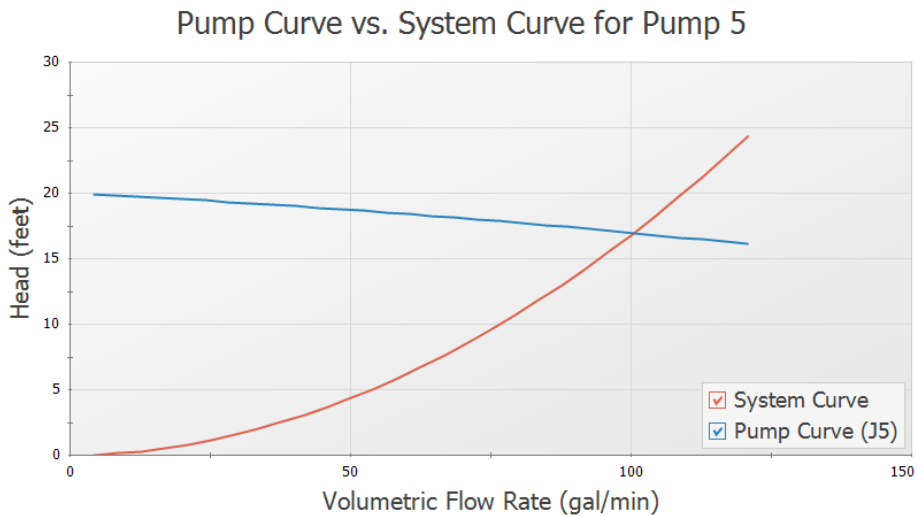
Figure 6: Pump Curve vs. System Curve for Heat Exchanger System

Spray Discharge System

Summary

The objective of this example is to find the minimum supply pressure needed to supply a system of eight spray discharge heads with at least 100 gal/min to each spray head. The system has one main supply line that branches into two supply lines that supply four sprayers each.

Topics Covered

- Using spray discharge junctions
- Iterating on a supply condition to reach a desired output condition
- Using Isometric Pipe Drawing Mode

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Spray Discharge System.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

      a.   Fluid Library = AFT Standard

      b.   Fluid = Water (liquid)

            i.   After selecting, click Add to Model

      c.   Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for Spray Discharge System

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1.   Pipe Model tab

      a.   Pipe Material = Steel - ANSI

      b.   Pipe Geometry = Cylindrical Pipe

      c.   Size = Use table below

      d.   Type = STD (schedule 40)

      e.   Friction Model Data Set = Standard

      f.   Lengths =

Pipe	Size	Length (feet)
1 & 2	8 inch	500
3	4 inch	9
4 - 11	1-1/2 inch	10

Note: The Global Edit tool found in the Edit menu can be used to enter the data for pipes P4-P11 all at once. The Guide at the top right of the window can be used to walk you through how to global edit if needed.

Junction Properties

1. J1 - J8 Spray Discharges
    1. Elevation = 10 feet
    2. Loss Model = Cd Spray (Discharge Coefficient)
    3. Geometry = Spray Nozzle
    4. Exit Properties = Pressure
    5. Exit Pressure = 0 psig
    6. Cd (Discharge Coefficient) = 0.6
    7. Discharge Flow Area = 0.5 inches2
2. J9 Assigned Pressure
    1. Elevation = 1 feet
    2. Pressure = 200 psig
    3. Pressure Specification = Stagnation
3. J10 Branch
    1. Elevation = 10 feet
4. J11 Branch & J12 Dead End
    1. Elevation 1 feet

Note that the supply pressure is what we are trying to solve for, but we must supply a guess to start. We will need to iterate on the pressure at junction J9 to determine what the minimum pressure should be.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open Output Control by selecting Output Control from the toolbar or Tools menu. Click the Junctions button, then add Vol. Flow Rate Net At Junction to the shown junction output. This output will show the volumetric flow rate discharged from the spray nozzle. Click OK to save the changes and close Output Control.

The Output window contains all the data that was specified in the Output Control window. In the Junction output window at the bottom of the page there is a special tab for Spray Discharge, as shown in Figure 2. Select this tab and examine the flow rate out of each of these junctions. All flow rates are above 100 gal/min so we must decrease the supply pressure and re-run the model. You should find that the supply pressure must be above 170.4 psig for the flow rate to meet the minimum flow rate at each spray nozzle.

This example may be solved directly using the GSC Module.

Note: The parameter Volumetric Flow Rate Through Junction is for junctions with a single inlet and outlet, and will thus display N/A for the Spray Discharge nozzles that aren't at the end of the branch. The parameter Volumetric Flow Rate Junction Net is used instead to display the total volumetric flow rate that is exiting each spray nozzle.

⌃ All Junctions	Assigned Pressure	Branch	Dead End	Spray Discharge						
Spray Discharge	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)	Vol. Flow Rate Jct Net (gal/min)
1	Spray Discharge	103.5	14.70	105.4	14.70	108.6	15.08	46.05	108.6
2	Spray Discharge	N/A	14.70	108.6	14.70	N/A	N/A	46.05	110.5
3	Spray Discharge	N/A	14.70	121.0	14.70	N/A	N/A	46.05	117.5
4	Spray Discharge	N/A	14.70	150.0	14.70	N/A	N/A	46.05	132.6
5	Spray Discharge	N/A	14.70	150.0	14.70	N/A	N/A	46.05	132.6
6	Spray Discharge	N/A	14.70	121.0	14.70	N/A	N/A	46.05	117.5
7	Spray Discharge	N/A	14.70	108.6	14.70	N/A	N/A	46.05	110.5
8	Spray Discharge	103.5	14.70	105.4	14.70	108.6	15.08	46.05	108.6

Figure 2: Junction Output for Spray Discharge System with the supply pressure set to 200 psig

Figure 3 below displays the results for the final supply pressure of 170.4 psig.

⌃ All Junctions	Assigned Pressure	Branch	Dead End	Spray Discharge						
Spray Discharge	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)	Vol. Flow Rate Jct Net (gal/min)
1	Spray Discharge	89.99	14.70	91.66	14.70	100.0	13.89	46.05	100.0
2	Spray Discharge	N/A	14.70	94.33	14.70	N/A	N/A	46.05	101.7
3	Spray Discharge	N/A	14.70	104.89	14.70	N/A	N/A	46.05	108.3
4	Spray Discharge	N/A	14.70	129.57	14.70	N/A	N/A	46.05	122.2
5	Spray Discharge	N/A	14.70	129.57	14.70	N/A	N/A	46.05	122.2
6	Spray Discharge	N/A	14.70	104.89	14.70	N/A	N/A	46.05	108.3
7	Spray Discharge	N/A	14.70	94.33	14.70	N/A	N/A	46.05	101.7
8	Spray Discharge	89.99	14.70	91.66	14.70	100.0	13.89	46.05	100.0

Figure 3: Junction Output for Spray Discharge System with the supply pressure set to 170.4 psig

Step 6. Use the Isometric Grid

AFT Fathom allows the user to place pipe or junction objects anywhere in the Workspace. Objects are placed on a 2D grid by default, as was the case with this example. At times it may be convenient to demonstrate the three-dimensional nature of a system. For example, if you are building a model based on isometric reference drawings. AFT Fathom includes an isometric grid for these cases. The isometric grid has three gridlines that are offset by 60 degrees, representing the *x*, *y*, and *z* axes. Figure 4 shows the spray discharge system built on an isometric grid. The completed model file drawn in isometric can be seen in the Examples folder and is named Spray Discharge Isometric.
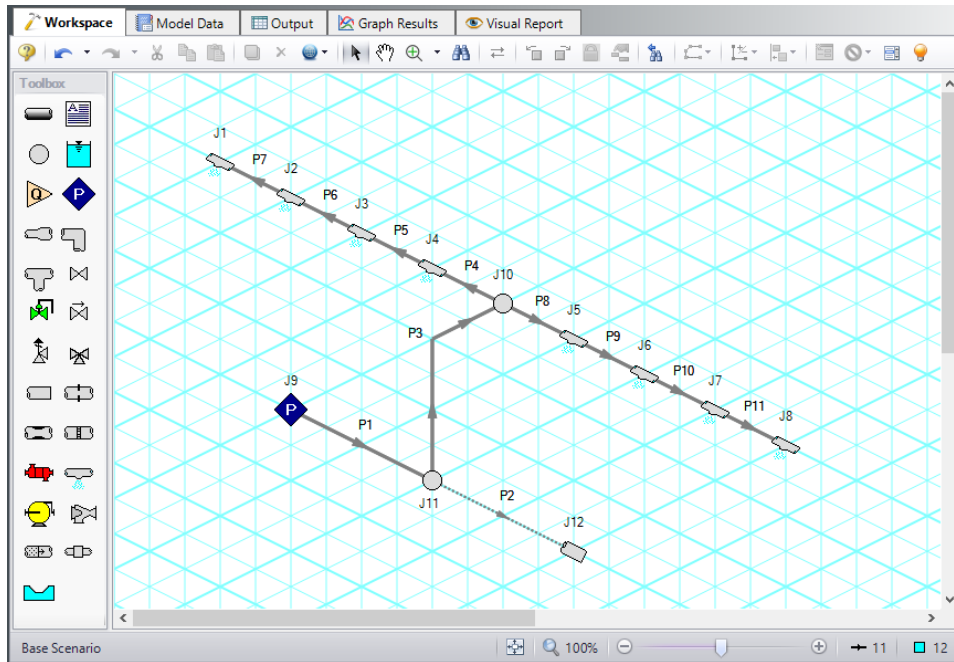
Figure 4: Spray discharge system with the isometric grid shown

You can enable the isometric grid in AFT Fathom by going to the Arrange menu. Under Pipe Drawing Mode, there are three options: 2D Freeform (default), 2D Orthogonal, and Isometric.

Creating the spray discharge system on an isometric grid will demonstrate how to use this feature:

1. Go to the File menu and select New.
2. Go to the Arrange menu again and under Pipe Drawing Mode, select Isometric
3. Place a Spray Discharge junction, J1, on the Workspace. You will notice that placing junctions onto the Workspace follows the usual rules, however, the visual appearance of the icon is more complex than in a 2D grid. Due to the increased number of axes, the preferred icon and rotation must be selected to obtain visual consistency.
4. Right click on J1 and select Customize Icon. Select the icon shown in Figure 5.

Note: If a junction in isometric mode has at least two pipes connected to it, the Auto-rotate Icon option can be selected instead of Customize Icon. Auto-rotate icon will automatically choose the icon orientation based on the connected pipes.
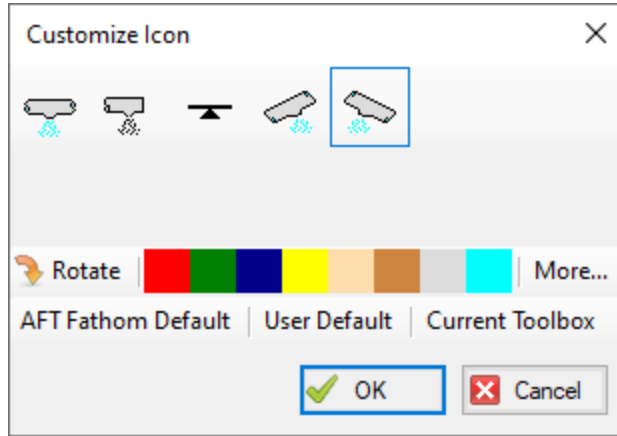
Figure 5: Right click on the junction to open the Customize Icon window and select the preferred icon and rotation

5. Copy J1 for the other spray discharges, J2-J8, to maintain the preferred icon and rotation.
6. Place the assigned pressure junction, J9, on the Workspace.
7. Place two branch junctions, J10 and J11, on the Workspace.
8. Place the dead end junction, J12, on the Workspace.
9. Right click on J12 and select Customize Icon. Notice that you may need to use the Rotate button to get the preferred icon and rotation, as shown in Figure 6.
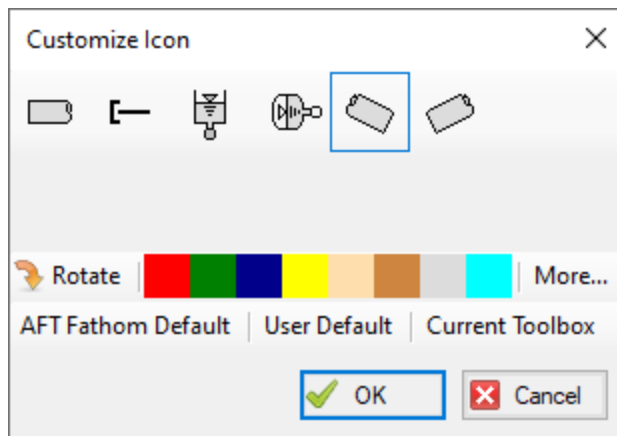
Figure 6: Use the Rotate button to obtain the preferred icon and rotation

10. Draw Pipes 1 and 2, as shown in Figure 4.
11. Draw Pipe 3 from J11 to J10. A red-dashed preview line will show how the pipe will be drawn on

the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse.

Figure 7 shows Pipe 3 being drawn with the preview line.
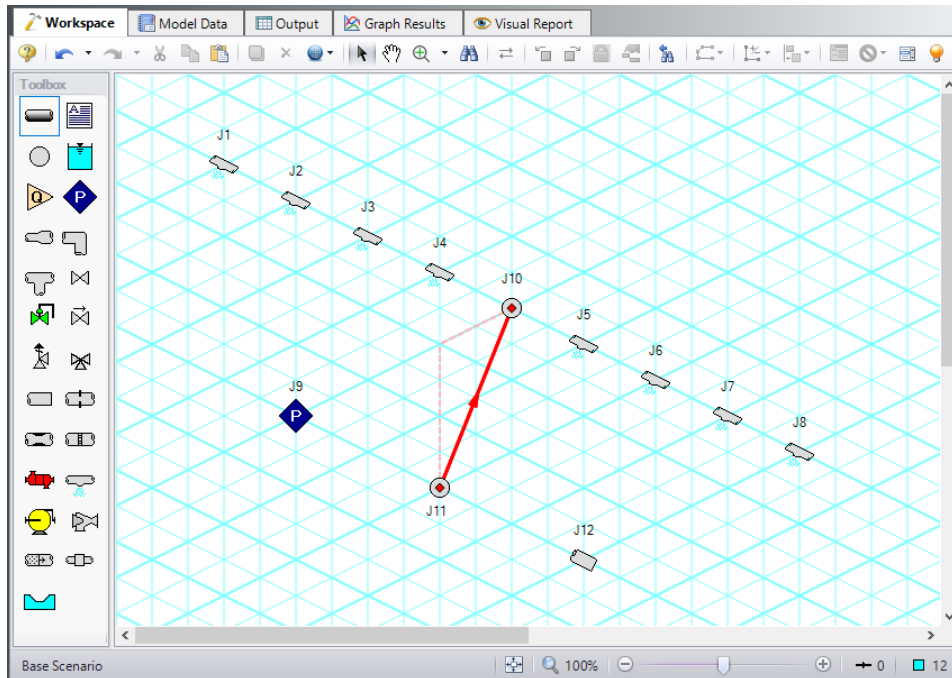
Figure 7: A preview line shows when drawing or adjusting pipes

Note: You can hold the ALT key while adjusting a pipe by the endpoint to add an additional vertex. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

12. Draw Pipes 4-11, as shown in Figure 4.

13. The grid can be shown or turned off in the Arrange menu. Figure 8 shows the completed model with the grid not shown.
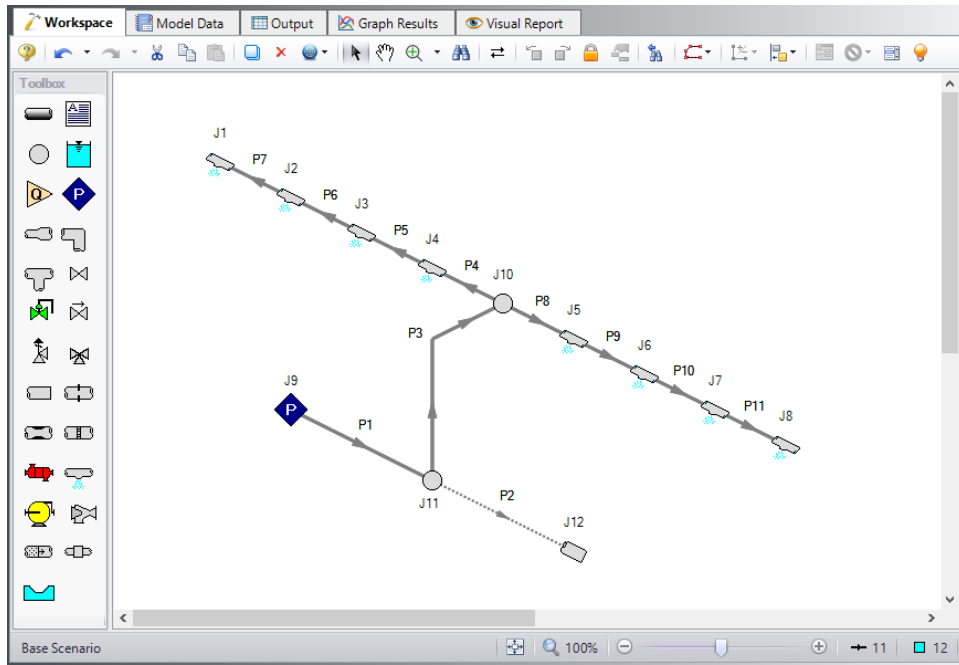
Figure 8: Isometric spray discharge system with the grid not shown

Pump Sizing and Selection with Flow Control Valves

Summary

This example will demonstrate how to size and select pumps in a heat exchanger system with flow control valves. The system will pump water through a heat exchanger system from a supply tank at one elevation and pressure to a receiving tank at a higher elevation and pressure. There will be two flow control valves operating in parallel. The control valves require a minimum pressure drop of 5 psi in order to control the flow rate.

Topics Covered

- Using flow control valves
- Entering and changing pump data

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Pump Sizing and Selection with Flow Control Valves.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

     a.   Fluid Library = AFT Standard

     b.   Fluid = Water (liquid)

          i.   After selecting, click Add to Model

     c.   Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for Pump Selection with Flow Control Valves Example

The system is in place but now the properties of the objects must be entered.

Pipe Properties

1.   Pipe Model tab

     a.   Pipe Material = Steel - ANSI

     b.   Pipe Geometry = Cylindrical Pipe

     c.   Size = 2 inch

d.   Type = STD (schedule 40)

e.   Friction Model Data Set = Standard

f.   Length =

Pipe	Length (feet)
1	20
2	50
3	60
4	20
5	100
6	20
7	60
8	20
9	100

Junction Properties

1.   J1 Reservoir
    a.   Name = Supply Tank
    b.   Liquid Surface Elevation = 5 feet
    c.   Liquid Surface Pressure = 10 psig
    d.   Pipe Depth = 5 feet

2.   J2 Pump
    a.   Inlet Elevation = 0 feet
    b.   Pump Model = Centrifugal (Rotodynamic)
    c.   Analysis Type = Sizing
    d.   Parameter = Volumetric Flow Rate
    e.   Fixed Flow Rate = 200 gal/min

3.   J3 Tee
    a.   Elevation = 0 feet
    b.   Loss Model = Simple

4.   J4 & J7 Control Valve
    a.   Inlet Elevation = 0 feet
    b.   Valve Type = Flow Control (FCV)
    c.   Control Setpoint = Volumetric Flow Rate
    d.   Flow Setpoint = 100 gal/min

5.   J5 & J8 Heat Exchanger

      a.   Inlet Elevation = 0 feet

      b.   Loss Model = Resistance Curve

      c.   Enter Curve Data =

Volumetric	Pressure
gal/min	psid
100	10

      d.   Click Fill as Quadratic

      e.   Curve Fit Order = 2

      f.   Click Generate Curve Fit Now

1. J6 Bend

      a.   Inlet Elevation = 0 feet

      b.   Type = Standard Elbow (knee, threaded)

      c.   Angle = 90 Degrees

2. J9 Reservoir

      a.   Name = Receiving Tank

      b.   Liquid Surface Elevation = 10 feet

      c.   Liquid Surface Pressure = 30 psig

      d.   Pipe Depth = 10 feet

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Resolve Reference Pressure Needed

Click Run Model on the toolbar or from the Analysis menu.

The Reference Pressure Needed window will appear (see Figure 2). This window is displayed when a model is constructed that does not have a unique solution. A common occurrence of this is when a model contains one or more sections that are completely bounded by known flow rates.

In this case, because the pump has been designated as an assigned flow and the downstream control valves are flow control valves, there is no way to uniquely determine the pressure between the pump and the control valves without an additional reference pressure. The reference pressure is the pressure from which the other pressures in the system are derived. There can be more than one reference pressure, but there always has to be at least one.

For a more in-depth discussion of reference pressures, see the Fathom help file topic: Role of Pressure Junctions - Detailed Discussion
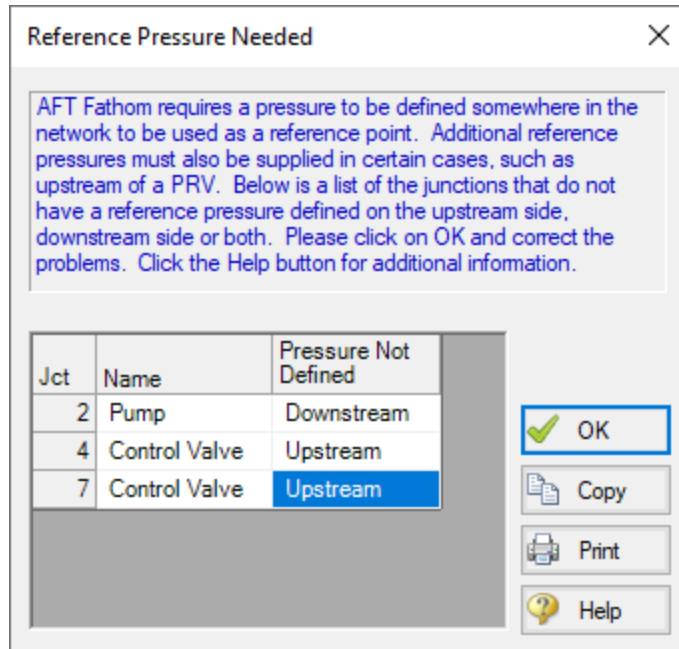
Figure 2: Reference Pressure Needed window

Open the J7 Control Valve Property window, and make the following changes.

1.   Valve Type = Constant Pressure Drop (PDCV)
2.   Control Setpoint = Pressure Loss
3.   Pressure Drop = 5 psid

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

We can look at the pump summary and see that the head at 200 gal/min needs to be at least 167.3 feet (see Figure 3). The control valves both have enough pressure drop to control flow (see Figure 4).

Figure 3: Pump Summary window

Jct	Name	Valve Type	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP Stag. (psid)	dH (feet)	P Static In (psia)	Cv	K	Valve State
4	Control Valve	FCV	100.0	13.89	6.839	15.79	74.19	38.23	11.118	Open
7	Control Valve	PDCV	100.0	13.89	5.000	11.55	72.35	44.71	8.129	Open

Tabs: General, Warnings, Design Alerts, Pump Summary, Valve Summary, Heat Exchanger Summary

Figure 4: Valve Summary window

Step 7. Enter a Pump Curve

Now we need to specify a real pump. We will enter in some pump data and create a pump curve with the curve fitting routines in Fathom. Use the Scenario Manager on the Quick Access Panel to create a child of the base scenario with a name of Pump Curve. Then in the Pump Curve scenario, open the Pump Properties window and enter the following data:

1. J2 Pump
   a. Pump Model = Centrifugal (Rotodynamic)
   b. Analysis Type = Pump Curve
   c. Enter Curve Data =

Volumetric	Head
gal/min	feet
0	175
200	168
400	120

   c. Curve Fit Order = 2
2. J7 Control Valve
   1. Valve Type = Flow Control (FCV)
   2. Control Setpoint = Volumetric Flow Rate
   3. Flow Setpoint = 100 gal/min

Re-run the model and verify that the flow rate and pressure drop across the control valves is still within the limits.

Freon Delivery System with Heat Transfer

Summary

This example will model a Freon delivery system to supply Freon at 75 deg. F. As the engineer you will evaluate two candidate refrigerants (Refrigerant 11 and 123) to determine the best choice. We will also calculate the temperature of the hot water supply (i.e., heat exchanger Secondary Fluid) needed for the heat exchangers.

Topics Covered

- Modeling heat transfer
- Entering a resistance curve for a Heat Exchanger

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Freon Delivery System with Heat Transfer.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

      a.   Fluid Library = NIST REFPROP

      b.   Fluid = R11 (after selecting, click Add to Model)

      c.   Fluid Phase = Liquid

      d.   Pressure = 150 psig

      e.   Temperature = -50 deg. F

3.   Open the Heat Transfer/Variable Fluids panel

      a.   Select Heat Transfer With Energy Balance (Single Fluid)

This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.

Note: When Heat Transfer is enabled the fluid properties in the Analysis Setup window are only used as an initial guess. The final fluid properties will be calculated by the Solver.

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
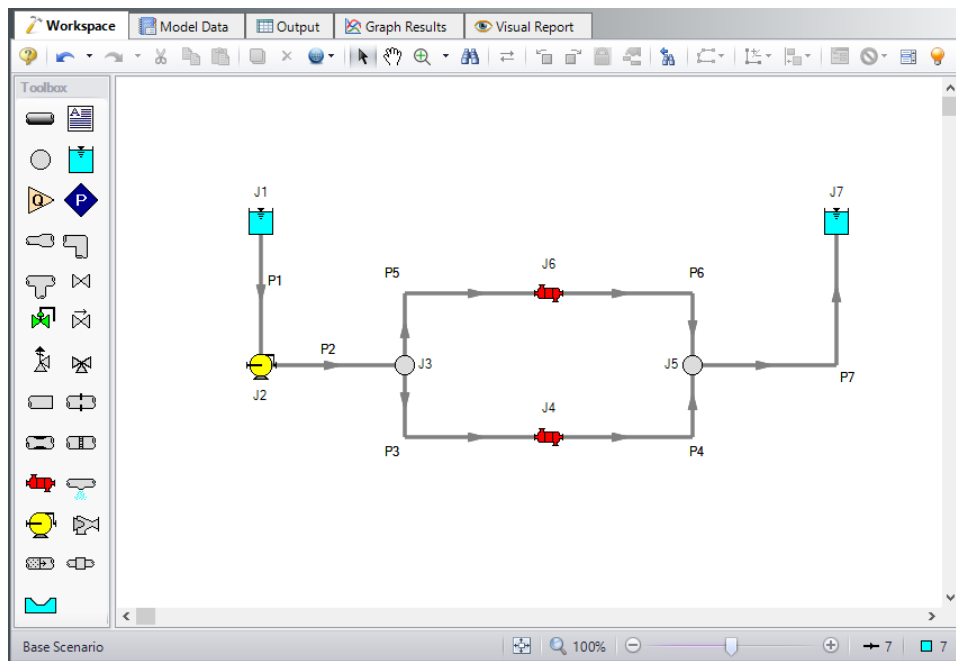
Figure 1: Model for Freon Delivery System

Now enter the properties for the pipes and the junctions.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 3 inch
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Lengths =

Pipe	Length (feet)
1 - 2	10
3 - 6	5
7	30

2. Fittings & Losses tab
    a. P3 - P6 only
        i. Click Specify Fittings & Losses
        ii. Bends tab
        iii. Standard Threaded, General, 90 deg. (C) Quantity = 1
        iv. Click OK
3. Heat Transfer tab (Figure 2)
    a. Heat Transfer Model = Convective Heat Transfer
    b. Resistance Geometry = Radial
    c. Number of Insulation Layers = 1 External
    d. Ambient Conditions
        i. Temperature = 60 deg. F
        ii. Fluid Velocity = 5 miles/hr
        iii. Fluid = Air
    e. Fluid Internal
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = Correlation
        iii. HT Area Ratio = 1
        iv. Convection Coefficient = Gnielinski
    f. Pipe Wall
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = Material Library
        iii. Conductivity = From Library
        iv. Thickness = 0.216 inches
        v. HT Area Ratio = 1
    g. Insulation #1 (external)
        i. Apply = Checked
        ii. Conductivity/Convection Data Source = User Specified

    iii. Conductivity = 0.4 Btu/hr-ft-R

    iv. Thickness = 1 inches

    v. HT Area Ratio = 1

  h. External

    i. Apply = Checked

    ii. Conductivity/Convection Data Source = Correlation

    iii. HT Area Ratio = 1

    iv. Convection Coefficient = Forced (Churchill-Bernstein)

Figure 2: Heat Transfer tab defined in the Pipe Properties window

Junction Properties

1. J1 Reservoir

  a. Name = Supply Reservoir

  b. Liquid Surface Elevation = 10 feet

  c. Liquid Temperature = -50 deg. F

  d. Update Density with Temperature = Checked

  e. Liquid Surface Pressure = 150 psig

  f. Pipe Depth = 0 feet

2. J2 Pump

  a. Inlet Elevation = 2 feet

  b. Pump Model = Centrifugal (Rotodynamic)

    c.   Analysis Type = Pump Curve

    d.   Enter Curve Data =

Volumetric	Head
gal/min	feet
0	25
200	24
500	15

    d.   Curve Fit Order = 2

3.   J3 & J5 Branches

    a.   Elevation = 2 feet

4.   J4 & J6 Heat Exchangers

    a.   Inlet Elevation = 2 feet

    b.   Loss Model tab

        i.   Loss Model = Resistance Curve

        ii.   Enter Curve Data =

Volumetric	Pressure
gal/min	psid
250	2

        iii.   Click Fill As Quadratic

        iv.   Curve Fit Order = 2

        v.   Click Generate Curve Fit Now

    c.   Thermal Data tab

        i.   Thermal Model = Counter-Flow

        ii.   Heat Transfer Area = 80 feet2

        iii.   Overall Heat Transfer Coefficient = 500 Btu/hr-ft-R

        iv.   Flow Rate = 30 lbm/sec

        v.   Specific Heat = 1 Btu/lbm-R

        vi.   Inlet Temperature = 200 deg. F

Note: This secondary fluid inlet temperature will be the initial guess.

5.   J7 Reservoir

    a.   Name = Receiving Reservoir

    b.   Liquid Surface Elevation = 10 feet

    c.   Liquid Temperature = 75 deg. F

    d.   Update Density with Temperature = Checked

e. Liquid Surface Pressure = 150 psig

f. Pipe Depth = 0 feet

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

When the Output is first shown, note that if the default Output Control parameters are being displayed, a warning will be shown stating the *HGL, EGL and head loss results may not be meaningful for variable density systems*. If all head output parameters are hidden using Output Control, this warning will no longer appear.

Open Output Control by selecting Output Control from the toolbar or Tools menu. Select the Pipes button to display the pipe output parameters, and remove the Head Loss (dH) from the list of shown output by highlighting the parameter and selecting Remove under the table. Next add the Temperature Inlet and Temperature Outlet to the shown output by double-clicking those parameters in the list on the left.

The Output window contains all the data that was specified in the output control window. The output from this model is shown in Figure 3. Now look at the outlet temperature of the last pipe (pipe 7). This is the temperature of the delivered Freon. The temperature is about 114.99 deg. F. This is significantly higher than the required temperature of 75 deg. F.

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	T Inlet (deg. F)	T Outlet (deg. F)
1	Pipe	388.2	16.849	165.0	161.6	10.000	2.000	-3.4708	-3.4708	-5.669	161.6	165.0	164.7	168.2	-50.00	-49.96
2	Pipe	388.2	16.848	178.8	176.6	2.000	2.000	2.2002	2.2002	0.000	178.8	176.6	181.9	179.7	-49.96	-49.92
3	Pipe	194.1	8.425	178.9	178.2	2.000	2.000	0.7049	0.7049	0.000	178.9	178.2	179.7	179.0	-49.92	-49.88
4	Pipe	222.3	9.648	176.9	176.1	2.000	2.000	0.7911	0.7911	0.000	176.9	176.1	177.8	177.0	115.06	115.04
5	Pipe	194.1	8.425	178.9	178.2	2.000	2.000	0.7049	0.7049	0.000	178.9	178.2	179.7	179.0	-49.92	-49.88
6	Pipe	222.3	9.648	176.9	176.1	2.000	2.000	0.7911	0.7911	0.000	176.9	176.1	177.8	177.0	115.06	115.04
7	Pipe	444.4	19.286	173.4	161.1	2.000	10.000	12.3188	12.3188	4.953	173.4	161.1	177.0	164.7	115.04	114.99

Figure 3: Pipe Output window for Freon Delivery System with water supply at 200 deg. F

Now iterate on the temperature of the secondary fluid in the heat exchangers to find the temperature required to produce an outlet temperature of 75.00 deg. F for pipe 7. The answer for this should be 138.5 deg. F. The output for this run in shown in Figure 4.

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	T Inlet (deg. F)	T Outlet (deg. F)
1	Pipe	389.1	16.887	165.0	161.6	10.000	2.000	-3.4609	-3.4609	-5.669	161.6	165.0	164.7	168.2	-50.00	-49.96
2	Pipe	389.1	16.887	178.7	176.5	2.000	2.000	2.2101	2.2101	0.000	178.7	176.5	181.9	179.7	-49.96	-49.92
3	Pipe	194.6	8.444	178.9	178.2	2.000	2.000	0.7081	0.7081	0.000	178.9	178.2	179.7	179.0	-49.92	-49.88
4	Pipe	214.7	9.319	176.9	176.1	2.000	2.000	0.7675	0.7675	0.000	176.9	176.1	177.8	177.0	74.98	74.97
5	Pipe	194.6	8.444	178.9	178.2	2.000	2.000	0.7081	0.7081	0.000	178.9	178.2	179.7	179.0	-49.92	-49.88
6	Pipe	214.7	9.319	176.9	176.1	2.000	2.000	0.7675	0.7675	0.000	176.9	176.1	177.8	177.0	74.98	74.97
7	Pipe	429.2	18.627	173.5	161.2	2.000	10.000	12.2925	12.2925	5.139	173.5	161.2	177.0	164.7	74.97	74.96

Figure 4: Output window for Freon Delivery System with water supply at 138.5 deg. F

Now change the fluid on the Fluid panel to R123 and repeat the analysis. For R123, the temperature water supply to the heat exchangers necessary to meet the temperature requirement should be higher, around 154.7 deg. F. If water supply temperature is the main criteria, Refrigerant 11 would be the best choice.

Cooling System

Summary

The objective of this example is to determine how much flow will be delivered to the heat exchangers in the system. A secondary objective is to determine the best pump for the system.

Topics Covered

- Modeling a closed loop system
- Using a pressure reducing valve
- Generating a pump curve
- Generating a resistance curve for a heat exchanger
- Importing a pump curve to obtain pump data

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Cooling System.fth*
- *US - Cooling System Pump A.txt*
- *US - Cooling System Pump B.txt*
- *US - Cooling System Pump C.txt*

Note: If you open the completed *US - Cooling System.fth* model file, you will notice a child scenario named **Initial Model for ANS Quick Start Chapter 3**. This child scenario is used as the starting point for a model in the ANS Quick Start Guide. This scenario can be disregarded when comparing your model to this completed model.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Dowtherm J (after selecting, click Add to Model)
   c. Temperature = 150 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

To add the segment into pipe P22, use the Pipe Segments tool on the Arrange menu.
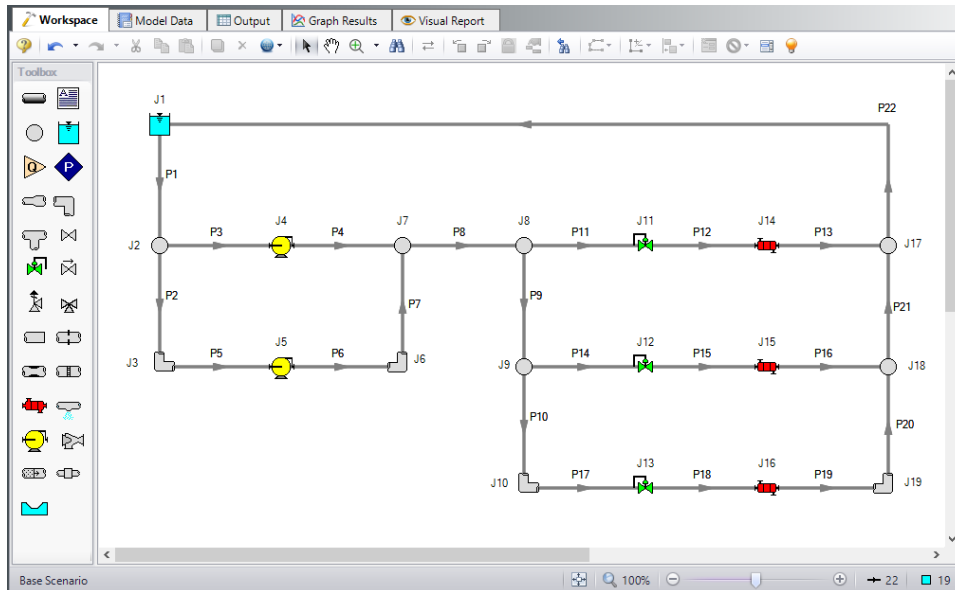
Figure 1: Workspace window for Cooling System Problem

Pipe Properties

1. Pipe Model tab
   a. Pipe Material = Steel - ANSI
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = Use table below
   d. Type = STD (schedule 40)
   e. Friction Model Data Set = Standard
   f. Lengths = Use table below

Pipe	Size	Length (feet)
1	6 inch	20
2, 7, 9-10, 20-21	4 inch	3
3-6, 12-13, 15-16, 18-19	4 inch	5
11, 14, 17	4 inch	10
8, 22	6 inch	200

Junction Properties

1. J1 Reservoir
   a. Liquid Surface Elevation = 10 feet
   b. Liquid Surface Pressure = 40 psig
   c. Pipe Depth = 10 feet
2. All Branches
   a. Elevation = 2 feet
3. All Bends
   a. Inlet Elevation = 2 feet
   b. Type = Standard Elbow (knee, threaded)
   c. Angle = 90 Degrees
4. J4 & J5 Pumps
   a. Inlet Elevation = 2 feet
   b. Pump Model = Centrifugal (Rotodynamic)
   c. Analysis Type = Pump Curve
   d. Enter Curve Data =

Volumetric	Head
gal/min	feet
0	70
500	60

Volumetric	Head
gal/min	feet
1000	40

      e.   Curve Fit Order = 2

5.   All Control Valves

      a.   Inlet Elevation = 2 feet

      b.   Valve Type = Pressure Reducing (PRV)

      c.   Control Setpoint = Pressure, Static

      d.   Setpoint = 50 psig

6.   All Heat Exchangers

      a.   Inlet Elevation = 2 feet

      b.   Loss Model = Resistance Curve

      c.   Enter Curve Data =

Volumetric	Pressure
gal/min	psid
500	10

      d.   Click Fill As Quadratic

      e.   Curve Fit Order = 2

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The output is shown below in Figure 2. The pump summary shows that the head requirement for the pumps is 60.48 feet and 62.42 feet. The heat exchanger summary shows that the flow rates through the heat exchangers range from 290.9 gal/min to 307.8 gal/min.

Figure 2: Output Summary Windows for Cooling System example

Step 6. Evaluate Other Pumps

The customer for this project wants an evaluation of three other pumps to determine which uses the least energy. The pump curve for each of these pumps is found in a text file.

For each of the pumps follow the following set of steps:

1. Create a child of the base scenario in the Scenario Manager on the Quick Access Panel for Pump A, Pump B, and Pump C.
2. Load scenario Pump A, open the Pump Properties window, click Enter Curve Data
3. Click Edit Table and select Import From File
4. Change the file format to Text (*.txt) then select the data file **US - Cooling System Pump A.txt** and click Open.
5. Using the Comma Separator, click OK.
6. With a Curve Fit Order of 2, click Generate Curve Fit Now, then click OK
7. Select the second pump and set all the properties equal to the first pump.

Run each scenario and take note of the overall pump power in the pump summary window. The data should match the following:

Pump	J4	J5
A	5.755 hp	4.509 hp
B	6.866 hp	5.489 hp
C	6.546 hp	4.403 hp

From this data, Pump A is the best choice, because it consumes the least power.

Housing Project

Summary

The objective of this example is to model a pipeline system to supply water to a new housing development. As the engineer you will evaluate the design to find the smallest pipe size that will still meet the water supply requirements.

No Fire Conditions:

Each house requires 10 gal/min at a minimum of 60 psig.

Fire Conditions:

Each house requires 2 gal/min with no minimum pressure. One fire hydrant may be open at a time and requires 100 gal/min at 90 psig static.

Topics Covered

- Using Scenario Manager

Required Knowledge

This example assumes the user has already worked through the <u>Beginner - Three Reservoir</u> example, or has a level of knowledge consistent with that topic. You can also watch the <u>AFT Fathom Quick Start Video Tutorial Series</u> on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Housing Project - Initial.fth***
- ***US - Housing Project.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the ***US - Housing Project- Initial.fth*** model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder**.

The model should appear as shown in Figure 1.
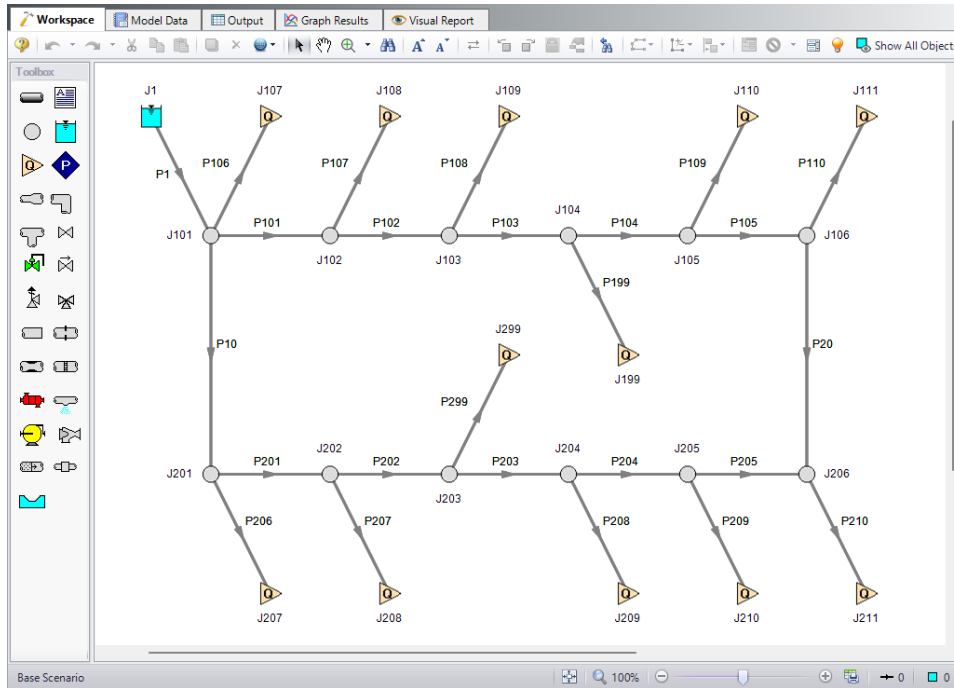
Figure 1: Initial model for Housing Project example

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 60 deg. F

Step 4. Define the Pipe and Junction Properties

The fire hydrants are both turned off to begin the problem. Select the Optional tab in the Assigned Flow Properties window and select Flow Off to close the hydrant. Do this for both hydrants.

Pipe Properties

All the mains are to be the same size. We will try 3, 3 ½ , and 4 inches to see which one meets the requirements. Enter 3-inch pipe for the main and hydrant supply lines in the base case.

1. Pipe Model tab
   a. Pipe Material = PVC - ASTM
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = Use table below
   d. Type = schedule 40
   e. Friction Model Data Set = Standard
   f. Lengths = Use table below

Pipe	Size	Length (feet)
1	3 inch	2000
10 & 20	3 inch	300
101-105 & 201-205	3 inch	200
199 & 299	3 inch	10
206	1 inch	150
108 & 207	1 inch	125
106, 110, 208, & 210	1 inch	100
107, 109, & 209	1 inch	75

Junction Properties

1. J1 Reservoir
   a. Name = West Lake
   b. Liquid Surface Elevation = 235 feet
   c. Liquid Surface Pressure = 0 psig
   d. Pipe Depth = 0 feet
2. Branches
   a. Elevation = Use table below

Branch	Elevation (feet)
203	-10

Branch	Elevation (feet)
102-105 & 204	-5
101, 106, 202, & 205	0
201 & 206	5

3. Assigned Flows
   a. Elevation = Use table below
   b. Type = Outflow
   c. Flow Specification = Volumetric Flow Rate
   d. Flow Rate = Use table below

Assigned Flow	Name	Elevation (feet)	Flow Rate (gal/min)	Special Conditions
107	Browns	10	10	None
108	Peters	20	10	None
109	Chambers	35	10	None
110	Fords	15	10	None
111	Kellys	20	10	None
199	North Hydrant	5	100	Flow Off
207	Norths	20	10	None
208	Stewarts	-15	10	None
209	Coles	30	10	None
210	Allens	10	10	None
211	Burns	5	10	None
299	South Hydrant	0	100	Flow Off

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

The flows will be varied based on the different scenarios examined. To do this, we need to create child scenarios. We will use the Scenario Manager to create the designs we need to evaluate.

Step 5. Manage the Scenarios

1. In the Scenario Manager, create a child of this scenario by right-clicking on the Base Scenario and selecting Create Child. Name this child scenario 3 inch Pipe. This will be the parent scenario for all the runs with 3 inch main pipes. Alternatively, you can select the Create Child icon 🔲 on the Scenario Manager on the Quick Access Panel to create this child scenario.

2. Select the 3 inch Pipe scenario and create another child, this one named Fire Flow. set the flow rate at each house to 2 gal/min instead of 10 gal/min. This will be the flow rate demand that needs to be met at the houses when either hydrant is open.

3. Next, create two children of this scenario named North Hydrant Fire and South Hydrant Fire. Load the North Hydrant Fire scenario from the Scenario Manager. Open the properties window of the North Hydrant and on the Optional tab, toggle the flow on by changing the special condition to None.

4. Now load the South Hydrant Fire scenario turn on the South Hydrant.

5. You now have all the scenarios that need to be evaluated for each pipe size but we still have two more pipe sizes that we need to investigate.

6. Right-click the 3 inch Pipe scenario and select Clone With Children.

7. Name this scenario 3.5 inch Pipe.

8. Repeat this process to create another set of scenarios named 4 inch Pipe. The Scenario Manager on the Quick Access Panel should look like Figure 2.
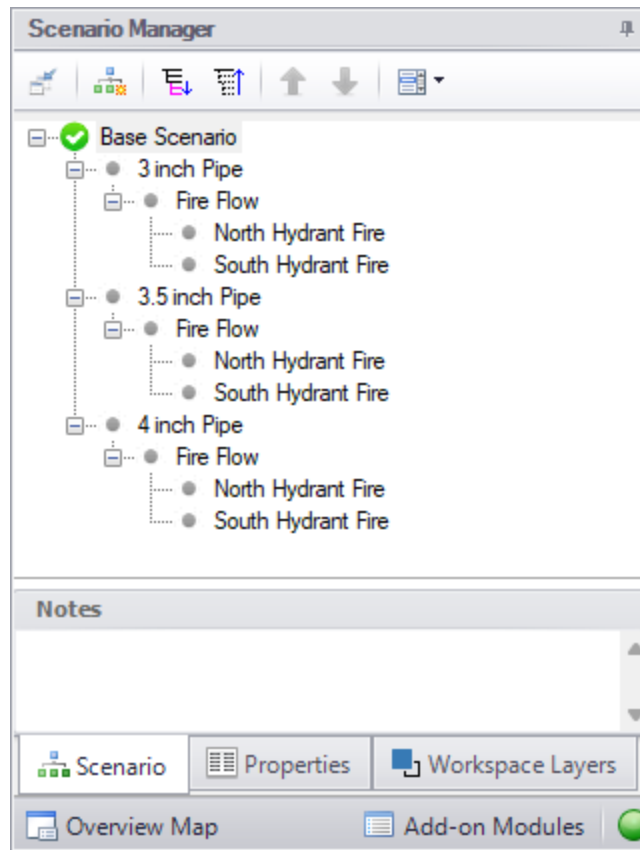
Figure 2: Scenario Manager for Water Supply to Housing Project

After creating the scenarios for the 3.5 inch pipe and 4 inch pipe, we need to open these scenarios and change the pipe sizes. This is accomplished most quickly with the Global Pipe Edit window.

1.   Load the 3.5 inch Pipe scenario.
2.   Select Global Edit then Global Pipe Edit from the toolbar or Edit menu.
3.   Click the radio button for Pipe Template at the top, if not already selected, and click Select Pipe Data.
4.   Change the Size to 3-1/2 inch, click OK.
5.   In the Parameters to Change (Select in List) area, check the box next to Nominal Size 3-1/2 inch.
6.   Since we only want to apply this to certain pipes, select Special above the Pipe List.
7.   Change the Selection Type to Pipe Input Properties.
8.   For Pipe Property, select Pipe Inner Diameter.
9.   For Range, enter From 2.9 to 3.1 inches.
10.   Click Select Pipes and note that 15 of 25 pipes were selected, these are all the pipes that are currently 3 inches. Click OK.
11.   Click Apply Selections. A message should appear to inform you that changes were successively applied.
12.   Click OK.

Repeat this process for the 4 inch Pipe scenario but changing the pipe size to 4 inch.

Step 6. Specify Output

Before running the model, load the Base Scenario to set the Output Control. Open the Output Control window by selecting Output Control from the toolbar or Tools menu. This window allows the user to specify the output from the Fathom simulation. The window starts with a default set of outputs. Often the default output is sufficient. You may also specify a title for the model. Click on the General tab and add a title of **Housing Project Example**. On the Display Parameters tab, select the Junctions button and change the static and stagnation pressure units to psig. Click OK to save and close the Output Control

Now load the 3 inch Pipe scenario.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

The output from this model is shown in Figure 3. For the 3 inch Pipe scenario, with no hydrants open, the flow to each house is 10 gal/min and the static pressure must be at least 60 psig. The pressure at each house is above 60 psig, so the system with 3 inch mains would be sufficient.

Assigned Flow	Name	P Static In (psig)	P Static Out (psig)	P Stag. In (psig)	P Stag. Out (psig)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
107	Browns	76.56	76.56	76.65	76.65	10.00	1.391	0
108	Peters	72.40	72.40	72.49	72.49	10.00	1.391	0
109	Chambers	64.31	64.31	64.40	64.40	10.00	1.391	0
110	Fords	73.92	73.92	74.01	74.01	10.00	1.391	0
111	Kellys	71.03	71.03	71.12	71.12	10.00	1.391	0
X199	North Hydrant	80.45	80.45	80.45	80.45	0.00	0.000	0
207	Norths	70.31	70.31	70.41	70.41	10.00	1.391	0
208	Stewarts	85.87	85.87	85.96	85.96	10.00	1.391	0
209	Coles	66.73	66.73	66.82	66.82	10.00	1.391	0
210	Allens	75.99	75.99	76.08	76.08	10.00	1.391	0
211	Burns	77.51	77.51	77.60	77.60	10.00	1.391	0
X299	South Hydrant	82.54	82.54	82.54	82.54	0.00	0.000	0

Figure 3: Output window for 3 inch mains with no fire

On the Scenario Manager, load the North Hydrant Fire scenario under the 3 inch Pipe scenario. Run the model, and look at the output. This output is shown in Figure 4. Although the hydrant receives 100 gal/min, the supply pressure is less than the required 90 psig.

Assigned Flow	Name	P Static In (psig)	P Static Out (psig)	P Stag. In (psig)	P Stag. Out (psig)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
107	Browns	71.98	71.98	71.99	71.99	2.000	0.2781	0
108	Peters	66.60	66.60	66.60	66.60	2.000	0.2781	0
109	Chambers	58.99	58.99	58.99	58.99	2.000	0.2781	0
110	Fords	66.95	66.95	66.96	66.96	2.000	0.2781	0
111	Kellys	64.98	64.98	64.98	64.98	2.000	0.2781	0
199	North Hydrant	70.99	70.99	71.11	71.11	100.000	13.9058	0
207	Norths	66.95	66.95	66.96	66.96	2.000	0.2781	0
208	Stewarts	81.79	81.79	81.79	81.79	2.000	0.2781	0
209	Coles	61.63	61.63	61.63	61.63	2.000	0.2781	0
210	Allens	70.02	70.02	70.03	70.03	2.000	0.2781	0
211	Burns	71.87	71.87	71.87	71.87	2.000	0.2781	0
X299	South Hydrant	75.14	75.14	75.14	75.14	0.000	0.0000	0

Figure 4: Output window for 3 inch mains with north hydrant fire

Because we know that 3 inch is sufficient for No Fire conditions, we only need to test the Fire conditions for each pipe size. The results of this testing are displayed in Table 1. After testing all the fire conditions, 4 inch pipe is the only size that will deliver 90 psig to the North Hydrant during fire conditions. Therefore, the housing development mains must be 4 inch pipe.

Table 1: Static pressure in psig at the open hydrant for each combination of pipe size and active hydrant

Fire Flow Location	3 inch	3.5 inch	4 inch
North Hydrant Fire	70.99	85.39	91.89
South Hydrant Fire	72.89	87.43	93.98

Beginner - Cost Calculation

Summary

This example demonstrates how to set up cost calculations in AFT Fathom.

Determine the cost of the heat exchanger system over a 10-year period including material, installation, and energy costs.

Topics Covered

- Estimating simple costs
- Connecting cost libraries
- Using Cost Settings
- Using the Cost Report

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Beginner - Cost Calculation.fth***
- ***Controlled Heat Exchanger Temperature.dat*** - engineering library
- ***Controlled Heat Exchanger Temperature - Junction Costs.cst*** - cost library for ***Controlled Heat Exchanger Temperature.dat***
- ***Controlled Heat Exchanger Temperature - Pipe Costs.cst*** - cost library for **Steel - ANSI**

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

If you have already worked the Controlled Heat Exchanger Temperature example for either the GSC or XTS modules, you can open the file you built for those examples, create a new scenario for this example, and then skip to Step 4.

Important: The XTS and GSC modules and cost calculations can be performed in the same run to simulate control functions over time, but we are not going to do this here. Before proceeding, make sure to disable GSC by selecting the radio button for Ignore under GSC in the Modules panel in Analysis Setup. Also, make sure to disable XTS by selecting Steady Only in the XTS section.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F
    d. This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.
3. Open the Heat Transfer/Variable Fluidspanel:
    a. Heat Transfer = Heat Transfer With Energy Balance (Single Fluid)

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
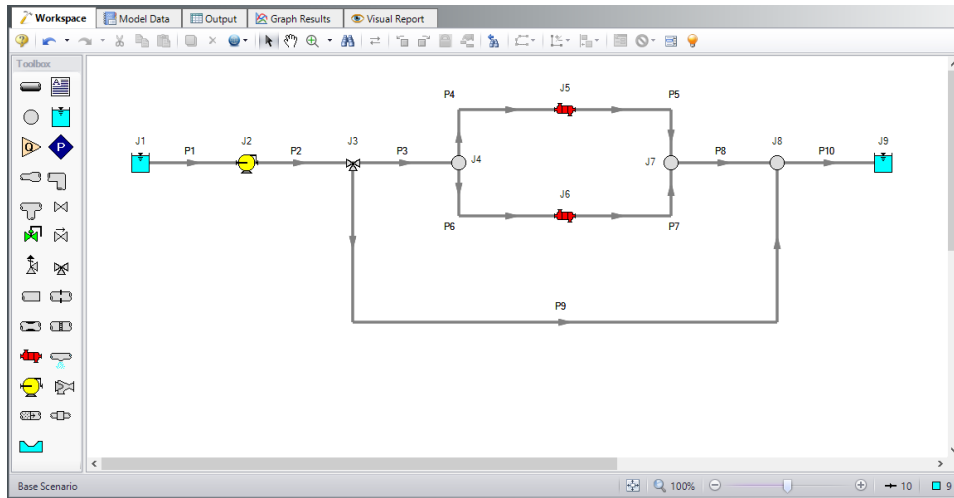
Figure 1: Workspace for Controlled Heat Exchanger Temperature example

Pipe Properties

1. All Pipes
   a. Pipe Material = Steel - ANSI
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = 8 inch
   d. Type = STD (schedule 40)
   e. Friction Model Data Set = Standard
   f. Length =

Pipe	Length (feet)
1 & 3-8	20
2	5
9	200
10	100

Junction Properties

1. J1 & J9 Reservoirs
   a. Liquid Surface Elevation = 10 feet
   b. Liquid Temperature = 200 deg. F
   c. Update Density with Temperature = Checked

      d.   Liquid Surface Pressure = 40 psig

      e.   Pipe Depth = 0 feet

2.   J2 Pump

      a.   Inlet Elevation = 0 feet

      b.   Pump Model = Centrifugal (Rotodynamic)

      c.   Analysis Type = Pump Curve

      d.   Enter Curve Data =

Volumetric	Head
gal/min	feet
0	70
500	60
1000	40

      e.   Curve Fit Order = 2

3.   J3 Three Way Valve

      a.   Elevation = 0 feet

      b.   Combined Flow = P2

      c.   Split Flow Pipe #1 = P3

      d.   Split Flow Pipe #2 = P9

      e.   Loss Model = Cv

      f.   Open Percentage Data =

Open Pct.	Cv Pipe #1	Cv Pipe #2
0	0	100
100	100	0

      g.   Actual Percent Open = 75%

4.   J4, J7, & J8 Branch Junctions

      a.   Elevation = 0 feet

5.   J5 & J6 Heat Exchangers

      a.   Inlet Elevation = 0 feet

      b.   Loss Model tab

          i.   Loss Model = K Factor

          ii.   K = 1400

      c.   Thermal Data tab

          i.   Thermal Model = Counter-Flow

          ii.   Heat Transfer Area = 1600 feet2

          iii.   Overall Heat Transfer Coefficient = 50 Btu/hr-ft2-R

          iv.   Flow Rate = 200 lbm/sec

v.   Specific Heat = 1 Btu/lbm-R

vi.   Inlet Temperature = 40 deg. F

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Define the Cost Settings group

To perform cost calculations they will first need to be enabled. Do this by opening Analysis Setup, navigating to the Cost Settings panel , then selecting Calculate in the Cost Calculation section. If it is only desired to estimate energy costs, the Pump Energy Only option could be selected instead to simplify the input.

The Cost Settings panel is where the types of costs to be included in the cost calculations are specified. The Cost Settings panel is shown in Figure 2.

The Cost Definitions section allows you to specify the type of costs to be determined. You may select from engineering parameter costs, such as pipe weight, or actual monetary costs, including material and installation costs. For this example, select Monetary Costs, if not already selected. Include Material, Installation, and Operation/Energy costs by selecting them from the list of available monetary costs, as shown in Figure 2.

Energy cost data can be specified through an energy cost library, or by directly entering an energy cost on the Cost Settings panel. Using an energy cost library allows you to specify multiple energy costs, such as energy costs at both peak and off-peak rates. For this example, select Use This Energy Cost Information and enter a Cost of 0.12 U.S. Dollars Per kW-hr, as shown in Figure 2.

The system lifetime to be applied for the cost calculations is specified in the Cost Time Period section. You can also specify interest and inflation rates to be applied to the cost calculations as well. For this example, enter a system life of 10 years, as shown in Figure 2, then close the Cost Settings panel by clicking the OK button.
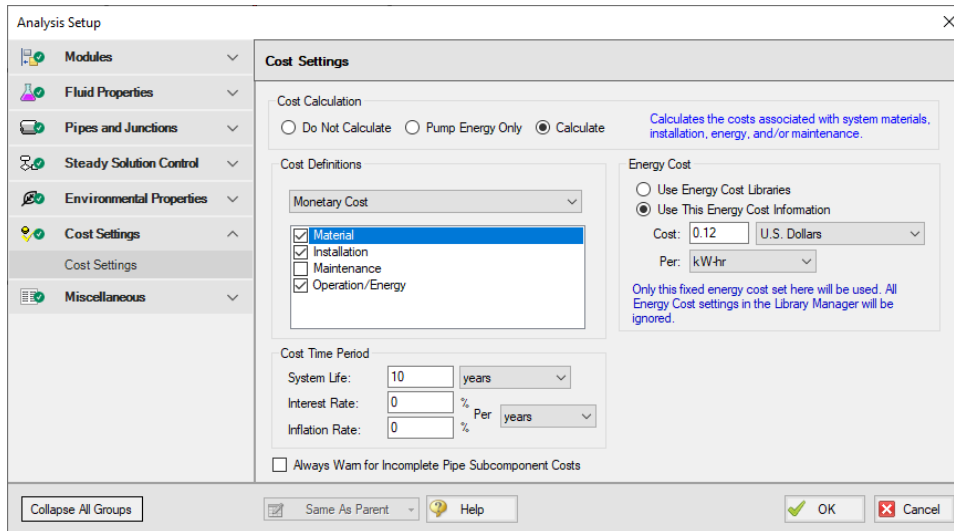
Figure 2: General cost calculation information is defined in the Cost Settings panel

Step 5. Engineering and Cost Libraries

The actual material and installation costs that would be used with cost calculations are contained in cost libraries. The cost libraries needed for this example already exist, and just need to be accessed.

The Library Manager (opened from the Library menu) shows all of the available and connected libraries. Libraries can either be engineering libraries or cost libraries. Cost libraries are always associated with an engineering library, and are thus displayed subordinate to an engineering library in the library lists.

Here we will summarize some key aspects of libraries:

- Cost information for a pipe is accessed from a cost library. Cost library items are based on corresponding items in an engineering library. (The engineering library also includes engineering information such as pipe diameters, hydraulic loss factors, etc.)

- To access a cost for a particular pipe or junction in a model, that pipe or junction must be based on items in an engineering library that is connected to the model file.

- There can be multiple cost libraries associated with and connected to an engineering library (such as regionally varying costs, or multiple foreign currencies). This makes it easier to manage costs of items.

Open the Library menu and select Library Manager. Click Add Existing Library, select the ***Controlled Heat Exchanger Temperature.dat*** engineering library from the AFT Fathom 13 Examples folder, and click Open. The engineering library titled **Controlled Heat Exchanger Temperature** should now appear in the list of available libraries.

Now add the cost libraries by clicking Add Existing Cost Library, and browsing to the ***Controlled Heat Exchanger Temperature - Junction Costs.cst*** and the ***Controlled Heat Exchanger Temperature - Pipe Costs.cst*** cost library files.

These libraries should connect automatically. If not, connect the newly added engineering and cost libraries by checking the box next to the name in the list of available libraries.

The Library Manager should appear as shown in Figure 3. The Library Manager window displays all of the available and connected libraries.
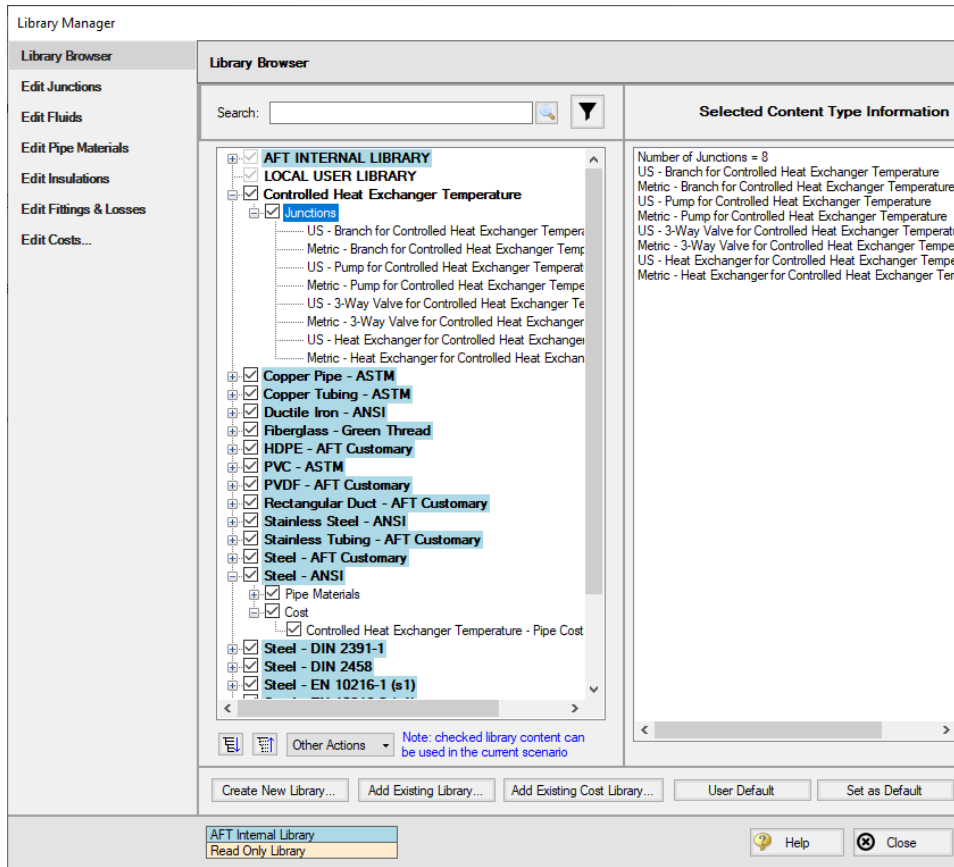

Figure 3: Library Manager with engineering and cost libraries added and connected

The engineering libraries associated with these two cost libraries are the **Steel - ANSI** pipe material library, and an external library called **Controlled Heat Exchanger Temperature**. For the cost data in the two cost libraries to be accessed by pipes and junctions in the model, the pipes and junctions must use these two engineering libraries.

The pipes for this example are already using data from the Steel - ANSI pipe material library, so no modifications need to be made for the pipes. All of the junctions except the reservoir junctions must be modified to use the junction data contained in the engineering libraries. Do this by opening the properties window for each junction, and selecting the junction name displayed in the Library list, as shown in Figure 4 for the Three-Way Valve Junction.
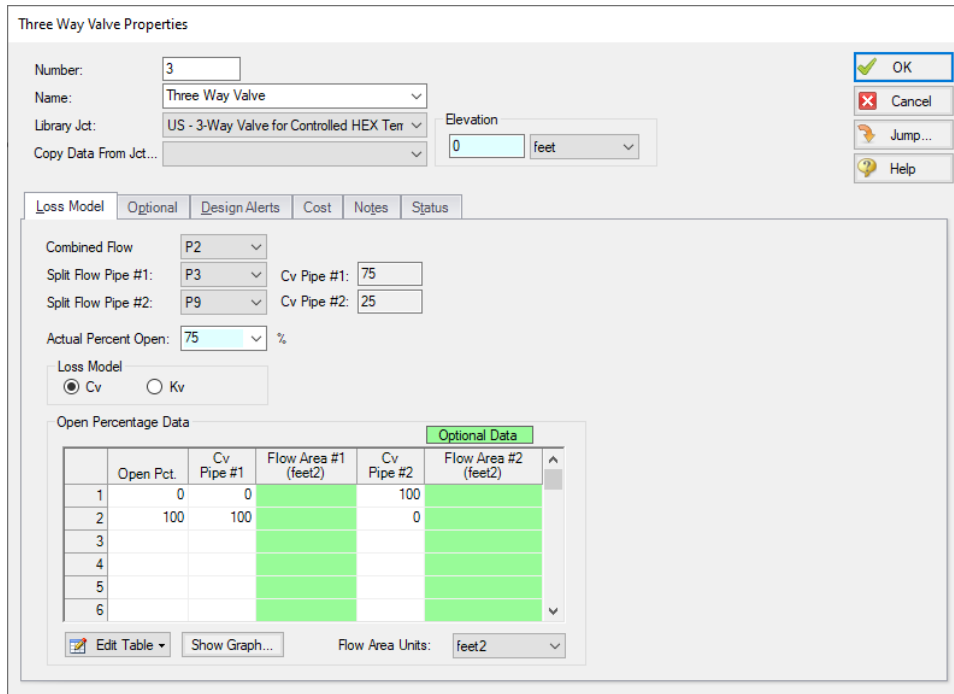
Figure 4: The junctions to be included in the cost analysis must be associated with an engineering library

Step 6. Including objects in the Cost Report

The final step before performing a cost analysis is to specify which pipes and junctions you want to be included in the final cost report. This is done by opening the properties window, navigating to the Cost tab, and choosing Include in Cost Report **for each pipe and junction to be included**, as shown for Pipe 1 in Figure 5.

The Global Edit feature may be used to update this information for all of the pipes and junctions.
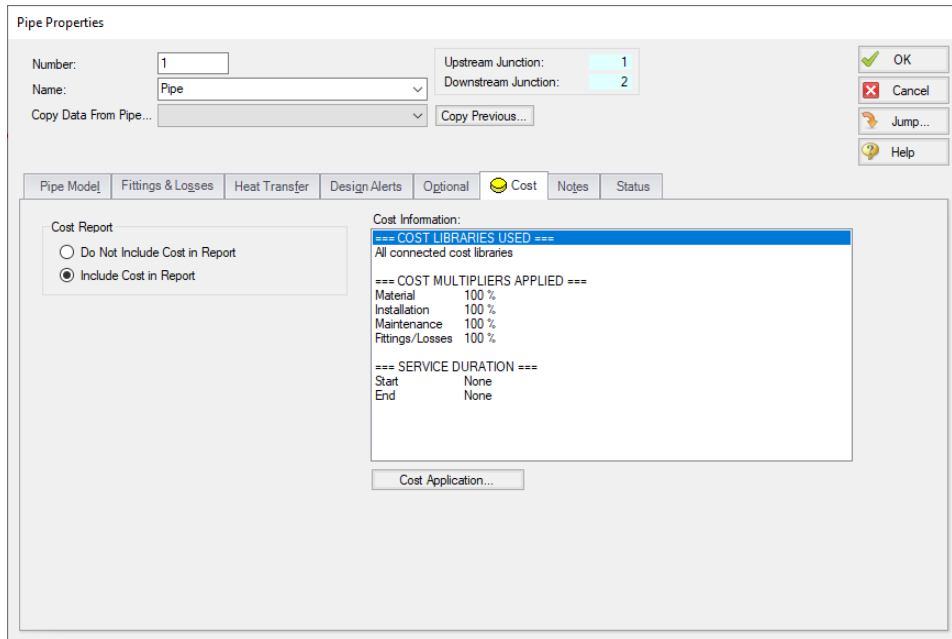
Figure 5: The Cost tab on the pipe and junction Properties windows is used to include the objects in the Cost Report

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Cost Report

The Cost Report is displayed in the General Results section of the Output Window. View the Cost report by selecting the Cost Report Tab. The content of the Cost Report can be controlled from the Output Control window. Figure 6 shows the Cost Report for this example with the Material, Installation, Energy, and Total costs displayed.

The Cost Report shows the total system cost, as well as the individual totals for the material, installation, and energy costs. In addition, the Cost Report displays the detailed cost for each pipe, junction, and fitting that was included in the report. The items are grouped together by type, and a subtotal for each category is listed.

| General | Warnings | Design Alerts | Cost Report | Pump Summary | Valve Summary | Heat Exchanger Summary | Reservoir Summary |

Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS								**100,724**
Total of All Shown Costs			**34,915**	**14,041**	**48,956**	**51,768**	**51,768**	**100,724**
Pipe Subtotal			**22,695**	**11,125**	**33,820**	**0**	**0**	**33,820**
P1	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P2	Pipe	Pipe	255	125	380	0	0	380
P3	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P4	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P5	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P6	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P7	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P8	Pipe	Pipe	1,020	500	1,520	0	0	1,520
P9	Pipe	Pipe	10,200	5,000	15,200	0	0	15,200
P10	Pipe	Pipe	5,100	2,500	7,600	0	0	7,600
Branch Subtotal			**1,620**	**666**	**2,286**	**0**	**0**	**2,286**
J4	Branch	Branch	540	222	762	0	0	762
J7	Branch	Branch	540	222	762	0	0	762
J8	Branch	Branch	540	222	762	0	0	762
Heat Exchanger Subtotal			**6,000**	**1,200**	**7,200**	**0**	**0**	**7,200**
J5	Heat Exchanger	Heat Exchanger	3,000	600	3,600	0	0	3,600
J6	Heat Exchanger	Heat Exchanger	3,000	600	3,600	0	0	3,600
Pump Subtotal			**1,500**	**500**	**2,000**	**51,768**	**51,768**	**53,768**
J2	Pump	Pump	1,500	500	2,000	51,768	51,768	53,768
Three Way Valve Subtotal			**3,100**	**550**	**3,650**	**0**	**0**	**3,650**
J3	Three Way Valve	Three Way Valve	3,100	550	3,650	0	0	3,650

Figure 6: The Cost Report in the General Output section shows the results of the cost analysis

Cost Analysis Summary

The cost analysis for this example shows the total system cost over 10 years for the heat exchanger system is $100,724. Of that cost, $34,915 was for Material costs, $14,041 was for Installation costs, and $51,768 was energy cost over the life of the system.

Non-Newtonian Phosphates Pumping

Summary

This example demonstrates how to perform Non-Settling Slurry calculations in AFT Fathom to size a pump as part of a system design process.

Topics Covered

- Entering rheological data
- Using homogeneous scale-up features
- Obtaining Power Law and Bingham Plastic constants

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Non-Newtonian Phosphates Pumping.fth*
- *US - Non-Newtonian Phosphates Pumping Scale-up Data.txt*
- *US - Non-Newtonian Phosphates Pumping Rheology Data.txt*

Problem Statement

The piping for a non-settling slurry system which moves phosphate is being designed. The system will pump 4,755 gal/min of phosphate through a horizontal pipe 2,300 feet long.

Use non-settling slurry calculations to size the pump for 4,755 gal/min and alternate potential flows of 3,170 and 6,340 gal/min.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Fluid panel
    a. Fluid Library = User Specified Fluid
    b. Name = Phosphate Slime
    c. Density = 70.5 lbm/ft3
    d. Dynamic Viscosity = 1 centipoise (note that the viscosity entered here will *not* be used once the non-Newtonian parameters are entered).
3. Viscosity Model panel
    a. Viscosity Model = Homogeneous Scale-up
    b. Entered Scaled Data: The data provided by Wilson, et al, 2006 for 203 mm (8 inch) pipe is as follows:
        i. Material = Steel - ANSI
        ii. Size = 8 inch
        iii. Type = STD (schedule 40)
        iv. Raw Data table =

Table 1: Pressure drop data on phosphate slime from Wilson, et al, 2006 page 76 (203 mm/8 inch pipe)

Parameters	Velocity	Jm	Flow Regime
Units	feet/sec	ft/ft	
1	1.74	0.089	Laminar
2	4.99	0.100	Laminar
3	6.56	0.102	Laminar
4	8.50	0.105	Laminar
5	10.6	0.108	Laminar
6	12.5	0.110	Laminar
7	14.5	0.113	Laminar
8	16.8	0.119	Turbulent
9	18.5	0.130	Turbulent

1. In the table choose Velocity in feet/sec and Jm in ft/ft and use the Edit Table button to import the data from the ***US - Non-Newtonian Phosphates Pumping Scale-up Data.txt*** file. Alternatively, enter the data shown in Table 1 above, with the Flow Regime set to Laminar for the first 7 entries,

and Turbulent for rows 8 and 9.

2. Click Create Graph at the bottom.

3. To see how the data will scale to 305 mm, choose the **Cross-Plot Scale-Up Curves** option, with **Plot a Curve for Specific Pipe Size** selected, specify the following parameters: **Steel - ANSI**, **12 inch**, and **STD**

4. Click Update Graph at the bottom (see Figure 2)

5. To see how Power Law and Bingham Plastic model would work for this data choose them in the **Cross-Plot Correlation for Test Pipe Diameter** area.

6. Select OK

Figure 1: Non-Settling Slurry Scale-up test data entry
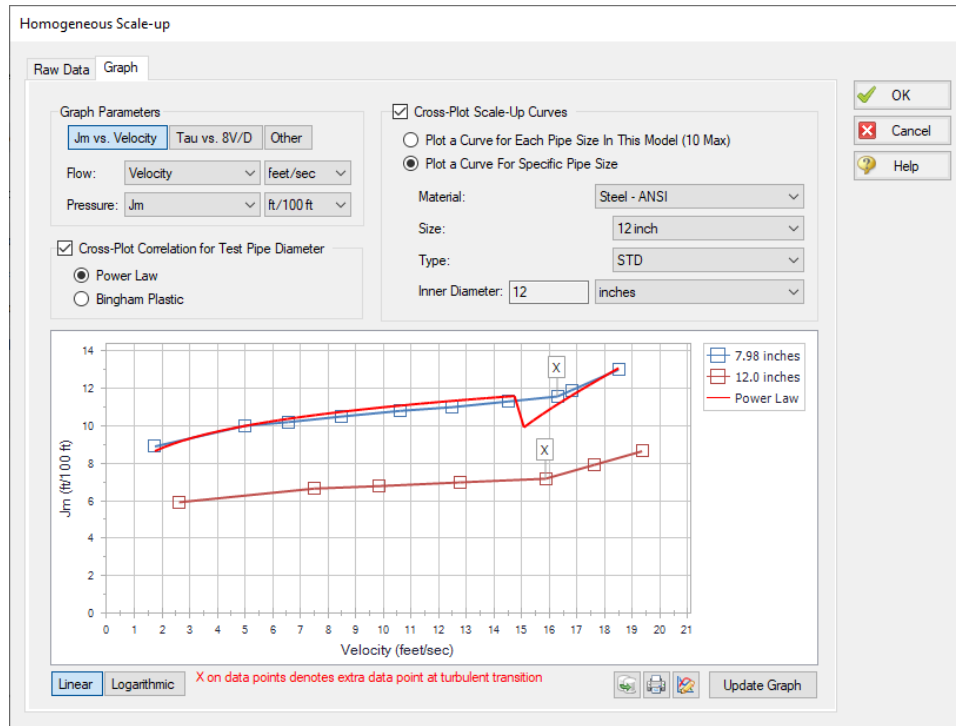
Figure 2: Graph of Non-Settling Slurry scale-up data

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
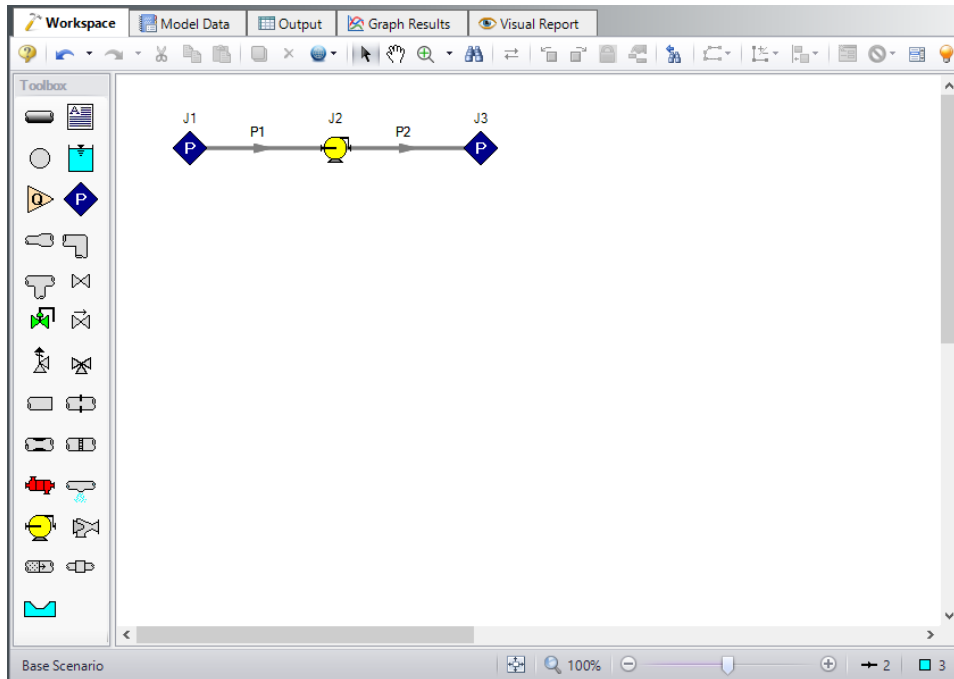
Figure 3: Layout of pipe system for Non-Newtonian Phosphates Pumping Example

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 12 inch
    d. Type = STD
    e. Length =

Pipe	Length
1	1 inches
2	2300 feet

Junction Properties

1. J1 & J3 Assigned Pressure
    a. Elevation = 0 feet
    b. Pressure = 0 psig
    c. Pressure Specification = Stagnation
2. J2 Pump

     a. Pump Model tab
          i. Name = 4755 gal/min
          ii. Inlet Elevation = 0 feet
          iii. Pump Model = Centrifugal (Rotodynamic)
          iv. Analysis Type = Sizing
          v. Parameter = Volumetric Flow Rate
          vi. Fixed Flow Rate = 4755 gal/min
          vii. Nominal Efficiency = 75 Percent
     b. Optional tab
          i. Additional Efficiency Data

Motor Power	Motor Eff. (%)
100	95

          ii. Power Units = hp

Step 4. Create Two Alternate Cases

The example in Wilson et al, 2006, also requires evaluation of alternate flow rates. To create the alternate cases do the following:

1. Go to the **Edit menu** and click **Select All**
2. From the **Edit menu** again, select **Duplicate Special**
3. Increment All Pipe and Junction Numbers By = **10**
4. Click **OK**
5. Move the duplicate below the first and click **Paste**
6. Repeat steps 2-5 on the pipes and junctions that were duplicated
7. For Pump J12, enter the following:
    a. Name = 3170 gal/min
    b. Fixed Flow Rate = 3170 gal/min
8. For the J22, enter the following:
    a. Name = 6340 gal/min
    b. Fixed Flow Rate = 6340 gal/min

To show the names of the pumps and their head rise, a new Workspace Layer will be created.

1. Select all pumps on the Workspace using a selection box (dragged from left to right) or holding CTRL while clicking each pump.
2. In the Quick Access Panel, open the Workspace Layers tab.
3. Create a new layer by selecting New Layer and then select the option From Workspace Selections in the drop-down menu. This creates a layer in which the selected objects and their labels are set to be visible.
4. Name the layer Pump Labels.

5. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window or by double clicking on the layer name.

6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.

7. Double-click **Junction Name** to add it to the list on the right-hand side. This will add the Junction Name to the pump labels.

8. In the same panel, add **Head Rise** (ft) to the label from the Pump Summary Parameters group.

9. Close the Layer Settings window.

Note: The Head Rise parameter will be grayed out in the right hand column since there is not yet output. After the model runs, this row will be white and each pump's head rise will be shown on the Workspace.

The workspace should resemble Figure 4.
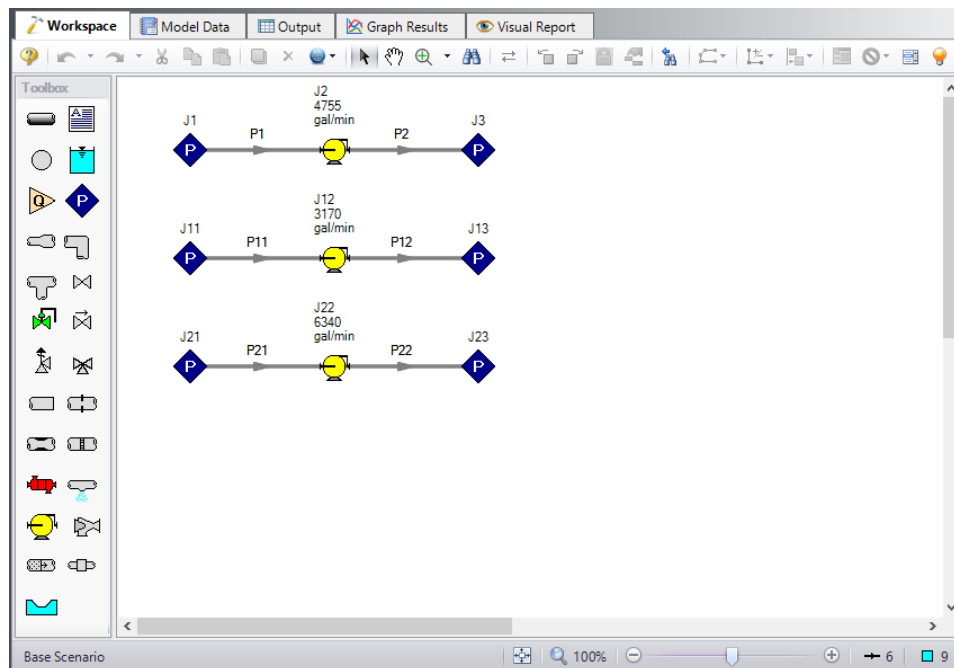
Figure 4: Layout of pipe system for Non-Newtonian Phosphates Pumping Example with all three cases

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The Output window contains all the data that was specified in the Output Control window.

Figure 5 shows the Output window and includes the Pump Summary tab.

Figure 5: The Output window with default output data

The Output Control window can be used to customize the output and to add output parameters.

1. Open Output Control from the toolbar or Tools menu
2. In the Pipes section choose **Head Gradient** and add it to shown output. Then change its units to ft/ft. Head gradient is the same thing as *Jm*. See Figure 6.
3. Next choose **Pressure Gradient Stag. Total** and add it. Then change its units to psi/ft.
4. Select the Pumps button that runs along the top of Output Control.
5. Choose **Pressure Rise Stagnation** and add it. Then change its units to ft. H2O std.. This is the head rise in terms of water head. See Figure 7.
6. Click OK. The Output now appears as in Figure 8.

Table 3 summarizes the results from all three cases which agree well with those published by Wilson et al, 2006.

Figure 6: The Output Control window Pipes report
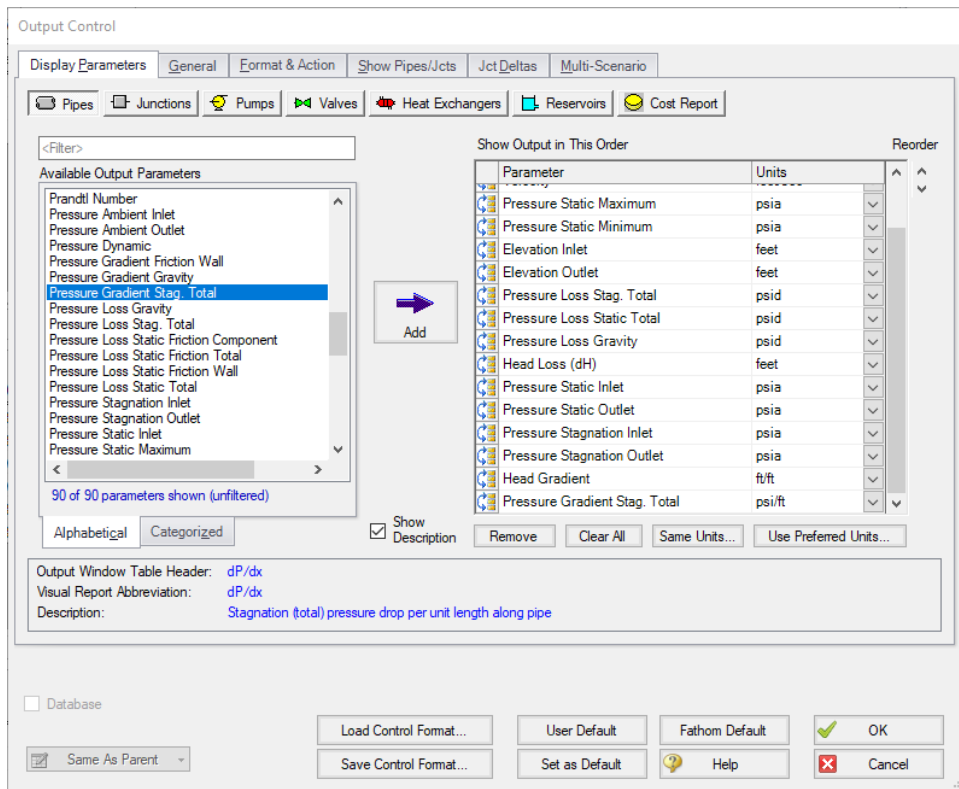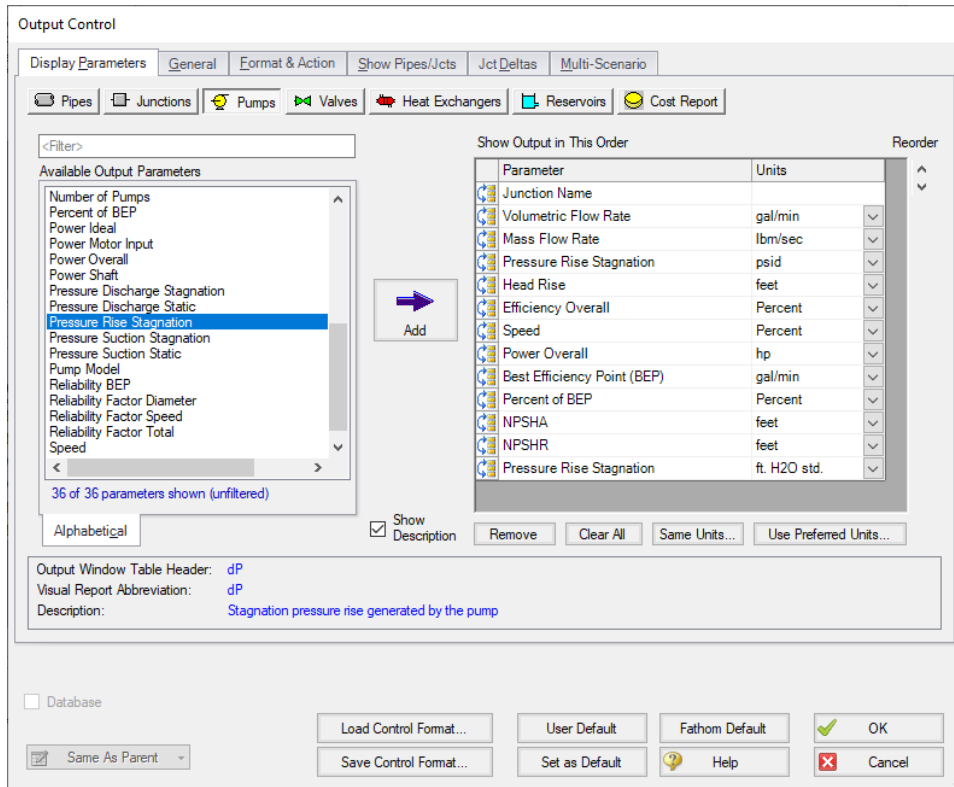
Figure 7: The Output Control window Pumps report

Figure 8: The Output showing head gradient, pressure gradient and pump water head rise

Also of interest is the system curve. Navigate to the Graph Results window by clicking the Graph Results tab on the toolbar, or by pressing Control+G. On the Graph Control tab in the Quick Access Panel, select the Pump vs. System Curve tab if not already selected. CRV for Pump 2 should be selected, see Figure 9. Click Generate.

The system curve is shown in Figure 10.
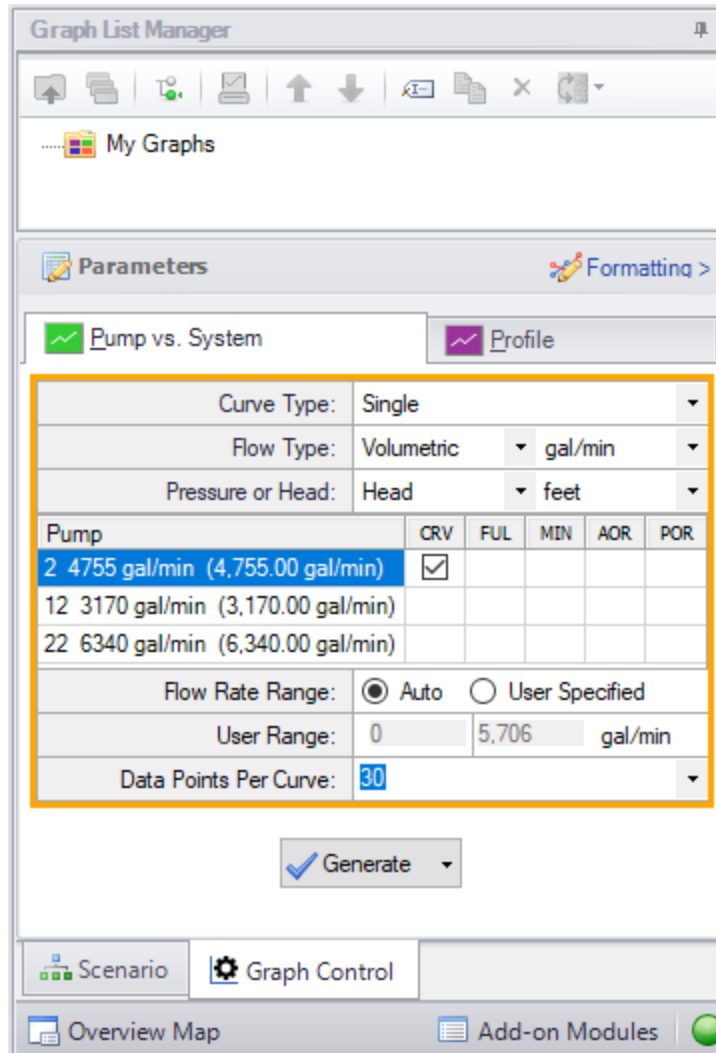
Figure 9: Data selection for graphing system curve
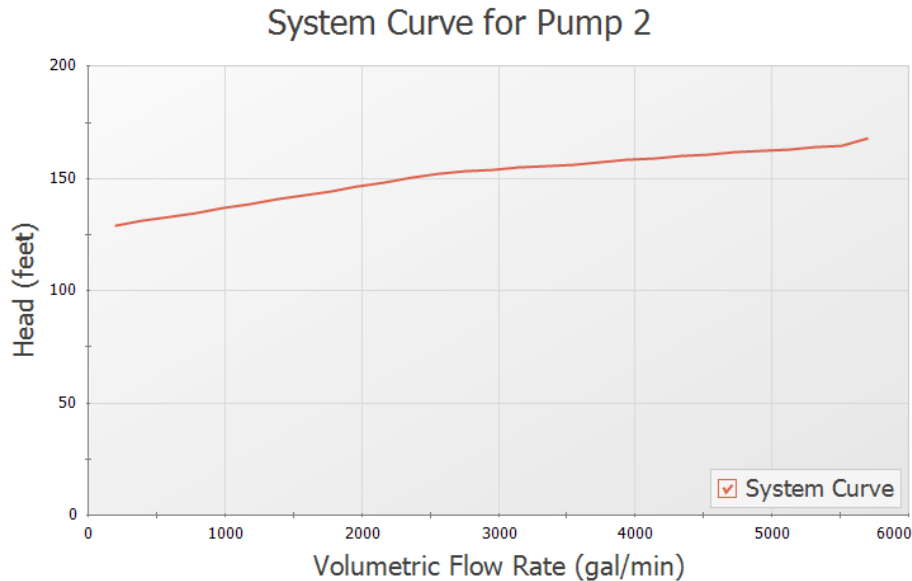
Figure 10: System curve in Graph Results window

Step 7. Compare Homogeneous Scale-Up to Power Law and Bingham Plastic Viscosity Models

Based on the fluid and flow regime and industry preference, a non-Newtonian model such as Power Law or Bingham Plastic may be preferred. How do those models compare to the Scale-up approach for the current example?

To perform the comparison, create two child scenarios. This can be done by opening the Scenario Manager on the Quick Access Panel. Create two child scenarios and call them Power Law and Bingham Plastic.

Here the Power Law scenario will be reviewed. The Bingham Plastic scenario will be very similar and hence is not shown. Results for all three scenarios will be compared later in this example.

1. Load the Power Law scenario
2. Open Analysis Setup
3. Open the Viscosity Model panel in the Fluid Properties group
4. Click the Enter Scaled Data button
5. Click the Graph tab
6. Select the Cross-Plot Correlation for Test Pipe Diameter option
7. Make sure the Power Law option is selected
8. If not, select it and click Update Graph

Figure 11 shows that the Power Law correlation matches the data well. The Bingham Plastic correlation diverges from test data at higher flow rates.
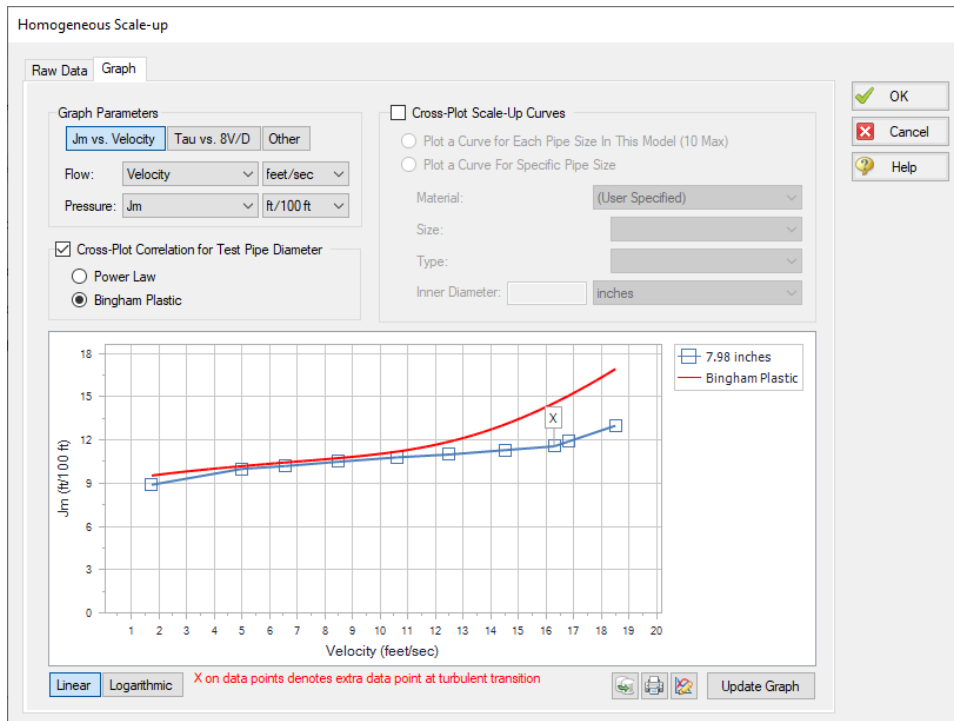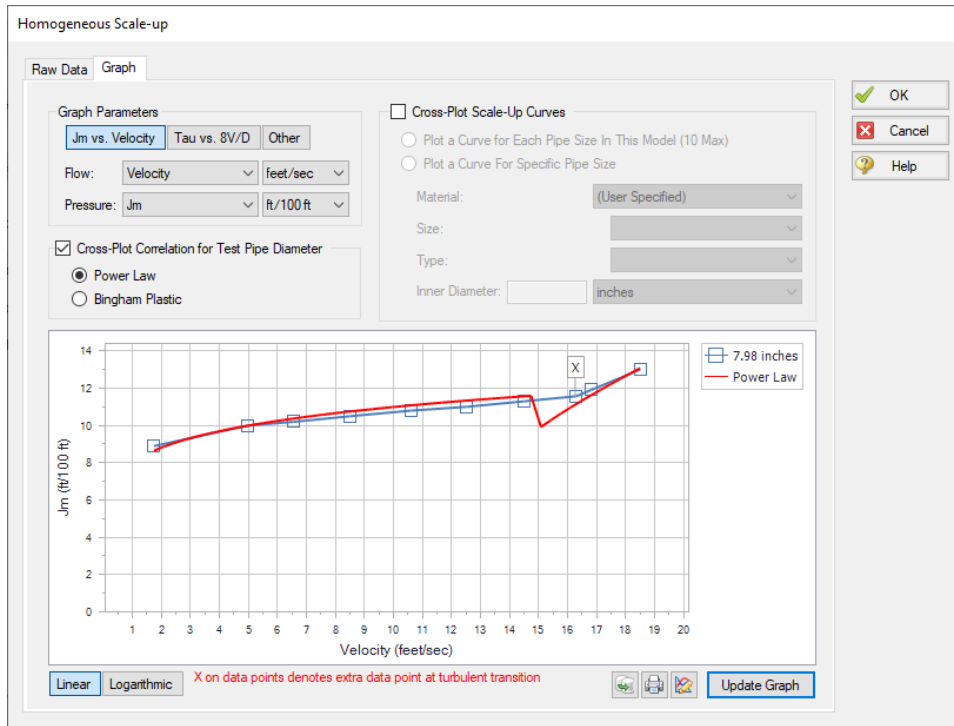
Figure 11: Non-Settling Slurry Scale-up data and Power Law (top) and Bingham Plastic (bottom) correlations

1. Click Cancel
2. Select Power Law from the drop down list of Viscosity Models
3. Select Calculate from Rheological Data
4. Click Calculate Constants
5. Make sure the Raw Data Type selection is Tube Flow Rheometer Data (8V/D)
6. Enter the shear rate and shear stress data from Table 2, below. This data is equivalent to the data from Table 1, entered previously. Alternatively, use the*US - Non-Newtonian Phosphates Pumping Rheological Data.txt* file to import the shear rate and shear stress data into Fathom.
7. Click Generate Curve Fit Now
8. Click OK

The data should be K = 69,240 lbm/hr-ft and n = 0.1341. See Figure 12. For the Bingham Plastic scenario the constants should be Plastic Viscosity= 86.63 lbm/hr-ft and Yield Stress= 0.007138 psia.

Table 2: Pressure drop data on phosphate slime from Wilson, et al, 2006 page 76 (203 mm/8 inch pipe) converted into shear rate (8V/D) and shear stress (Tau)

Run	V	Jm	Shear Rate	Shear Stress
	feet/sec	m/m	1/seconds	psia
1	1.74	0.089	20.89	0.0073
2	4.99	0.100	59.90	0.0082
3	6.56	0.102	78.82	0.0083
4	8.50	0.105	102.1	0.0086
5	10.6	0.108	127.7	0.0088
6	12.5	0.110	150.2	0.0090
7	14.5	0.113	174.6	0.0092
8	16.8	0.119	201.8	0.0097
9	18.5	0.130	222.3	0.0106
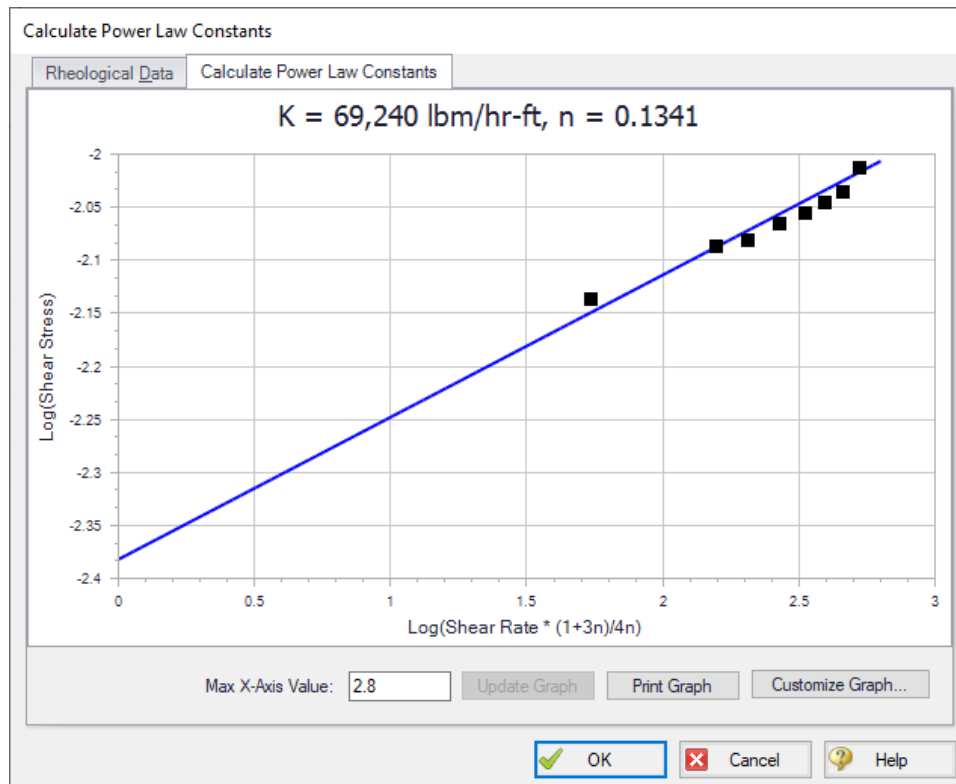
Figure 12: Power Law data curve fit

Each scenario can be loaded and run. Results are summarized in Table 3. It is clear that the Bingham Plastic model over predicts the required pump head and power at higher flow rates but is comparable at the lowest flow rate of 3170 gal/min. This is consistent with the lower part of Figure 11 where the Bingham Plastic correlation offers a poor match to the raw data as the flow rate increases.

Table 3: Comparison of results between Homogeneous Scale-up, Power Law, and Bingham Plastic models

Q	gal/min	3170 gal/min			4755 gal/min			6340 gal/min		
		Scale-Up	Power Law	Bingham Plastic	Scale-Up	Power Law	Bingham Plastic	Scale-Up	Power Law	Bingham Plastic
V	feet/sec	8.993	8.993	8.993	13.489	13.489	13.489	17.985	17.985	17.985
Jm	ft/ft	0.06735	0.06846	0.06985	0.07028	0.07228	0.08079	0.08071	0.08210	0.10506
dP/dx	psid/ft	0.03297	0.03351	0.03420	0.03441	0.03539	0.03955	0.03951	0.04019	0.05144
dP	psid	75.84	77.09	78.66	79.14	81.39	90.97	90.89	92.45	118.31
dH Water	feet	174.9	177.8	181.4	182.6	187.7	209.8	209.6	213.2	272.9
dH Fluid	feet	154.9	157.5	160.7	161.7	166.3	185.8	185.6	188.8	241.7
Power	hp	196.8	200.0	204.1	308.1	316.8	354.1	471.7	479.8	614.0

Analysis Summary

A non-settling phosphate slurry pipeline was modeled and the pump sized for a range of flow rates. AFT Fathom non-settling slurry calculations allow evaluation of the non-Newtonian viscous behavior of the slurry.

NFPA Firewater System

Summary

This problem walks through building a hydraulic model of a fire water sprinkler system using Fathom's NFPA features. This example is based off of figure B.2 (a) and associated calculations from NFPA 15 standard 8.5.3.1.

In this example, a firewater spray nozzle system is hydraulically modeled. The system consists of 28 spray nozzles and is supplied by a fixed pressure source. As the design engineer, it is your responsibility to ensure that all spray nozzles receive at least 40 gal/min, as well as the minimum supply pressure required to achieve these flows. To ensure that the proposed design is adequate, you will model the system and use the hydraulically most remote nozzle to ensure that all spray nozzles are satisfied.

Topics Covered

- Modeling a fire water spray system
- Making groups
- Using NFPA features
- Generating an NFPA report
- Using Excel Change Data

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - NFPA Firewater System - Initial.fth***
- ***US - NFPA Firewater System - Final.fth***
- ***US - NFPA AFT Transfer Spreadsheet.xlsx***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - NFPA Firewater System - Initial.fth** model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder**.

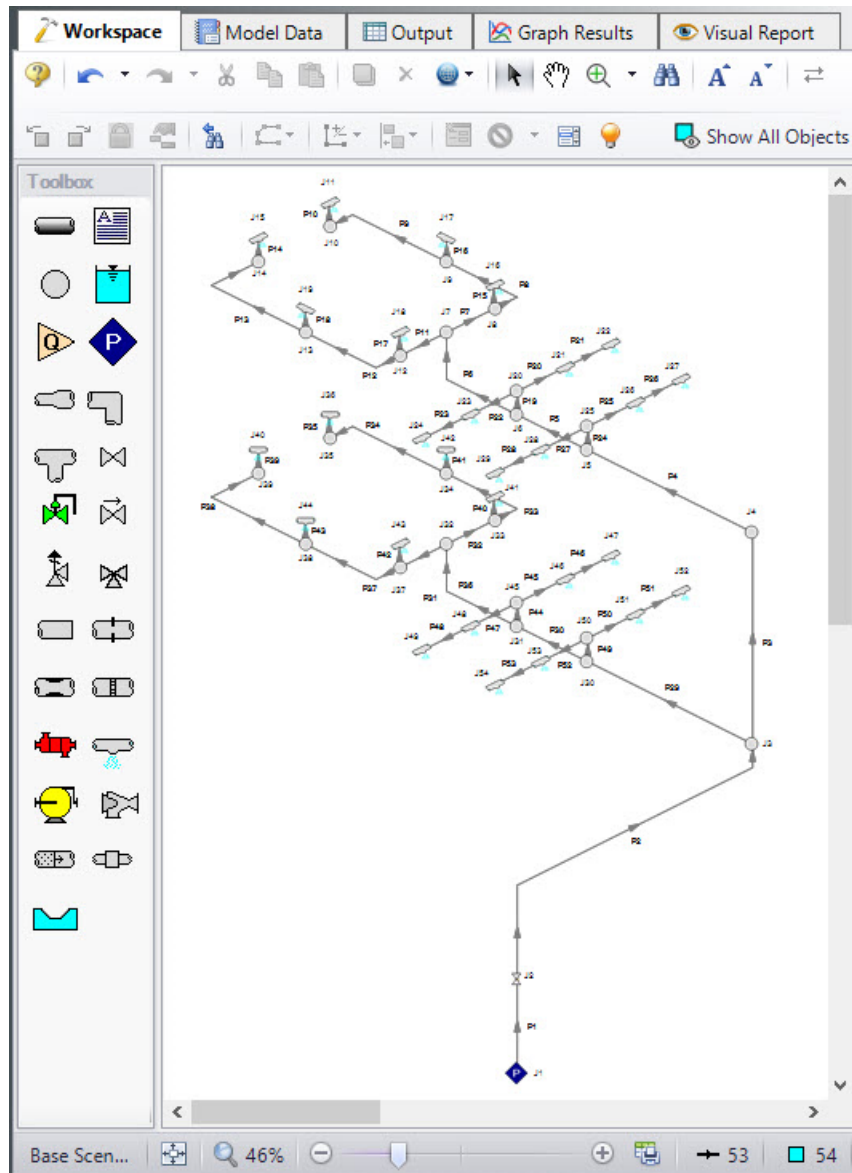The model should appear as shown in Figure 1.

Figure 1: Initial model for NFPA Firewater System example

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F

Step 4. Define the Miscellaneous group

While still in the analysis setup, expand the Miscellaneous drop down and open the NFPA panel. Check the box next to Enable NFPA. This will enable various features throughout the software such as NFPA internal fittings & losses for pipes, equivalent length loss model for junctions, NFPA report, etc. Keep the Initial Values the same. A group for the NFPA report will be selected later.

Step 5. Define the Pipe and Junction Properties

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click the "Ok" button to save changes to the analysis setup and return to the workspace. At this point a user would need to draw their own network using the tools in the toolbox, but to save some time the piping layout for this model has already been created.

With the system in place, we now need to enter the properties of the objects. Because this model is large, all input has been added to a specially-configured Excel Change data spreadsheet that is included in the Examples folder and will be used to change the input to match the values in this example. Before importing this spreadsheet, it is necessary to define values in each pipe and junction because the Excel Change data spreadsheet is only capable of changing input data, not entering new data.

To quickly enter input data before using the Excel Change data spreadsheet, we will use the Global Pipe and Junction Editing features. If you are going to use Excel Change data to change the input you already have in your model, the values that you initially input in the model act simply as place holders and the value itself is not important. It is crucial, though, that you ensure that the units for each value that will be changed are specified correctly, because the Excel Change data spreadsheet will not change these units (it will only change the value itself).

Pipe Properties

To globally edit the pipes, select Global Pipe Edit from the Edit menu. Click All to select all pipes. Next, click Select Pipe Data and enter the following inputs in the pipe properties window.

1. Pipe Material = Steel - ANSI
2. Pipe Geometry = Cylindrical Pipe
3. Size = 1 inch
4. Type = STD (schedule 40)
5. Friction Model = User Specified
6. Data Set = Hazen-Williams Factor
7. Value = 120
8. Length = 10 feet

In the Parameters to Change section, click All to apply all changes to the selected pipes, then click Apply Selections at the bottom. Then click OK.

Next, fittings a losses will need to be added to various pipes throughout the system. On the fittings & losses tab of the pipe properties window move the slider from "Use K Factor" to "Use Equivalent Lengths" then specify the Fittings for each pipe according to the table below.

Note: Pipes that have Fittings & Losses entered in them will now have a red & displayed next to them. The symbol and display settings can be customized in User Options.

Table 1: Fittings and Losses for pipe input in NFPA model

Pipes	Fitting Type	Quantity
4, 6, 8, 10, 12, 14, 31, 33, 35, 37, 39	Elbow, 90 deg. standard	1
2, 9, 13, 34, 38	Elbow, 90 deg. standard	2
7, 11, 19, 20, 22, 24, 25, 27, 29, 32, 36, 40, 41, 42, 43, 44, 45, 47, 49, 50, 52	Tee or Cross, (flow turned 90 deg.)	1
15, 16, 17, 18, 40, 41, 42, 43	Elbow, 90 deg. standard and Tee or Cross, (flow turned 90 deg.)	1 Tee and 1 Elbow

Junction Properties

To globally edit the junctions, select Global Junction Edit from the Edit menu. Next, click the appropriate junction based on what you are inputting. For example, Spray Discharge. Click All to select all junctions in the list. Then, click Select Spray Discharge Data and enter the following inputs in the pipe properties window. Repeat this process for all junction types in the model using the inputs below.

1. All Spray Discharges
   a. Elevation = 10 feet
   b. Loss Model = K (Fire Sprinkler)
   c. Exit Properties = Pressure
   d. Exit Pressure = 0 psig

      e.   K (Fire Sprinkler) = 3 gal/min per square root of psid

      f.   Number of Sprinklers = 1

2.   All Branches

      a.   Elevation = 5 feet

3.   J1 Assigned Pressure

      a.   Elevation = .5 feet

      b.   Pressure = 63.8 psig

      c.   Pressure Specification = Static

4.   J2 Valve

      1.   Inlet Elevation = 2.5 feet

      2.   Outlet Elevation = 4 feet

      3.   Valve Data Source = Equivalent Length (English)

      4.   Equivalent Length = User Defined

      5.   Equivalent Length (User) = 13 feet

Import Excel Change Data

After you have input for all pipes and junctions, import the Excel Change data spreadsheet. To do this, import the spreadsheet in the Examples Help file entitled ***US - NFPA AFT Transfer Spreadsheet.xlsx*** by clicking on File > Import Excel Change Data, and then browsing to this spreadsheet. After all changes were made, a window will appear that lets you know that all changes were made.

Step 6. Create the NFPA Group

In order to generate an NFPA report, AFT Fathom needs to know the path for which you wish to generate the report. Generally, this report is developed for the hydraulically most remote path. For this model the hydraulically most remote path is either the path to J11 or J15. For Fathom to generate a path it will first need a group containing all pipes and junctions along the desired flow path.

To create a group, select all pipes and junctions that are along the flow path. For this example you will need to select pipes P1, P2, P3, P4, P5, P6, P7, P8, P9, and P10, and junctions J1, J2, J3, J4, J5, J6, J7, J8, J9, J10, and J11 (see this path selected in Figure 2). Note, to select multiple objects, hold the shift key while selecting each with this mouse. Alternatively, you can select these pipes and junctions directly from the Group Manager. Once these pipes and junctions are selected, open the Group Manager by going to Edit > Groups > Manager. Click on Create Group and name the group *NFPA Path*. Make sure this path is selected under Existing Groups, then under both the Pipes in Group and Junctions in Group columns, select Workspace. Press OK to confirm this group.
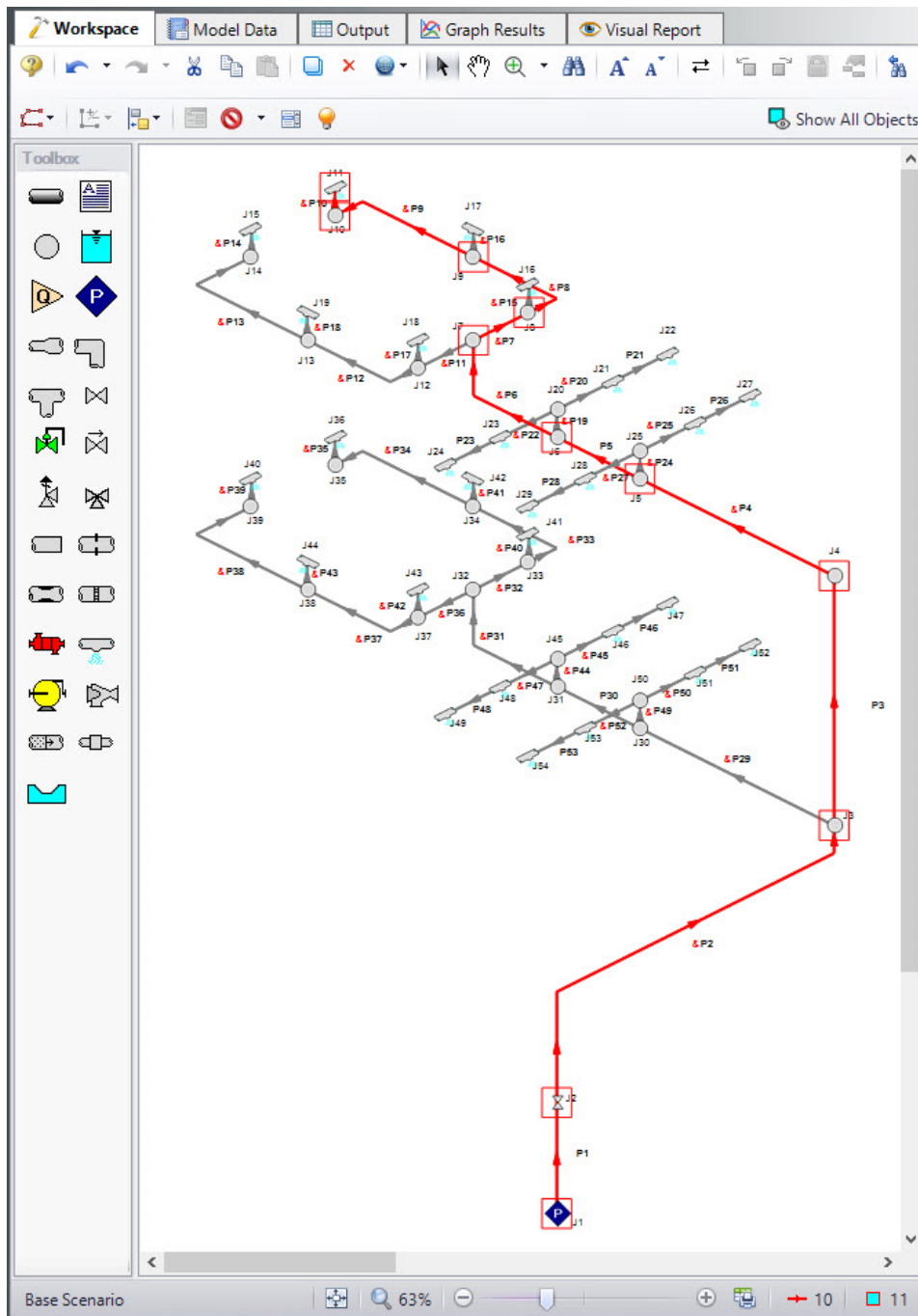
Figure 2: NFPA Group selected on the Workspace

The group we just selected contains the hydraulically most remote path, but this information was simply provided for the sake of convenience in this example and is not frequently known without modeling the system. If you have the GSC module, you can use it to determine the hydraulically most remote path. See the Example problem NFPA Firewater System - GSC if you are interested in seeing how to use the GSC module for this. Otherwise, if the hydraulically most remote path is not known, you will need to dig through Fathoms output to determine which spray nozzle is receiving the least flow.

Step 7. Specify the NFPA Path

Once you know the hydraulically most remote path, you will generate an NFPA report for this path. To specify which path in AFT Fathom you want to generate the NFPA report for, open Analysis Setup and go to the NFPA panel. Select the group in the Select a Group for the NFPA Report list.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

You will notice there are numerous cautions about the pipe lengths being shorter than the elevation change. This model used pipe lengths and elevations based off a NFPA example. In the NFPA example, the length of the spray discharges raised the outlet height of the junction. Fathom treats junctions as single points in space, so to keep the inputs identical the pipe length had to be a little bit short.

Step 9. Examine the Output

The Output window contains the NFPA Report, as well as the standard AFT Fathom output parameters. To view the NFPA Report, click on the NFPA Report tab in the General section of the Output window. You can expand this section by dragging the border down. The NFPA Report for this model is shown in Figure 3a and Figure 3b.

Figure 3a: NFPA Report Page 1

Figure 3b: NFPA Report Page 2

Conclusion

AFT Fathom allows you to display the hydraulic calculations for an NFPA report. In this example, you have built a model of a firewater system and generated an NFPA hydraulic report. In this report we can see that the minimum pressure for this system is 63.8 psig in order to see 40 gal/min at the most hydraulically remote nozzle.

To those familiar with the NFPA Example this was based off, you may notice a slight difference in the minimum pressures calculated by Fathom. This is because Fathom uses a more detailed model to account for various behavior in the system. You can read more about these differences on the NFPA-Panel page.

Crude Oil Pipeline

Summary

The objective of this example is to model a pipeline system to investigate the effects of a viscous fluid. We will answer a number of questions about the pipeline system such as:

1. How does viscosity affect pump performance?
2. What is the flow rate?
3. What is the maximum pressure in the pipe?
4. Is the flow laminar or turbulent?
5. How high can the hill be until a vacuum (i.e., less than atmospheric pressure) is created?
6. How does the hill height affect flow rate?
7. How much can the pump speed be slowed down and still not pull a vacuum?

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Crude Oil Pipeline.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

    a.   Fluid Library = User Specified Fluid

    b.   Name = Petroleum Product

    c.   Density = 0.8 S.G. water

    d.   Dynamic Viscosity = 500 centipoise

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
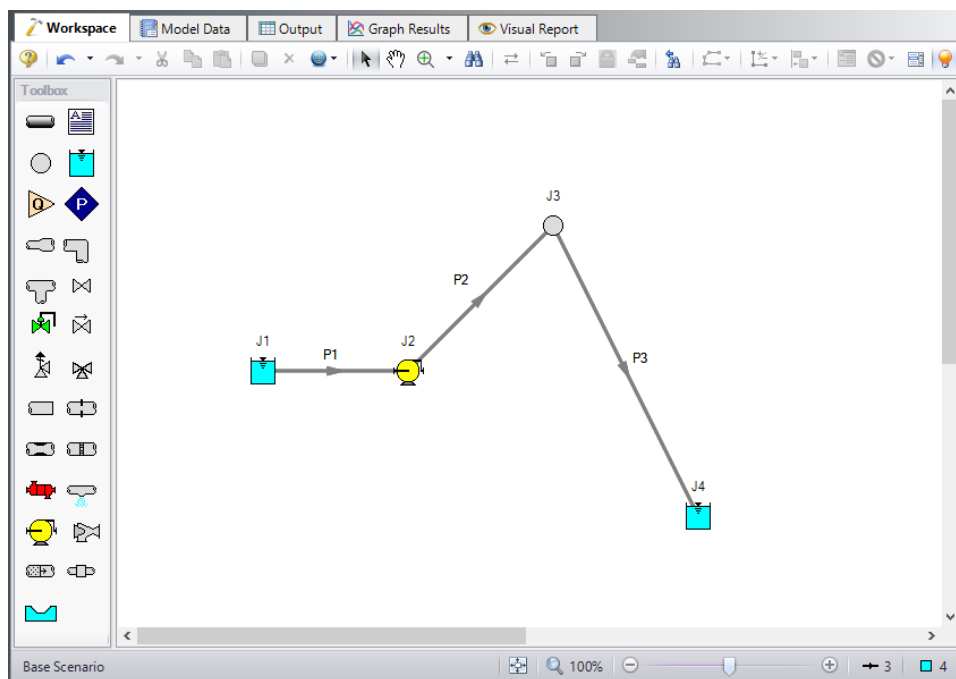
Figure 1: Model for Pipeline Problem

Pipe Properties

1.   Pipe Model tab

    a.   Pipe Material = Steel - ANSI

    b.   Pipe Geometry = Cylindrical Pipe

    c.   Size = 24 inch

    d.   Type = STD (schedule 20)

    e.   Friction Model Data Set = Standard

    f.   Lengths =

Pipe	Length
1	100 feet
2	25 miles
3	50 miles

Junction Properties

Enter the properties of the junctions in the workspace as follows:

1. J1 Reservoir
    a. Liquid Surface Elevation = 750 feet
    b. Liquid Surface Pressure = 0 psig
    c. Pipe Depth = 10 feet
2. J2 Pump
    a. Inlet Elevation = 720 feet
    b. Pump Model = Centrifugal (Rotodynamic)
    c. Analysis Type = Pump Curve
    d. Enter Curve Data =

Volumetric	Head
barrels/day	feet
0	2000
100,000	1800
200,000	1400

    e. Curve Fit Order = 2

Note: If barrels/day is not an available unit, select User Options from the Tools menu, click on Preferred Units, and select the radio button for All Units.

4. J3 Branch
    a. Elevation = 1250 feet
5. J4 Reservoir
    a. Liquid Surface Elevation = 10 feet
    b. Liquid Surface Pressure = 0 psig
    c. Pipe Depth = 10 feet

In the Pump Properties window on the Optional tab, Fathom includes an option to Use Viscosity Correction. We will turn this option on later to determine how much this option affects the answers. Leave the option off for now.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the toolbar or Tools menu. On the Display Parameters tab, select the Pumps button to display the list of output parameters that are specific to pumps. Then add the three correction parameters to pump efficiency CE, pump head rise CH, and pump flow CQ. These are the pump viscosity correction/pump de-rating factors that will de-rate the pump according to the Hydraulic Institute method, ANSI/HI Standard 9.6.7-2010. Change the Volumetric Flow Rate units to barrels/day.

Now select the Pipes button to display the pipe output parameters, then change the units for Volumetric Flow Rate to barrels/day. Also, add Reynolds Number to the list of output variables. Click OK to save the changes and close Output Control.

The Output window contains all the data that was specified in the output control window. The output from this model is shown in Figure 2. Note that the flow rate in all the pipes is 151,571 barrels/day.

Figure 2: Output Window for Pipeline example

Return to the workspace and open the pump properties window. Navigate to the Optional tab and check the box for Use Viscosity Correction. Input the following:

1. Head Correction (ANSI/HI 9.6.7-2015) = Assume Calculated Flow Is At BEP
2. Rated Speed (rpm) = 3600

Click OK and run the model again. The output from this run is shown below. Now notice that the flow rate is 143,822 barrels/day. Click the Pump Summary tab in the General Output section to see the values for *CE*, *CH*, and *CQ*. The output from this model is shown in Figure 3.

Figure 3: Output Window for Pipeline example with viscosity correction

Question 1. How does viscosity affect pump performance?

The flow rate, pump head, and overall power all decreased when the viscosity corrections were used.

Question 2. What is the flow rate?

Before the viscosity correction was added the flow rate was 151,571 barrels/day. After the viscosity correction was added the flow rate became 143,822 barrels/day. Therefore, the viscosity correction factor reduced the flow rate by approximately 5.1%.

Question 3. What is the maximum pressure in the pipe?

The maximum pressure can be found in the pipe summary in the P Static Max column. It occurs in Pipe 2 and it occurs at the inlet of the pipe (outlet of the pump). The pressure is 543.93 psia or 529.24 psig.

Question 4. Is the flow laminar or turbulent?

In the pipe summary window we can see that the Reynolds number is 912.2. Therefore the flow is laminar. The general cut-off for laminar flow is Reynolds number greater than 2,300.

Question 5. How high can the hill be until a vacuum is created?

To answer this question we have to change the elevation of branch J3 and re-run the model. If the hill height is raised to 1500 feet, the minimum pressure in pipe 2 and 3 and at junction J3 is 14.98 psia, or nearly atmospheric pressure. If the hill is higher than 1500 feet, a vacuum would be created. This output can be seen in Figure 4.

Figure 4: Output Window for Pipeline example with 1500 foot hill

Question 6. How does the hill height affect flow rate?

Compare Figure 3 and Figure 4 and you will see that the flow rate is exactly the same. Because the pipe length is the same and the net elevation change is the same, the flow rate will be the same.

Question 7. How much can the pump speed be slowed down and still not pull a vacuum?

The pump can be slowed down to 86.6% before the minimum pressure in pipes 2 and 3 goes below atmospheric (14.7 psia). The output of this run is shown in Figure 5.

Figure 5: Output Window for Pipeline example with pump operating at 86.6% speed

Hot Water System

Summary

The objective of this example is to select the best pump configuration for a hot water system. The design goals of the system are:

- Minimum 6.5 psid drop across flow control valves
- Maximum 10 feet/sec velocity in all pipes
- No NPSH violations
- All pumps must operate between 70% and 100% of BEP
- The selected pump must also work in a special operating mode with one pump off
- The best pump will meet all the requirements and use the least power.

Topics Covered

- Using Junction Libraries to enter data
- Using multiple pump configurations
- Using NPSH and Efficiency data for pumps
- Turning off parts of a model to simulate different operating conditions

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Hot Water System.fth*
- *Hot Water System.dat* - engineering library

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 150 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Model Layout for Hot Water System

Note: A safety factor of 10% will be added to all pipe and junction friction losses. This is added by typing in 1.1 in the Pipe Friction field for Design Factors found on the Optional tab in the Pipe Properties window. The Design Factor of 1.1 will be added to the Junction Friction Losses on the Optional tab in the Junction Properties windows for every junction except for the three pumps.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = Use table below
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Lengths = Use table below

Pipe	Size	Length (feet)
12, 13	4 inch	30
6, 8 ,10, 18, 20, 22, 24	4 inch	50
5, 7, 9, 17, 21	4 inch	100
19, 23	4 inch	150
11, 14	6 inch	30
3, 15	6 inch	50
1, 4, 16	6 inch	100
2	6 inch	300

2. Optional tab
    a. Design Factors
        i. Pipe Friction = 1.1

Junction Properties

1. J1 Reservoir
    a. Name = Topping Tank
    b. Liquid Surface Elevation = 10 feet
    c. Liquid Surface Pressure = 0 psig
    d. Pipe Depth = 10 feet
2. All Tees

      a.   Elevation = 0 feet

      b.   Loss Model = Simple

3.   All Bends

      a.   Inlet Elevation = 0 feet

      b.   Type = Standard Elbow (knee, threaded)

      c.   Angle = 90 Degrees

4.   J14 & J17 Control Valves

      a.   J14 Name = FCV A

      b.   J17 Name = FCV B

      c.   Inlet Elevation = 0 feet

      d.   Valve Type = Flow Control (FCV)

      e.   Control Setpoint = Volumetric

      f.   Flow Setpoint = 220 gal/min

5.   All Pumps & Heat Exchangers

      a.   J4 Name = Pump A

      b.   J7 Name = Pump B

      c.   J10 Name = Pump C

      d.   J15 Name = Heat Exchanger A

      e.   J18 Name = Heat Exchanger B

      f.   Inlet Elevation = 0 feet

To show the names of the pumps, control valves, heat exchangers, and reservoirs, a new Workspace Layer will be created.

1.   Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2.   Name the layer Junction Labels.
3.   Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4.   Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5.   Toggle the visibility icon next to the pumps, heat exchangers, control valves, and reservoir to on. This will cause their labels to be shown.
6.   Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7.   Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8.   Close the Layer Settings window.

To enter the pump and heat exchanger data, we will use a pre-defined library:

1.   Open the Library menu and select the Library Manager.
2.   At the bottom, click Add Existing Library. Navigate to the AFT Fathom 13 Examples folder and select the file titled **Hot Water System.dat** and Open. The library *Hot Water System* will be added to the list of available libraries.

> **Note:** Typically when a Library is first added to the list in Library Manager, it will automatically be connected and will appear with a checkmark next to it. Step 3 below is only needed if the library is not automatically connected.

3. If not already, check the box next to the library name Hot Water System, this will connect the library to the model.

4. Close Library Manager

5. Now open the Pump Properties window for Pump A, and in the Library Jct field at the top of the properties window, select US - Hot Water Pump with Multiple Configs from the drop down menu.

6. In the Pump Curve area, make sure RPM = 1800 RPM and Impeller = 6 inches.

7. Repeat this process for Pump B and Pump C.

8. Open the Heat Exchanger Properties Window for Heat Exchanger A, and select US - Hot Water Heat Exchanger in the library drop down box. This library includes the pressure loss data for the heat exchangers.

9. Repeat these steps for Heat Exchanger B.

Finally, use Global Junction Edit to add the 10% safety factor.

1. Select Global Edit then Global Junction Edit from either the toolbar or Edit menu.

2. Select Common Data

3. Click Select Common Junction Data

4. In the Design Factor area, set Junction Friction Loss = 1.1

5. Click OK

6. Click All to select all junctions in the list

7. Check the box in the Parameters to Change (Select in List) area next to Design Factor 1.1

8. Click Apply Selections. A message should appear to inform you that changes were successively applied.

9. Click OK.

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

After running the model, select the Pump Summary tab and review these results. The pump summary data is shown in Figure 2.

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)
4	Show ...	Pump A	145.4	19.85	16.86	39.63	75.99	100.0	1.882	176.2	82.54	33.25	9.476
7	Show ...	Pump B	149.2	20.37	16.77	39.42	76.55	100.0	1.907	176.2	84.71	33.01	9.546
10	Show ...	Pump C	145.4	19.85	16.86	39.63	75.99	100.0	1.882	176.2	82.54	32.58	9.476

Figure 2: Pump Summary for First Configuration

Now we must review these results to determine if the requirements are met. The pumps operate at 82% to 85% of BEP and there are no NPSH violations. Although the pump meets the requirements, we can see from the valve summary in Figure 3 that the flow control valves do not maintain a minimum of 10 psid pressure drop. The velocity in all the pipes is less than 10 feet/sec. The pipe summary is shown in Figure 4.

Jct	Name	Valve Type	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP Stag. (psid)	dH (feet)	P Static In (psia)	Cv	K	Valve State
14	FCV A	FCV	220.0	30.02	4.481	10.534	31.89	103.0	22.05	Open
17	FCV B	FCV	220.0	30.02	3.083	7.249	31.19	124.2	15.17	Open

Figure 3: Valve Summary for First Configuration

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	dH (feet)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)
1	Pipe	0.0	0.000	18.95	18.95	0	0	0.00000	0.00000	0	0.0000	18.95	18.95	18.95	18.95
2	Pipe	440.0	4.886	20.82	19.15	0	0	1.66826	1.66826	0	3.9219	20.82	19.15	20.97	19.31
3	Pipe	440.0	4.886	19.07	18.79	0	0	0.27804	0.27804	0	0.6537	19.07	18.79	19.23	18.95
4	Pipe	440.0	4.886	18.79	18.24	0	0	0.55608	0.55608	0	1.3073	18.79	18.24	18.95	18.39
5	Pipe	145.4	3.665	18.30	17.78	0	0	0.52684	0.52684	0	1.2386	18.30	17.78	18.39	17.87
6	Pipe	145.4	3.665	34.64	34.37	0	0	0.26342	0.26342	0	0.6193	34.64	34.37	34.73	34.46
7	Pipe	149.2	3.761	18.22	17.67	0	0	0.55368	0.55368	0	1.3017	18.22	17.67	18.32	17.76
8	Pipe	149.2	3.761	34.44	34.16	0	0	0.27667	0.27667	0	0.6504	34.44	34.16	34.53	34.25
9	Pipe	145.4	3.665	18.02	17.49	0	0	0.52684	0.52684	0	1.2386	18.02	17.49	18.11	17.58
10	Pipe	145.4	3.665	34.35	34.09	0	0	0.26315	0.26315	0	0.6187	34.35	34.09	34.44	34.18
11	Pipe	294.7	3.273	18.32	18.25	0	0	0.07716	0.07716	0	0.1814	18.32	18.25	18.39	18.32
12	Pipe	145.4	3.665	34.32	34.17	0	0	0.15779	0.15779	0	0.3709	34.32	34.17	34.41	34.25
13	Pipe	145.4	3.665	18.23	18.07	0	0	0.15805	0.15805	0	0.3716	18.23	18.07	18.32	18.16
14	Pipe	294.7	3.273	34.18	34.11	0	0	0.07716	0.07716	0	0.1814	34.18	34.11	34.25	34.18
15	Pipe	440.0	4.886	34.02	33.74	0	0	0.27804	0.27804	0	0.6537	34.02	33.74	34.18	33.90
16	Pipe	440.0	4.886	33.66	33.11	0	0	0.55609	0.55609	0	1.3073	33.66	33.11	33.82	33.27
17	Pipe	220.0	5.545	33.06	31.89	0	0	1.16965	1.16965	0	2.7497	33.06	31.89	33.27	32.10
18	Pipe	220.0	5.545	27.41	26.83	0	0	0.58482	0.58482	0	1.3749	27.41	26.83	27.62	27.03
19	Pipe	220.0	5.545	22.53	20.77	0	0	1.75447	1.75447	0	4.1246	22.53	20.77	22.73	20.97
20	Pipe	220.0	5.545	33.06	32.48	0	0	0.58482	0.58482	0	1.3749	33.06	32.48	33.27	32.68
21	Pipe	220.0	5.545	32.36	31.19	0	0	1.16965	1.16965	0	2.7497	32.36	31.19	32.57	31.40
22	Pipe	220.0	5.545	28.11	27.53	0	0	0.58482	0.58482	0	1.3749	28.11	27.53	28.31	27.73
23	Pipe	220.0	5.545	23.22	21.47	0	0	1.75447	1.75447	0	4.1246	23.22	21.47	23.43	21.67
24	Pipe	220.0	5.545	21.36	20.77	0	0	0.58482	0.58482	0	1.3749	21.36	20.77	21.56	20.97

Figure 4: Pipe Output for First Configuration

After examining this pump configuration, this process needs to be repeated for each configuration, in order to find which pump best meets the specified requirements. A tool that can be used for this kind of evaluation is the Scenario Manager. Open the Scenario Manager on the Quick Access Panel and create

five scenarios, one for each pump configuration. Below each scenario, create a child scenario for each configuration named Pump A Off. The Scenario Manager on the Quick Access Panel should look like Figure 5. Alternatively, you can access the Scenario Manager from the Tools menu.
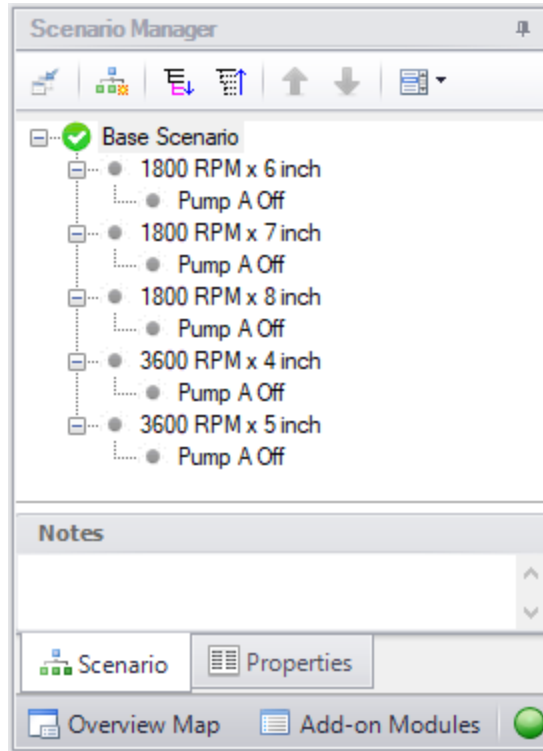
Figure 5: Scenario Manager on the Quick Access Panel for Hot Water System

Now, work your way through each scenario and change the pump configuration to the pump configuration for each scenario. For the scenarios where Pump A is off, use the Special Condition Pump Off No Flow. Run each scenario, note the sum of the three pump's power usage, and determine if the pump meets all the requirements. It would help to create a table of the requirements and configurations as shown in Table 1.

Table 1: Summary Table for Hot Water System

Pump	1800 x 6	1800 x 7	1800 x 8	3600 x 4	3600 x 5
Normal Mode					
BEP OK?	Yes	Yes	No	No	No
NPSH OK?	Yes	Yes	Yes	Yes	Yes
FCV dP OK?	No	Yes	Yes	Yes	Yes
Pipe Velocity OK?	Yes	Yes	Yes	Yes	Yes

Pump	1800 x 6	1800 x 7	1800 x 8	3600 x 4	3600 x 5
Power Used (hp)	5.671	8.401	11.948	11.948	21.899
Special Mode (Pump A Off)					
NSPH OK?	Yes	Yes	Yes	Yes	Yes
FCV dP OK?	No	Yes	Yes	Yes	Yes
Pipe Velocity OK?	Yes	Yes	Yes	Yes	Yes

From Table 1, it can be seen that the only configurations that meet all the requirements are the 1800 RPM x 7-inch impeller. One thing to note is that the 1800 RPM x 8-inch and 3600 RPM x 4-inch configurations are actually the *same* pump configuration due to the affinity laws. The power usage of both configurations help show that they are the same pump.

Note: The 1800 RPM x 6-inch configuration with Pump A turned off is a case where the flow control valves will have to add pressure in order to maintain their set point. A flow control valve adding pressure is physically unrealistic, however, it is useful for troubleshooting purposes because it shows how close a control valve is to being able to maintain set point. The fact that the FCVs fail with this particular scenario is an example of how a certain pump configuration and operating condition may not be able to provide the system pressure and meet the requirements.

Plant Cooling Cost Calculation

Summary

This example demonstrates the fundamental concepts of doing a cost analysis with AFT Fathom. The example illustrates how to calculate initial and life cycle costs for a given system design.

After designing a plant cooling system, it is necessary to calculate the system cost over a 10-year period to determine the feasibility of the design. The system model consists of four circulating water pumps, schedule 20 steel - ANSI pipes and fittings, and two sets of cooling tower cells.

To determine the cost of the cooling system for a 10-year period, include material, installation, and energy costs for the pumps, pipes, and fittings. The cooling towers costs will be determined apart from the system cost (cooling tower cost calculations are not included in this example).

Topics Covered

- Creating cost libraries
- Entering pipe, junction and fitting cost data into libraries
- Connecting cost libraries
- Using Cost Settings
- Using the Cost Report

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition, it is assumed that the user has worked through the Beginner - Cost Calculation example and is familiar with the basics of doing cost calculations.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Plant Cooling Cost Calculation - Initial.fth*
- *US - Plant Cooling Cost Calculation - Final.fth*
- *Plant Cooling.dat* - existing engineering library
- *Plant Cooling - Pump Costs.cst* - completed cost library for *Plant Cooling.dat*

- *Plant Cooling - Elbow Costs.cst* - completed cost library for **AFT INTERNAL LIBRARY**
- *Plant Cooling - Pipe Costs US.cst* - completed cost library for **Steel - ANSI**

Note: The cost libraries above will be recreated as part of this example, but they are also included as reference material.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the *US - Plant Cooling Cost Calculation - Initial.fth* model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder**.

The model should appear as shown in Figure 1.
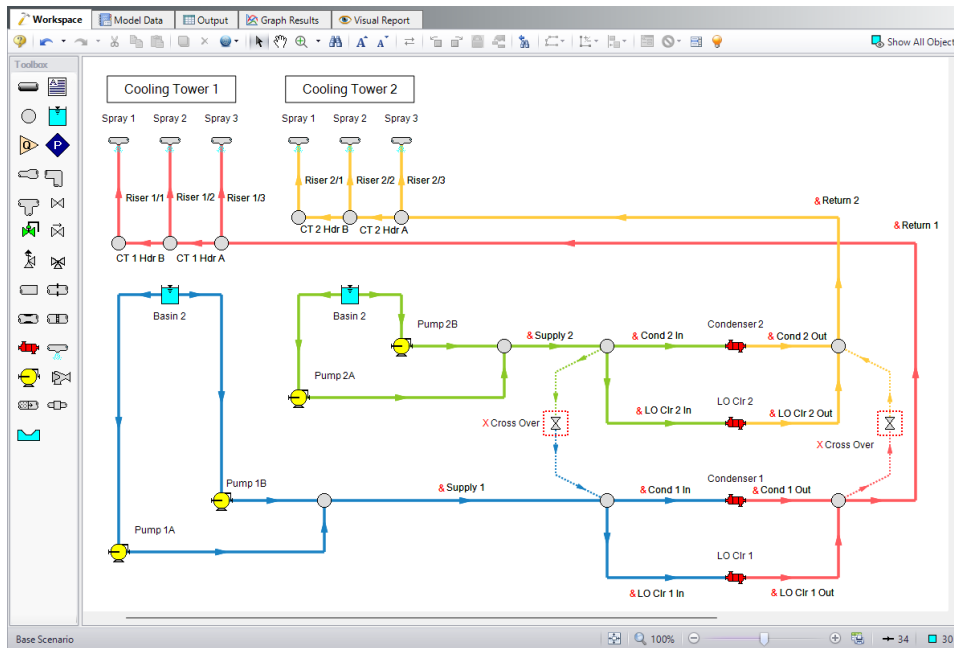
Figure 1: The Plant Cooling Cost Calculation model (from the Examples folder in the AFT Fathom application directory)

Step 3. Define the Cost Settings group

By default, the cost calculations are turned off. To turn on the cost calculations, open Analysis Setup, navigate to the Cost Settings panel, and select Calculate in the Cost Calculation section. The Cost Settings panel is shown in Figure 2.

The Cost Definitions section allows you to specify the type of costs to be determined.

In the Cost Definitions area, select Material, Installation, and Operation/Energy costs by checking the box next to the name in the list.

In the Energy Cost section, select Use This Energy Cost Information, and enter a Cost of 0.12 U.S. Dollars Per kW-hr.

In the Cost Time Period section, enter a System Life of 10 years.

The Cost Settings panel should look like Figure 2. Close the Analysis Setup window by clicking OK.
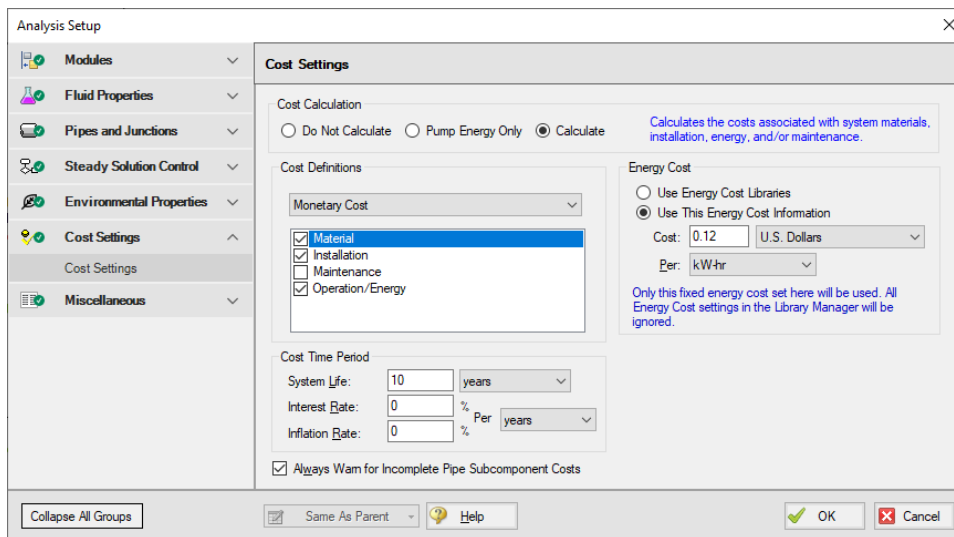
Figure 2: The cost calculations are selected on the Cost Settings window

Step 4. Create the Cost Libraries

Refer to the AFT Fathom Help Content for detailed information regarding creating and using cost libraries.

The energy cost for the pumps was specified in the Cost Settings as a fixed cost rate. Now, the material and installation cost for the pumps, pipes, and fittings must be included. This will be done by creating **three new cost libraries**.

The **first** cost library will be for the pipe material and installation costs and it will be associated with the engineering library that contains the pipe material data, which is the Steel - ANSI pipe material library.

The **second** cost library will be for the additional pipe fittings and losses costs for the elbow losses that are lumped into the pipes. The second cost library will be associated with the AFT INTERNAL LIBRARY.

This is because the AFT Internal Library contains the loss information for the additional fittings and losses tab in the Pipe Properties window.

The **third** cost library will be for the pump material and installation costs, and it will be associated with the engineering library that contains the pump information (i.e., the *Plant Cooling.dat* engineering library).

Create a cost library for the pipe material and installation costs

Create a new cost library for the pipe costs by opening the Library menu and selecting Cost Library. Create a new cost library by clicking New at the bottom. When you click New, you are prompted to choose the engineering library with which the cost library will be associated.

The piping material data used in the model comes from the default Steel - ANSI Pipe Material Library, so the pipe cost library will be connected to this library, as well. Select **Steel - ANSI** from the library list shown in the Select Library window, then click Select.

Important: Make this choice carefully, because once you have made the association you cannot change it.

Once the library is created, you must enter a meaningful description and select the cost units. Enter the following data on the General tab:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Plant Cooling - Pipe Costs US
4. Notes = This library is for the Plant Cooling Cost Calculation example file

Save the cost library to a file by clicking Save. Specify a file name such as, *Plant Cooling - Pipe Costs US.cst*, and click Save. After saving, if prompted, choose to add the new library to the list of available and connected libraries by clicking Yes.

The information on the General tab should appear as shown in Figure 3. The selected library file information, cost type and monetary units, a description of the cost library, and any descriptive notes you have added will be displayed.

Be sure to keep the Edit Cost Library window open for the next step. For this cost library, you will enter pipe material and installation costs on the Pipe Material tab.

Note: The library filenames may be different than those shown in Figure 3.
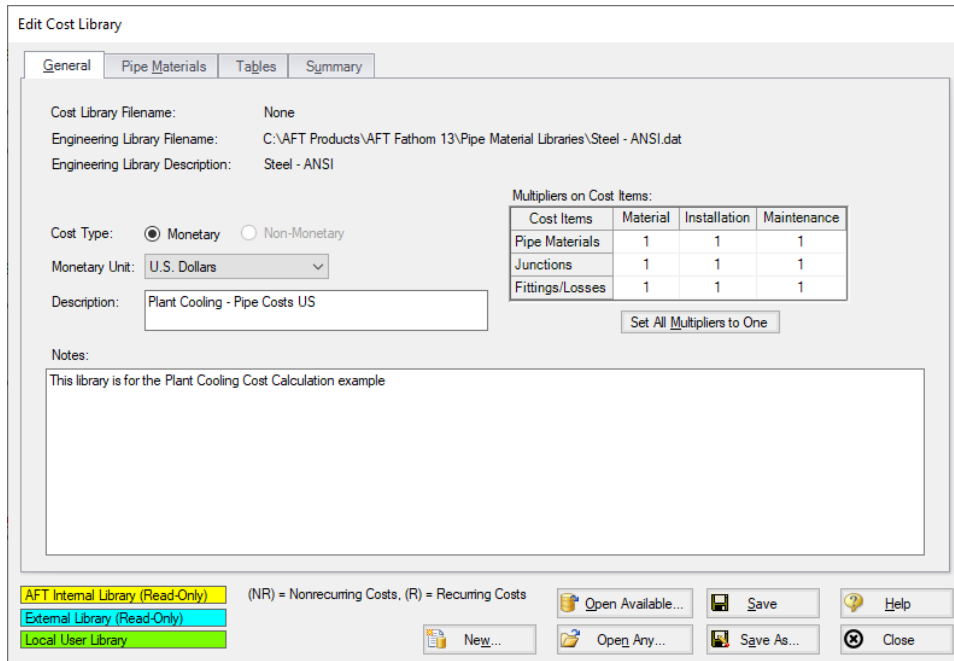
Figure 3: The Edit Cost Library window is used to enter cost library data

Enter the pipe material costs

After the pipe cost library has been created, select the Pipe Materials tab. This is where you will enter the material and installation costs for the piping.

The Pipe Materials tab shows all the pipes in the engineering library (AFT Steel - ANSI Pipe Material Library). Costs can be entered at several levels. You can enter costs at the material level, the nominal size level, and finally at the type (i.e., schedule) level. Costs entered at the material level apply to all nominal sizes and types in that material type. Costs entered at the nominal size level apply to all schedules within that nominal size. Costs entered at the type (schedule) level apply only to that type. **For this example, all of the pipe material costs will be entered at the type (schedule) level for specific sizes.**

To enter a new cost, start by switching the Sort selection to Material, Type or Schedule, Nominal Size. Navigate to Steel - ANSI, STD, 8 inch. Next click New Cost to create a new cost item in the table. The new cost item will appear as a new column.

Each pipe size in this example has two non-recurring (NR) costs associated with it. The costs to be entered are the material and installation costs. You will need to make a second column and chang Cost Type to Installation (NR). Figure 4 shows the costs entered for 8-inch STD Steel - ANSI.

Enter the non-recurring pipe costs for the pipe in this example, as shown below.

All pipes are STD Steel - ANSI:

Nominal Size (inches)	Material Cost (dollars/foot)	Installation Cost (dollars/foot)
8	10.50	24.00
14	22.50	51.50
18	27.40	70.00
20	35.40	85.50
24	40.00	96.50
28	58.00	110.00
30	60.00	129.00
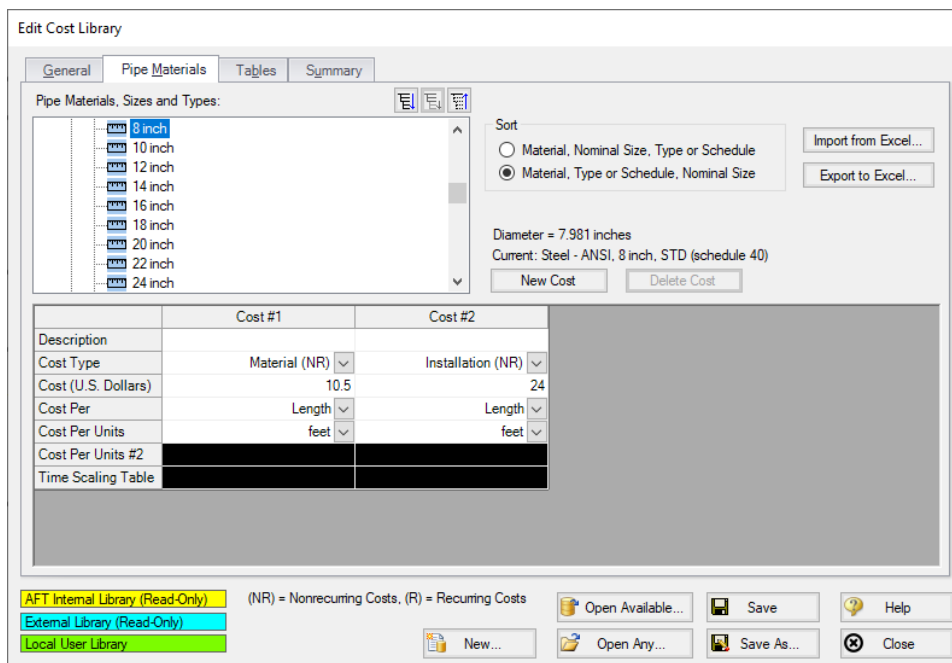
Figure 4: Non-recurring pipe costs, such as material and installation costs, are entered on the Pipe Materials tab

After the material and installation costs are entered for each STD type for the sizes listed above, click Save to save the information in the cost library file, and then click Close to close the Edit Cost Library window.

Create a cost library for the elbow fittings with cost scale tables

Several pipes in the model have 90 degree elbows specified in the Fittings and Losses tab in the respective Pipe Properties windows. The cost of these 90 degree elbow fittings can be included in the cost calculation. The costs for these items will be accounted for in scale tables. Scale tables can be used to vary a cost with a parameter such as diameter (i.e., size). Once created, this scale table can be applied to fittings and losses items.

Create a cost library for the elbow fittings and losses costs by opening the Library menu and selecting Cost Library. Create a new cost library by clicking **New**. You are prompted to choose the engineering library with which the cost library will be associated.

➤**The cost library for the elbow fittings and losses will be associated with the AFT INTERNAL LIBRARY**, as this is the engineering library in which the pipe fittings and losses data is kept for the default elbow pipe fittings and losses. Select AFT INTERNAL LIBRARY and click Select.

On the General tab, specify the following information:

1.　Cost Type = Monetary
2.　Monetary Unit = U.S. Dollars
3.　Description = Plant Cooling - Elbow Costs
4.　Notes = This library is for the elbows included as additional Fittings and Losses for the pipes

Click Save and save this cost library with a file name such as *Plant Cooling - Elbow Costs.cst*

The first table to create is the scale table for the 90-deg. elbow installation costs. Select the Tables tab, and click New Table.

Enter the following data for the scale table on the New Scale Table window:

1.　Name = 90 Elbows Installation
2.　Table Type = Diameter
3.　Table Format = Cost

After entering the data, click OK.

Now enter the cost data in the table using the values in the table below, as shown in Figure 5.

Diameter (inches)	Installation Cost (US dollars)
7.981	185.50
12.25	298.00
17.376	446.00
23.25	595.00
29.00	744.00
33.00	843.00

You do not need to enter data in the scale tables for every diameter in the model. If a diameter falls between two data points in the table, AFT Fathom will use the points on either side to linearly interpolate for a value. If the cost function is non-linear, you may need to add additional data points to achieve a more accurate cost value.

Create another scale table for the material costs using the following data:

1. Name = 90 Elbows Material
2. Table Type = Diameter
3. Table Format = Cost

Diameter (inches)	Material Cost (US dollars)
7.981	68.00
12.25	151.50
17.376	262.50
23.25	373.50
29.00	484.50
33.00	558.50

After the scale tables have been created, and the cost data entered, the Tables tab should appear as shown in Figure 5.
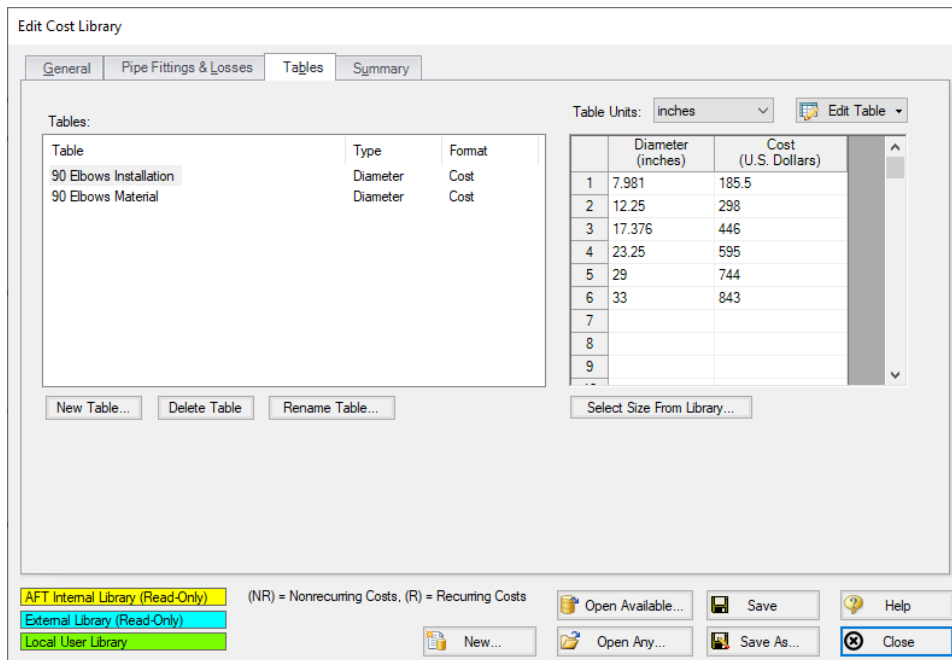
Figure 5: The Tables tab on the Edit Cost Library window is used to create cost scale tables

Click Save after the two cost tables are created and keep the Edit Cost Library window open for the next step.

Add the costs for pipe fittings

With the cost scale tables for the fittings defined, the costs for the pipe fittings can be added. Select the Pipe Fittings & Losses tab. A list of all of the available fittings and losses is displayed in the Pipe Fittings

& Losses list.

The fittings in the model are Elbow/Bend, Smoothed Flanged, r/D=1, 90 deg. (C). Navigate through the list until you find the proper selection for these fittings.

Add the material cost by clicking New Cost, and entering the following data:

1. Description = Elbow Material Costs
2. Cost Type = Material (NR)
3. Material = Steel - ANSI
4. Material Size = All Sizes
5. Material Type = All Types
6. Use Size Table = Table of Costs
7. Multiplier = 1
8. Size Scaling Table = 90 Elbows Material

Now add the installation cost by clicking New Cost again and entering the following data:

1. Description = Elbow Installation Costs
2. Cost Type = Installation (NR)
3. Material = Steel - ANSI
4. Material Size = All Sizes
5. Material Type = All Types
6. Use Size Table = Table of Costs
7. Multiplier = 1
8. Size Scaling Table = 90 Elbows Installation

The actual cost values used for the fittings are the values that were entered in the scale tables on the Tables tab. The scale table to use for each cost is specified when the cost is defined. After the cost data is entered, the Pipe Fittings & Losses tab should appear as shown in Figure 6.
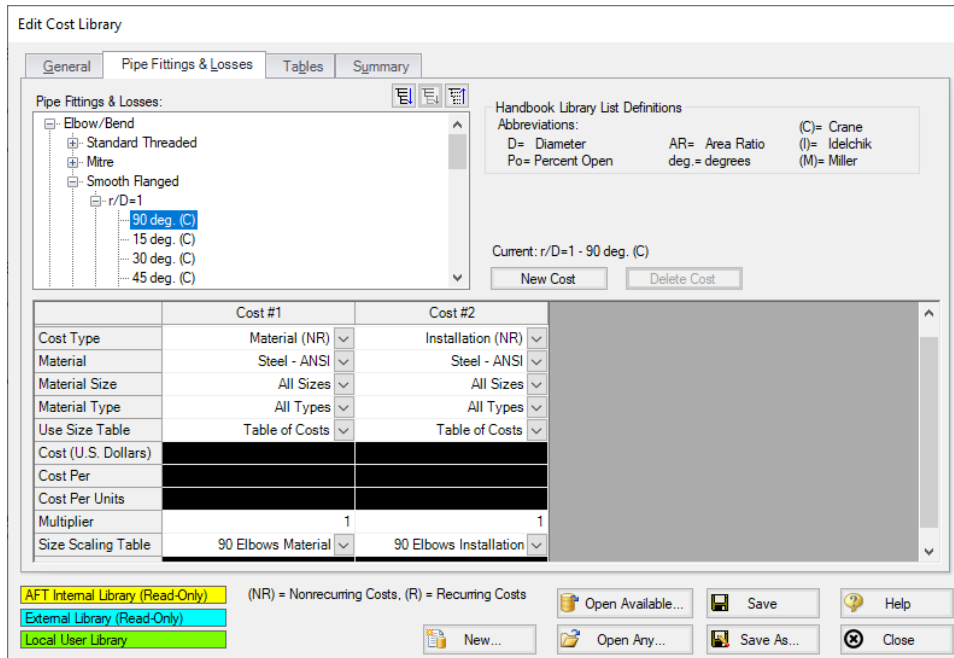
Figure 6: The Pipe Fittings & Losses tab in the Edit Cost Library window is used to specify costs for pipe fittings and losses items

Click Save and close the Edit Cost Library window, as now, the cost library for the elbow fittings and losses is now completed.

Create a cost library for the pumps

To enter cost for a pump, or any other junction type, the junction must first be added to the an engineering library in the Library Manager. Cost data is then entered in a cost library associated with that engineering library. The pumps used in this example have already been added to an engineering library, but no costs have been created for it yet. Create a new cost library associated with the Plant Cooling engineering library by opening the Library menu and selecting Cost Library.

Click **New**, then select the **Plant Cooling** library and click **Select**. Specify the following general information on the General tab:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Plant Cooling - Pump Costs
4. Notes = This library is for the pumps in the Plant Cooling engineering library

Save the cost library to a file by clicking Save and giving the file a name such as ***Plant Cooling - Pump Costs.cst***. When prompted, choose to add the new library to the list of available and connected libraries. Keep the Edit Cost Library window open for the next step.

Enter the pump costs

After the pump cost library has been created, select the Junctions tab. This is where you will enter the material and installation costs for the pumps.

All of the junctions that are available in the selected engineering library will be listed. Select the **US - Plant Cooling Pump** from the list of pumps.

Add the material cost by clicking New Cost, and entering the following data:

1. Description = Pump Material Cost
2. Cost Type = Material (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 28500
5. Cost Per = Item

Add the installation cost by clicking New Cost again and entering the following data:

1. Description = Pump Installation Cost
2. Cost Type = Installation (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 12500
5. Cost Per = Item

Figure 7 shows the pump costs entered in the cost library.

Click Save and then click Close.
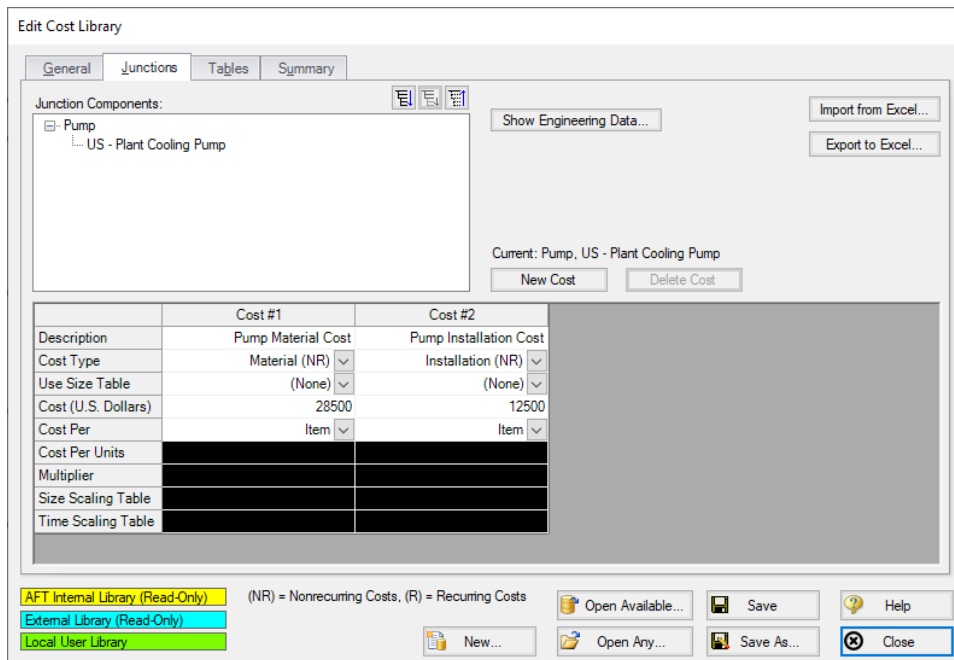
Figure 7: The Junctions tab on the Edit Cost Library window is used to specify costs for junctions

Connecting the cost libraries

The Plant Cooling engineering and cost libraries should have been added to the list of available libraries and been connected to the example model. If they were not, you can use the Library Manager to connect the cost libraries now (see Figure 9). The **Plant Cooling - Pump Costs** is connected subordinate to the **Plant Cooling**. The **Plant Cooling - Pipe Costs US** is connected subordinate to the **Steel - ANSI** pipe material library. Lastly, the **Plant Cooling - Elbow Costs** is connected subordinate to the **AFT INTERNAL LIBRARY**. Refer to the AFT Fathom Help Content for information on how to connect libraries using the Library Manager.

Make sure that the pumps are utilizing the engineering library junction data, otherwise the costs will not be calculated.
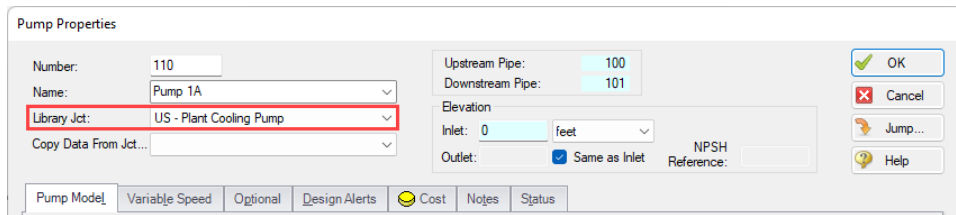
Figure 8: Pump using the engineering library data
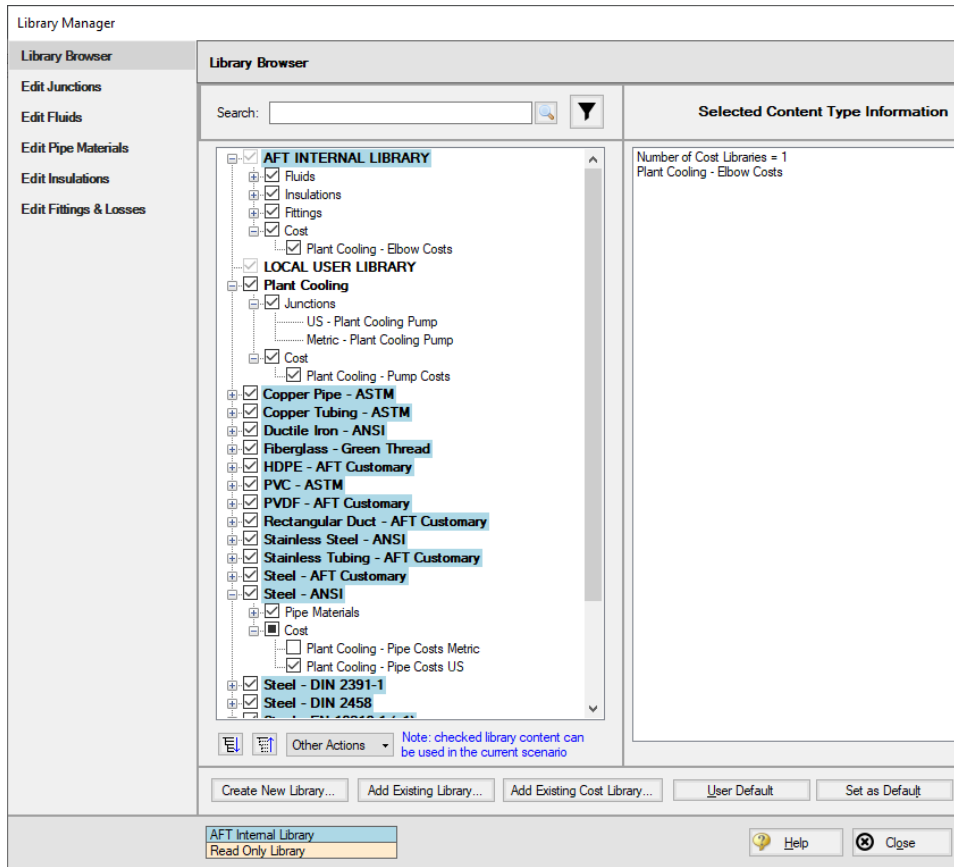
Figure 9: Library Manager with connected engineering and cost libraries

Step 5. Including objects in the Cost Report

The final step before performing a cost analysis is to specify which pipes and junctions you want to be included in the final cost report. This is done by opening the properties window, navigating to the Cost tab, and choosing Include in Cost Report for each pipe and junction to be included, as shown for Pipe 100 in Figure 10.

Include the four circulating water pumps and all of the pipes in the cost report. The Global Edit feature may be used to update this information for all of the pumps and pipes.
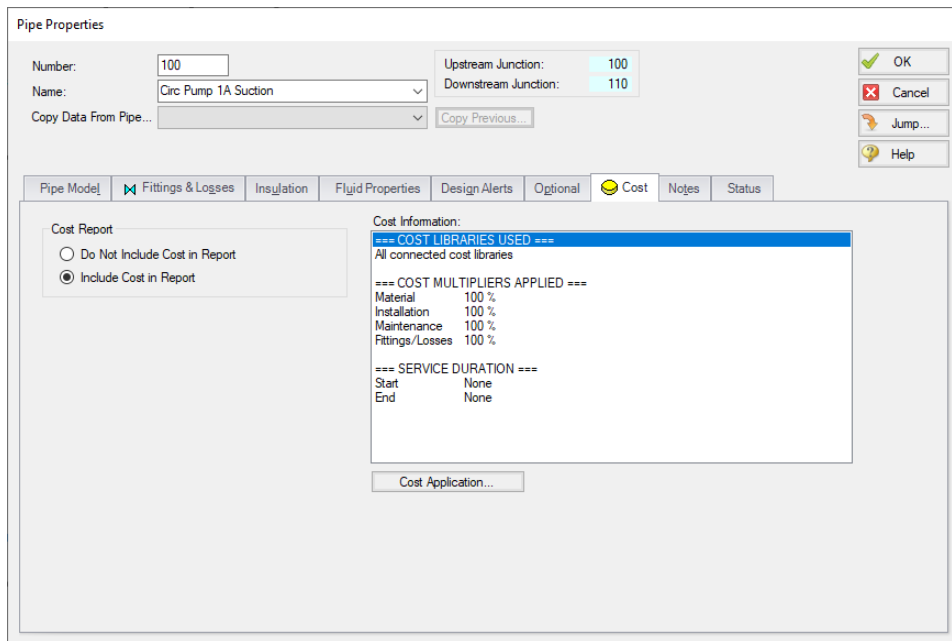
Figure 10: The Cost tab on the Pipe and Junction Properties windows is used to include the objects in the Cost Report

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 7. Examine the Cost Report

The Cost Report is displayed in the General Results section of the Output window. View the Cost report by selecting the Cost Report tab. The content of the Cost Report can be modified from the Output Control window. Figure 11 shows the Cost Report for this example with the Material, Installation, Operation/Energy, and Total costs displayed.

The Cost Report shows the total system cost, as well as the individual totals for the material, installation, and energy costs. In addition, the Cost Report displays the detailed cost for each pipe, junction, and fitting that was included in the report. The items are grouped together by type, and a subtotal for each category is listed.

Figure 11: The Cost Report in the General Output section shows the results of the cost analysis

Analysis Summary

The cost analysis for this example shows the following costs for the plant cooling system design:

Total of All Model Costs	$5,296,174
Total Material Cost	$417,883
Total Installation Cost	$697,484
Total Operation/Energy Cost	$4,180,806
Total Cost of Pipe	$918,940
Total Cost of Pumps	$4,344,806
Total Cost of Fittings	$32,426

Minimizing Cost with AFT Fathom Automated Network Sizing Module™

The AFT Fathom Automated Network Sizing Module (ANS) is a tool built on AFT Fathom technology. With the ANS module you can automatically size all pipes or ducts in your system to minimize monetary cost, weight, volume, or surface area. In addition, you can concurrently size the pumps and pipes to obtain the absolute lowest cost system that satisfies your design requirements. Finally, by accounting for non-recurring and recurring costs, you can size pipe and duct systems to minimize life cycle costs over some specified duration.

By using the AFT Fathom Automated Network Sizing Module (ANS) on this example, not only could the system cost be determined, the cost could also be minimized by varying selected pipe and junction sizes in the system.

Non-Newtonian Mill Discharge Slurry

Summary

This example demonstrates how one can take small-scale raw rheological data for a mill slurry at a minerals plant and use it on a full-scale pipe calculation.

Topics Covered

- Reviewing and entering rheological data
- Using homogeneous scale-up features
- Obtaining and using Power Law and Bingham Plastic constants

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition the user should have worked through Non-Newtonian Phosphates Pumping example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Non-Newtonian Mill Discharge Slurry.fth*
- *Non-Newtonian Mill Discharge Slurry Data.xlsx*

Problem Statement

The piping for a non-settling slurry for a mill discharge at a minerals plant is being designed. The system will supply 880 gal/min of slurry through a 12-inch horizontal pipe 300 feet long.

Rheological data using the actual fluid in a 1/4 inch pipe has been obtained. Perform non-settling slurry calculations to evaluate how well the Bingham Plastic and Power Law models follow the raw data. Then determine the pressure drop in the full-scale pipe using Bingham Plastic, Power Law and Homogeneous Scale-up models.

Step 1. Review Raw Data in Excel

The data taken for 1/4 inch STD schedule (0.364 inch ID), 2.5 foot pipe is as follows:

Density = 87.4 lbm/ft3

Table 1: Pressure drop data on mill slurry (1/4 inch pipe - 0.364 inch ID, 2.5 foot long)

Run	Q (gal/min)	dP (psid)
1	0.013	3.6
2	0.066	5.2
3	0.110	6.4
4	0.220	7.7
5	0.440	8.7

The Bingham Plastic and Power Law models typically process data in terms of **shear rate** and **shear stress**. The above data needs to be changed into a form for input into AFT Fathom.

Open the provided Excel spreadsheet **Non-Newtonian Mill Discharge Slurry Data.xlsx**. Click the Excel tab for English units. Here one can see the raw data and calculated shear rate and shear stress. See Figure 1 for a view of the data. The shear rate and shear stress data is in the yellow columns.

Tube Diameter	0.364 inches
Tube Length	2.5 feet
Slurry Density	87.4 lbm/ft3

	Test Data		Calculated Data			Data Input to Fathom		Fathom Results		
						Shear Rate	Shear Stress (Tau)		Bingham Plastic	Power Law
Run #	Flow rate	ΔP	Flow rate	Velocity	Jm	8v/D	PD/4L	Vol. Flow	dP Stag.	dP Stag.
	gal/min	psid	ft3/sec	feet/sec	ft/ft	1/seconds	psia	gal/min	psid	psid
1	0.013	3.6	0.000029	0.0401	2.37	10.6	0.011	0.013	4.927	3.610
2	0.066	5.2	0.000147	0.2035	3.43	53.7	0.016	0.066	5.605	5.430
3	0.110	6.4	0.000245	0.3391	4.22	89.4	0.019	0.110	6.016	6.173
4	0.220	7.7	0.000490	0.6783	5.07	178.9	0.023	0.220	6.884	7.348
5	0.440	8.7	0.000980	1.3566	5.73	357.8	0.026	0.440	8.396	8.746

Figure 1: Mill slurry raw and calculated data from Excel file

Step 2. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel:
   a. Fluid Library = User Specified Fluid
   b. Name = Mill Discharge Slurry
   c. Density = 87.4 lbm/ft3
   d. Dynamic Viscosity = 10 lbm/hr-ft (note that the viscosity entered here will not be used once the non-Newtonian parameters are entered)
3. Open the Viscosity Model panel
   a. Viscosity Model = Bingham Plastic
   b. Select Calculate from Rheological Data
   c. Click Calculate Constants
   d. Raw Data Type = Tube Flow Rheometer Data (8V/D)
   e. Raw Data =

Parameter	Shear Rate	Shear Stress
Units	1/seconds	psia
1	10.6	0.011
2	53.7	0.016
3	89.4	0.019
4	178.9	0.023
5	357.8	0.026

   f. Curve Fit Order = 1

Figure 2: Bingham Plastic data entry
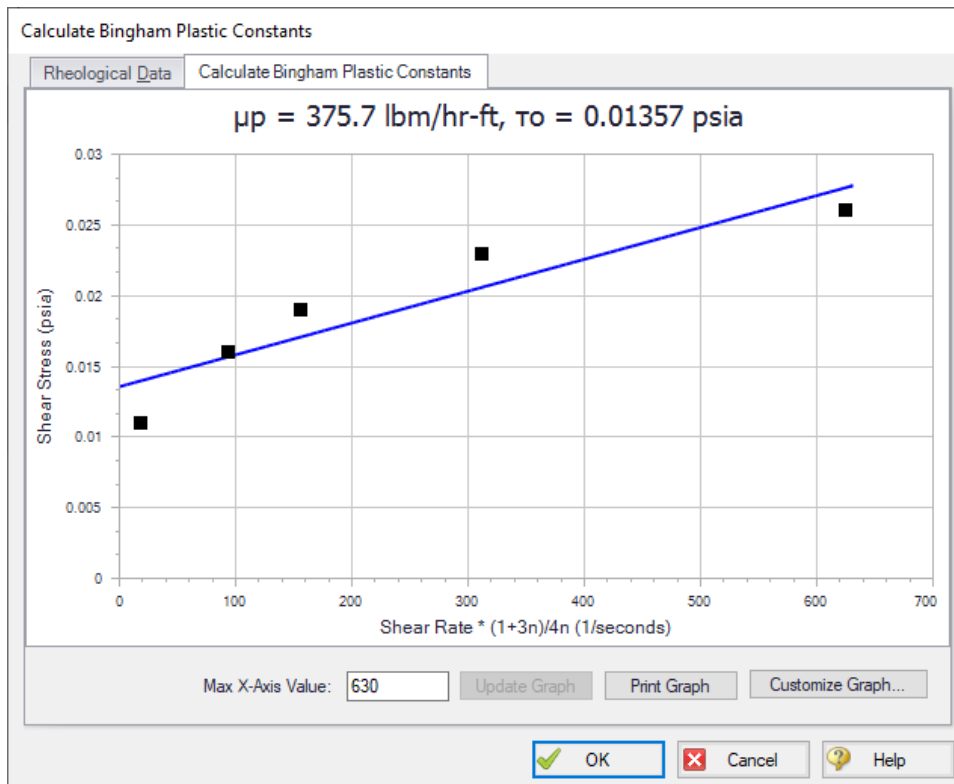
Figure 3: Bingham Plastic curve fit and constants calculation

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
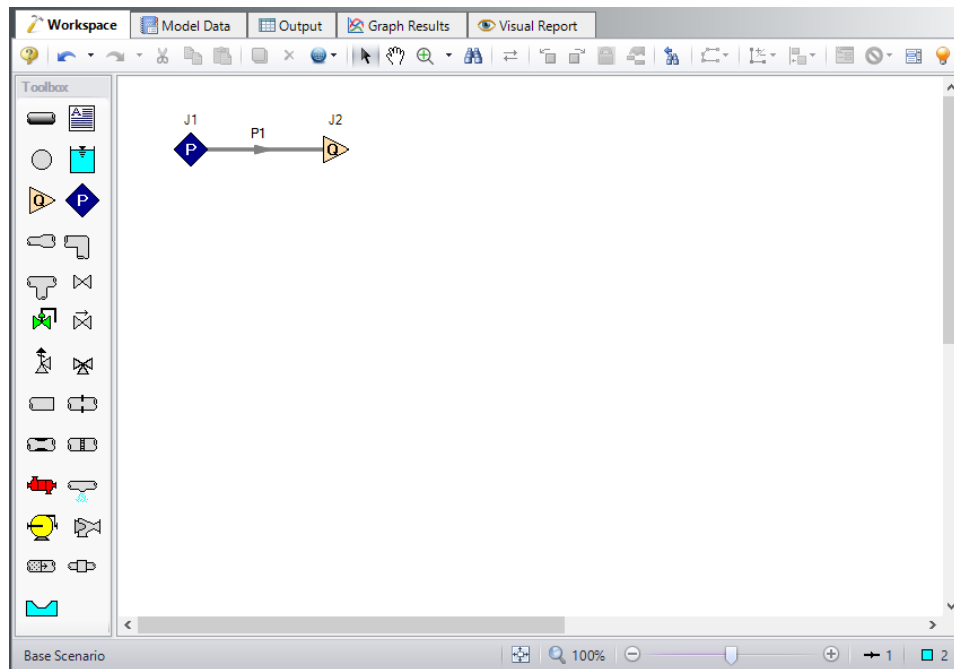
Figure 4: Layout of pipe system for Mill Discharge Slurry example

Pipe Properties

1. Pipe Material = Steel - ANSI
2. Pipe Geometry = Cylindrical Pipe
3. Size = 1/4 inch
4. Type = STD (schedule 40)
5. Length = 2.5 feet

Junction Properties

1. J1 Assigned Pressure
   a. Elevation = 0 feet
   b. Pressure = 100 psia
   c. Pressure Specification = Stagnation
2. J2 Assigned Flow
   a. Elevation = 0 feet
   b. Type = Outflow
   c. Flow Specification = Volumetric Flow Rate
   d. Flow Rate = 0.013 gal/min

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will

have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Figure 5 shows the Output window. It can be seen that the pressure drop is 4.927 psid for 0.01300 gal/min, compared to the raw data value of 3.6 psid.

Figure 5: The Output window with default output data

Step 7. Create Scenarios

From the Base Scenario, make a child scenario named *Bingham Plastic*.

Step 8. Run Other Bingham Plastic Scenarios

Load the Bingham Plastic scenario and use the Duplicate Special feature to create five identical runs. Enter flow rates of 0.066, 0.11, 0.22 and 0.44 gal/min at the Assigned Flow junctions. Run the model and note the pressure drop for each pipe.

Step 9. Create and Run Power Law Scenarios

In the Scenario Manager, right click the Bingham Plastic scenario and select **Clone Without Children**. Name this scenario *Power Law*.

1. Load the Power Law Scenario
2. Open Analysis Setup
3. Open the Viscosity Model panel

      a.   Viscosity Model = Power Law

      b.   Select Calculate from Rheological Data

      c.   Click Calculate Constants

      d.   Raw Data Type = Tube Flow Rheometer Data (8V/D)

      e.   Raw Data = should still be loaded from before

      f.   Curve Fit Order = 1

      g.   Click Generate Curve Fit Now

      h.   Click OK

4.   Click OK

Run the model again.

Figure 7 shows the output for each of the ten cases. Figure 8 plots the test data against the results of Bingham Plastic and Power Law. From Figure 8 it is clear the Power Law model fits the mill discharge slurry data better than Bingham Plastic.
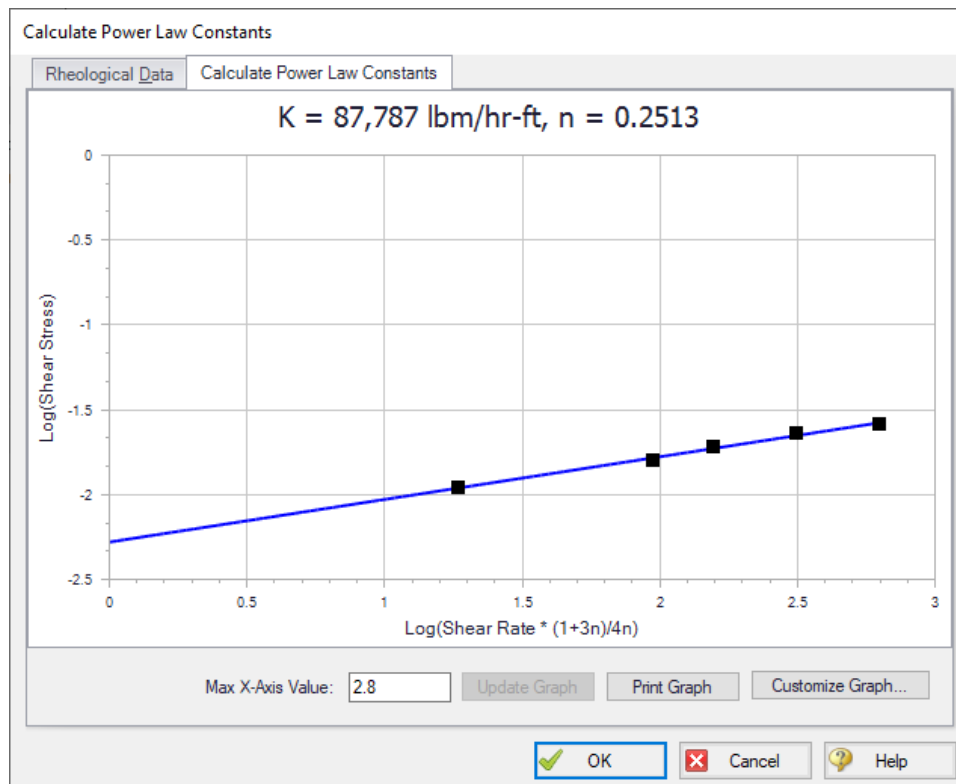
Figure 6: Power Law curve fit from Viscosity Model panel

Tube Diameter	0.364 inches
Tube Length	2.5 feet
Slurry Density	87.4 lbm/ft3

	Test Data		Calculated Data			Data Input to Fathom		Fathom Results		
						Shear Rate	Shear Stress (Tau)	Vol. Flow	Bingham Plastic dP Stag.	Power Law dP Stag.
Run #	Flow rate	ΔP	Flow rate	Velocity	Jm	8v/D	PD/4L			
	gal/min	psid	ft3/sec	feet/sec	ft/ft	1/seconds	psia	gal/min	psid	psid
1	0.013	3.6	0.000029	0.0401	2.37	10.6	0.011	0.013	4.927	3.610
2	0.066	5.2	0.000147	0.2035	3.43	53.7	0.016	0.066	5.605	5.430
3	0.110	6.4	0.000245	0.3391	4.22	89.4	0.019	0.110	6.016	6.173
4	0.220	7.7	0.000490	0.6783	5.07	178.9	0.023	0.220	6.884	7.348
5	0.440	8.7	0.000980	1.3566	5.73	357.8	0.026	0.440	8.396	8.746

Figure 7: Summary of AFT Fathom output and test data
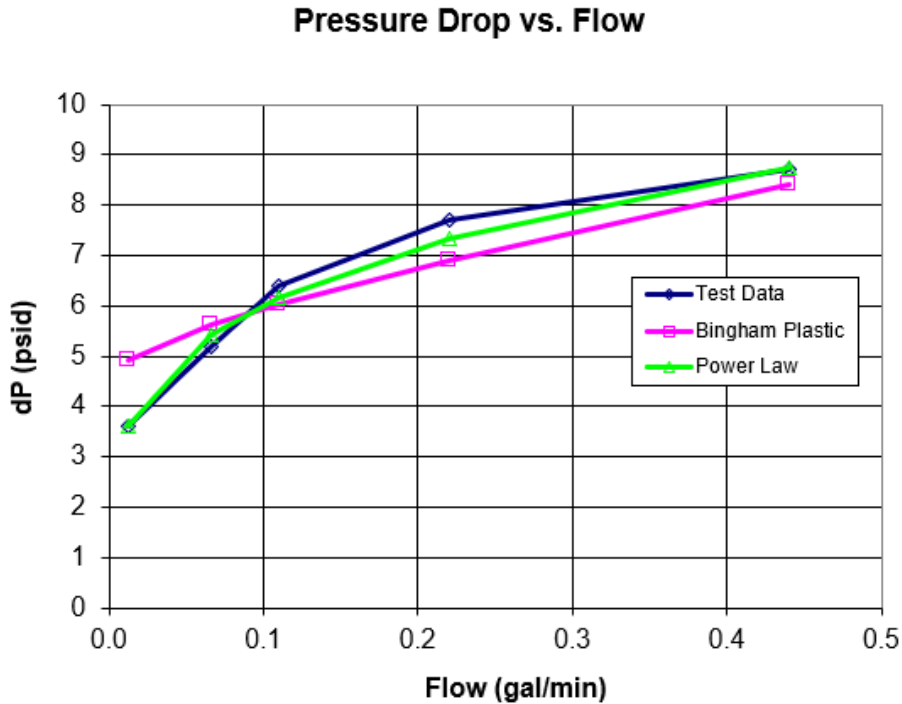
Figure 8: Graph comparing AFT Fathom output and test data

Step 10. Evaluate Full Scale Data

1. Make another child scenario called Full Scale - 12 inch
2. Change pipe P1 to 300 feet length and 12 inch diameter
3. Change junction J2 to a flow rate of 880 gal/min
4. Make three child scenarios below Full Scale - 12 inch: one for each, **Bingham Plastic**, **Power Law**, and **Homogeneous Scale-up**

5. Load the Bingham Plastic scenario, open Analysis Setup, and perform a curve fit for Bingham Plastic

6. Run the Bingham Plastic scenario and note the pressure drop

7. Repeat the above for the Power Law scenario

8. For the Homogeneous Scale-up scenario, use the test data from the Excel spreadsheet. Make sure to enter the data as Volumetric in gal/min and Jm in ft/ft. All 5 data points are laminar flow, as shown in Figure 9.
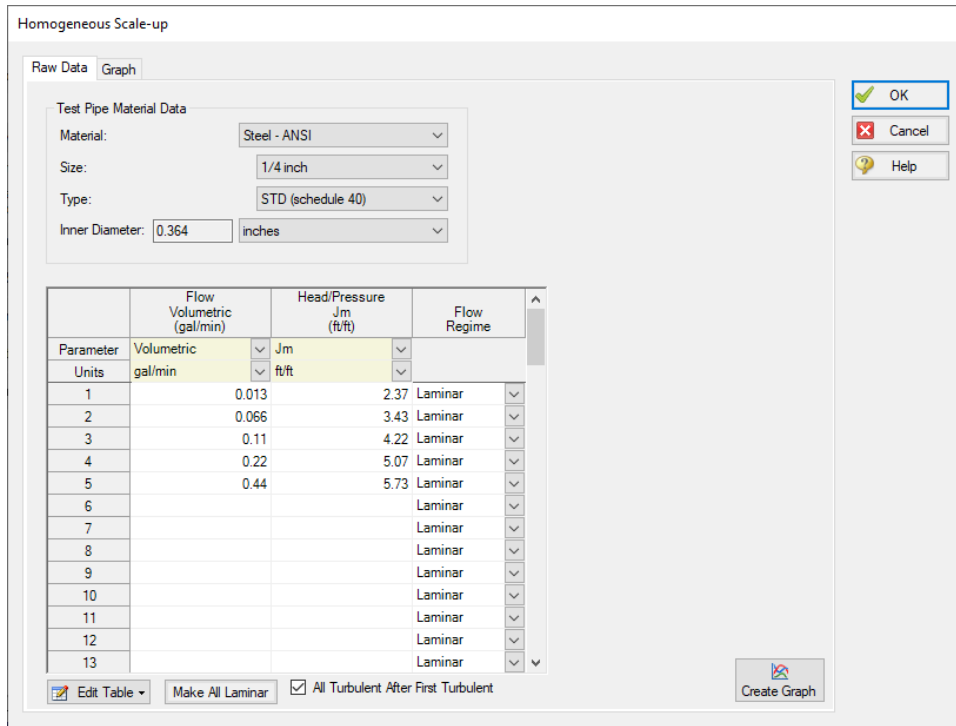
Figure 9: Homogeneous Scale-up window with test data input

Figure 10 shows a comparison of pressure drop results using multi-scenario output for the three children. It can be seen that there is closer agreement between Power Law and Homogeneous Scale-up. The Bingham Plastic results are not as close which was true earlier in this example comparing to actual test data (Figure 8). Therefore the Bingham Plastic pressure drop should be excluded and the design pressure drop should be based on Power Law and Homogeneous Scale-up.

Figure 10: Full scale pressure drop comparison of Bingham Plastic, Power Law, and Homogeneous Scale-up

Analysis Summary

A non-settling mill discharge slurry for a minerals plant was modeled using non-settling slurry calculations in AFT Fathom. Runs for small scale and full-scale calculation were made and pressure drop was determined for these cases.

Module Walk-Through Examples

The examples in the Fathom Module Help System are designed to get new users trained to use the Fathom modules correctly in the shortest amount of time.

If this is your first time using Fathom, please start with the Walk-Through Examples to learn how to create and run Fathom models. Start with the Three Reservoir Model, because it provides the most complete, basic description on how to set up and run a Fathom model. After this example, it is recommended that you work through the remaining Walk-Through examples.

If you are just learning to work with the Fathom modules, start with the Beginner model for each module, then choose an example from the tables on the Module pages that most closely matches your application. The examples are arranged in order of increasing complexity.

Fathom GSC Examples

Fathom XTS Examples

Fathom SSL Examples

Fathom ANS Examples

Fathom Combined Module Examples

Goal Seek and Control (GSC) Examples

The examples below utilize the Goal Seek and Control module.

Fathom GSC Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Heat Transfer in a Pipe	Beginner	Water (liquid)	1	0	Learn the basics of the GSC module. See how to use Analysis Setup to activate and enable the GSC module and to use the Goal Seek and Control group to define variables and goals. Learn where to find the GSC Variables and GSC Goals tabs in the Output window.
Pump Sizing and Selection with Flow Control Valves	Beginner	Water (liquid)	9	1	Learn how to use the GSC module to size a pump in a system with a pump and flow control valves in series.
Cooling System	Beginner	Water (liquid)	20	2	Learn how to link GSC variables together to apply multiple variables and goals in a single simulation.
Model Calibration with Pipe Variables	Beginner	Water (liquid)	6	0	Learn how to use the GSC module to vary pipe friction to calibrate a model to match a measured volumetric flow rate.
Spray Discharge System	Intermediate	Water (liquid)	11	0	Learn how to create groups and use the Group Max/Min and Group Sum goals using the GSC module.
Controlled Heat Exchanger Temperature	Intermediate	Water (liquid)	10	1	Learn how to use the GSC module with heat transfer to vary the open percentage of a three-way valve to achieve a desired temperature.
NFPA Firewater System	Intermediate	Water (liquid)	71	0	Learn how to use the GSC module to determine the most hydraulically remote

Example	Complexity	Fluid	Pipes	Pumps	Description
					nozzle for NFPA Reports.

Beginner - Heat Transfer in a Pipe - GSC

Summary

The objective of this example is to demonstrate how goals and variables are used in the GSC module to achieve an objective.

For this example, we will use the GSC module to determine the flow rate required such that the fluid at the outlet of a 100-foot section of pipe, with heat transfer, reaches a temperature of 120 deg. F.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Beginner - Heat Transfer in a Pipe.fth*
- *US - Beginner - Heat Transfer in a Pipe - GSC.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Heat Transfer in a Pipe.fth** example file listed above, which is located in the Examples folder in the AFT Fathom application folder. Save the file to a different folder. The workspace should look like Figure 1 below:
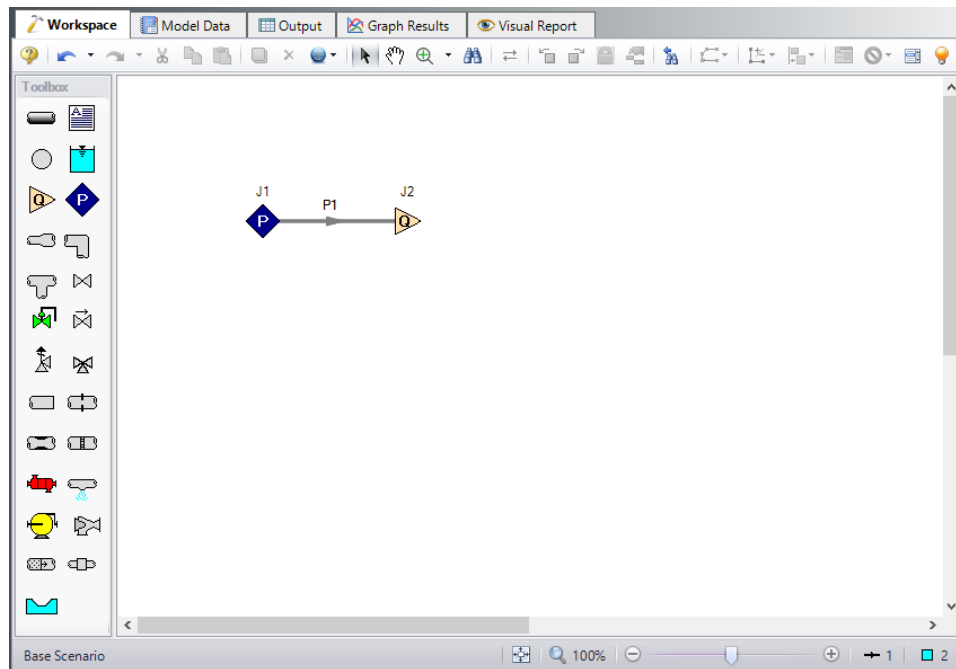
Figure 1: Workspace for GSC Heat Transfer example

Step 3. Define the Modules group

The GSC module is controlled through the Goal Seek and Control group in Analysis Setup. Open Analysis Setup and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use. Alternatively, GSC can be enabled from the Analysis menu by going to the Goal Seek & Control option and selecting Use, as shown in Figure 2.
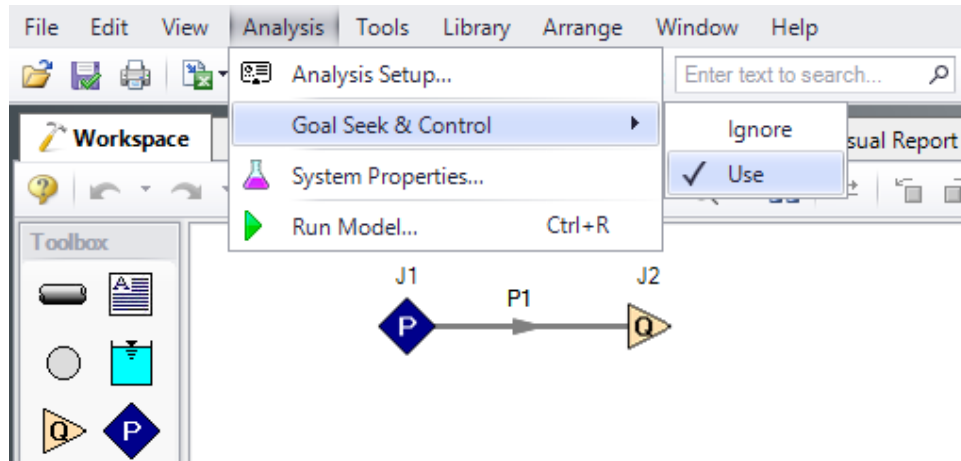
Figure 2: GSC can be enabled and disabled from the Analysis menu

Step 4. Define the Goal Seek and Control Group

The Goal Seek and Control group is made up of three items: Variables, Goals, and Numerical Controls. The default settings in Numerical Controls will suffice for most applications and no input is necessary in this panel for this example. The variables and goals are defined in their respective panels. The Variables panel is shown in Figure 3 below:
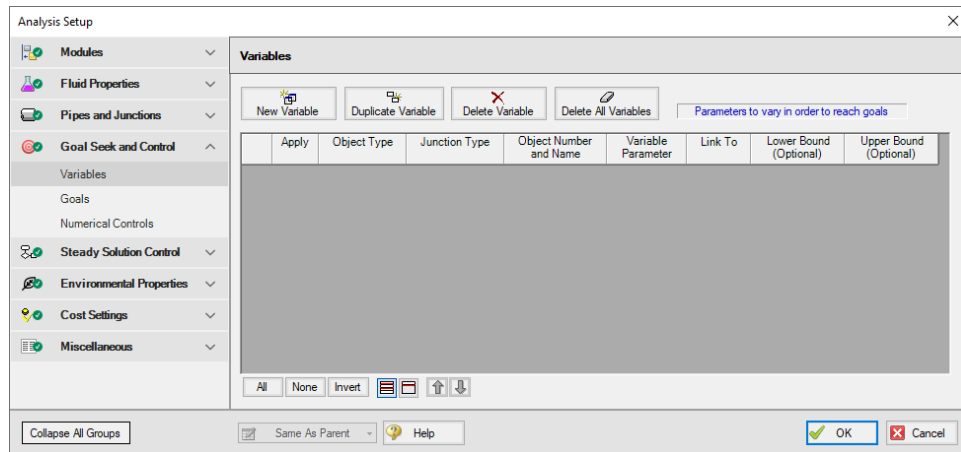
Figure 3: Variables panel

Variables Panel

In the GSC module, variables are the parameters that the solver will modify in order to achieve the specified goals. There should be one variable defined for each goal.

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. The Variables panel allows you to create and modify the system variables. The object type and junction type are selected, then the name and number of the object to which the variable applies, and the object parameter that is to be varied are specified all on the Variables panel.

To create a new variable, follow these steps:

1. Click New Variable
2. Apply = Checked
3. Object Type = Junction
4. Junction Type = Assigned Flow
5. Junction Number and Name = J2 (Assigned Flow)
6. Variable Parameter = Flow
7. Link To = (None)
8. Lower Bound (Optional) = 0

The Apply column allows users to specify which of the variables that have been defined will be used. This allows the flexibility of creating multiple variable cases, while only applying selected variables for any given run.

The Link To column allows users to apply the same variable to multiple objects. This allows users to force parameters for several objects to be varied identically.

Upper and Lower Bounds provide logical extremes during the goal search. For this case, leave both the lower bound and the upper bound values blank.

After entering the data, the Variables panel should appear as shown in Figure 4:
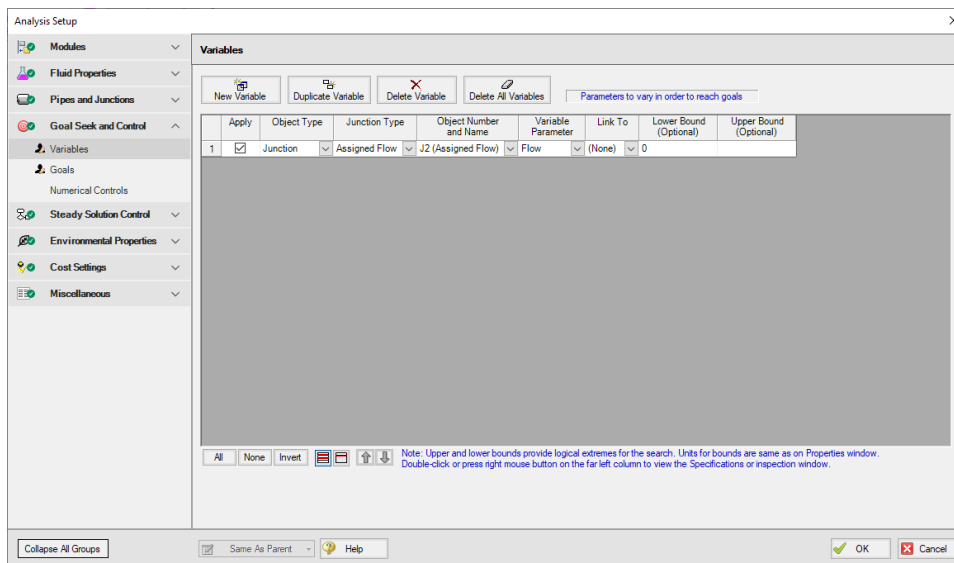
Figure 4: The Variables panel is used to specify the variables

Goals Panel

Goals are the parameter values set by the user that Fathom adjusts the variables to achieve. There should be one goal defined for each variable.

Open the Goals panel in the Goal Seek and Control group. The Goals panel allows you to create and modify the system goals. The goal type, object type, and the goal parameter are selected. A criterion for determining if the goal has been met is then specified, along with a value and units for the goal parameter. You can select the object to which the goal applies, and, if applicable, the location on the object at which the goal applies (i.e. the inlet or outlet of a pipe object).

Do the following:

1. Click New Goal
2. Apply = Checked
3. Goal Type = Point
4. Object Type = Pipe
5. Goal Parameter = Temperature
6. Criteria = =
7. Goal Value = 120
8. Goal Units = deg. F
9. Object ID = P1 (Pipe)
10. Object Location = Outlet

After entering the data, the Goals panel should appear as shown in Figure 5:
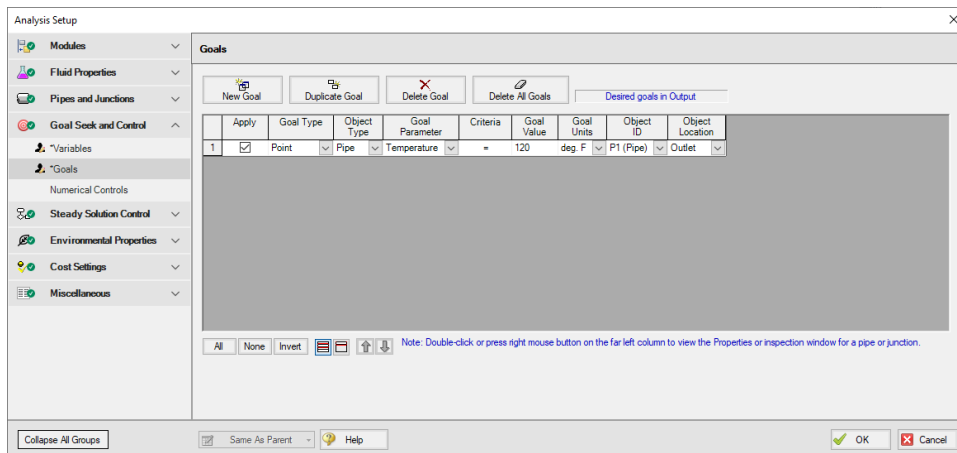
Figure 5: The Goals panel is used to specify the goals

As variables and goals are added to a model, Fathom will display symbols beside the pipes and junctions that have variables or goals applied to them. The default is a V for variables, and a G for goals. The goal symbol is not displayed next to objects that are part of a group goal. This is illustrated in Figure 6:
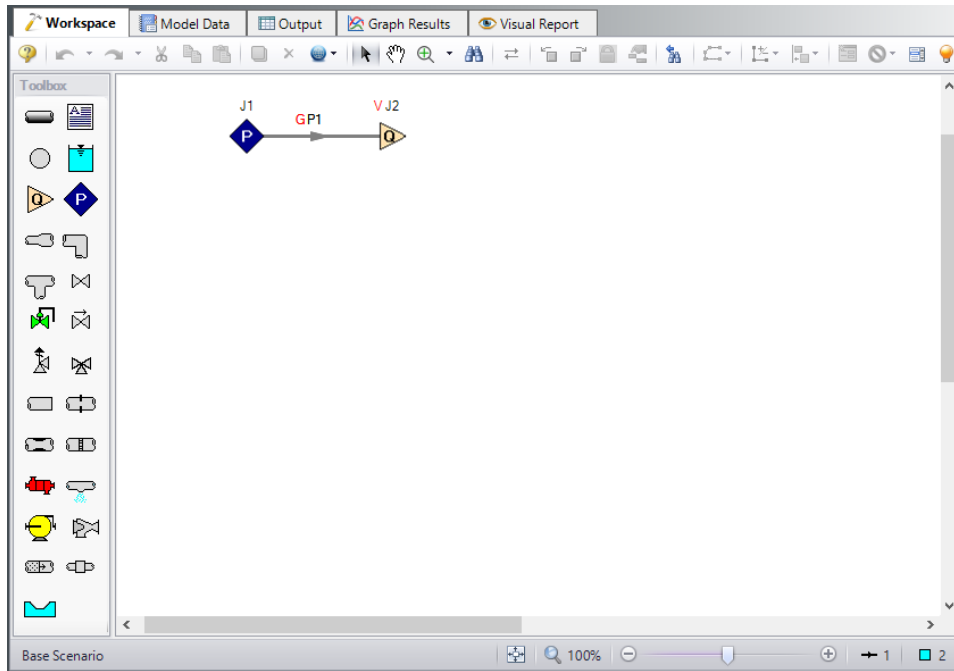
Figure 6: Workspace with Goal and Variable symbols displayed

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Figure 7 shows the Output window with the General, Pipes, and Junctions sections displayed.

Figure 7: The Output window contains all the data that was specified in the output control window

The results of the GSC analysis are shown in the General Output section in the GSC Variables and GSC Goals tabs. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 8. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 9.
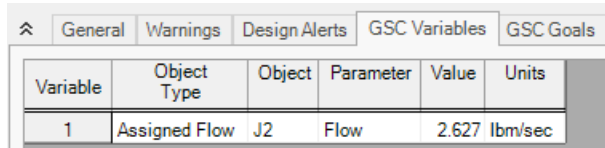
	Variable	Object Type	Object	Parameter	Value	Units
	1	Assigned Flow	J2	Flow	2.627	lbm/sec

Figure 8: GSC Variable Results are displayed in the General section of the Output Window

	Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
	1	Point	Pipe	P1 (Pipe)	Outlet	Temperature	120	120.0	deg. F

Figure 9: GSC Goal Results are displayed in the General section of the Output Window

The GSC module analysis determined that a mass flow rate of 2.627 lbm/sec (19.16 gal/min) results in a pipe outlet temperature of 120 deg. F. In order to achieve the desired temperature, the assigned flow rate was varied from the initial value of 10 lbm/sec, to the final value of 2.628 lbm/sec. In addition to the GSC module tabs in the General Output section, the resulting pipe flow and temperature results can be seen in the Pipes section of the Output Window.

Pump Sizing and Selection with Flow Control Valves - GSC

Summary

The piping for a heat exchanger system is being designed. The system will pump supply water from a tank pressurized to 10 psig with a liquid surface elevation of 5 feet to a receiving tank with a pressure of 30 psig and a liquid surface elevation of 10 feet. The heat exchangers operate in parallel. The flow through the heat exchangers is controlled to 100 gal/min by two flow control valves. There is a design requirement that the control valves have a minimum pressure drop of 5 psid.

Use the GSC module to size the pump.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Using Pump head rise as a GSC Variable
- Using Group Max/Min Goals

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner: Heat Transfer in a Pipe - GSC example and the Spray Discharge System - GSC example, which demonstrates how to use Group Max/Min goals.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Pump Sizing and Selection with Flow Control Valves.fth*
- *US - Pump Sizing and Selection with Flow Control Valves - GSC.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Pump Sizing and Selection with Flow Control Valves.fth** example file and save it as a new file. Right click on the Base Scenario and select Create Child. Name the child scenario Goal Seek Control.

Set the J2 Pump Sizing Parameter to Head Rise, with a Fixed Head Rise of 20 feet. In this example, the head rise will become the GSC variable.

The workspace should look like Figure 1 below:
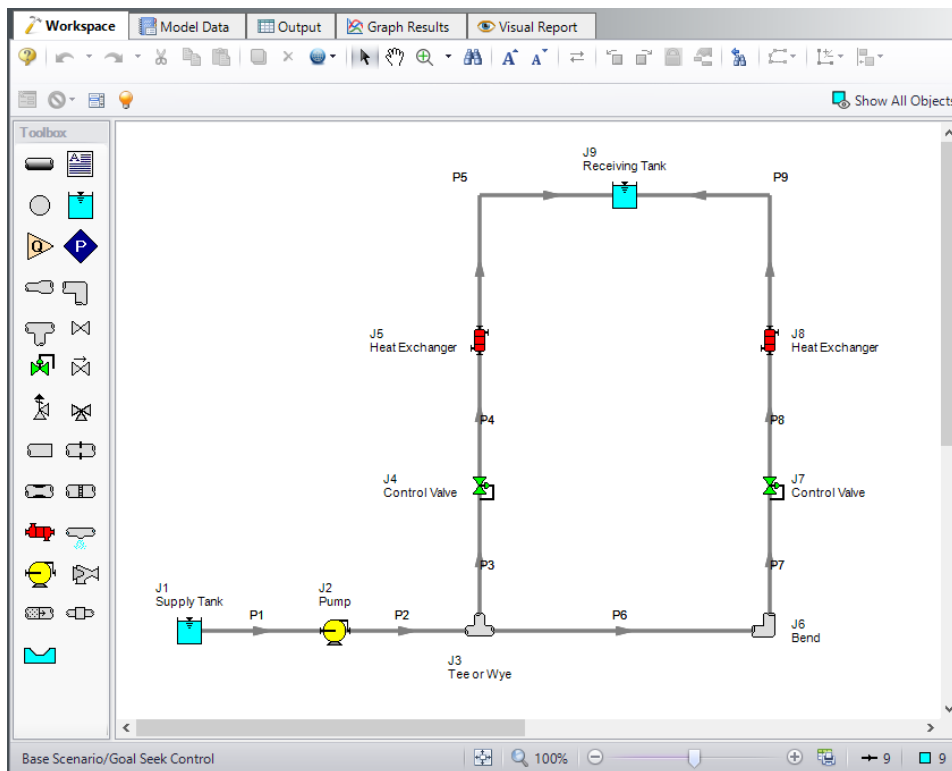
Figure 1: Workspace for GSC Pump Selection with Flow Control Valves example

Step 3. Create a Group

Create a group named *Flow Control Valves* by opening the Edit menu, hover over Groups, and select Create. Add both Control Valve junctions J4 and J7 to the group.

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Select the Variables panel in the Goal Seek and Control group. For this example, click New Variable and enter the following variable data:

1. Apply = Checked
2. Object Type = Junction
3. Junction Type = Pump
4. Junction Number and Name = J2 (Pump)
5. Variable Parameter = Head Rise
6. Link To = (None)

After entering the data, the Variables panel should appear as shown in Figure 2:
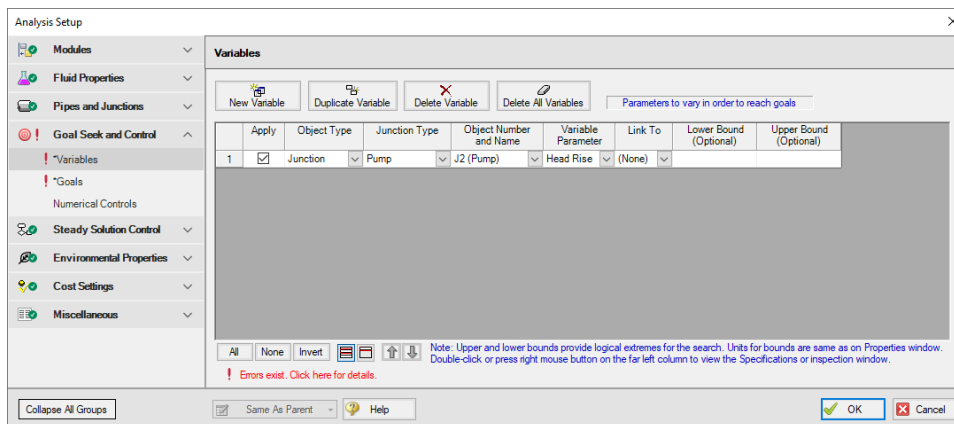
Figure 2: The Variables panel is used to specify the variables

Goals Panel

Select the Goals panel in the Goal Seek and Control group.

Create a new goal for the pressure drop across the flow control valves as defined below (if the option for a group goal is not visible, you may have forgotten to create a group as discussed above). Click New Goal and enter the following:

1. Apply = Selected
2. Goal Type = Group
3. Object Type = Group Max/Min
4. Goal Parameter = Pressure Loss
5. Criteria = >=
6. Goal Value = 5
7. Goal Units = psid
8. Object ID = Flow Control Valves
9. Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 3:
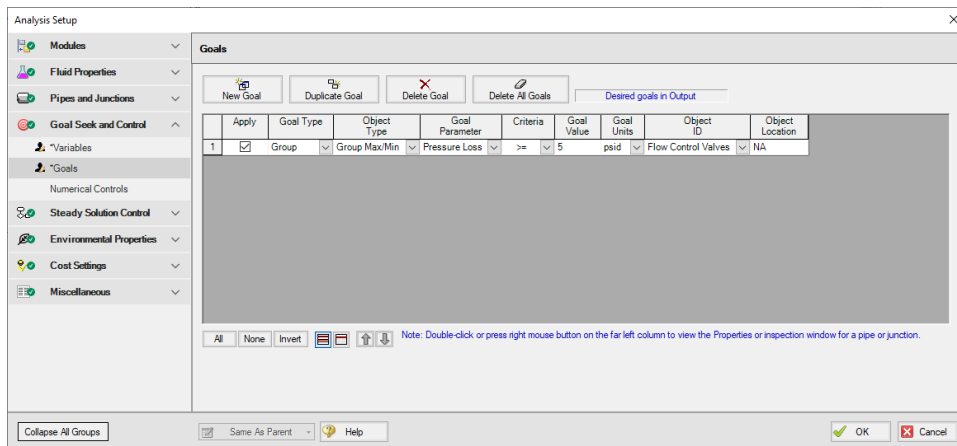
Figure 3: The Goals panel is used to specify the goals

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 4. The pump head rise required for this system to supply a minimum of 5 psid pressure drop at each of the Flow Control Valve junctions is 167.3 ft. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5. This shows the goal of a minimum 4.999 psid pressure drop was obtained for both valves.

Note that the results for this example are same as the results for the original Pump Selection with Flow Control Valves example.

Variable	Object Type	Object	Parameter	Value	Units
1	Pump	J2	Head Rise	167.3	feet

Figure 4: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Flow Control Valves	N/A	Pressure Loss	5	4.999	None	J7 (Control Valve)

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

Cooling System - GSC

Summary

The objective of this example is to use the GSC module to vary the required pump speed and control pressure such that a minimum flow of 150 gal/min is maintained through the Heat Exchanger, and the control valve open percentage remains above 30%.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Applying multiple variables and goals.
- Using linked variables

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner - Heat Transfer in a Pipe - GSC example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Cooling System.fth*
- *US - Cooling System - GSC.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - Cooling System.fth** example file and save it as a new file.

Load the Pump A scenario and turn off Pump J5 by opening the properties window, navigating to the Optional tab, and selecting the Pump Off No Flow special condition.

Open the properties window for Control Valves J11. On the Control Valve Model tab, under Loss When Fully Open, select Cv and then select From Open % Table (on Optional Tab) for the Loss Source.

Go to the Optional tab and add the following data to Open Percentage Data table:

Open Pct.	Cv
0	0
100	150

Click OK. Make these same changes to Control Valves J12 and J13. The workspace should look like Figure 1 below:
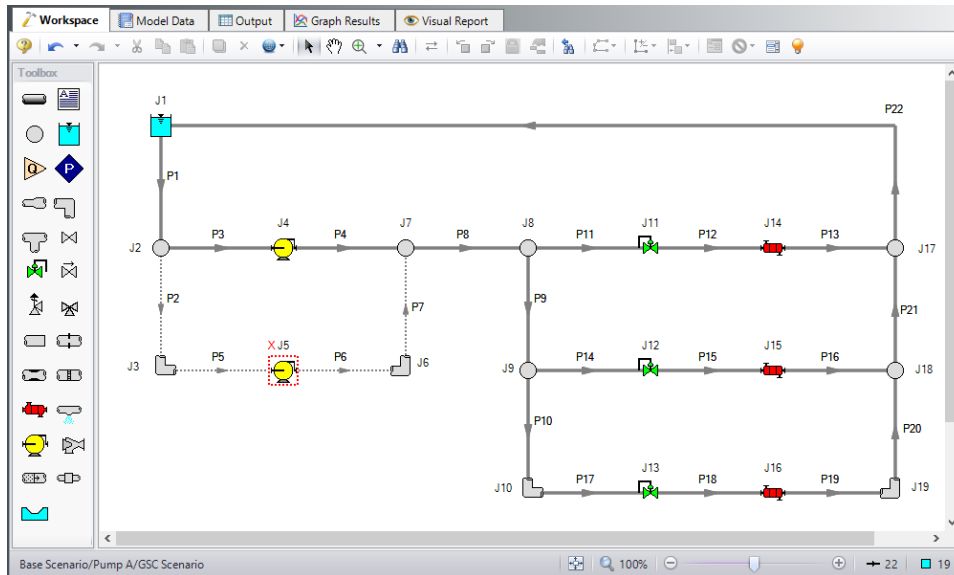
Figure 1: Workspace for GSC Cooling System example

Step 3. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 4. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. For this example, we will be adding four variables: one pump variable and three control valve variables which will be linked. Enter the variable inputs shown in Figure 2:
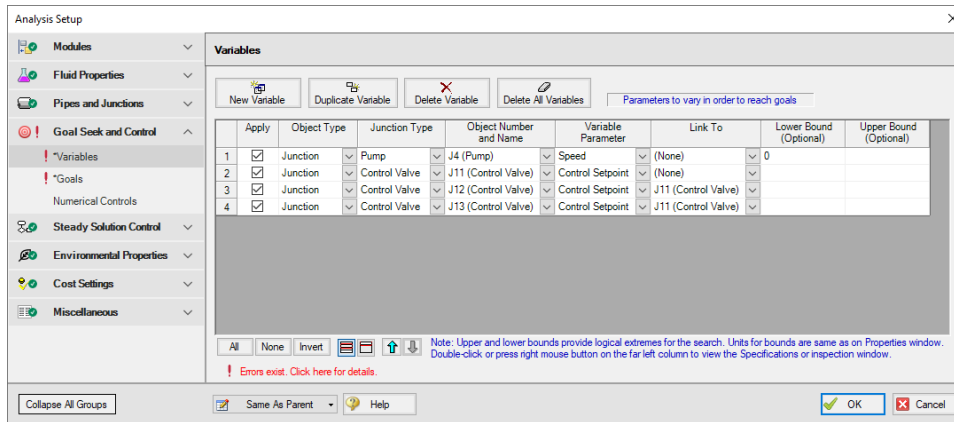
Figure 2: The Variables panel is used to specify the variables

Goals Panel

There should be one goal defined for each variable. In this case we essentially have two variables, since all of the control valve variables are linked, causing them to act as a single variable.

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Create two new goals as defined below in Figure 3:
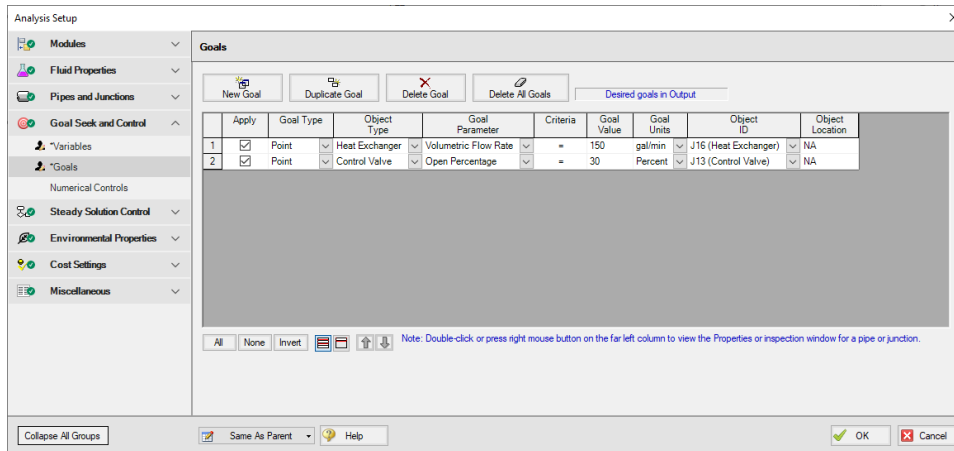
Figure 3: GSC Manager Goals tab is used to specify the goals

After entering the data, the Goals panel should appear as shown in Figure 3. Note that by setting the most hydraulically remote heat exchanger and control valve equal to the minimum requirements, we are ensuring that the other heat exchangers and control valves will exceed the minimum values.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final values for the variable parameters, as shown in Figure 4. The pump speed was determined to be 81.98%, and the control setpoint for the valves was determined to be 44.82 psig. The GSC Goals tab shows the final values achieved for the goals, as shown in Figure 5.

Viewing the Valve Summary tab will show that control valves J11 and J12 have open percents above the minimum of 30%. The Heat Exchanger Summary tab will similarly show that heat exchangers J14 and J15 have volumetric flow rates above the required minimum of 150 gal/min.

Variable	Object Type	Object	Parameter	Value	Units
1	Pump	J4	Speed	81.98	Percent
2	Control Valve	J11	Control Setpoint	44.82	psig
3	Control Valve	J12	Control Setpoint	44.82	psig
4	Control Valve	J13	Control Setpoint	44.82	psig

Figure 4: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Heat Exchanger	J16 (Heat Exchanger)	N/A	Volumetric Flow Rate	150	149.9	gal/min
2	Point	Control Valve	J13 (Control Valve)	N/A	Open Percentage	30	30.02	Percent

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

Model Calibration with Pipe Variables - GSC

Summary

A supply reservoir feeds a water treatment pond through gravity flow and the pipeline has been in existence for about 15 years. The original pipeline delivered about 2,000 gal/min and recently, the operators working at the treatment pond have reported that the flow has dropped by about 20%. Their lead engineer has determined that the cause of the flow reduction is most likely due to pipe corrosion.

The objective of this example is to demonstrate how to use the GSC module to determine the amount of additional frictional resistance in the pipeline due to years of corrosion. This could factor into the decision on whether or not to replace the pipeline and when.

For this example, the GSC module will be used to approximate the absolute roughness due to corrosion in the pipeline which has reduced the flow rate from 2,000 gal/min to 1,600 gal/min.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Setting Variables and Goals
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner - Heat Transfer in a Pipe - GSC example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Model Calibration with Pipe Variables - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for GSC Model Calibration Using Pipe Variables Example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
    a.   Pipe Material = Steel - ANSI
    b.   Pipe Geometry = Cylindrical Pipe
    c.   Size = 8 inch
    d.   Type = STD (schedule 40)
    e.   Friction Model Data Set = Standard
    f.   Length =

Pipe	Length (feet)
1 & 5	5280
2, 3, & 4	2640
6	2000

Junction Properties

1. J1 Reservoir
    a.   Name = Supply Reservoir
    b.   Liquid Surface Elevation = 1201 feet
    c.   Liquid Surface pressure = 0 psig
    d.   Pipe Elevation = 1000 feet
2. J2 Branch
    a.   Elevation = 500 feet
3. J3 Branch
    a.   Elevation = 600 feet
4. J4 Branch
    a.   Elevation = 100 feet
5. J5 Branch
    a.   Elevation = 300 feet
6. J6 Branch
    a.   Elevation = 50 feet
7. J7 Reservoir
    a.   Name = Treatment Pond
    b.   Liquid Surface Elevation = 10 feet
    c.   Liquid Surface Pressure = 0 psig
    d.   Pipe Depth = 0 feet

To show the names of the reservoirs a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.

2. Name the layer Reservoir Labels.

3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.

4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.

5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.

6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.

7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.

8. Close the Layer Settings window.

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

After running the model based upon the original pipeline operation with brand new clean steel pipe, we find that the flow rate is 2000 gal/min, as shown in Figure 2.

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	dH (feet)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)
1	Pipe	2,000	12.83	184.15	100.61	1,000.00	500.00	-83.54	-83.54	-216.48	307.1	100.61	184.15	101.72	185.26
2	Pipe	2,000	12.83	184.15	74.38	500.00	600.00	109.77	109.77	43.30	153.5	184.15	74.38	185.26	75.49
3	Pipe	2,000	12.83	224.39	74.38	600.00	100.00	-150.01	-150.01	-216.48	153.5	74.38	224.39	75.49	225.50
4	Pipe	2,000	12.83	224.39	71.33	100.00	300.00	153.06	153.06	86.59	153.5	224.39	71.33	225.50	72.44
5	Pipe	2,000	12.83	71.33	46.63	300.00	50.00	24.70	24.70	-108.24	307.1	71.33	46.63	72.44	47.73
6	Pipe	2,000	12.83	46.63	13.59	50.00	10.00	33.04	33.04	-17.32	116.3	46.63	13.59	47.73	14.70

Figure 2: Original pipeline results based upon clean pipe

Step 6. Create a child scenario named Flow Reduction from Corrosion

In the child scenario, we are going to use the GSC module to determine how much additional frictional resistance due to corrosion is in the pipeline. In this new scenario, we know that the flow rate has been reduced to 1600 gal/min in the physical system by corrosion and therefore the GSC goal in this case is going to be the volumetric flow rate.

The GSC module is a very useful tool that allows one to calibrate their models so that the predicted results can match measured data. There are four different GSC pipe variables that can be used to assist with this calibration including the specified pipe Friction (e.g. absolute roughness), pipe Design Factor, pipe ID (scaling), and Insulation Thickness.

In order to use GSC to determine the amount of additional frictional resistance due to corrosion, an input parameter must be varied. In this case, we are going to vary each pipe's absolute roughness in order to achieve the desired goal flow rate of 1600 gal/min.

Step 7. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 8. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Select the Variables panel within the Goal Seek and Control group. The Variables panel allows users to create and modify the system variables.

In this scenario, we are going to vary the current pipe friction model (i.e. absolute roughness) for each pipe in the system. Since there are six individual pipes, there will be six individual pipe variables. If all six variables were to be able to vary independently of each other and there is only one goal associated, then this could lead to a less unique solution (as there could be several different combinations of pipe roughness values that would lead to the desired result). Therefore, we are going to assume that the entire pipeline has the same level of additional frictional resistance due to corrosion. In order to accomplish this, all pipe variables can be linked together. Since only one goal is present, we will only need one effective variable and so all of the pipe variables will be linked to the first variable.

Since there are six pipes, we will set pipes P2 - P6 to be linked to pipe P1.

For pipe P1, set the following variable data:

1. Apply = Selected
2. Object Type = Pipe
3. Junction Type = NA
4. Object Number and Name = P1 (Pipe)
5. Variable Parameter = Friction
6. Link To = (None)

Now, add new variables for pipes P2 - P6. The only differences between the variables for these pipes and the variable for pipe P1 will be the Object Number and Name and instead of selecting (None) under the Link To column, select P1 (Pipe). This will link pipes P2 - P6 to pipe P1 which will force all six pipe

variables to be varied identically, requiring only one goal to be specified. Figure 3 below shows all the linked pipe variables.
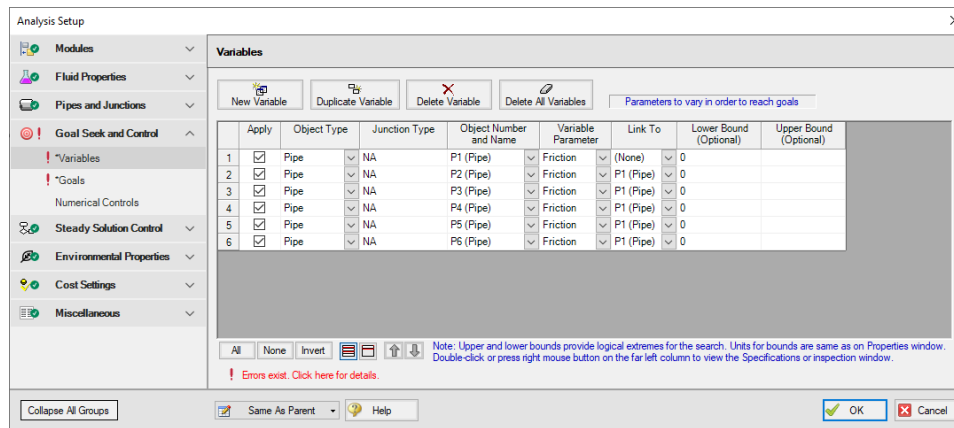
Figure 3: Six linked pipe variables (working effectively as a single variable) to vary the pipe friction model (i.e. absolute roughness)

Goals Panel

Goals are the parameter values set by the user that the solver tries to achieve by adjusting the variables. There should be one goal defined for each unlinked (and thus independent) variable. Since variables 2 - 6 are all linked to variable 1, only one goal will be required.

Select the Goals panel in the Goal Seek and Control group.

Create a new goal and set up the following goal information:

1.   Apply = Selected
2.   Goal Type = Point
3.   Object Type = Pipe
4.   Goal Parameter = Volumetric Flow Rate
5.   Criteria = =
6.   Goal Value = 1600
7.   Goal Units = gal/min
8.   Object ID = P6 (Pipe)
9.   Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 4.

On the Numerical Control panel, the Search Method can be modified for Goals between two options, Beta Method and Least Squares. The Beta Method is useful in situations where there are multiple variables and multiple goals. For this model, make sure that the Least Squares method is selected.
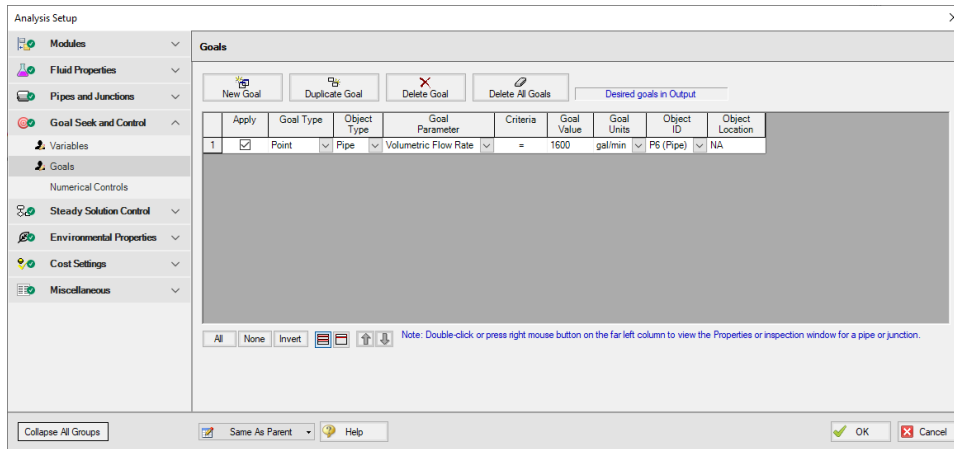
Figure 4: GSC Manager Goals tab is used to specify the goals

Step 9. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 10. Examine the Output

The results of the GSC module analysis are shown in the General section of the Output window. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 5. The resulting absolute roughness value that would decrease the flow rate to 1600 gal/min would be about 0.01584 inches. The additional frictional resistance in the pipeline due to corrosion is about nine times larger than the absolute roughness for clean steel pipe. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 6.

Figure 5: GSC Variable Results are displayed in the General section of the Output Window

| General | Warnings | Design Alerts | Reservoir Summary | GSC Variables | GSC Goals |

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Pipe	P6 (Pipe)	N/A	Volumetric Flow Rate	1600	1,600	gal/min

Figure 6: GSC Goal Results are displayed in the General section of the Output Window

Spray Discharge System - GSC

Summary

The objective of this example is to demonstrate how the Group Max/Min and Group Sum goal types can be used in the GSC module to achieve a goal for a group of objects.

For this example, we will use the GSC module to solve for the supply pressure required to:

Supply a minimum of 125 gal/min to each spray discharge junction using a Group Max/Min goal.

Supply a total flow rate of 1,000 gal/min through the system using a Group Sum goal.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Creating Groups
- Using Group Max/Min and Group Sum Goals

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner - Heat Transfer in a Pipe - GSC example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Spray Discharge System.fth*
- *US - Spray Discharge System - GSC.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - Spray Discharge System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select Delete All Children. The workspace should look like Figure 1 below:
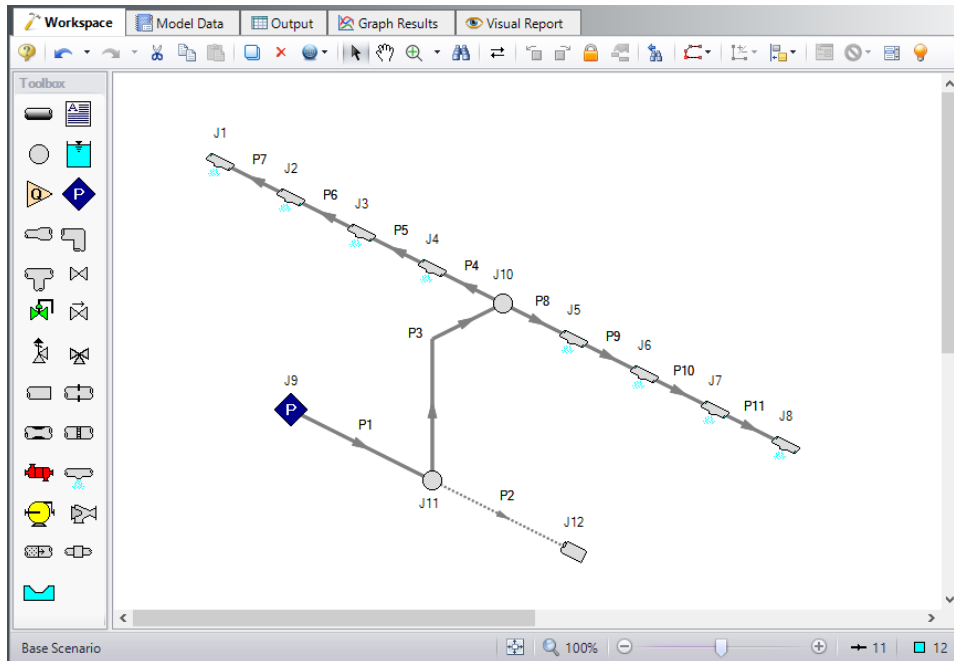
Figure 1: Workspace for GSC Spray Discharge system example

Step 3. Create a Group

In order to use the Group Max/Min and Group Sum goal types, the junctions included in the goal must be added to a group. To create a group, go to the Edit menu and select the Groups -> Create option. Give the new group the name Spray Discharges, and click OK. The Group Manager will be displayed. Select all of the Spray Discharge junctions from the junction list, as shown in Figure 2. After choosing the Spray Discharge junctions, click OK on the Group Manager form.

Note: If the Spray Discharge junctions were selected in the Workspace before Create Group was selected from the Edit menu, then they would be automatically selected for us in the Group Manager window.
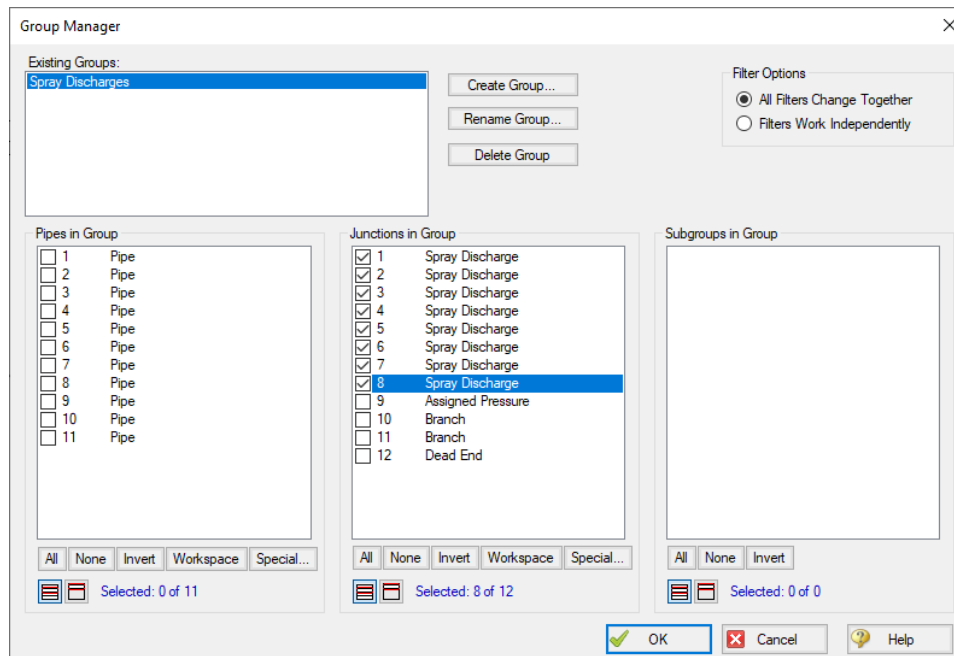
Figure 2: Groups are Created Using the Group Manager

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group. For this example select New Variable, then choose following variable data:

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Assigned Pressure
4. Junction Number and Name = J9 (Assigned Pressure)
5. Variable Parameter = Pressure
6. Link To = (None)

After entering the data, the Variable panel should appear as shown in Figure 3:
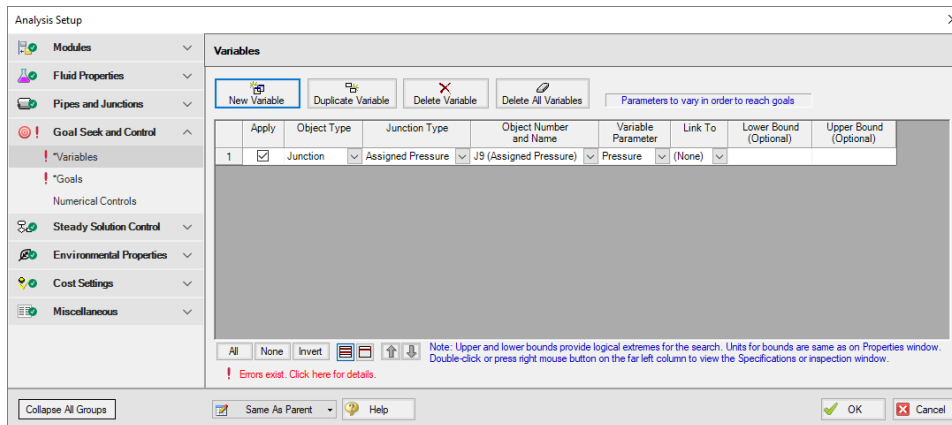
Figure 3: The Variables panel is used to specify the variables

Goals Panel

Open the Goals panel next. For this example, we are going to define two goals. We will apply one goal at a time for the sake of comparing the impact that each goal has on the results.

The first goal type is a Group Max/Min goal. This type of goal allows a single goal to be applied to a group of objects. Fathom applies a Group Max/Min goal by ensuring the final goal value is either greater than or equal to (Min) or less then or equal to (Max) the specified value. For this example, a Group Max/Min goal will be applied to ensure the minimum flow rate out of all of the spray discharge junctions is at least 125 gal/min.

The second goal type is a Group Sum goal. This type of goal is used to set the sum of the goal parameter for all the members in the group to a specified value. For this example, the Group Sum goal will be applied to ensure the sum of the flow rates through the spray discharges equals 1000 gal/min.

Create two new goals defined as follows:

Group Max/Min:

1.   Apply = Checked
2.   Goal Type = Group
3.   Object Type = Group Max/Min
4.   Goal Parameter = Volumetric Flow Rate Out
5.   Criteria = >=
6.   Goal Value = 125
7.   Goal Units = gal/min
8.   Object ID = Spray Discharges
9.   Object Location = NA

Group Sum:

1. Apply = Not Checked
2. Goal Type = Group
3. Object Type = Group Sum
4. Goal Parameter = Volumetric Flow Rate
5. Criteria = =
6. Goal Value = 1000
7. Goal Units = gal/min
8. Object ID = Spray Discharges
9. Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 4 in Step 7.

Step 6. Create Scenarios

Using the Scenario Manager on the Quick Access Panel, create two child scenarios, one named *Group Max Min* and another named *Group Sum*.

Step 7. Select Group Max/Min Goal

Load the Group Max Min scenario.

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Make sure the Group Max/Min goal is applied, indicated by the check mark in the Apply column for that goal. Make certain Group Sum goal is NOT applied by unchecking the check box in the Apply column. The GSC goals should appear as shown in Figure 4:

Figure 4: Group Max Min Goal Applied in the Goals panel

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 5. The supply pressure required to supply a minimum of 125 gal/min at each of the Spray Discharge junctions is 263.3 psig. The GSC Goals tab shows the value achieved for the goal, as shown in Figure 6. The flow rates determined for the Spray discharge junctions are shown in Figure 7. The spray discharge flow rates shown in Figure 7 illustrate how the Group Max/Min resulted in a the flow rate through each of the spray discharges meeting or exceeding the specified minimum flow rate of 125 gal/min. The total system flow is 1079.9 gal/min.

Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Pressure	J9	Pressure	263.3	psig

Figure 5: GSC Variable Results are displayed in the General section of the Output Window

General | Warnings | Design Alerts | GSC Variables | **GSC Goals**

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Spray Discharges	N/A	Volumetric Flow Rate Out	125	125.0	gal/min	J1 (Spray Discharge)

Figure 6: GSC Goal Results are displayed in the General section of the Output Window

All Junctions | Assigned Pressure | Branch | Dead End | **Spray Discharge**

Spray Discharge	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
1	Spray Discharge	132.3	14.70	134.9	14.70	125.0	17.36	46.05
2	Spray Discharge	N/A	14.70	139.1	14.70	N/A	N/A	46.05
3	Spray Discharge	N/A	14.70	155.5	14.70	N/A	N/A	46.05
4	Spray Discharge	N/A	14.70	193.8	14.70	N/A	N/A	46.05
5	Spray Discharge	N/A	14.70	193.8	14.70	N/A	N/A	46.05
6	Spray Discharge	N/A	14.70	155.5	14.70	N/A	N/A	46.05
7	Spray Discharge	N/A	14.70	139.1	14.70	N/A	N/A	46.05
8	Spray Discharge	132.3	14.70	134.9	14.70	125.0	17.36	46.05

Figure 7: Flow Rates Calculated for the Spray Discharge junctions for Group Max/Min goal

Step 10. Select Group Sum Goal

Load the Group Sum scenario.

Open the Goals panel in the Goal Seek and Control group from Analysis Setup. Apply the Group Sum goal by clicking the check box next to the Group Sum goal in the Apply column. Make certain Group Max/Min goal is NOT applied by unchecking the check box in the Apply column. The GSC goals should appear as shown in Figure 8:
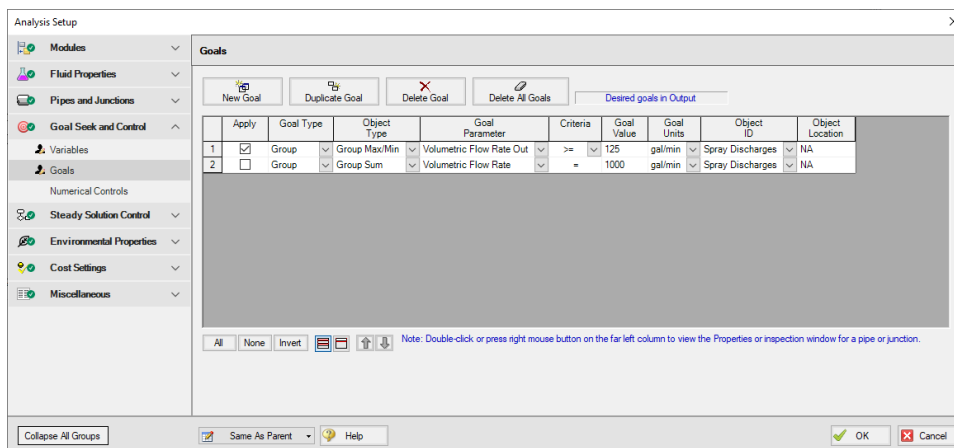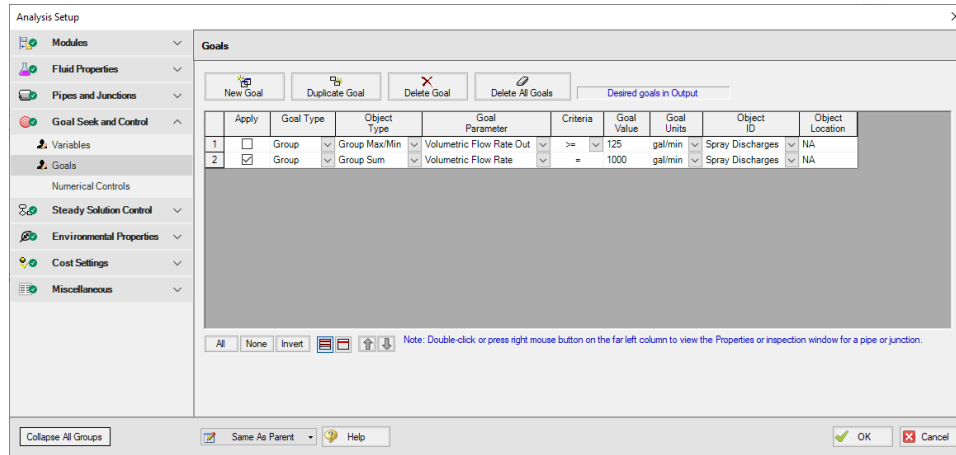

Figure 8: Group Sum Goals are applied in the GSC Manager

Step 11. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 12. Examine Output

The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 9. The supply pressure required to supply a total system flow of 1000 gal/min is 226.5 psig. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 10. The goal of 1000 gal/min for the system flow in this example was achieved. The flow rates determined for the Spray Discharge junctions are shown in Figure 11. Note that overall, the flows through the spray discharges in the Group Sum scenario are less than the flows determined for the Group Max/Min scenario. This is because in the Group Max/Min scenario, the restriction was placed on the flow rate at each of the spray discharges, resulting in a system flow of 1079.9 gal/min, while the restriction in the Group Sum scenario was on the total system flow equaling 1000 gal/min.

Figure 9: GSC Variable Results are displayed in the General section of the Output Window

Figure 10: GSC Goal Results are displayed in the General section of the Output Window

Figure 11: Flow Rates Calculated for the Spray Discharge junctions for Group Sum Goal

Controlled Heat Exchanger Temperature - GSC

Summary

The objective of this example is to use the GSC module to vary a Three Way Valve percent open to divert heat exchanger flow to maintain a downstream temperature of 120 deg. F for a specified upstream supply temperature.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Using GSC to control flow splits

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner - Heat Transfer in a Pipe - GSC example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Controlled Heat Exchanger Temperature.fth*
- *US - Controlled Heat Exchanger Temperature - GSC.fth*
- *Controlled Heat Exchanger Temperature.dat*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - Controlled Heat Exchanger Temperature.fth** example file and save it as a new file. The workspace should look like Figure 1 below:
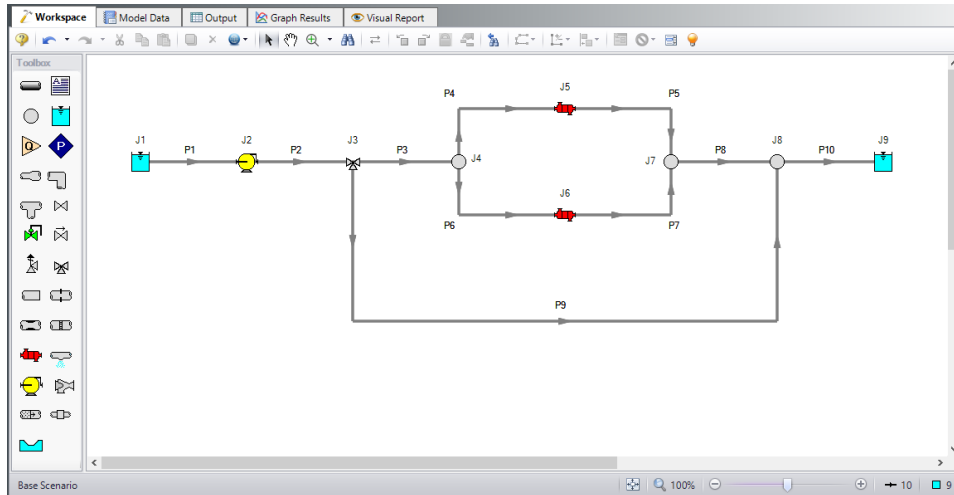
Figure 1: Workspace for GSC Controlled Heat Exchanger Temperature example

Step 3. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 4. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group. For this example, we will be adding a variable for the Three Way Valve % Open. Click New Variable and enter the following data:

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Three Way Valve
4. Junction Number and Name = J3 (Three Way Valve)
5. Variable Parameter = Open Percentage
6. Link To = (None)

7. Lower Bound = 0
8. Upper Bound = 100
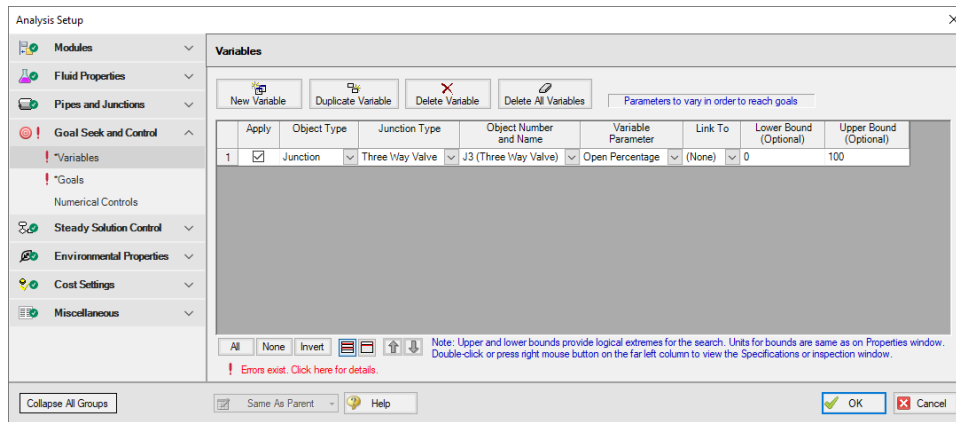
Figure 2: GSC Manager Variables tab is used to specify the variables

Goals Panel

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Click New Goal for the downstream temperature and define it with the data below:

1. Apply = Selected
2. Goal Type = Point
3. Object Type = Branch
4. Goal Parameter = Temperature
5. Criteria = =
6. Goal Value = 120
7. Goal Units = deg. F
8. Object ID = J8 (Branch)
9. Object Location = Outlet

After entering the data, the Goals panel should appear as shown in Figure 3:
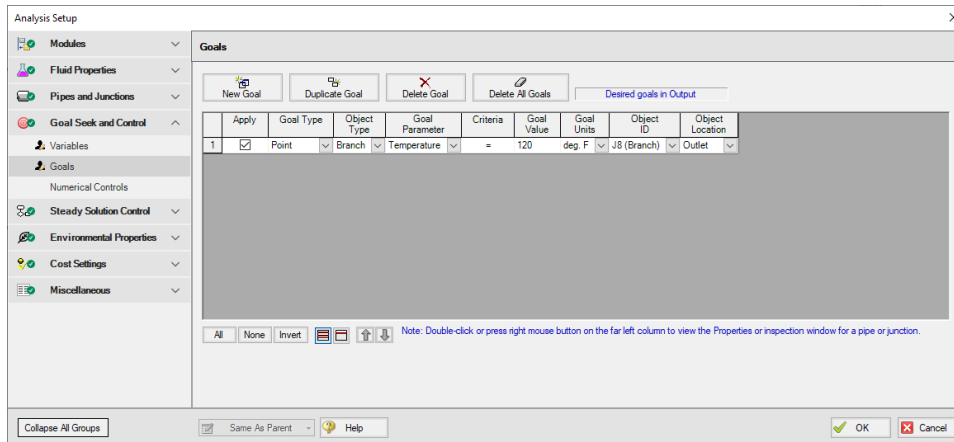
Figure 3: The Goals panel is used to specify the goals

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 4. The open percentage required for the Three Way Valve to achieve a temperature of 120 deg. F at Branch J8 is 85.94%. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5. This shows the goal of 120 deg. F was achieved for the downstream temperature.

Figure 4: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Branch	J8 (Branch)	Outlet	Temperature	120	120.0	deg. F

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

NFPA Firewater System - GSC

Summary

The goal of this example is to determine how to use the GSC module to determine the hydraulically most remote path to generate an NFPA report.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Setting Variables and Goals
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the Beginner - Heat Transfer in a Pipe - GSC example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - NFPA Firewater System.fth***
- ***US - NFPA Firewater System - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - NFPA Firewater System - Final.fth** file and save under a different name.

The workspace should look like Figure 1 below:
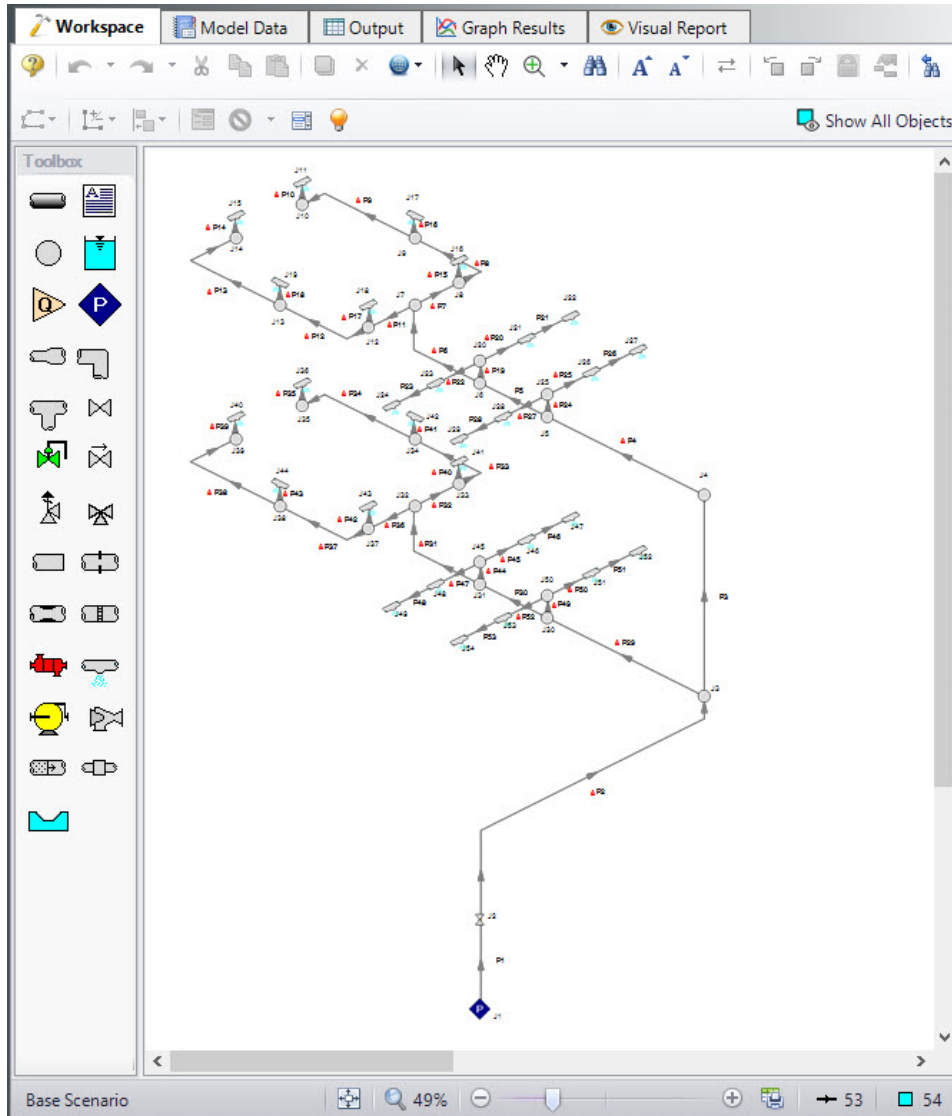
Figure 1: Workspace window for NFPA Firewater System - GSC Example

Step 3. Create the Spray Nozzles group

In this example, we will use the GSC module to determine the hydraulically most remote path.

To do this, you first need to create a group of all spray nozzles. Make this group by going to the Edit menu and selecting groups, and then clicking on Create. Name the group *Spray Nozzles* and under the

Junctions in Group section, click on Special, specify Selection Type as Junction Type, and choose Spray Discharge as the Junction Type. Click on Select Junctions and then press OK to confirm the selected junctions, then press OK to exit the Group Manager. 28 junctions should be selected.

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Next, ensure that GSC is both activated and enabled in the Modules panel of Analysis Setup. Then open the Variables panel. Click on New Variable and specify the Assigned Pressure J1 as the variable. This pressure boundary will be varied in order to achieve the goal, which in this case is a goal of at least 40 gal/min through each sprinkler. See Figure 2 to see this variable specified.
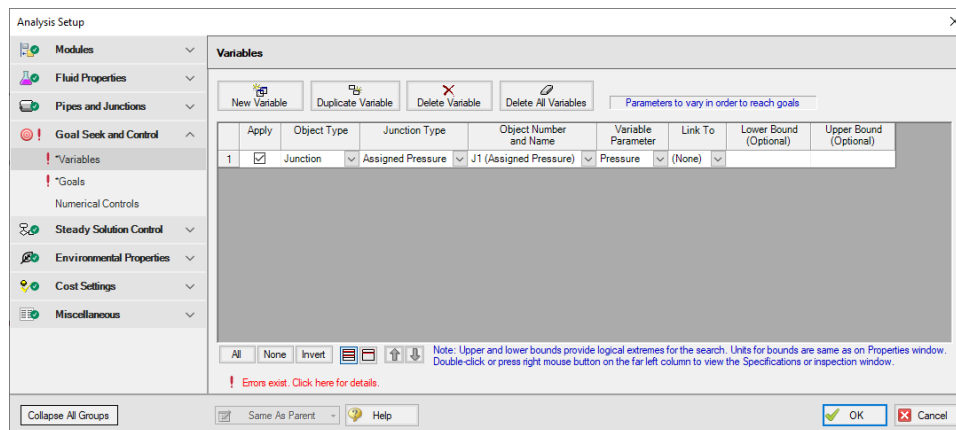
Figure 2: Assigned Pressure Variable on Goal Seek and Control Manager window

Goals Panel

Open the Goals panel and select New Goal. Specify the goal as shown in Figure 3.
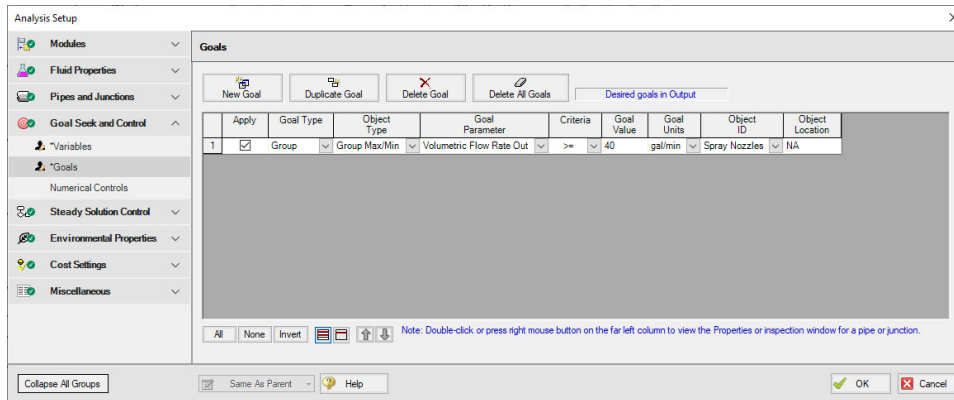
Figure 3: Minimum Flow Rate Group Goal on Goal Seek and Control Manager window
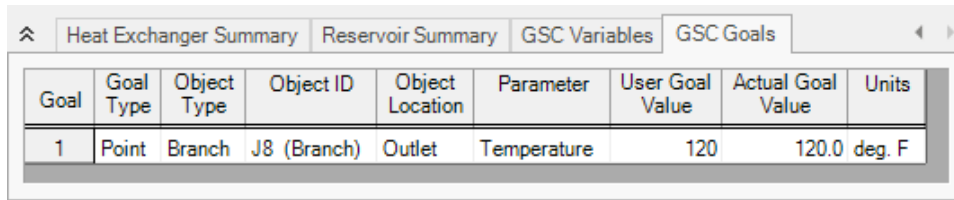
Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

You will notice there are numerous cautions about the pipe lengths being shorter than the elevation change. This model used pipe lengths and elevations based off a NFPA example. In the NFPA example, the length of the spray discharges raised the outlet height of the junction. Fathom treats junctions as single points in space, so to keep the inputs identical the pipe length had to be a little bit short.

Step 7. Examine the Output

The GSC module displays the value of the variable required in order to achieve the specified goal, as well as the object that was used for the group goal (in this case, this object is the spray discharge that receives the minimum amount of flow and, therefore, is the spray nozzle that contains the hydraulically most remote path). You can see that the pressure needed at J1 in order to ensure that all spray nozzles receive 40 gal/min of flow is 63.77 psig, and that the hydraulically most remote spray nozzle is J15 (see Figure 4 and Figure 5).

Figure 4: Calculated value for the GSC Variable in the Output

Figure 5: Calculated value for the GSC Goal when the GSC Variable is applied in the Output

Extended Time Simulation (XTS) Examples

The examples below utilize the Extended Time Simulation module.

Fathom XTS Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Filling a Tank	Beginner	Water	28	1	Learn the basics of the XTS module such as defining a transient junction and specifying the Transient Control group in Analysis Setup. See how to generate a Quick Graph from the Output window to see how a tank fills up over time.
Reservoir Balance	Beginner	Water	2	0	Learn how to use the XTS module to see the effects of a system with multiple finite open tanks.
Valve Closing	Beginner	Water	3	1	Learn how to use the XTS module to define an initiating transient event, such as a valve closing when a reservoir liquid height reaches a specified value.
Controlled Heat Exchanger Temperature	Beginner	Water	10	1	Learn how to use the XTS module to model a system with a pump that shuts off once the liquid level of a tank reaches a specified value.
Housing Project	Intermediate	Water	28	1	Learn how to use the XTS module to specify a Dual Event Cyclic transient initiation, such as a pump turning on or off depending on a reservoir liquid height.
Variable Demand	Intermediate	Water	19	2	Learn how to use the XTS module to define a Single Event Initiation of Transient for multiple junctions within a system.

Beginner - Filling a Tank - XTS

Summary

The objective of this example is to introduce the user to the Extended Time Simulation (XTS) module by modeling the filling of a tank. A liquid transfer system will be modeled. The pump will run for a specified time, and the change in fluid height in the delivery tank will be examined using the transient output data.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Specifying Transient Control
- Understanding Transient Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Beginner - Filling a Tank - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A

new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for XTS Filling a Tank example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
   a. Pipe Material = Steel - ANSI
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = 4 inch
   d. Type = STD (schedule 40)
   e. Friction Model Data Set = Standard
   f. Lengths =

Pipe	Length (feet)
1	10
2	990

Junction Properties

1. J1 Reservoir
   a. Name = Lower Reservoir
   b. Tank Model = Infinite Reservoir
   c. Liquid Surface Elevation = 10 feet
   d. Liquid Surface Pressure = 0 psig
   e. Pipe Elevation = 0 feet
2. J2 Pump
   a. Inlet Elevation = 0 feet
   b. Pump Model = Centrifugal (Rotodynamic)
   c. Analysis Type = Pump Curve
   d. Enter Curve Data =

Volumetric	Head
gal/min	feet
0	300
500	380
1000	250

   e. Curve Fit Order = 2
3. J3 Reservoir
   a. Name = Upper Reservoir
   b. Tank Model = Finite Open Tank
   c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)

    d.   Cross-Sectional Area = Constant, 150 feet2

    e.   Liquid Level Specification = Height from Bottom

    f.   Liquid Height = 0 feet

    g.   Liquid Surface Pressure = 0 psig

    h.   Tank Height = 30 feet

    i.   Pipe Elevation = 200 feet

    j.   Tank Bottom Elevation = 200 feet

To show the names of the reservoirs a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2. Name the layer Reservoir Labels.
3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8. Close the Layer Settings window.

Step 5. Define the Transient Control Group

XTS module transient simulations are controlled through the Transient Control group in Analysis Setup. Access the Transient Control group by opening the Simulation Duration panel. When Fathom is set to perform a transient analysis, Transient Control becomes one of the items required to run the model.

The Simulation Duration panel in the Transient Control group is used to specify transient start and stop times, as well as the time step controls and solution parameters for finite tank liquid level calculations. Start times, stop times, and time steps can be specified in any units from seconds to years. From the Simulation Duration panel, users can specify the number of output points that are saved to the transient output file. This allows XTS to do transient calculations with a small time step while limiting the size of the transient data output file. Enter the following inputs in the Simulation Duration panel, as shown in Figure 2.

1. Start Time = 0 minutes
2. Stop Time = 30
3. Time Step = 1 minutes
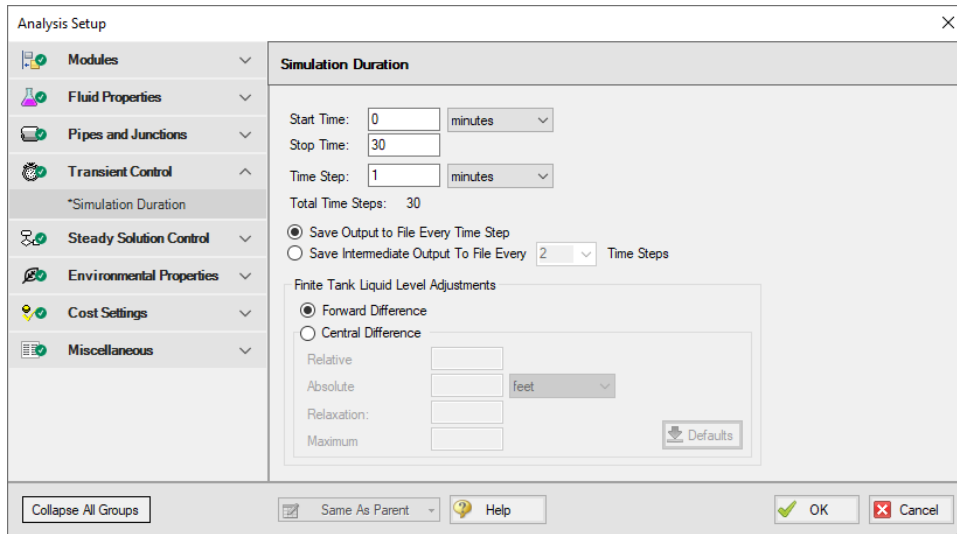
Figure 2: The Transient Control panel is used to control transient simulations

In addition to Transient Control, it is possible in define transient events for individual junctions. This allows the user to define events such as pump trips or start-ups, or valve closures. For this example, we will not specify any junction transients. The transient data for this example will be related to the fluid being pumped into the tank over the transient time period we have defined in Transient Control.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

After all of the pipes and junctions have been defined, and all of the transient data and transient control items have been specified, the transient simulation may be executed.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

When using the XTS module, the Solution Progress window displays the progress through the transient analysis. This information is displayed below the solution tolerance data, as shown in Figure 3:
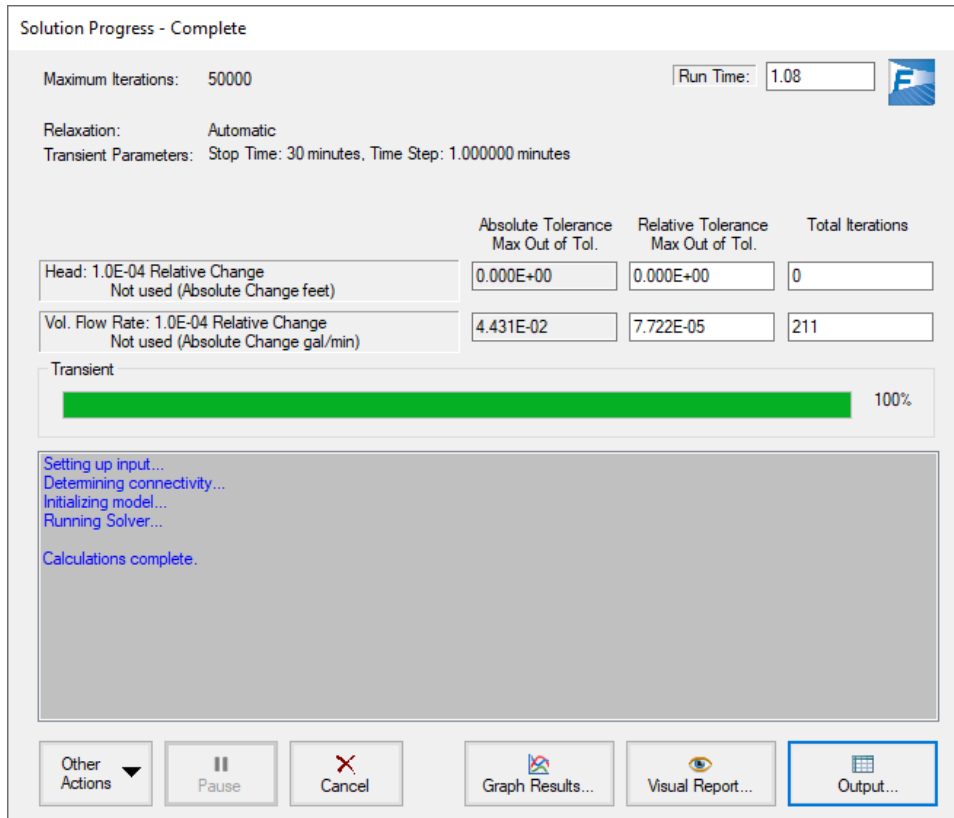
Figure 3: Solution Progress Window shows the convergence progress, as well as the transient solution progress when running transient cases

Step 8. Examine the Output

AFT Fathom displays transient output for all Pipes and Junctions in the Output Window, as shown in Figure 4.
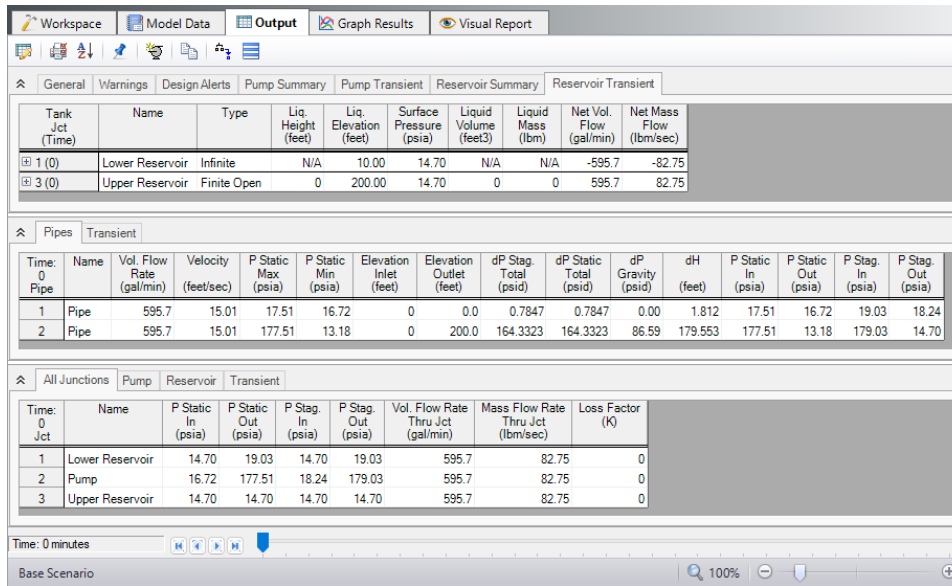
Figure 4: Transient output is displayed in the Output Window

Each of the summary tables in the General Output section has a companion transient summary tab which displays the summary data at each time step in the transient run. Each junction included in the summary is included in the transient summary. The transient summary data for each junction may be expanded or collapsed by clicking the + or – sign in beside the junction data list. The entire list may be expanded or collapsed by clicking the button in the top left-hand corner of the transient summary window. Figure 5 shows the Reservoir Transient summary tab, with all of the data sets collapsed. Figure 6 shows the Reservoir Transient summary tab, with the data for the J3 Reservoir expanded.

Figure 5: Transient Summaries are displayed in the General Output section

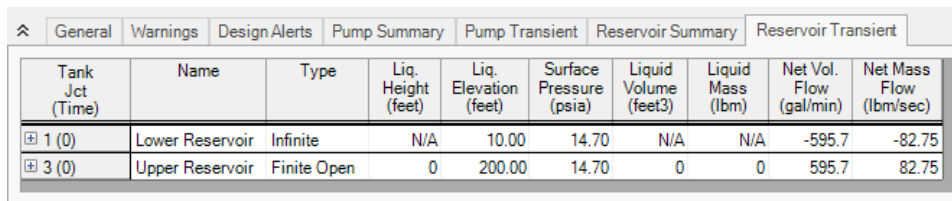| Tank Jct (Time)	Name	Type	Liq. Height (feet)	Liq. Elevation (feet)	Surface Pressure (psia)	Liquid Volume (feet3)	Liquid Mass (lbm)	Net Vol. Flow (gal/min)	Net Mass Flow (lbm/sec)
⊞ 1 (0)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-595.7	-82.75
⊟ 3 (0)	Upper Reservoir	Finite Open	0	200.00	14.70	0	0	595.7	82.75
3 (1)	Upper Reservoir	Finite Open	0.5309	200.53	14.70	79.64	4,965	595.3	82.70
3 (2)	Upper Reservoir	Finite Open	1.061	201.06	14.70	159.2	9,927	594.6	82.60
3 (3)	Upper Reservoir	Finite Open	1.591	201.59	14.70	238.7	14,883	593.9	82.49
3 (4)	Upper Reservoir	Finite Open	2.121	202.12	14.70	318.1	19,832	593.1	82.39
3 (5)	Upper Reservoir	Finite Open	2.649	202.65	14.70	397.4	24,776	592.4	82.29
3 (6)	Upper Reservoir	Finite Open	3.177	203.18	14.70	476.6	29,713	591.7	82.19
3 (7)	Upper Reservoir	Finite Open	3.704	203.70	14.70	555.7	34,644	590.9	82.08
3 (8)	Upper Reservoir	Finite Open	4.231	204.23	14.70	634.7	39,569	590.2	81.98
3 (9)	Upper Reservoir	Finite Open	4.757	204.76	14.70	713.6	44,488	589.4	81.88
3 (10)	Upper Reservoir	Finite Open	5.282	205.28	14.70	792.4	49,401	588.7	81.78
3 (11)	Upper Reservoir	Finite Open	5.807	205.81	14.70	871.1	54,307	588.0	81.67
3 (12)	Upper Reservoir	Finite Open	6.331	206.33	14.70	949.7	59,208	587.2	81.57
3 (13)	Upper Reservoir	Finite Open	6.854	206.85	14.70	1,028	64,102	586.5	81.47
3 (14)	Upper Reservoir	Finite Open	7.377	207.38	14.70	1,107	68,990	585.7	81.36
3 (15)	Upper Reservoir	Finite Open	7.899	207.90	14.70	1,185	73,872	585.0	81.26
3 (16)	Upper Reservoir	Finite Open	8.420	208.42	14.70	1,263	78,747	584.2	81.16
3 (17)	Upper Reservoir	Finite Open	8.941	208.94	14.70	1,341	83,617	583.5	81.05
3 (18)	Upper Reservoir	Finite Open	9.461	209.46	14.70	1,419	88,480	582.8	80.95
3 (19)	Upper Reservoir	Finite Open	9.980	209.98	14.70	1,497	93,337	582.0	80.85
3 (20)	Upper Reservoir	Finite Open	10.50	210.50	14.70	1,575	98,188	581.3	80.74
3 (21)	Upper Reservoir	Finite Open	11.02	211.02	14.70	1,653	103,033	580.5	80.64
3 (22)	Upper Reservoir	Finite Open	11.53	211.53	14.70	1,730	107,871	579.8	80.54
3 (23)	Upper Reservoir	Finite Open	12.05	212.05	14.70	1,808	112,703	579.0	80.43
3 (24)	Upper Reservoir	Finite Open	12.57	212.57	14.70	1,885	117,529	578.3	80.33
3 (25)	Upper Reservoir	Finite Open	13.08	213.08	14.70	1,962	122,349	577.5	80.22
3 (26)	Upper Reservoir	Finite Open	13.60	213.60	14.70	2,040	127,162	576.8	80.12
3 (27)	Upper Reservoir	Finite Open	14.11	214.11	14.70	2,117	131,970	576.0	80.02
3 (28)	Upper Reservoir	Finite Open	14.62	214.62	14.70	2,194	136,771	575.3	79.91
3 (29)	Upper Reservoir	Finite Open	15.14	215.14	14.70	2,271	141,565	574.5	79.81
3 (30)	Upper Reservoir	Finite Open	15.65	215.65	14.70	2,347	146,354	573.8	79.70

Figure 6: Transient Summary data can be expanded to show the junction data for every Time Step

The Quick Graph feature can be used to quickly plot transient data for quick examination. To use the Quick Graph feature, place the mouse cursor over the column of data you wish to examine. Then, right click with the mouse, and select Quick Graph from the list. Figure 7 illustrates how to do this for J3 Reservoir Liquid Height vs. Time. Figure 8 shows the resulting graph. The graph illustrates how the liquid level in the Reservoir rises with time, as would be expected. The Graph Results window may also be used to create a full range of graphs related to the XTS module transient output.

Figure 7: Use Quick Graph to easily plot J3 Reservoir liquid height

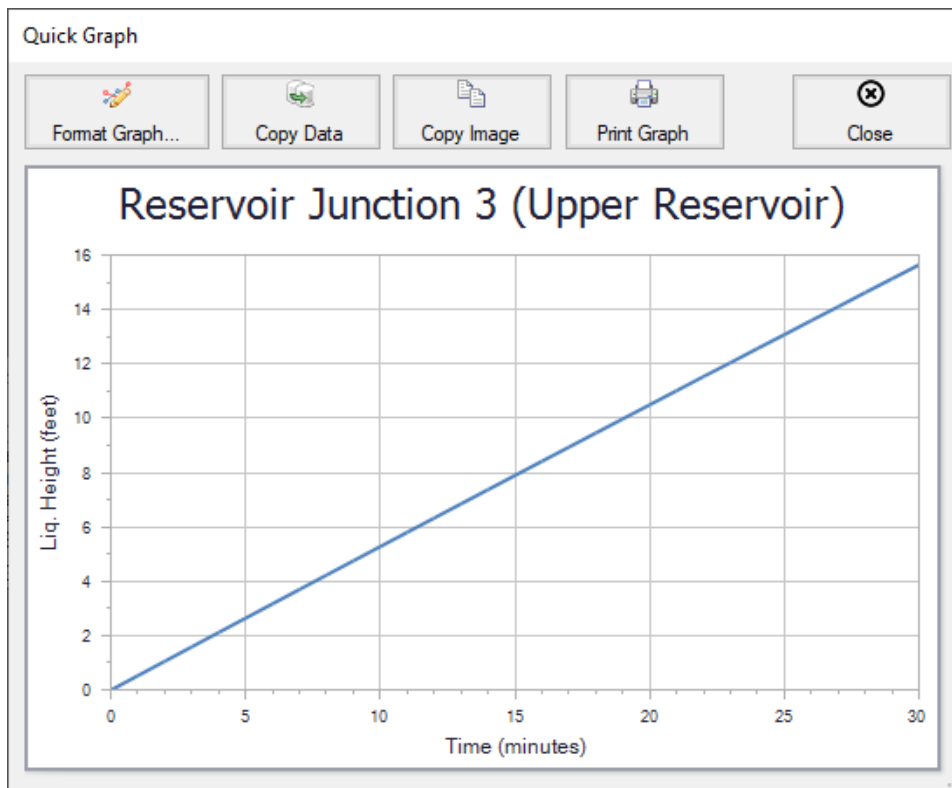

Figure 8: Quick Graph Plot of J3 Reservoir liquid height

Similar to the summary transient data, the transient tabs in the Pipe and Junctions sections of the Output Window are used to display the transient data for each pipe and junction at every time step, as shown in Figure 9. This transient data can be expanded or collapsed in the same manner as in the General Output transient summaries.

Pipe (Time)	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	dH (feet)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)
⊞ 1 (0)	Pipe	595.7	15.01	17.51	16.72	0	0.0	0.7847	0.7847	0.00	1.812	17.51	16.72	19.03	18.24
⊟ 2 (0)	Pipe	595.7	15.01	177.51	13.18	0	200.0	164.3323	164.3323	86.59	179.553	177.51	13.18	179.03	14.70
2 (1)	Pipe	595.3	15.00	177.54	13.41	0	200.0	164.1244	164.1244	86.59	179.073	177.54	13.41	179.05	14.93
2 (2)	Pipe	594.6	14.99	177.58	13.64	0	200.0	163.9384	163.9384	86.59	178.643	177.58	13.64	179.09	15.16
2 (3)	Pipe	593.9	14.97	177.63	13.88	0	200.0	163.7521	163.7521	86.59	178.213	177.63	13.88	179.14	15.38
2 (4)	Pipe	593.1	14.95	177.68	14.11	0	200.0	163.5659	163.5659	86.59	177.783	177.68	14.11	179.18	15.61
2 (5)	Pipe	592.4	14.93	177.72	14.34	0	200.0	163.3798	163.3798	86.59	177.353	177.72	14.34	179.22	15.84
2 (6)	Pipe	591.7	14.91	177.77	14.58	0	200.0	163.1938	163.1938	86.59	176.923	177.77	14.58	179.27	16.07
2 (7)	Pipe	590.9	14.89	177.82	14.81	0	200.0	163.0078	163.0078	86.59	176.494	177.82	14.81	179.31	16.30
2 (8)	Pipe	590.2	14.87	177.86	15.04	0	200.0	162.8220	162.8220	86.59	176.065	177.86	15.04	179.35	16.53
2 (9)	Pipe	589.4	14.86	177.91	15.27	0	200.0	162.6363	162.6363	86.59	175.636	177.91	15.27	179.39	16.76
2 (10)	Pipe	588.7	14.84	177.95	15.50	0	200.0	162.4507	162.4507	86.59	175.207	177.95	15.50	179.43	16.98

Jct (Time)	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
⊞ 1 (0)	Lower Reservoir	14.70	19.03	14.70	19.03	595.7	82.75	0
⊞ 2 (0)	Pump	16.72	177.51	18.24	179.03	595.7	82.75	0
⊟ 3 (0)	Upper Reservoir	14.70	14.70	14.70	14.70	595.7	82.75	0
3 (1)	Upper Reservoir	14.70	14.93	14.70	14.93	595.3	82.70	0
3 (2)	Upper Reservoir	14.70	15.16	14.70	15.16	594.6	82.60	0
3 (3)	Upper Reservoir	14.70	15.38	14.70	15.38	593.9	82.49	0
3 (4)	Upper Reservoir	14.70	15.61	14.70	15.61	593.1	82.39	0
3 (5)	Upper Reservoir	14.70	15.84	14.70	15.84	592.4	82.29	0
3 (6)	Upper Reservoir	14.70	16.07	14.70	16.07	591.7	82.19	0
3 (7)	Upper Reservoir	14.70	16.30	14.70	16.30	590.9	82.08	0

Figure 9: Output Window showing the Pipe and Junction Transient tabs

Figure 10 shows the output data for the pipes and junctions, found on the Pipes and All Junctions tabs, at the initial time step, Time = 0 minutes. The output for all of the pipes and junctions can be displayed at any time step by using the slider bar located at the bottom of the Output window. Figure 11 shows the pipe and junction data at Time = 5 minutes.

Time: 0 Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	dH (feet)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)
1	Pipe	595.7	15.01	17.51	16.72	0	0.0	0.7847	0.7847	0.00	1.812	17.51	16.72	19.03	18.24
2	Pipe	595.7	15.01	177.51	13.18	0	200.0	164.3323	164.3323	86.59	179.553	177.51	13.18	179.03	14.70

Time: 0 Jct	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
1	Lower Reservoir	14.70	19.03	14.70	19.03	595.7	82.75	0
2	Pump	16.72	177.51	18.24	179.03	595.7	82.75	0
3	Upper Reservoir	14.70	14.70	14.70	14.70	595.7	82.75	0

Time: 0 minutes

Figure 10: Pipe and Junction transient data at Time = 0 minutes

Figure 11: Pipe and Junction transient data at Time = 5 minutes

Step 9. Create a New Scenario

Create a new scenario called Variable Speed Pump.

Open the Pump Properties window for J2. On the Variable Speed tab, set the pump to Controlled Pump (Variable Speed) and select Volumetric Flow Rate with a Control Setpoint of 450 gal/min.

Rerun the model.

Step 10. Examine the Output

Figure 12 shows a Quick Graph plot of the Pump Speed vs. Time for the controlled flow pump. Note how the pump speed increases over time to maintain the controlled flow. This is because the hydrostatic pressure downstream of the pump is increasing because the tank is slowly filling over time.
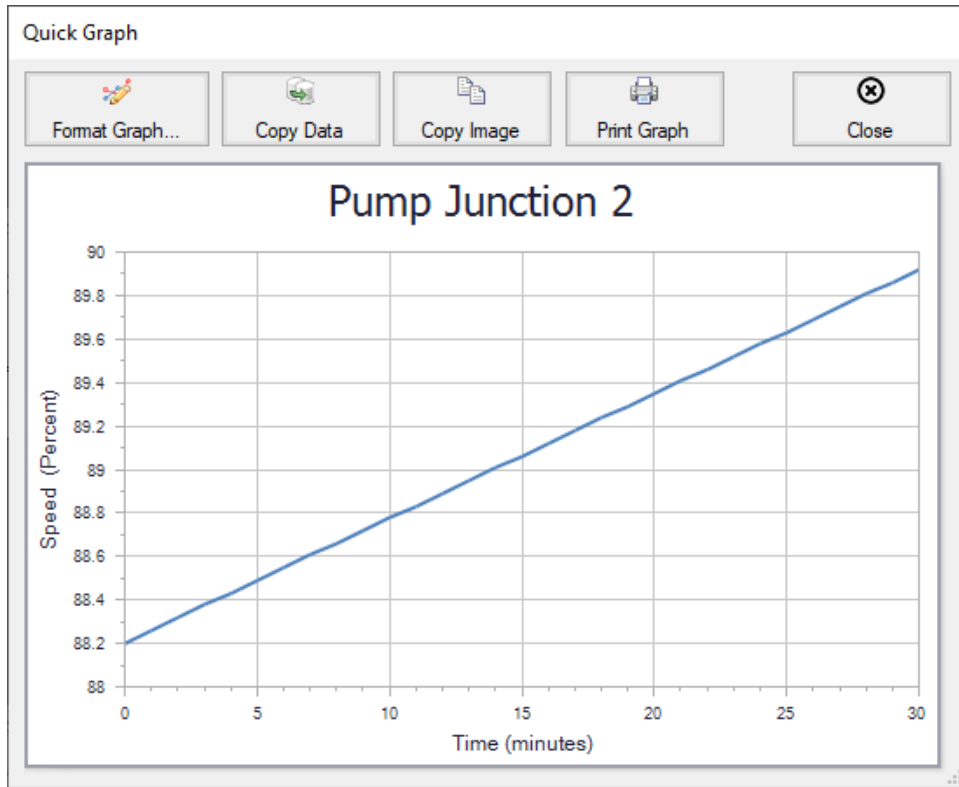
Figure 12: Quick Graph plot of Controlled Pump Speed vs. Time

Reservoir Balance - XTS

Summary

The objective of this example is to demonstrate how the interaction of different hydrostatic pressures between can be analyzed using the XTS module. This example will use three finite reservoirs connected in series, with different initial liquid elevations.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Using Finite Reservoirs Junctions

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Filling a Tank - XTS example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- *US - Reservoir Balance - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 70 deg. F

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for XTS Reservoir Balance example

Pipe Properties

Enter the following for both pipes P1 and P2.

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 4 inch
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Length = 250 feet

Junction Properties

Figure 2 shows the Reservoir Properties window for junction J1. In the Fathom XTS module, reservoir junctions can have finite dimensions. The reservoirs can also be open, or closed. If the tank is closed, the user must specify the gas thermodynamics inside the closed tank. Users may also specify different initial known parameters for the reservoir. If only the Liquid Surface level is known, Fathom will calculate the Surface Pressure. Alternatively, if only the Surface Pressure is known, Fathom will calculate the Liquid Height. When using a reservoir as a finite tank, users must also specify the tank height, the initial liquid height in the tank, as well as the tank cross sectional area.
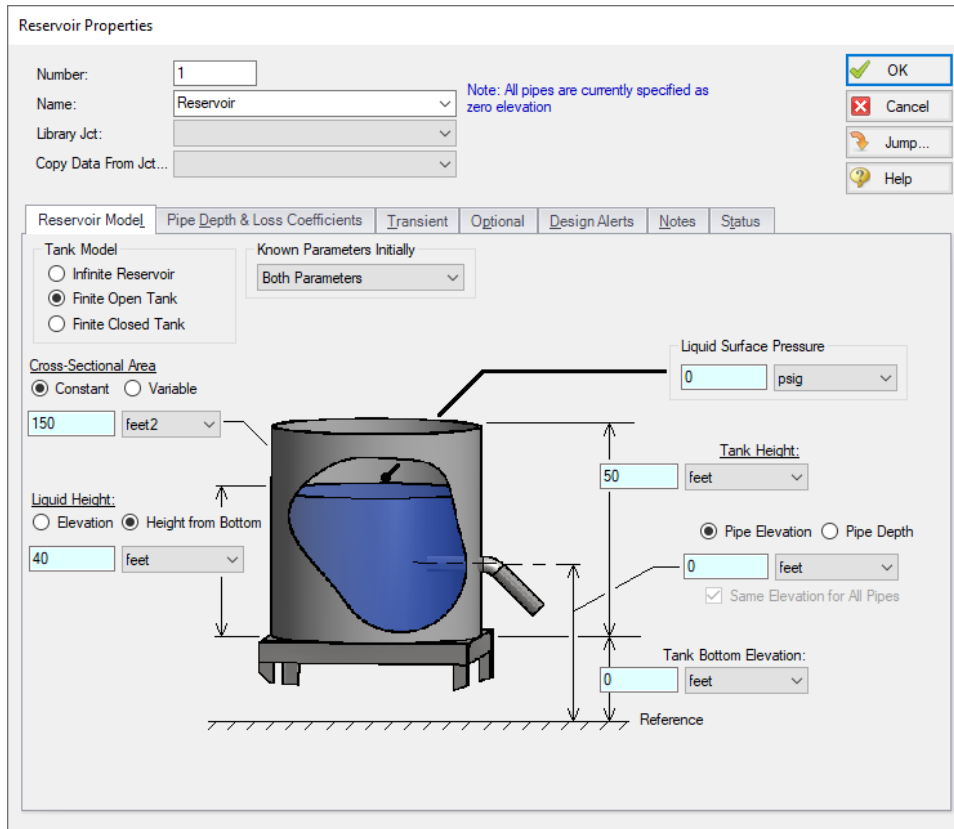
Figure 2: Reservoir Properties Window in Fathom XTS

Next, define the reservoir junctions.

1. J1 Reservoir
    a. Tank Model = Finite Open Tank
    b. Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)
    c. Cross-Sectional Area = Constant, 150 feet2
    d. Liquid Level Specification = Height from Bottom
    e. Liquid Height = 40 feet
    f. Liquid Surface Pressure = 0 psig
    g. Tank Height = 50 feet
    h. Pipe Elevation = 0 feet
    i. Tank Bottom Elevation = 0 feet
2. J2 Reservoir
    a. Tank Model = Finite Open Tank
    b. Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)
    c. Cross-Sectional Area = Constant, 150 feet2

      d.   Liquid Level Specification = Height from Bottom

      e.   Liquid Height = 0 feet

      f.   Liquid Surface Pressure = 0 psig

      g.   Tank Height = 50 feet

      h.   Pipe Elevation = 0 feet

      i.   Tank Bottom Elevation = 0 feet

3.   J3 Reservoir

      a.   Tank Model = Finite Open Tank

      b.   Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)

      c.   Cross-Sectional Area = Constant, 150 feet2

      d.   Liquid Level Specification = Height from Bottom

      e.   Liquid Height = 10 feet

      f.   Liquid Surface Pressure = 0 psig

      g.   Tank Height = 50 feet

      h.   Pipe Elevation = 0 feet

      i.   Tank Bottom Elevation = 0 feet

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 120 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

For this model, the output will be easiest to view as a graph, so select the Graph Results tab. Use the Graph Results window to plot the liquid height in each of the three reservoirs on the same plot. Figure 3 shows the Graph Control tab of the Quick Access Panel for simultaneously plotting the transient liquid height in all three reservoirs. After selecting the Transient graphing tab, select Reservoir from the Pipes or Junctions drop-down menu.

Figure 4 show the resulting graph. Note that, as one would expect because the tanks are the same sizes and at the same elevations, the liquid heights between the tanks eventually reach the same equilibrium value.
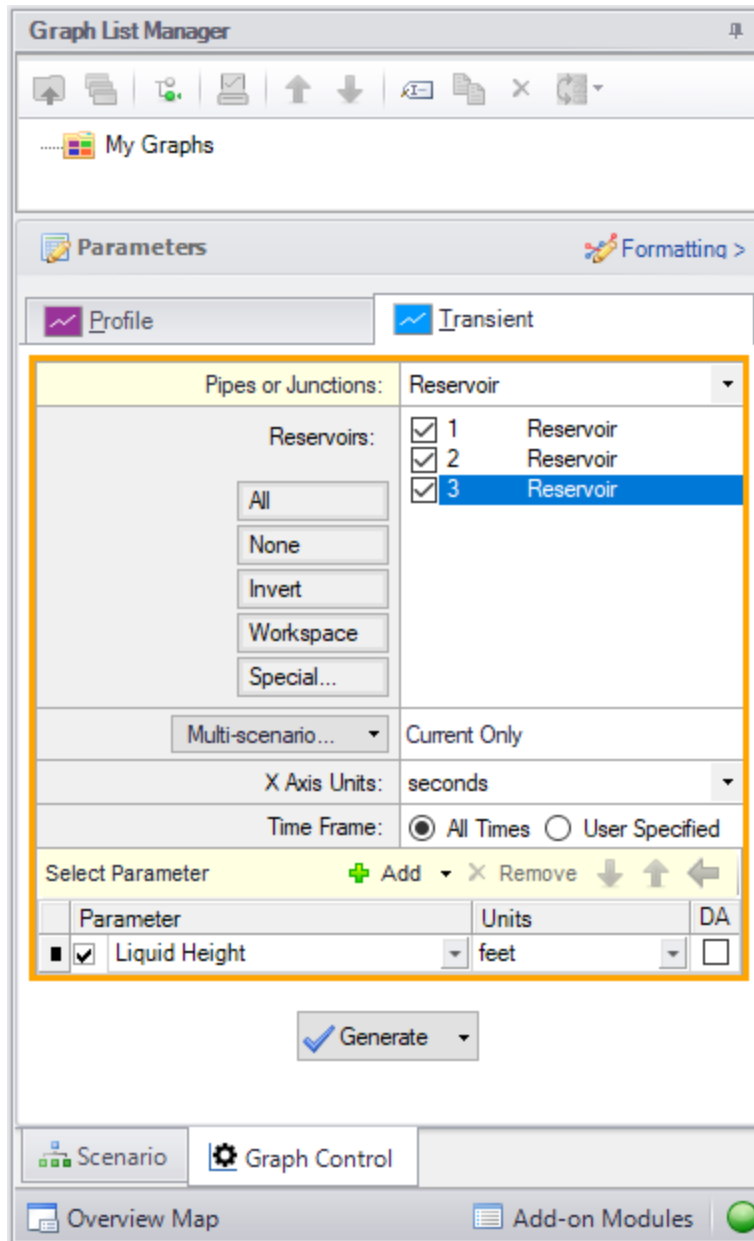
Figure 3: The transient liquid height in all three reservoirs can be selected in the Select Graph Data window
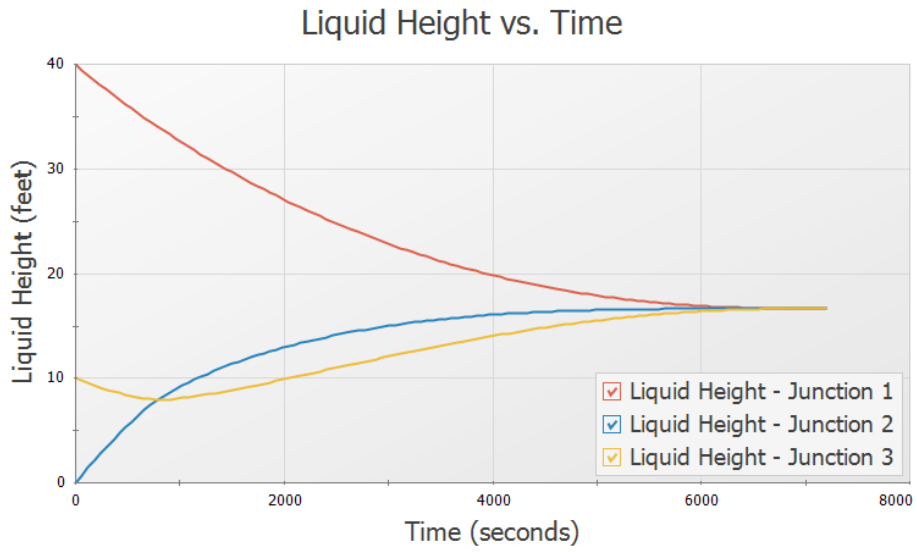
Figure 4: Reservoir liquid heights can be plotted using the Graph Results window

Valve Closing - XTS

Summary

The objective of this example is to demonstrate how to set up an event based transient in the XTS module for a valve junction. A valve is set to close when the liquid height in discharge tank reaches a specified level.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Junction Transients
- Understanding Transient Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Filling a Tank - XTS example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Valve Closing - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 70 deg. F

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 1: Workspace for XTS Valve Closing example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 4 inch
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Lengths =

Pipe	Length (feet)
1	10
2	980
3	10

Junction Properties

1. J1 Reservoir
   a. Name = Lower Reservoir
   b. Tank Model = Infinite Reservoir
   c. Liquid Surface Elevation = 10 feet
   d. Liquid Surface pressure = 0 psig
   e. Pipe Depth = 10 feet
2. J2 Pump
   a. Inlet Elevation = 0 feet
   b. Pump Model = Centrifugal (Rotodynamic)
   c. Analysis Type = Pump Curve
   d. Enter Curve Data =

Volumetric	Head
gal/min	feet
0	300
500	380
1000	250

   e. Curve Fit Order = 2
1. J3 Valve
   a. Inlet Elevation = 200 feet
   b. Valve Data Source = User Specified
   c. Loss Model = Cv
   d. Loss Source = Fixed Cv
   e. Cv = 100
3. J4 Reservoir
   a. Name = Upper Reservoir
   b. Tank Model = Finite Open Tank
   c. Known Parameters Initially = Both Parameters
   d. Cross-Sectional Area = Constant, 150 feet2
   e. Liquid Height Specification = Height from Bottom
   f. Liquid Height = 0 feet
   g. Liquid Surface Pressure = 0 psig
   h. Tank Height = 30 feet
   i. Pipe Elevation = 200 feet
   j. Tank Bottom Elevation = 200 feet

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 60 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 7. Enter the Valve Transient Data

For this example, we will set an event-based transient on the valve junction such that when the liquid elevation in the reservoir reaches a height of 15 feet, the valve will close over a period of 30 seconds. Junction transient events are defined on the Transient Data tabs in the Junction Properties windows. Open the J3 Valve Properties window, and select the Transient tab.

Transients for junctions can be defined as either time based or event based transients. A time based transient starts at a specified absolute time. Event based transients are initiated when a defined criteria, such as when a pressure in a particular pipe is reached or exceeded. There are three types of event transients for junctions. Single event transients are events that occur only once in a simulation. Dual cyclic events are two transient events that repeat one after the other as many times as the event triggers occur during a simulation (such as a valve closing then reopening). Dual sequential transient events are two transients events which occur one after the other, without repeating.

For this example, we will define a single transient event. The event will occur when the liquid height in the upper reservoir reaches 15 feet. When the liquid height reaches 15 feet, the valve will close over a period of 30 seconds.

Figure 2 shows the Transient Data tab for the Valve Properties window. The transient event is defined in the Initiation of Transient area. This is where the event that will trigger the transient event is defined. The profile of what occurs when the transient is initiated is defined in the Transient Data area. For this example, the transient data consists of the valve closing profile. The valve Cv profile is specified over the time period that the valve is to close. In this example, the Cv profile is linear over the 30-second closing time. Enter the transient data as shown below and in Figure 2, then select OK.

1. Transient tab
    a. Transient Special Condition = None
    b. Initiation of Transient = Single Event
    c. Event Type = Reservoir Liquid Height
    d. Condition = Greater Than
    e. Value = 15 feet
    f. Junction = 4 (Upper Reservoir)
    g. Transient Data = Absolute Values
    h. Data Points =

Time (seconds)	Cv
0	100

Time (seconds)	Cv
30	0
3600	0

Figure 2: Valve closing Event Transient definitions are made on the Transient tab in the Valve Properties window

Fathom will display a T next to the junction label on the workspace when a transient has been defined for a junction.

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

When junction event transients occur, these events are recorded and displayed in the general output section of the Output Window. Open the Output Control and navigate to Format & Action. Change the Time Simulation Formatting to minutes. The events are displayed on two different tabs. The first tab sorts the events by junction number, and the second tab sorts the events by time, in the order in which they occur. The tabs are shown in Figure 3:
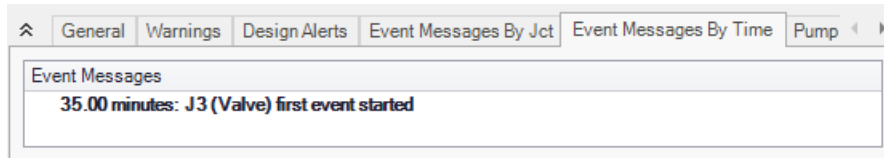
Figure 3: Event Transient Tabs in are shown in the General Output section of the Output Window

A Quick Graph plot of the liquid height in the reservoir, shown in Figure 4, shows that the final liquid height in the Upper Reservoir is about 16 feet. This is because the valve closes over a 30-second period. The valve starts to close when the liquid reaches 15 feet. During the 30-second period that the valve is closing, liquid continues to flow into the tank. Thus, there is some *overshoot* which results in approximately 10 additional inches of fluid being added to the tank. This can also be seen by looking at the liquid height at the final time step shown on the Reservoir Transient tab.
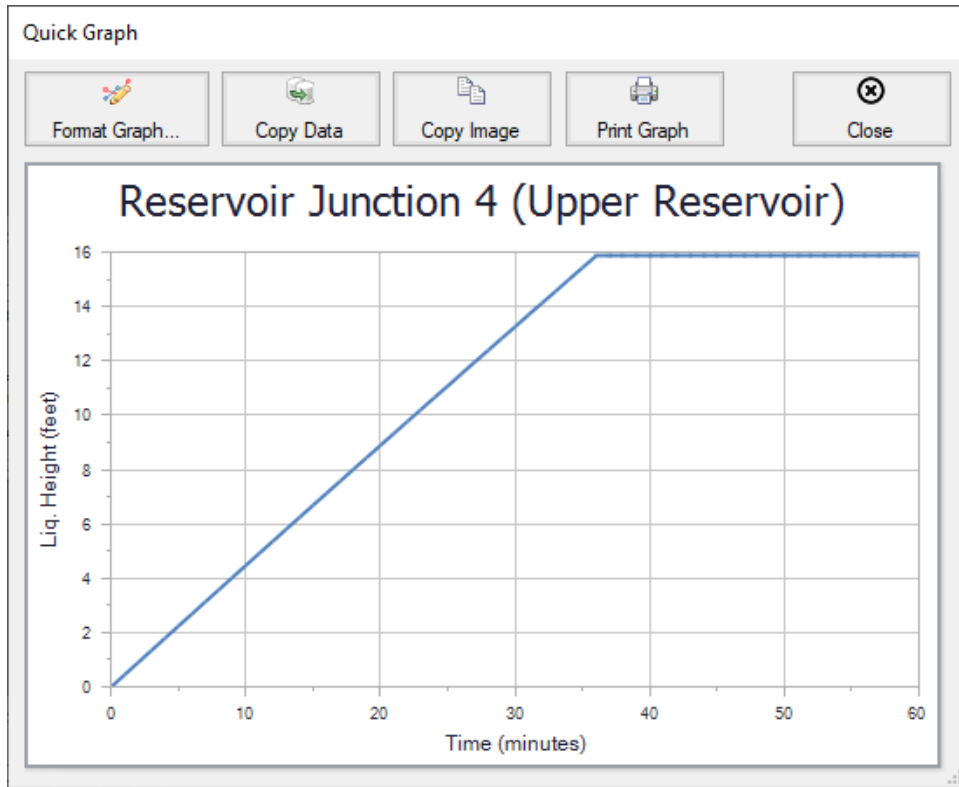
Figure 4: A Quick Graph plot can be used to plot reservoir liquid height showing overshoot due to valve closure time

Controlled Heat Exchanger Temperature - XTS

Summary

Use the Extended Time Simulation module to determine the length of time required to fill the discharge reservoir, and to turn off the pump when the tank is full.

The discharge reservoir is a finite size tank, 20 feet high, with a cross sectional area of 80 feet2. The pump is to shut off when the tank is filled. How long does it take to fill the tank? What happens after the pump is turned off?

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Transient Control
- Defining system transients

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Filling a Tank - XTS example, and is familiar with the basics of XTS transient analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Controlled Heat Exchanger Temperature.fth*
- *US - Controlled Heat Exchanger Temperature - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **US - Controlled Heat Exchanger Temperature.fth** example file and save it as a new file. The workspace should look like Figure 1 below.

Important: The XTS and GSC modules and cost calculations can be used in the same run to simulate control functions over time, but we are not going to do this here.
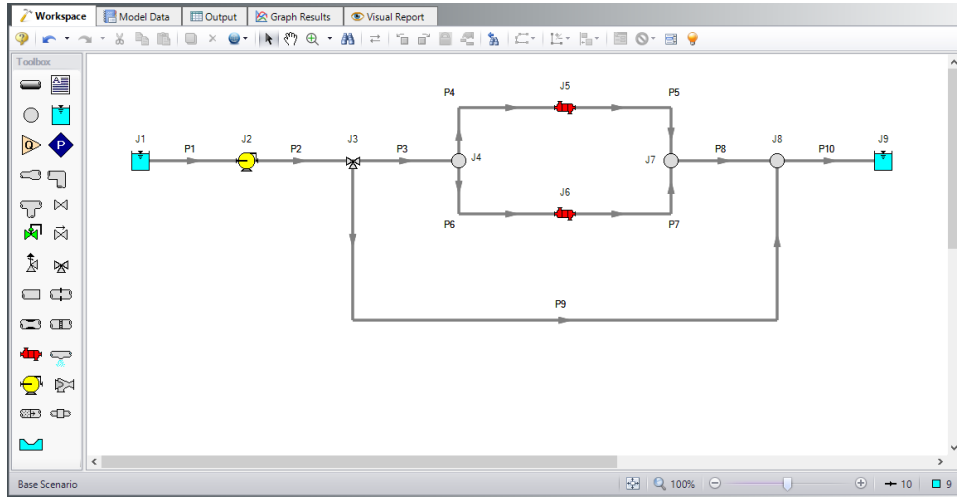
Figure 1: Workspace for XTS Controlled Heat Exchanger Temperature example

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Note: The XTS module does not perform heat transfer calculations. You will be prompted to disable heat transfer calculations in order to proceed with modeling a transient. Click Yes.

Step 4. Define Pipes and Junctions group

Most of the input is already entered. The only exception is for Reservoir J9. Enter the following junction properties.

1. J3 Three Way Valve
   a. Library Jct = (None)
   b. Actual Percent Open = 86 %
      i. This value comes from the Actual Percent Open value calculated from the GSC example
2. J9 Reservoir
   a. Name = Discharge Reservoir
   b. Tank Model = Finite Open Tank
   c. Known Parameters Initially = Liquid Surface Level & Surface Pressure
   d. Cross-Sectional Area = Constant, 80 feet2
   e. Liquid Level Specification = Height from Bottom
   f. Liquid Height = 0 feet
   g. Liquid Surface Pressure = 40 psig
   h. Tank Height = 20 feet
   i. Pipe Elevation = 10 feet
   j. Tank Bottom Elevation = 10 feet

Step 5. Setup the Transient Data

For this example, we will set an event-based transient on the pump so the pump will turn off when the liquid level in the discharge tank reaches 20 feet.

You will define a single transient event. The event will occur when the liquid height in the reservoir reaches 20 feet. When the liquid height reaches 20 feet, the pump speed will drop from 100% to 0% in 10 seconds.

Open the Pump Properties window, and enter the following information. Then the Transient tab should appear as shown in Figure 2.

1. Library Jct = (None)
2. Transient tab
   a. Transient Type = Speed vs. Time
   b. Transient Special Condition = None
   c. Initiation of Transient = Single Event
   d. Event Type = Reservoir Liquid Height
   e. Condition = Greater Than or Equal To
   f. Value = 20 feet
   g. Junction = 9 (Discharge Reservoir)
   h. Transient Data = Absolute Values
   i. Data Points =

Time (seconds)	Speed (Percent)
0	100
10	0
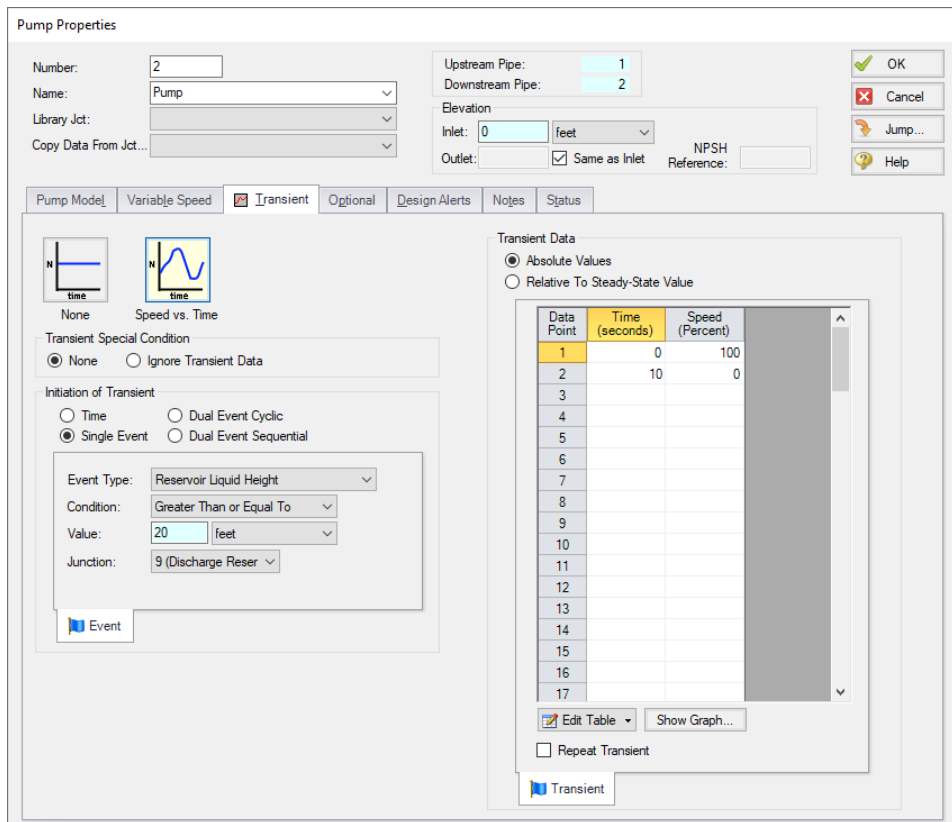
Figure 2: Junction transients are specified on the Transient tabs on the junction Properties windows

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 6. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of **0 minutes**, set the Stop Time to **45 minutes** with a time step of **1 minute**. The output will be saved to the output file for every time step.

Step 7. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

Open the Output Control and navigate to Format & Action. Change the Time Simulation Formatting selection to minutes and click OK. The transient reservoir data shows that the water in the discharge tank reached a height of 20 feet in 32 minutes. This is confirmed by the event message indicating the pump transient also initiated at this time (Figure 4).

The Quick Graph plot (Figure 3) shows the tank liquid height over time. To create this graph, go to the Reservoir Transient tab in the top section of the window, then right-click on any of the Liq. Height cells for the Discharge Reservoir, junction J9, and select the Quick Graph option. There is an increase in the tank liquid height until the time when the pump turns off. The graph then shows that there is actually back flow into the system, due to the hydrostatic pressure of the fluid in the tank. This explains why there is a Critical Warning, which states that the pump head cannot be predicted due to reverse flow through the pump. A check valve would need to be added to the system to prevent this situation in the case of pump shutdown.
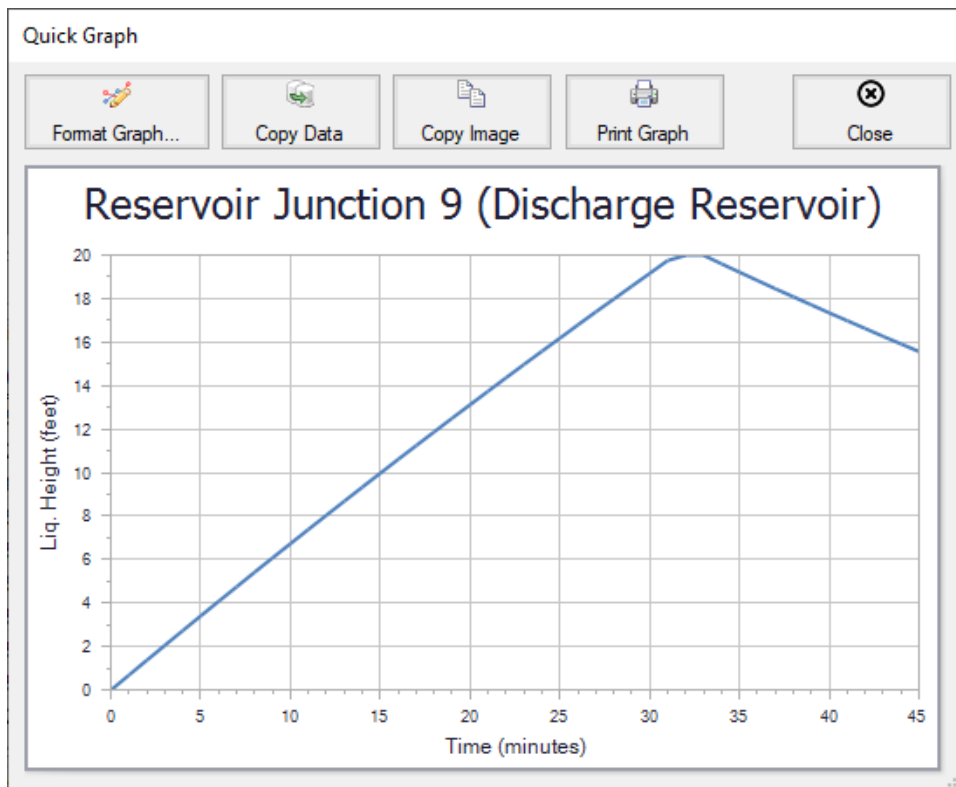

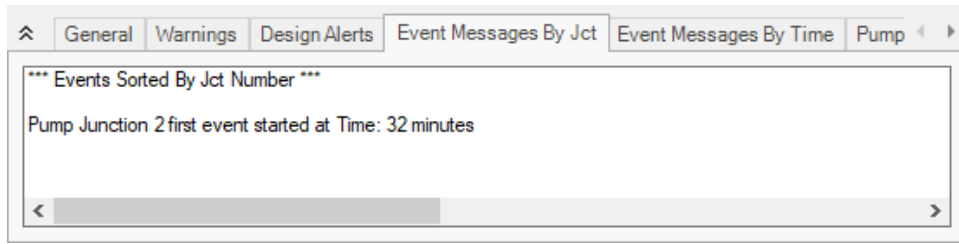*Figure 3: The liquid level in a finite tank junction can change with time*

Figure 4: Event transients occurrences are recorded in the General section of the Output Window

Housing Project - XTS

Summary

This example uses the XTS module to simulate a supply pump used to replenish a water storage tank that is used to supply water to a housing development. There is a requirement that the height of the liquid in the water supply tank is not to drop below 10 feet. Use the flow conditions specified in the **4 inch Pipe** scenario (not the fire flow scenarios) of the Housing Project walk-through example as the basis for this example.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Using Dual Cyclic Junction Transients
- Understanding Transient Output
- Resolving Time Step issues

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Filling a Tank - XTS example, and is familiar with the basics of XTS transient analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Housing Project.fth*
- *US - Housing Project - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Housing Project.fth** example file. Right-click on the 4 inch Pipe scenario and select Save Scenario to File Without Children and save this scenario to a new model file.

The Workspace should look like Figure 1 below:
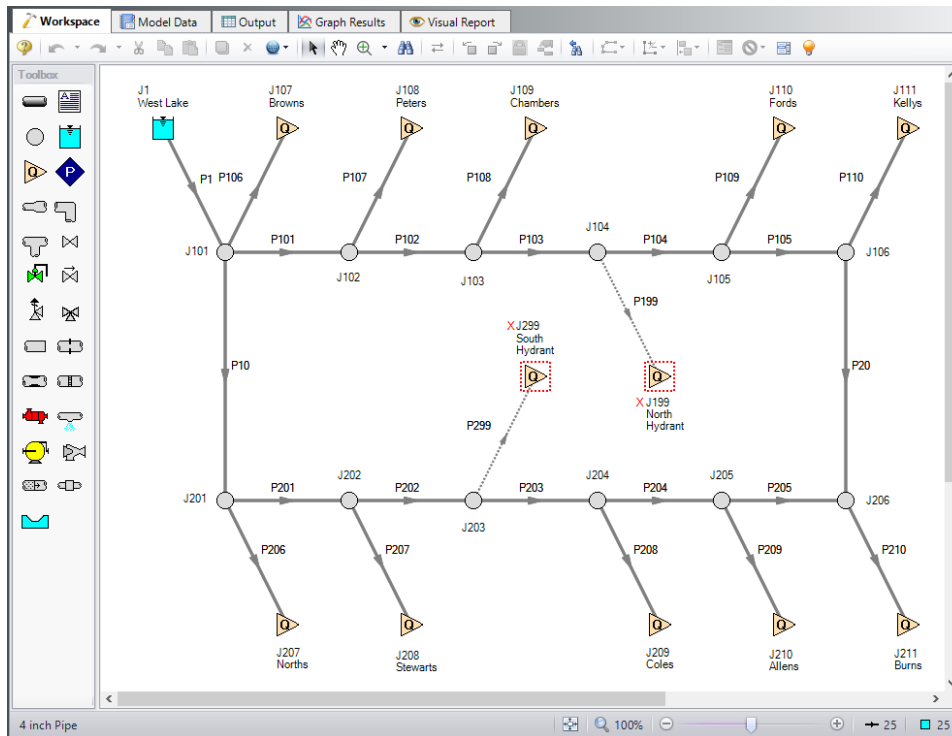
Figure 1: Workspace for XTS Water to Housing Project Example

For this example, we will be changing the West Lake junction (J1) from an infinite reservoir to a finite water supply tank. We will then add a new supply reservoir, as well as a supply pump and a check valve to the system. The new supply pump will replenish the supply tank as the water level in the tank is depleted.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 4. Modify the existing model
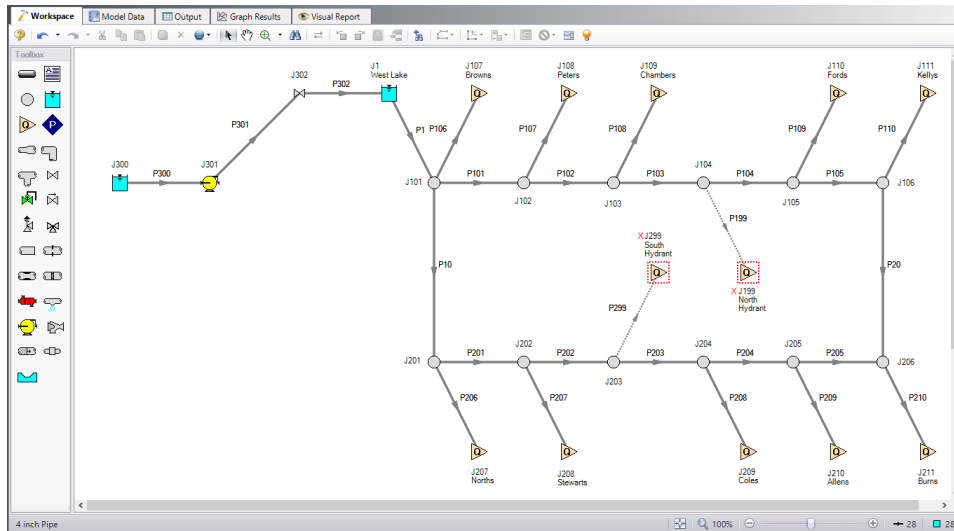
Figure 2: Workspace for XTS water to Housing Project example with supply pump system added

Modify the workspace to include the new supply reservoir, supply pump, check valve, and piping as shown in Figure 2. You may need to move the objects to the right on the Workspace to create room for the new pipes and junctions. Do this by selecting all pipes and junctions by either using CTRL+A or click-and-drag and selection box encompassing all objects. Then click, hold, and drag the objects to the right. Now define the new objects.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = PVC - ASTM
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 4 inch
    d. Type = schedule 40
    e. Friction Model Data Set = Standard
    f. Lengths =

Pipe	Length (feet)
300	25
301	500
302	5

Junction Properties

1. J300 Reservoir
    a. Name = West Lake
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 0 feet
    d. Liquid Surface Pressure = 0 psig
    e. Pipe Depth = 0 feet
2. J301 Pump
    a. Inlet Elevation = 0 feet
    b. Pump Model = Centrifugal (Rotodynamic)
        i. Analysis Type = Pump Curve
        ii. Enter Curve Data =

Volumetric	Head
gal/min	feet
0	1000
200	1150
400	800

        iii. Curve Fit Order = 2
    c. Optional tab
        i. Special Condition = Pump Off With Flow Through
1. J302 Check Valve
    a. Inlet Elevation = 200 feet
    b. Loss Model = Cv
    c. Cv = 10
    d. Forward Velocity to Close Valve = 0 feet/sec
    e. Delta Pressure/Head to Reopen = Pressure
    f. Delta Pressure = 0 psid
2. J1 Reservoir
    a. Name = Supply Tank
    b. Tank Model = Finite Open Tank
    c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)
    d. Cross-Sectional Area = Constant, 80 feet2
    e. Liquid Level Specification = Height from Bottom
    f. Liquid Height = 14.9 feet
    g. Liquid Surface Pressure = 0 psig
    h. Tank Height = 15 feet

i. Pipe Depth = 15 feet

j. Tank Bottom Elevation = 200 feet

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 120 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 7. Enter the Pump Transient Data

For this example, the pump will be set to turn on when the liquid level in the supply tank drops below 10 feet, and turn off when the water level returns to 15 feet. To do this, we will use a Dual Event Cyclic event transient.

The Dual Event Cyclic event requires that two transients be defined. In this case, the first transient will be the pump turning on when the water level drops below 10 feet, and the second transient will be the pump turning off when the water level returns to 15 feet. The pump speed transients need to be included for both transient events as well. Enter the following in the pump properties window.

1. Transient tab
   a. Transient Type = Speed vs. Time
   b. Transient Special Condition = None
   c. Initiation of Transient = Dual Event Cyclic
   d. First Event
      i. Event Type = Reservoir Liquid Height
      ii. Condition = Less Than or Equal To
      iii. Value = 10 feet
      iv. Junction = 1 (Supply Tank)
   e. First Transient =

Time (seconds)	Speed (Percent)
0	0
30	100

e. Second Event
   i. Event Type = Reservoir Liquid Height
   ii. Condition = Greater Than or Equal To
   iii. Value = 15 feet
   iv. Junction = 1 (Supply Tank)
e. Second Transient =

Time (seconds)	Speed (Percent)
0	100
30	0

Figure 3: First Event and First Transient Data for Water to Housing Project XTS example
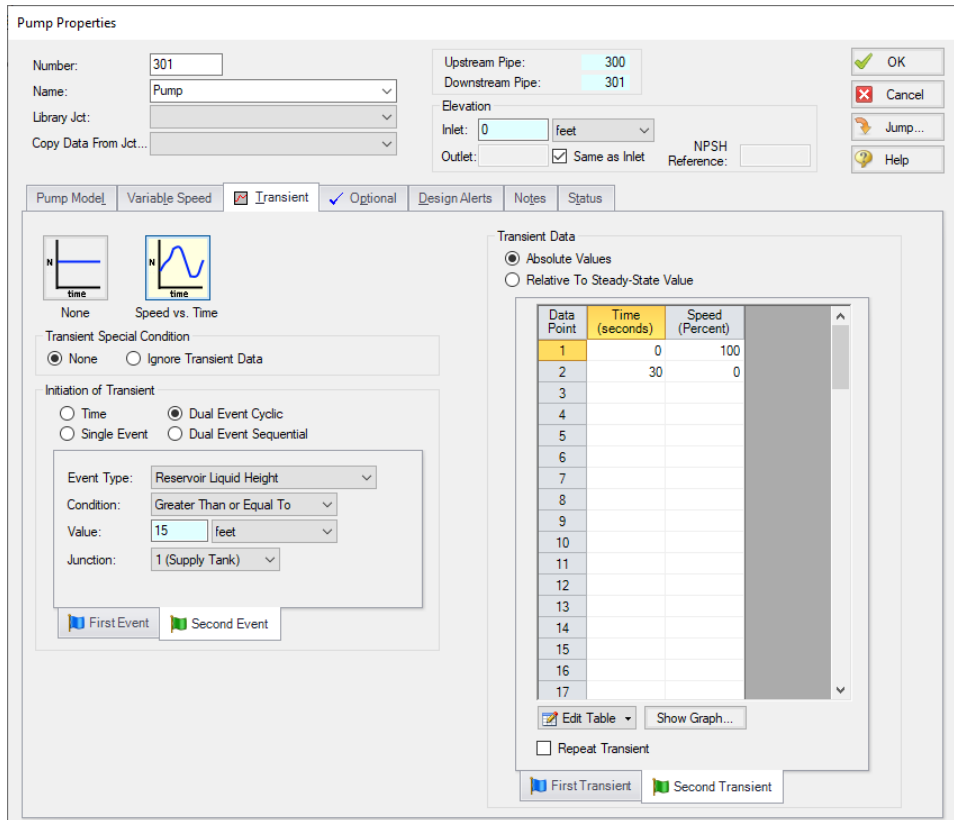
Figure 4: Second Event and Second Transient Data for Water to Housing Project XTS example

For this example, at the beginning of the simulation, the storage tank is full, so the pump will not be running initially. To account for this, the pump Special Condition on the Optional Tab must be set to Pump Off With Flow Through. In order to be consistent, the First Transient defined must be defined with an initial speed of 0, indicating a pump startup.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

Figure 5 shows a Quick Graph plot of the liquid height in the J1 Supply Tank. The plot illustrates how the Dual Event Cyclic transient will repeat continuously as the event to trigger each transient is met, in turn. The plot shows the tank draining due to the flow to the housing project. When the liquid level reaches 10 feet, the pump starts, causing the liquid level in the tank to rise again. When the level reaches 15 feet, the transient to stop the pump is initiated. Because the liquid level reaches the top of the tank at this time, the Warnings tab will display a warning about the tank overflowing at 65 minutes. With the pump turned off, the tank drains until the level drops to 10 feet and the pump is started again.

The plot shows the liquid level dropping below 10 feet. This is due, primarily with the time step resolution that was set in Transient Control. The time step was set to 1 minute, but the pump startup transient is 30 seconds. The size of the time step leads to *overshoot* in the calculations. Open the Simulation Duration panel, and set the time step to 10 seconds. Set the output to save to file every 6th time step (write output once every minute). The results from this case are shown in Figure 6. The results using the 10-second time step clearly show the effect of using the smaller time step.

The user should always carefully consider the time step being used in relation to the length of the transient events defined for their analysis.

Note: You may notice a warning message indicating that the static pressure is lower than vapor pressure at the outlet of pipe 301. Since this example is for demonstration purposes only, this warning may be ignored. However, warnings should otherwise never be ignored.
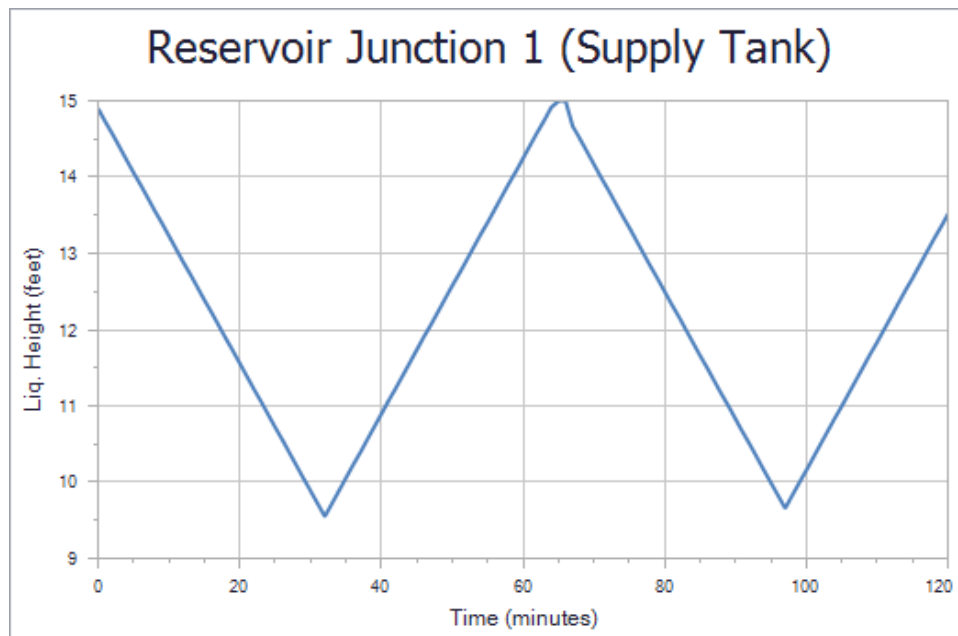

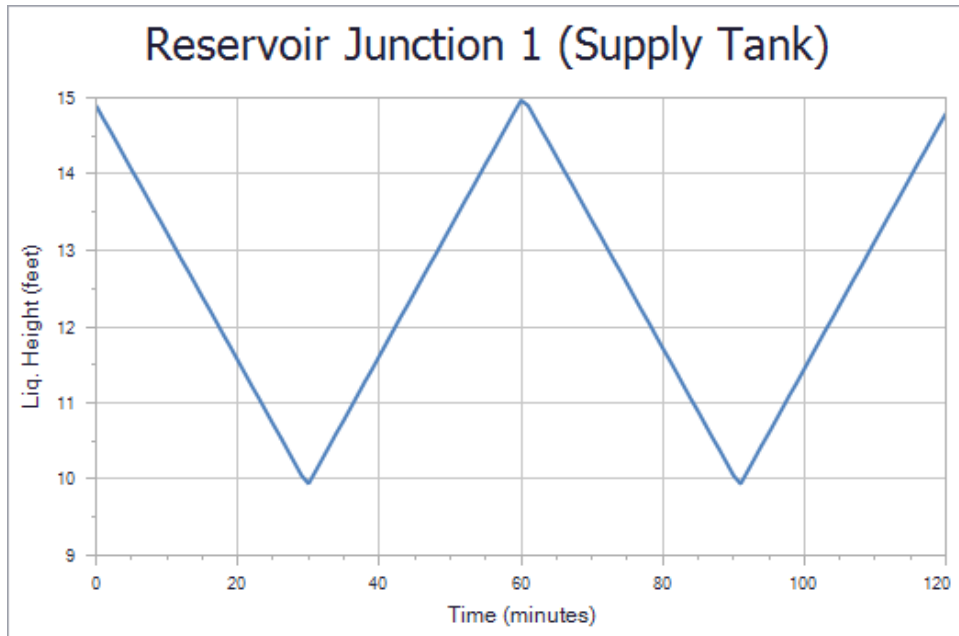***Figure 5: Plot of Supply Tank liquid level showing Dual Event Cyclic transient***

Figure 6: Plot of Supply Tank liquid level with 10-second time step

Variable Demand - XTS

Summary

This example uses a pump transient to initiate auxiliary pump start up for a system with variable demands to ensure adequate pressure drop across a control valve. Additional loads will be added to the system every 30 seconds, until the control valve open percentage exceeds 50%. At that time, the auxiliary pump will start to handle the additional capacity.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Control Valve Event Transient
- Understanding Transient Output
- Using Transient Graphs

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Filling a Tank - XTS example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- *US - Variable Demand - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
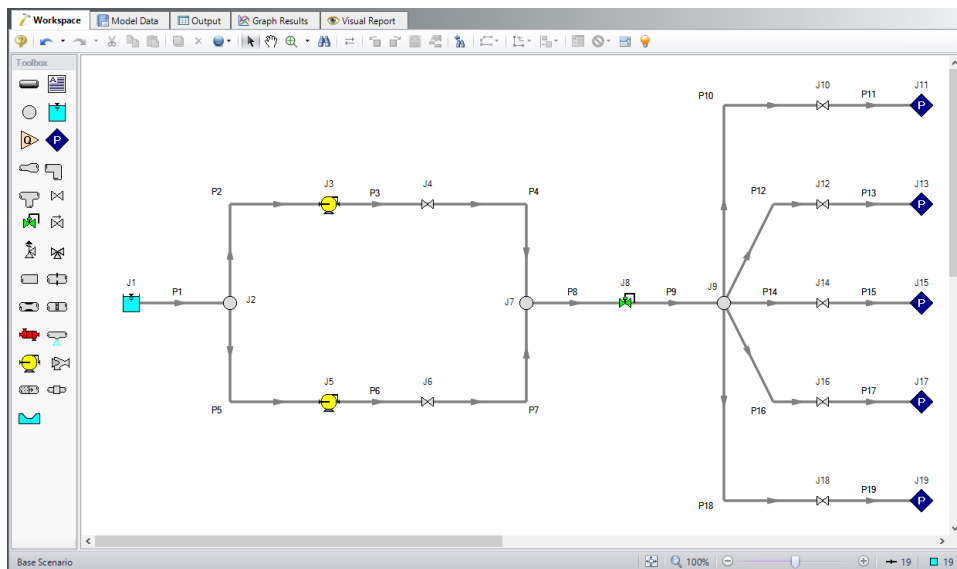

Figure 1: Workspace for XTS Variable Demand example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
   a. Pipe Material = PVC - ASTM
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = 4 inch
   d. Type = schedule 40
   e. Friction Model Data Set = Standard
   f. Lengths =

Pipe	Length (feet)
1-7 & 10-19	10
8 & 9	50

Junction Properties

1. J1 Reservoir
   a. Tank Model = Finite Open Tank
   b. Known Parameters Initially = Both Parameters
   c. Cross-Sectional Area = Constant, 20 feet2
   d. Liquid Level Specification = Height from Bottom
   e. Liquid Height = 5 feet
   f. Liquid Surface Pressure = 0 psig
   g. Tank Height = 5 feet
   h. Pipe Depth = 5 feet
   i. Tank Bottom Elevation = 0 feet
2. J2, J7, & J9 Branches
   a. Elevation = 0 feet
3. J3 & J5 Pumps
   a. Inlet Elevation = 0 feet
   b. Pump Model = Centrifugal (Rotodynamic)
   c. Analysis Type = Pump Curve
   d. Pump Curve Data =

Volumetric	Head
gal/min	feet
0	80

Volumetric	Head
gal/min	feet
125	90
250	50

     e.   Curve Fit Order = 2

4.   J4 & J6 Valves

     a.   Inlet Elevation = 0 feet

     b.   Valve Data Source = User Specified

     c.   Loss Model = Cv

     d.   Loss Source = Fixed Cv

     e.   Cv = 100

5.   J8 Control Valve

     a.   Inlet Elevation = 0 feet

     b.   Control Valve Model tab

          i.   Valve Type = Pressure Reducing (PRV)

          ii.   Always Control (Never Fail) = Unchecked

          iii.   Control Setpoint = Pressure, Static

          iv.   Setpoint = 45 psia

          v.   Loss When Fully Open = Cv

          vi.   Loss Source = From Open % Table (on Optional Tab)

     c.   Optional tab

          i.   Open Percentage Data =

Open Pct.	Cv
0	0
100	100

6.   J10, J12, J14, J16, & J18 Valves

     a.   Inlet Elevation = 10 feet

     b.   Valve Data Source = User Specified

     c.   Loss Model = Cv

     d.   Loss Source = Fixed Cv

     e.   Cv = 10

7.   J11, J13, J15, J17, & J19 Assigned Pressures

     a.   Elevation = 10 feet

     b.   Pressure = 0 psig

     c.   Pressure Specification = Stagnation

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 3 minutes with a time step of 5 seconds. The output will be saved to the output file for every time step.

Step 6. Enter the Transient Data

Valve Transients

Initially, the only flow through the system will be through the J10 Valve. Valves J12, J14, J16, and J18 will start closed, and they will open 30 seconds apart, in sequence. Set the Special Condition for Valves J12, J14, J16, and J18 to Closed on the Valve Properties Optional tab.

The valve opening transient for Valve J12 is shown in Figure 2. Valves J14, J16, and J18 should be set up with the same opening transient, and by adding 30 seconds to the Time Absolute event for each valve so that valve J14 opens at 60 seconds, valve J16 opens at 90 seconds, and valve J18 opens at 120 seconds.

Note that a red XT will now be displayed next to the junction labels to show that these valves are closed, and have transients applied to them.
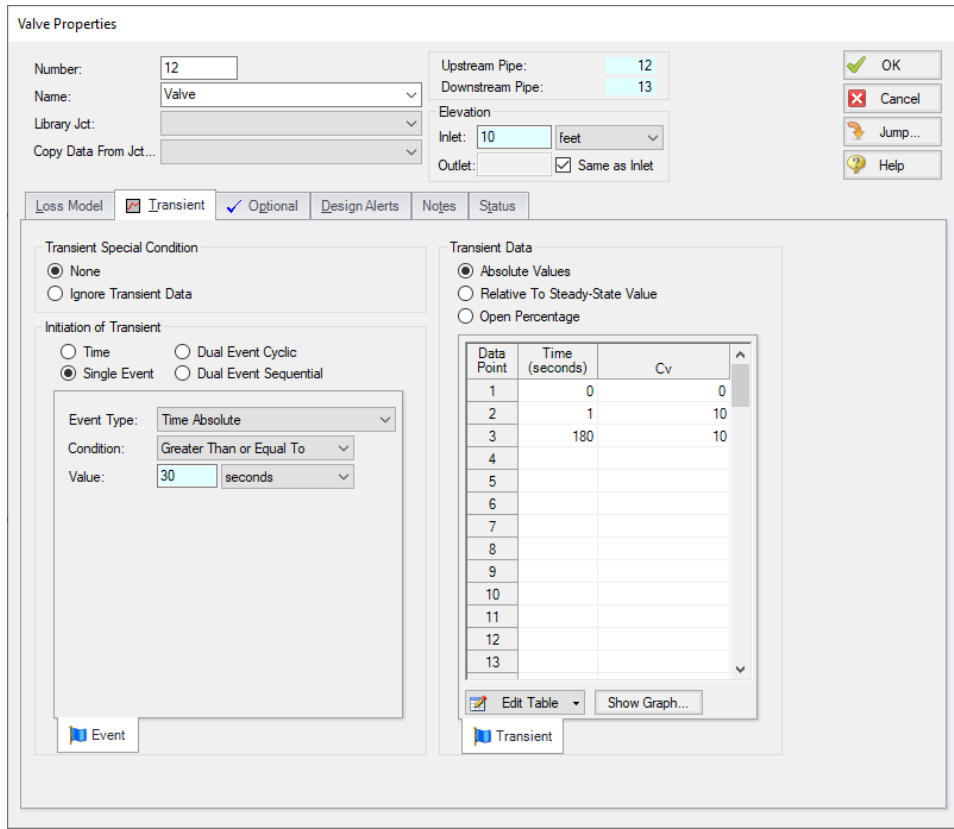
Figure 2: Valve J12-J18 opening transient profiles are defined on the Transient tab on the Valve Properties window

Pump Transient

Pump J5 is not operating at the beginning of this simulation. On the Optional tab of the Pump Properties window set the Pump J5 Special Condition to Pump Off With Flow Through. The pump is set to start when the open percentage of Control Valve J8 exceeds 50%. The transient event definition and pump startup profile are shown in Figure 3.
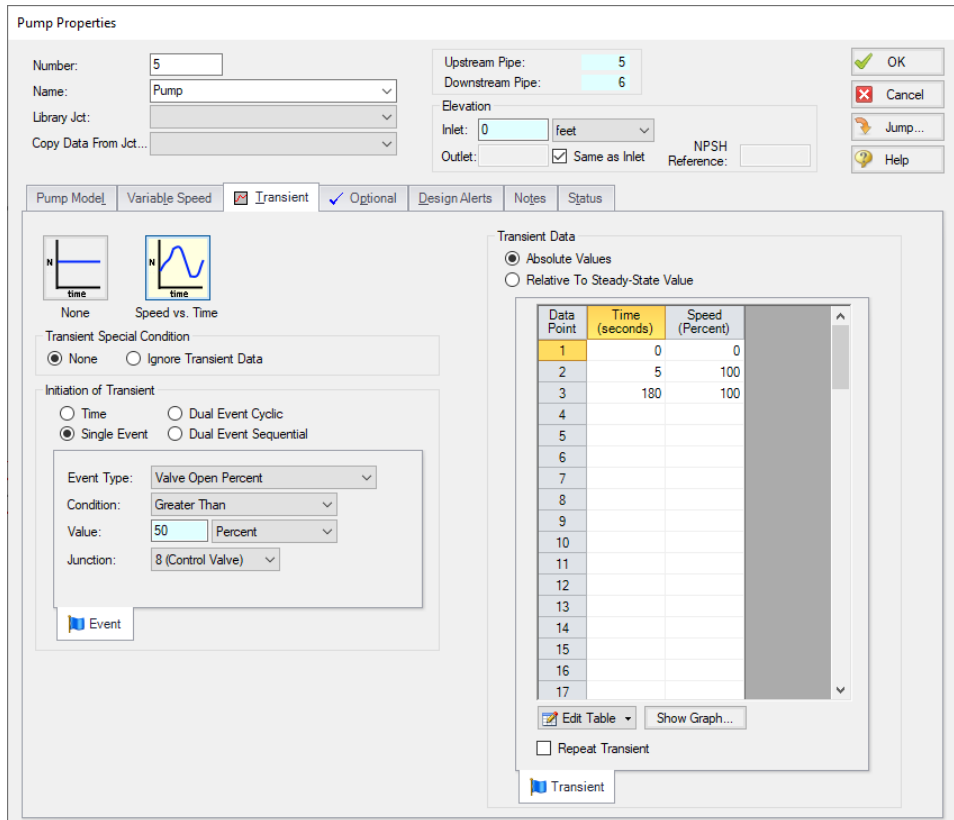
Figure 3: Pump J5 transient startup profile is defined on the Transient tab on the Pump Properties window

Isolation Valve J6 should be closed initially to prevent flow through the leg while Pump J5 is not operating. Do this by setting the Valve J6 Special Condition to Closed on the Optional tab. Valve J6 will open when Pump J5 starts. To do this, use the same transient event as defined for Pump J5. This is shown in Figure 4.
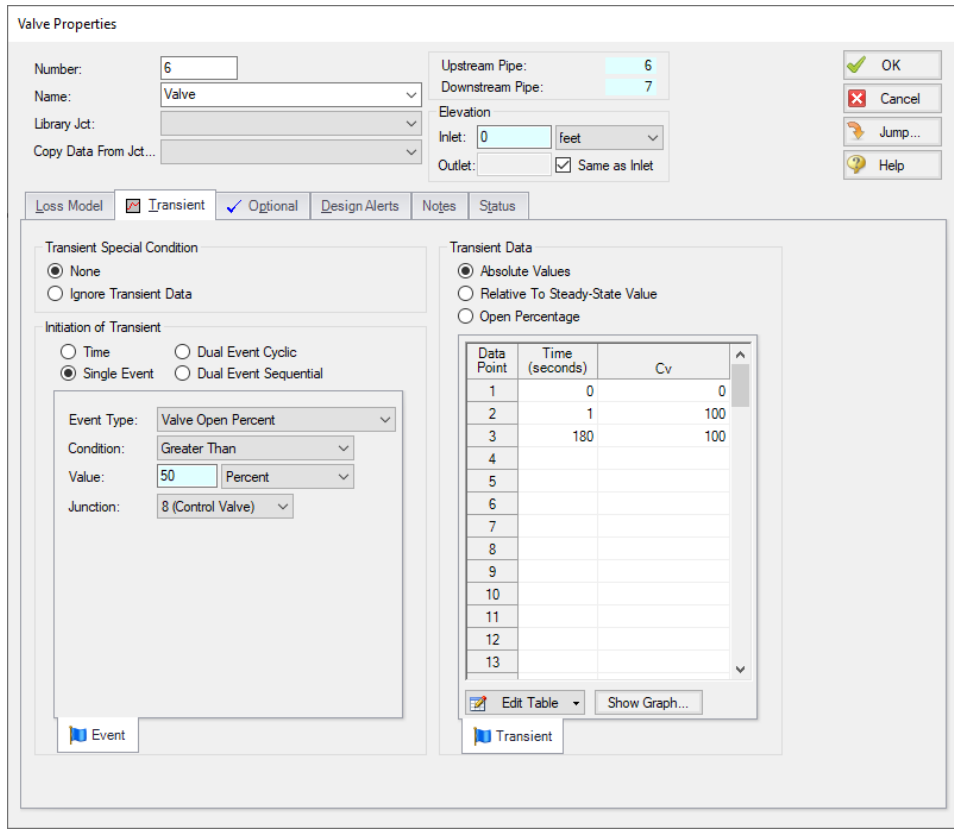
Figure 4: Valve J6 opening transient profile is defined on the Transient tab on the Valve Properties window

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

The occurrence of the transient events is recorded on two tabs in the General Output section. The first tab sorts the events by junction, and the second tab sorts the events chronologically. Figure 5 shows the chronological ordering of the transient events for this run. This output shows the downstream valves opening at 30-second intervals. It also shows that Pump J5 started 65 seconds into the run, and the isolation valve J6 also opened at 65 seconds.
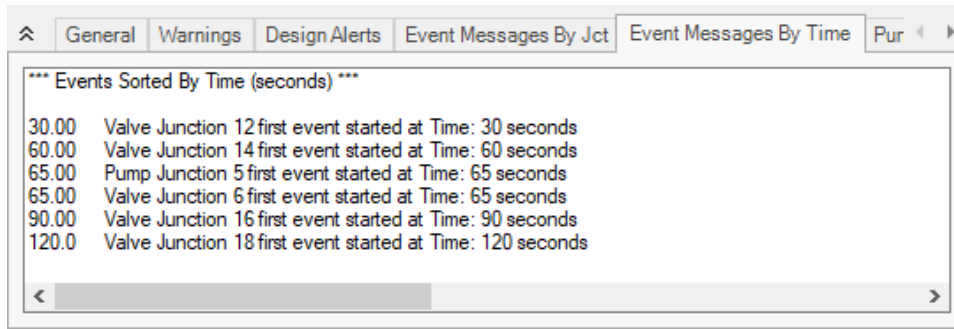
Figure 5: A chronological list of Transient Events is shown in the General Output section of the Output Window

Plotting the speed of Pump J5 vs. time as a Quick Graph from the Output shows the pump starting up at 65 seconds, and coming up to full speed over 5 seconds. This is shown in Figure 6.
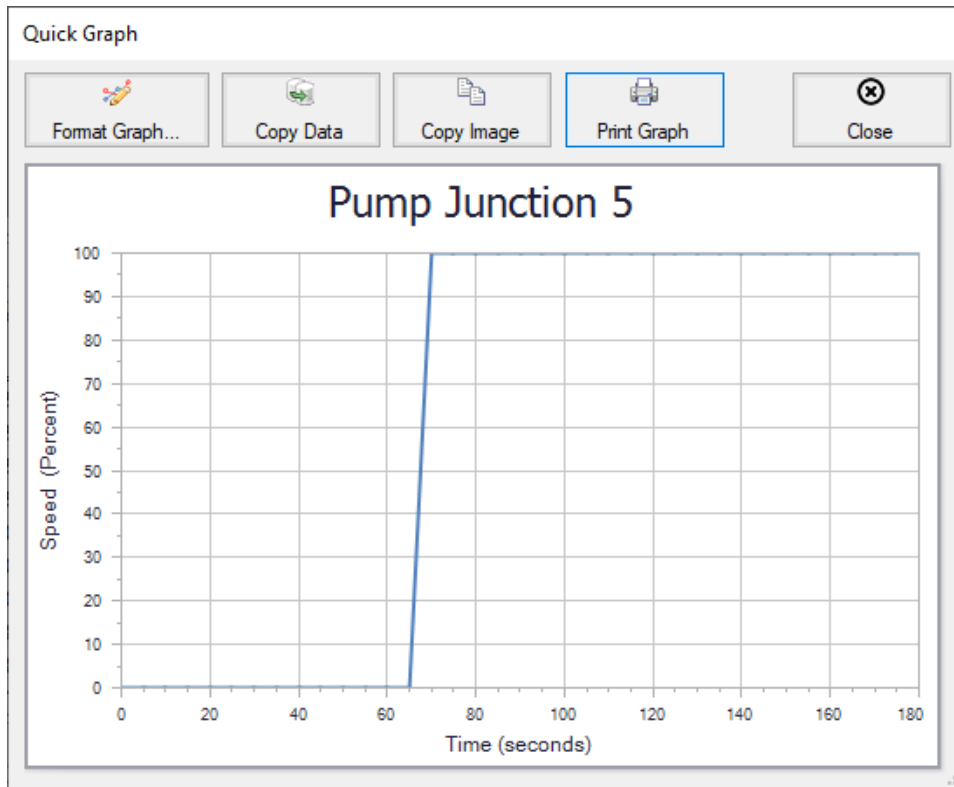

Figure 6: Quick Graph can be used to plot the Pump J5 Speed vs. Time

Now go to the Graph Results window. A stacked graph can be made for the Control Valve J8 as shown in Figure 7 to see the affect of the opening valves on the flow rate at the control valve, as well as how the open percent at the control valve responds. It can be seen here that the control valve open percent reaches 50% at 65 seconds, which correlates with the beginning of the pump startup.

To create these stacked graphs, go to the Transient tab in Graph Control. Change the Pipes or Junctions selection to Valves and check the box next the 8, Control Valve. At the bottom, click the Add button next to Select Parameter to add a second parameter to graph. Set the first parameter to Volumetric Flow Rate with gal/min Units, and set the second parameter to Open Percentage with Percent Units. Now click Generate.
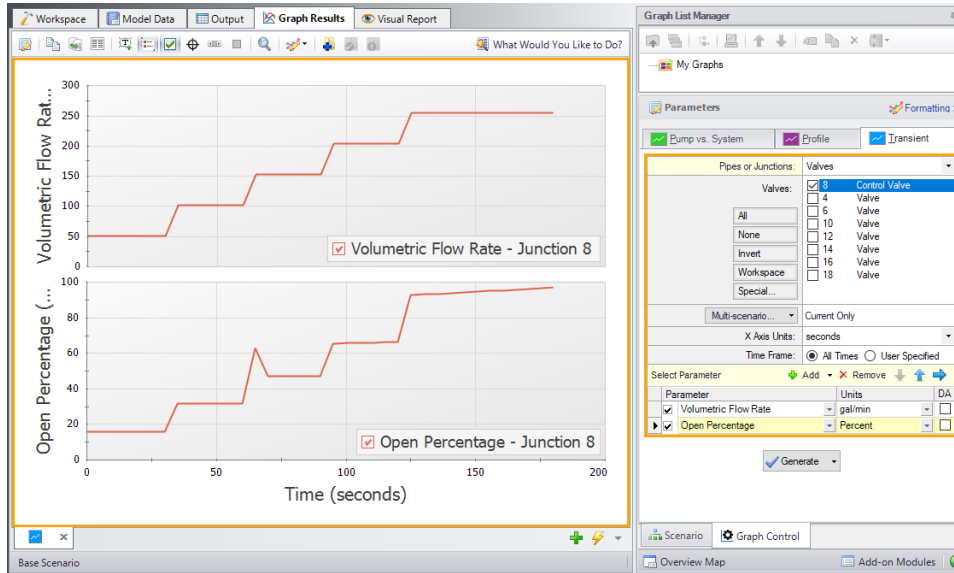

Figure 7: Stacked graph on the Graph results tab showing the Volumetric Flow Rate and Open Percentage over time at Control Valve J8

Settling Slurry (SSL) Examples

The examples below utilize the Settling Slurry module.

Fathom SSL Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Pump Sizing for Sand Transfer System	Beginner	Slurry	3	1	Learn the basics of the SSL module such as defining a Detailed slurry calculation model in Analysis Setup, using the Slurry De-Rating feature in the Pump Properties window, adding Slurry parameters using the Output Control window, and generating Slurry System Curves in the Graph Results window.
Slurries with Variable Fluid Properties	Intermediate	Water/Slurry	2-3	1	Learn how to use the SSL module with Variable Fluid Properties. See how to use the Volume Balance junction and the Simplified slurry calculation model.
Slurry System Feasibility Study	Advanced	Slurry	1	0	Learn how to use the SSL module and the Scenario Manager to determine the feasibility of selected operating conditions. See how to use the Multi-Scenario output feature in the Output window and how to perform a Batch run for a model with multiple scenarios.

Beginner - Pump Sizing for Sand Transfer System - SSL

Summary

This example demonstrates the fundamental concepts of the Settling Slurry (SSL) add-on module by way of example. The example shows how SSL can be used to size a slurry pump as part of a system design process.

The piping for a slurry system that moves sand is being designed. The system will pump 25% sand by volume from an open supply vessel with a liquid surface elevation of 5 feet to an open receiving tank with a liquid surface elevation of 10 feet.

Use the SSL module to size the pump.

> **Note:** This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Reviewing calculated slurry output
- Creating slurry system curves

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Pump Sizing for Sand Transfer System - SSL.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid - Figure 1
     a. Fluid Model = Basic (Water)
     b. Temperature = 70 deg. F
2. Solids Definition - Figure 2
     a. Slurry Calculation Method = Detailed
     b. Solids Specifications = User Specified Solids Added
     c. M (Stratification Ratio Exponent) = Calculated
     d. Terminal Velocity Parameter = Vt / Vts (Xi)
     e. Value = 1
     f. Density = 2.9 S.G. water
     g. d50 = 0.015 inches
     h. d85 = 0.03 inches
3. Slurry Definition - Figure 3
     a. Slurry Calculation Method = Detailed (this selection will be the same as on the Solids Definition panel)
     b. Concentration Type = Volume Fraction
     c. Amount Solids Added = 25 Percent
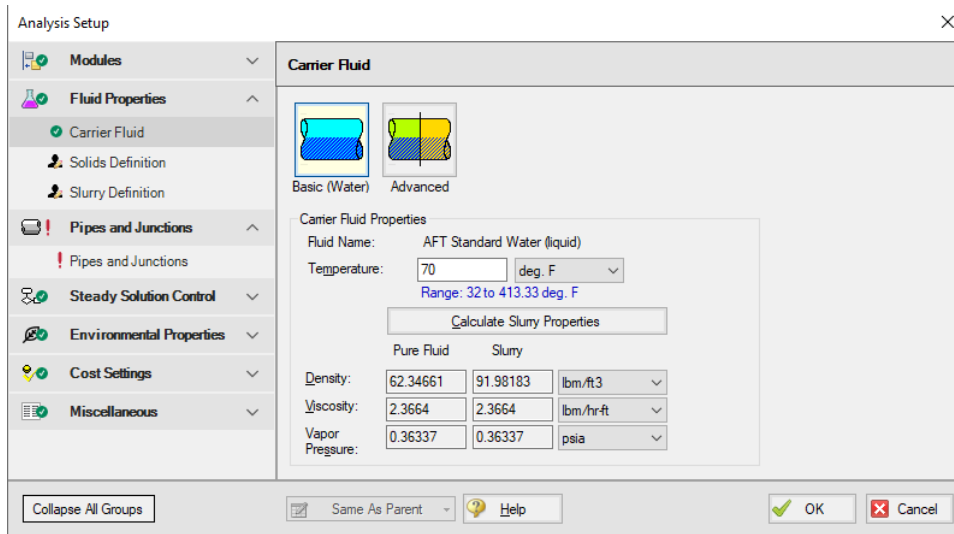   d. Name Alias = Water slurry

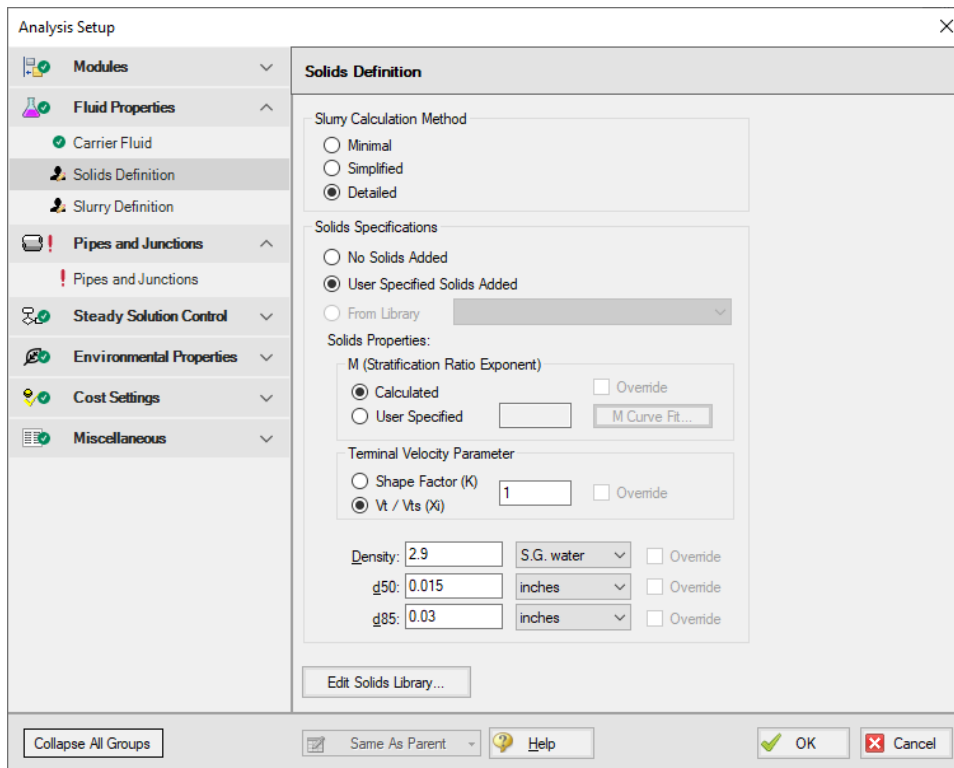*Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated*

Figure 2: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated
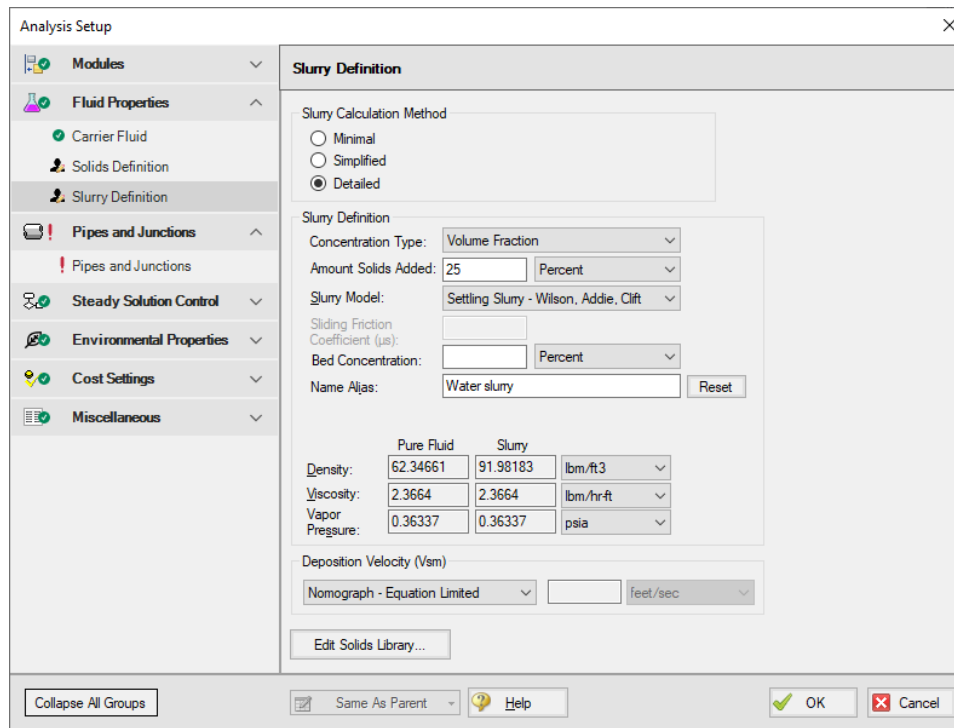
Figure 3: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
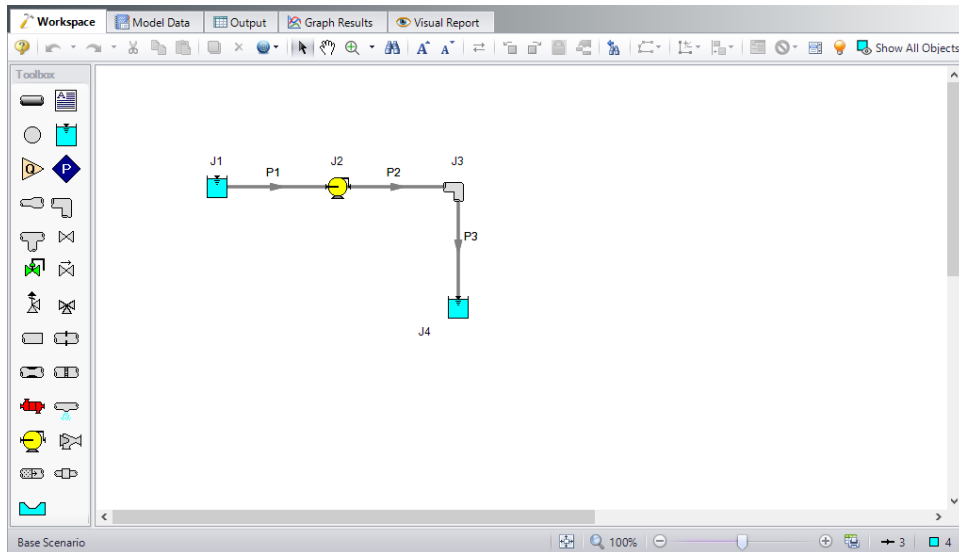
Figure 4: Layout of pipe system for Pump Sizing for Sand Transfer System example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 12 inch
    d. Type = STD
    e. Friction Model Data Set = Standard
    f. Length =

Pipe	Length (feet)
1	20
2	500
3	50

Junction Properties

1. J1 Reservoir
    a. Name = Supply Tank
    b. Liquid Surface Elevation = 5 feet

  c. Liquid Surface Pressure = 10 psig

  d. Pipe Depth = 5 feet

2. J2 Pump

  a. Inlet Elevation = 0 feet

  b. Pump Model tab

    i. Pump Model = Centrifugal (Rotodynamic)

    ii. Analysis Type = Sizing

    iii. Parameter = Mass Flow Rate

    iv. Fixed Flow Rate = 1400 ton/hr Solids

  c. Slurry De-Rating tab

    i. De-Rating Method = ANSI/HI Standard 12.1-12.6-2016

    ii. Percent Fines = 0 %

    iii. Head De-Rating Data = Specify Impeller Size Directly

    iv. Impeller Size = 8 inches

3. J3 Elbow

  a. Inlet Elevation = 0 feet

  b. Type = Standard Elbow (knee, threaded)

  c. Angle = 90 Degrees

4. J4 Reservoir

  a. Name = Receiving Tank

  b. Liquid Surface Elevation = 10 feet

  c. Liquid Surface Pressure = 30 psig

  d. Pipe Depth = 10 feet

To show the names of the reservoirs, a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.

2. Name the layer Reservoir Labels.

3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.

4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.

5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.

6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.

7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.

8. Close the Layer Settings window.

➢**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Open the **Output Control** window by selecting Output Control from the Output toolbar or Tools menu.

On the Display Parameters tab, click the **Pumps** button, and add the De-rating Head Correction factor (**CH**) to the Pump Summary output.

The Output Control window can also be used to configure the slurry output data, as is shown in Figure 5. Click OK when you are finished.
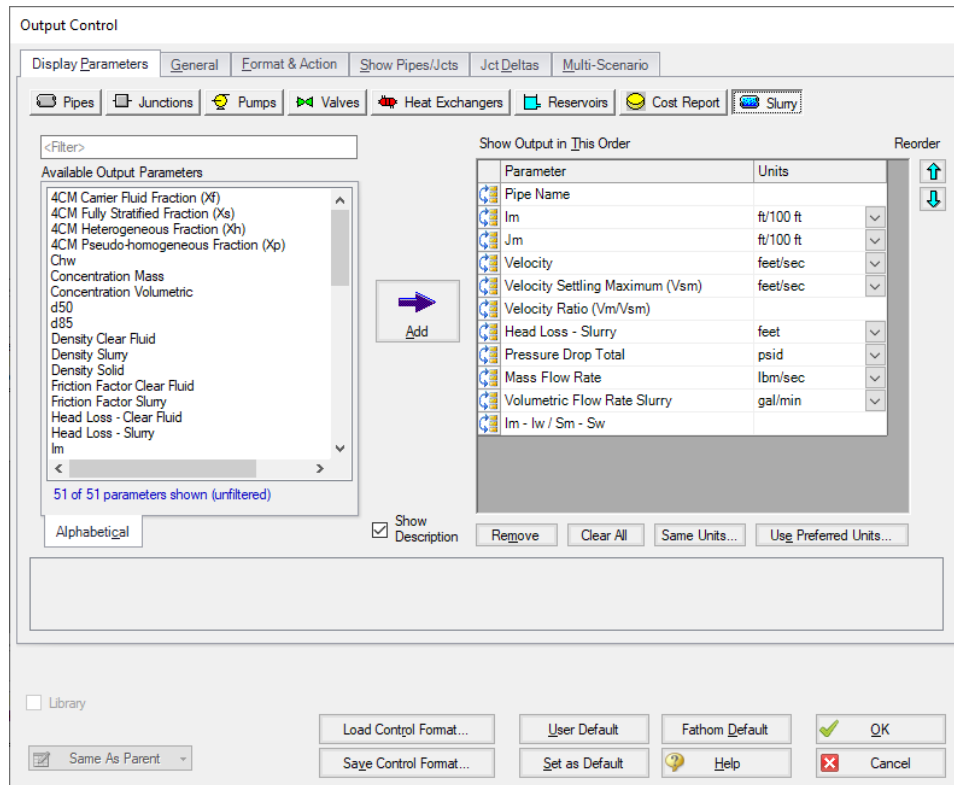

Figure 5: The Output Control window can be used to configure the slurry output data

The Slurry tab in the pipes section shows the slurry results, as shown in Figure 6. Here one can see that the velocity is 21.90 feet/sec, which is safely above the 11.48 feet/sec settling velocity. Other pressure drop and relevant slurry data can be viewed here.

Also shown in Figure 6 is the Pump Summary in the General Section. There the required pump head is shown as 93.98 feet, and the de-rating correction factor is 0.7917. Hence a pump should be selected that can generate 93.98/0.7917 = 118.7 feet of head based on water.

Figure 6: The Output window shows the Slurry results table in the center Pipes section and the pump de-rating in the Pump Summary in the upper General section

Also of interest is the system curve. Select the Graph Results window tab then in the Graph Control tab in the Quick Access Panel select the Slurry System Curve tab. See Figure 7.

By default the parameter to graph is Jm vs. Velocity. Select the units of Jm as ft/ft that will make it completely dimensionless as is typically done. Check the box for Water System Curve which cross-plots the water curve and then click Generate. The curves are shown in Figure 8.

Graph List Manager

My Graphs

Parameters   **Formatting >**

| Pump vs. System | Profile | Slurry System Curve |

Slurry System Curve

Flow Parameter:	Velocity	feet/sec
Pressure/Head Parameter:	Jm	ft/ft
Operating Point:	21.90 feet/sec	
Vsm:	11.48 feet/sec	
Range Start/End:	8.607	26.28	feet/sec
Data Points Per Curve:	30	
Water System Curve:	☑ Show	
Pump Curve:	☐ Show	

✓ Generate

Scenario   Graph Control

Overview Map   Add-on Modules

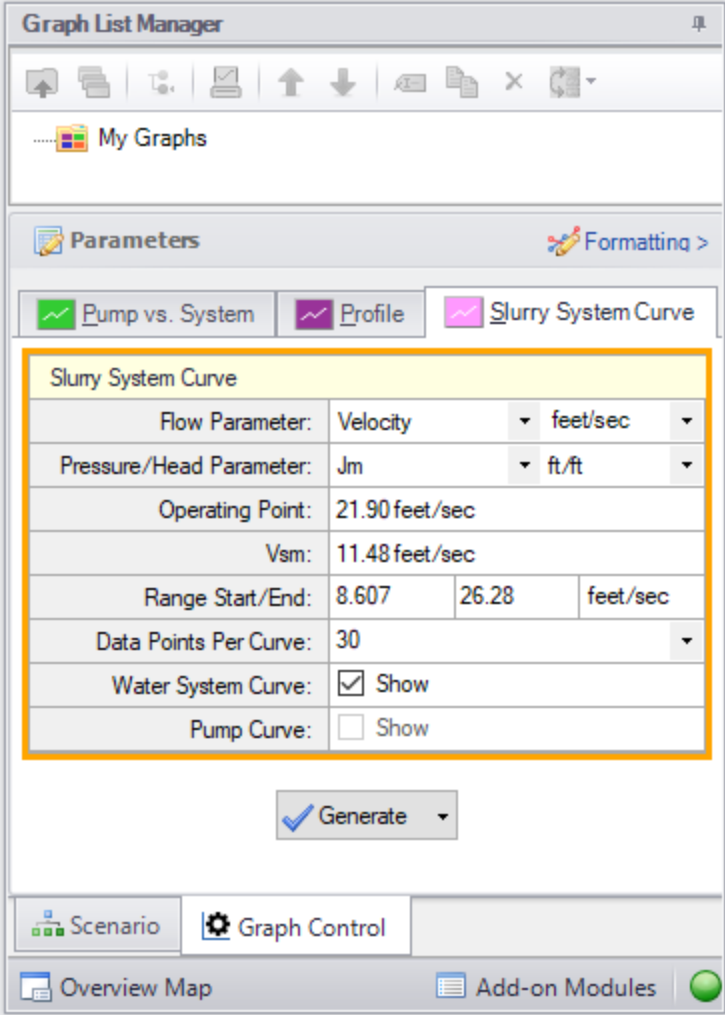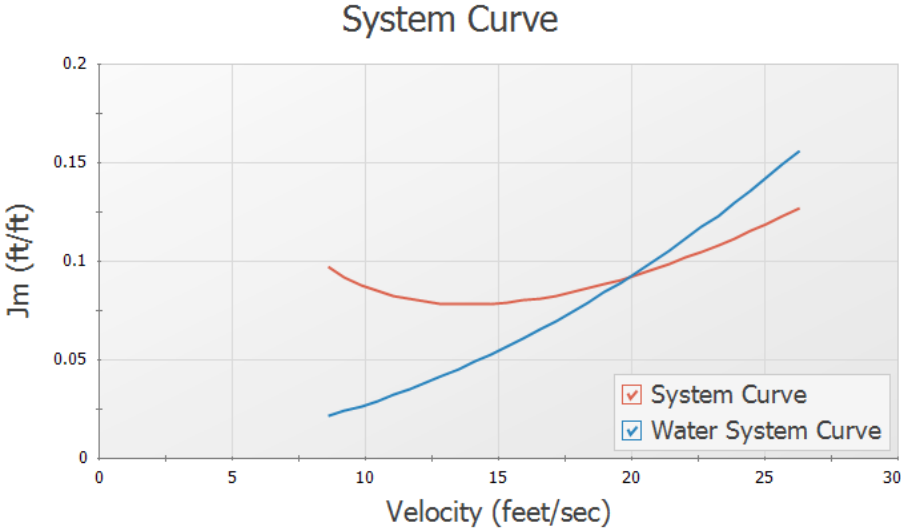*Figure 7: Data selection for graphing slurry system curve*

Figure 8: Slurry system curve and water curve in Graph Results window

A slurry of sand and water flowing through 570 feet of pipe was modeled. Total pipe head loss was determined and pump head requirements were calculated. A slurry system curve was created to allow the user to see where the system was operating with respect to solids settling.

Slurries with Variable Fluid Properties - SSL

Summary

This example demonstrates the fundamental concepts of the Settling Slurry (SSL) add-on module by way of example. The example shows how SSL can be used to model the introduction of solids into a clear fluid stream.

Evaluate pump and system performance for a slurry system that will pump 30% sand by volume for three cases:

1. Pumping clear water (no solids in system)
2. Solids introduced into the system, and propagated through to the pump discharge
3. Solids have propagated through the entire system

This example is essentially modeling four snapshots in time as the slurry is injected and pumped through the pipeline. These four snapshots in time can be modeled through the usage of the Volume Balance junction.

Note: This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Using Volume Balance junctions with variable fluid properties
- Creating slurry system curves and evaluating pump performance

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through Beginner - Pump Sizing for Sand Transfer System - SSL example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- **US - Slurries with Variable Fluid Properties - SSL.fth**

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid - Figure 1
   a. Fluid Model = Advanced
   b. Fluid Library = AFT Standard
   c. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   d. Temperature = 35 deg. F
   e. Select Variable Fluid Properties
2. Solids Definition - Figure 2
   a. Slurry Calculation Method = Simplified
   b. Solids Specifications = User Specified Solids Added
   c. M (Stratification Ratio Exponent) = User Specified
   d. Value = 1.7
   e. Density = 2.65 S.G. water
   f. d50 = 0.016 inches
3. Slurry Definition - Figure 3
   a. Slurry Calculation Method = Simplified (this selection will be the same as on the Solids Definition panel)
   b. Concentration Type = Volume Fraction
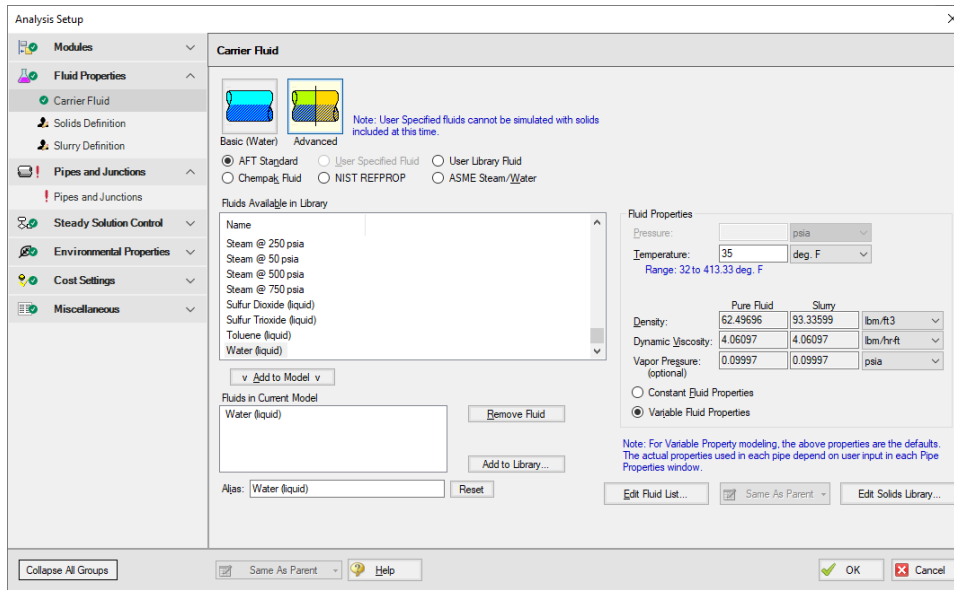   c. Amount Solids Added = 30 Percent

Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated

Figure 2: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Figure 3: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Figure 4: Layout of pipe system for Pump Sizing for Sand Transfer System example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
    a. Pipe Material = Steel - ANSI
    b. Pipe Geometry = Cylindrical Pipe
    c. Size = 6 inch
    d. Type = STD (schedule 40)
    e. Friction Model Data Set = Standard
    f. Length =

Pipe	Length (feet)
1	50
2	1000

Junction Properties

1. J1 & J3 Assigned Pressures
   a. Elevation = 0 feet
   b. Pressure = 150 psig
   c. Pressure Specification = Stagnation
2. J2 Pump
   a. Inlet Elevation = 0 feet
   b. Pump Model = Centrifugal (Rotodynamic)
   c. Analysis Type = Pump Curve
   d. Enter Curve Data =

Volumetric	Head	Efficiency
gal/min	feet	Percent
0	200	-
500	188	45
800	175	64
1100	155	75
1400	130	77
1700	100	74

   e. Curve Fit Order = 2

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Create the Scenarios

For this example, there are four cases to be examined. Create a scenario for each of the four cases, as shown in Figure 5:
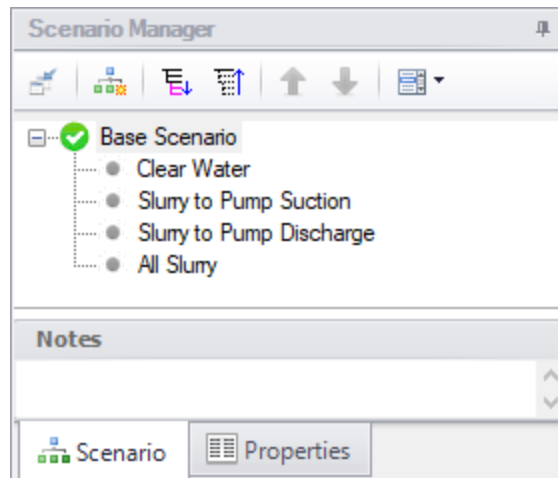
Figure 5: Use the Scenario Manager on the Quick Access Panel to create the four analysis cases

Clear Water Scenario

Load the **Clear Water** scenario by double-clicking the scenario name in the Scenario Manager.

This case requires that the pipes use clear water as their fluid. As pipes are drawn on the workspace, they automatically assume the same fluid type that is defined in the Analysis Setup window. As a result, at this point, all of the pipes are defined as using the sand slurry. Reset **both** pipes to use only water by opening the **pipe properties window** and inputting the following:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.
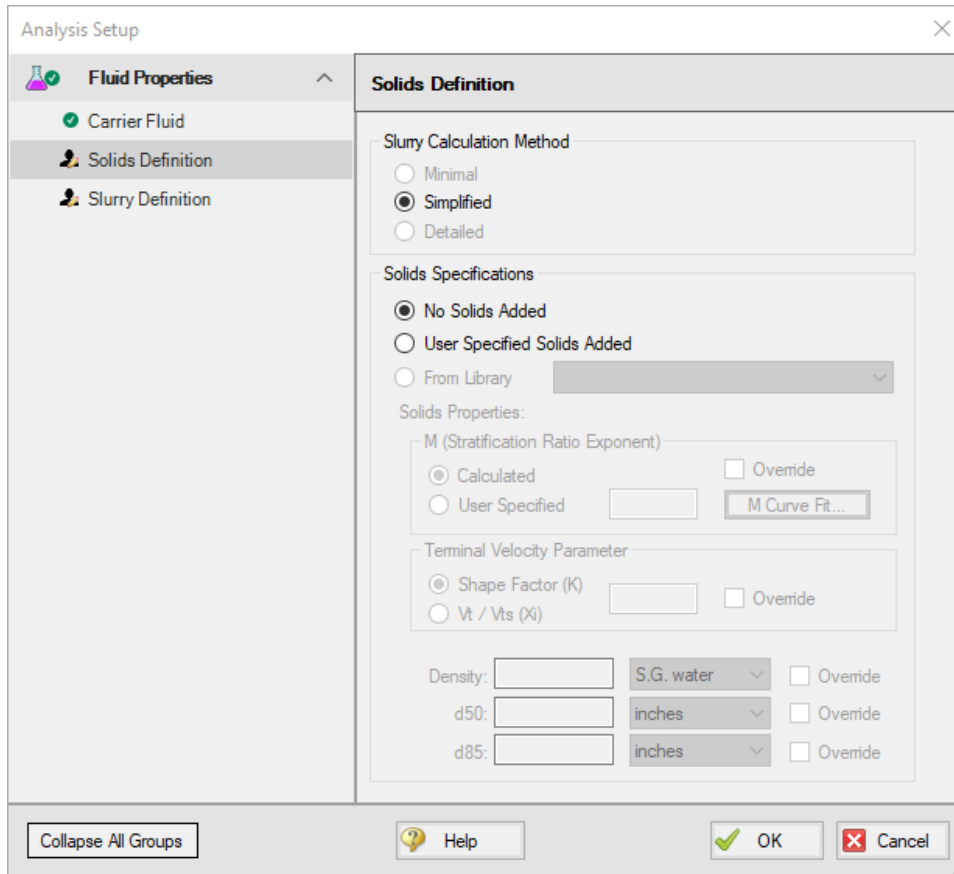5. Make sure to do this to both pipes.

Figure 6: Fluids can be treated as clear fluid by selecting No Solids Added as the Solids Specification

Slurry to Pump Suction Scenario

Now load the **Slurry to Pump Suction** as the active scenario in the Workspace.

For the case where the fluid far upstream of the pump suction is slurry, and the fluid just upstream of the pump is clear water, a Volume Balance junction must be used to account for the difference in fluid density. The Volume Balance junction balances the flow based on volume rather than mass, ensuring that the velocity across the interface is the same. This junction must be used any time there is a system interface with fluids of different densities.

Add a Volume Balance junction just upstream of the pump suction by splitting the pipe as follows:

1.   While holding the **Shift** key drag and drop the **Volume Balance junction** over pipe **P1**.
2.   In the dialogue box set Pipe A as **49.9 feet**, then select **OK**. The Workspace should now appear as shown in Figure 7. This will insert the fluid interface shortly before the pump suction.
3.   Open the **Volume Balance properties window** for junction J4 and check that the elevation is **0 feet**.

This case requires that the short pipe just upstream and the pipe downstream of the pump use clear water as their fluid. Set **both** pipes P2 and P3 to use only water as follows:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.
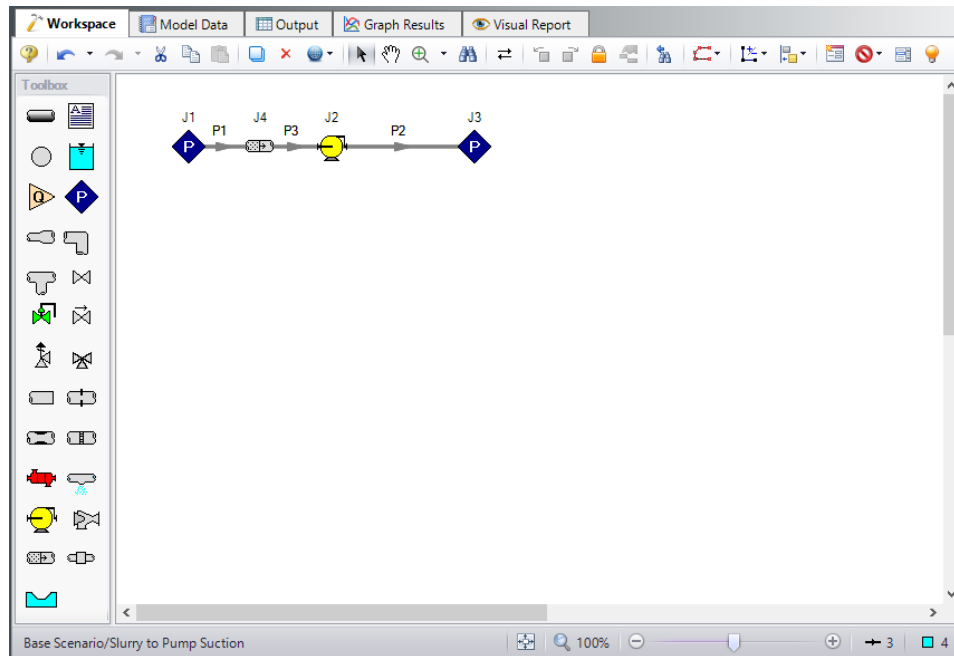5. Make sure to do this to **both** pipes.

Figure 7: Layout of pipe system for partial slurry case with slurry at the pump suction showing the Volume Balance junction for the Slurries with Variable Fluid Properties example

Slurry to Pump Discharge Scenario

Load the **Slurry to Pump Discharge** scenario as the active scenario.

For the case where the fluid upstream and just downstream of the pump discharge is slurry, and the rest of the system downstream is clear water, a Volume Balance junction must be used again to account for the difference in fluid density.

Add a Volume Balance junction just downstream of the pump discharge by splitting pipe 2.

1. While holding the **Shift** key drag and drop the **Volume Balance junction** over pipe **P2**.
2. In the dialogue box set Pipe A as **0.1 feet**, then select **OK**. The Workspace should now appear as shown in Figure 8. This will insert the fluid interface shortly after the pump discharge.
3. Open the **Volume Balance properties window** for junction J5 and check that the elevation is **0 feet**.

This case requires that the pipe P4 downstream of the pump use clear water as its fluid. Set pipe **P4** to use only water as follows:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.
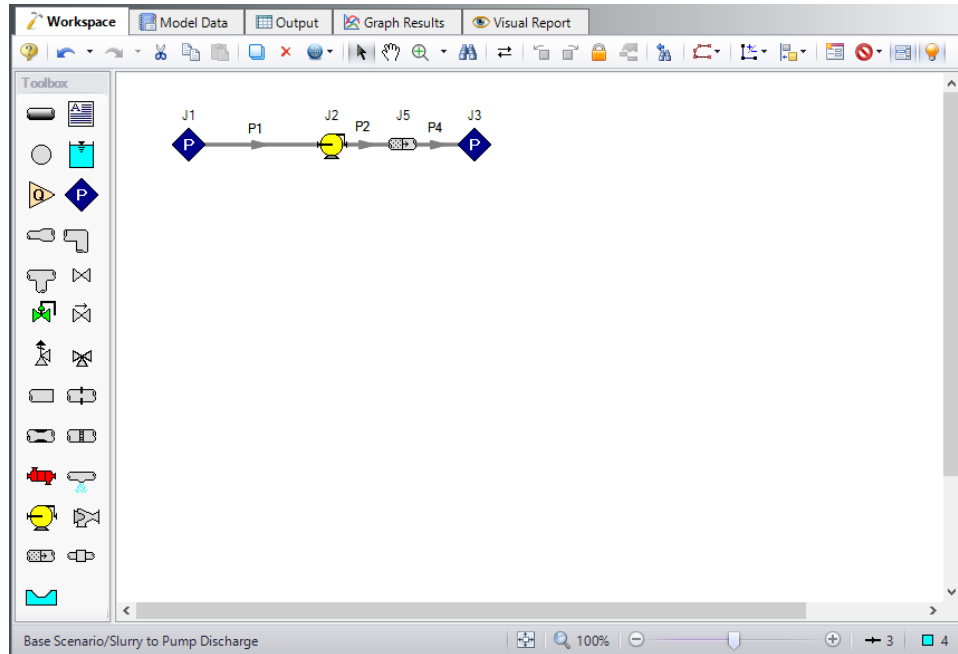
Figure 8: Layout of pipe system for partial slurry case with slurry at the pump discharge showing the Volume Balance junction for the Slurries with Variable Fluid Properties example

All Slurry Scenario

For this case, all of the pipes contain the sand slurry. The system layout is the same as is shown in Figure 4.

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Note: All scenarios can be run automatically by Fathom by completing a Batch Run from the File menu.

Step 7. Examine the Output

Table 1 shows the variation of the system flows, velocities, and pump performance as the slurry progresses through the system.

Figure 9 shows the pump curve plotted along with both the water and slurry system curves. This graph was generated in the All Slurry scenario using the Slurry System Curve graph type.

As the slurry progresses from the system inlet to the pump suction, the system volumetric flow rate, pump head, and power are not significantly changed (compare Table 1 output for Clear Water to Slurry to Pump Suction).

After the slurry arrives at the pump discharge (compare Table 1 output for Slurry to Pump Suction to Slurry to Pump Discharge), there is a notable increase in the system volumetric flow rate (1,401 to 1,595 gal/min). This is a result of the new density passing into the pump, which results in a different velocity in the pipe line (15.56 to 17.71 feet/sec), a different pressure loss (which varies as velocity squared, and the velocity is now higher), and hence a different head rise on the pump. This shifts the pump operation out on the curve to a higher flow rate and lower head (130.1 to 111.2 feet).

Eventually the slurry fills the entire pipe (compare Table 1 output for Slurry to Pump Discharge to All Slurry). Because of the increased overall head loss of the slurry compared to water, the system shifts back on the pump curve to a lower flow rate (1,595 to 1,497 gal/min) and higher head rise (111.2 to 121.0 feet). The system operating velocity also drops (17.71 to 16.63 feet/sec).

The power requirement on the pump increases from the Clear Water to All Slurry scenario as the system progresses from all water to all slurry.

Of significance here is the fact that, perhaps counter to intuition, the clear water and all slurry cases do not encompass the entire flow and system performance cases for the pump. The highest and lowest flow cases actually occur as the system is only partially full of slurry. Care should be taken to examine intermediate slurry flow cases to determine system feasibility for situations where slurry is being introduced into the flow, as the maximum and minimum flow cases will not necessarily occur when the system is flowing all clear fluid, or all slurry.

It is difficult to quantify the intermediate cases through the use of pump vs. system curves. This is a result of the variation in density that occurs in the system, which confuses the meaning of Head which itself depends on constant density. Thorough analysis is needed in these cases. Figure 9 shows the clear water and all slurry system curves.

Finally, note that to keep the example clearer, the effect of pump de-rating was not addressed. In real systems when the slurry progresses from the Slurry to Pump Suction case to the Slurry to Pump Discharge case, there will be a further decrease in pump performance which will effectively lower the pump curve in Figure 9. This would serve to shift operation to even higher flow rates for the Slurry to Pump Discharge and All Slurry cases.

Table 1: System flow and pump performance for different scenarios. The velocity and volumetric flow results are for the inlet of the pipe at the pump discharge

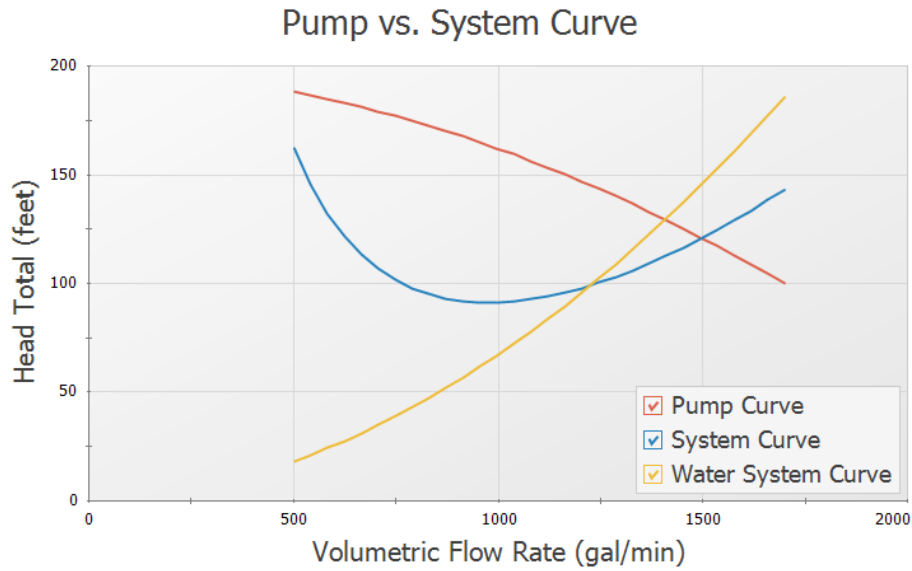| Parameter	Vol. Flow Rate	Velocity	dP	dH	Overall Power
Units	gal/min	feet/sec	psid	feet	hp
Clear Water	1,408	15.63	56.20	129.5	59.25
Slurry to Pump Suction	1,401	15.56	56.46	130.1	59.24
Slurry to Pump Discharge	1,595	17.71	72.07	111.2	88.20
All Slurry	1,497	16.63	78.42	121.0	88.54

Figure 9: Plotting the pump curve along with the water and slurry system curves illustrates how the pump performance will vary as the slurry progresses through the system

Slurry System Feasibility Study - SSL (Metric Only)

Summary

Note: Because this example is presented in SI units in the reference from which it was taken, it is presented in the same units here for consistency when discussing the results.

This example demonstrates how the Settling Slurry (SSL) add-on module can be used to evaluate the feasibility of a slurry system operating point. The example is based on Case Study 13.1 found on page 329 of *Slurry Transport Using Centrifugal Pumps*, 3rd Edition, by Wilson, Addie, Sellgren, and Clift. Here only the 0.65 m case is evaluated.

An operating condition from a previous case study was selected:

Condition A:

D = 0.65 meters

Vsm = 4.84 meters/sec

Cvd = 0.20

Vm = 6.30 meters/sec

It was noted in the previous study that increasing Cvd and decreasing Vm reduced the specific energy consumption, and a revised operating point was also selected:

Condition B:

D = 0.65 meters

Vsm = 4.84 meters/sec

Cvd = 0.221

Vm = 5.70 meters/sec

Using these two operating points, determine whether either operating point would be feasible for the system.

Note: This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Reviewing calculated slurry output
- Evaluating stability of system operability

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through Beginner - Pump Sizing for Sand Transfer System - SSL example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Slurry System Feasibility Study - SSL.fth*
- *Slurry System Feasibility Study - SSL.xlsx*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid - Figure 1
   a. Fluid Model = Basic (Water)
   b. Temperature = 4 deg. C
2. Solids Definition - Figure 2
   a. Slurry Calculation Method = Detailed
   b. Solids Specifications = User Specified Solids Added

    c.   M (Stratification Ratio Exponent) = Calculated

    d.   Terminal Velocity Parameter = Vt / Vts (Xi)

    e.   Value = 0.56

    f.   Density = 2.65 S.G. water

    g.   d50 = 0.7 mm

    h.   d85 = 1 mm

3.   Slurry Definition - Figure 3

    a.   Slurry Calculation Method = Detailed (this selection will be the same as on the Solids Definition panel)

    b.   Concentration Type = Volume Fraction
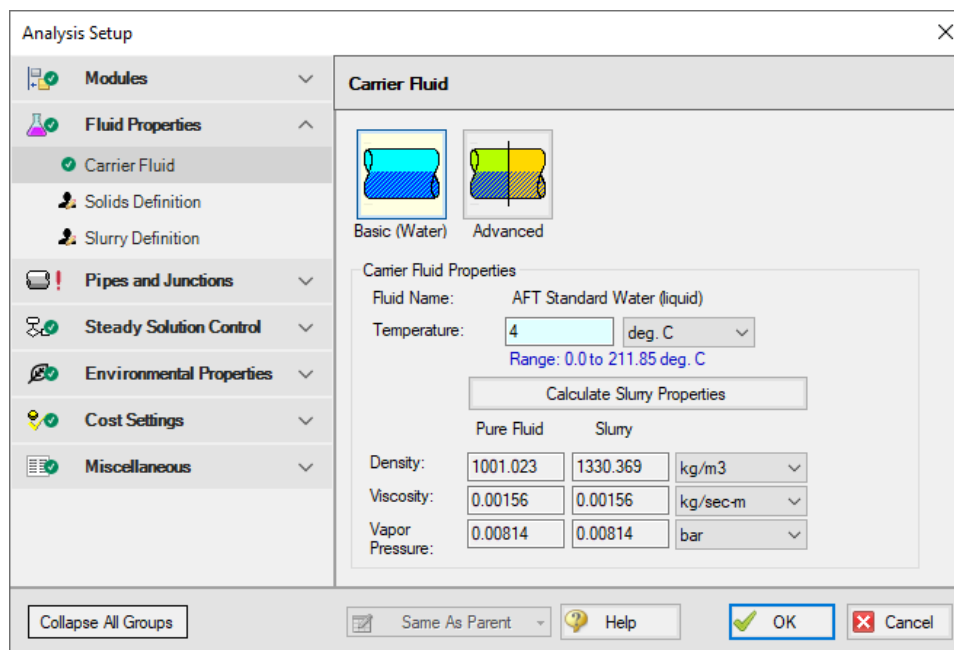
    c.   Amount Solids Added = 20 Percent

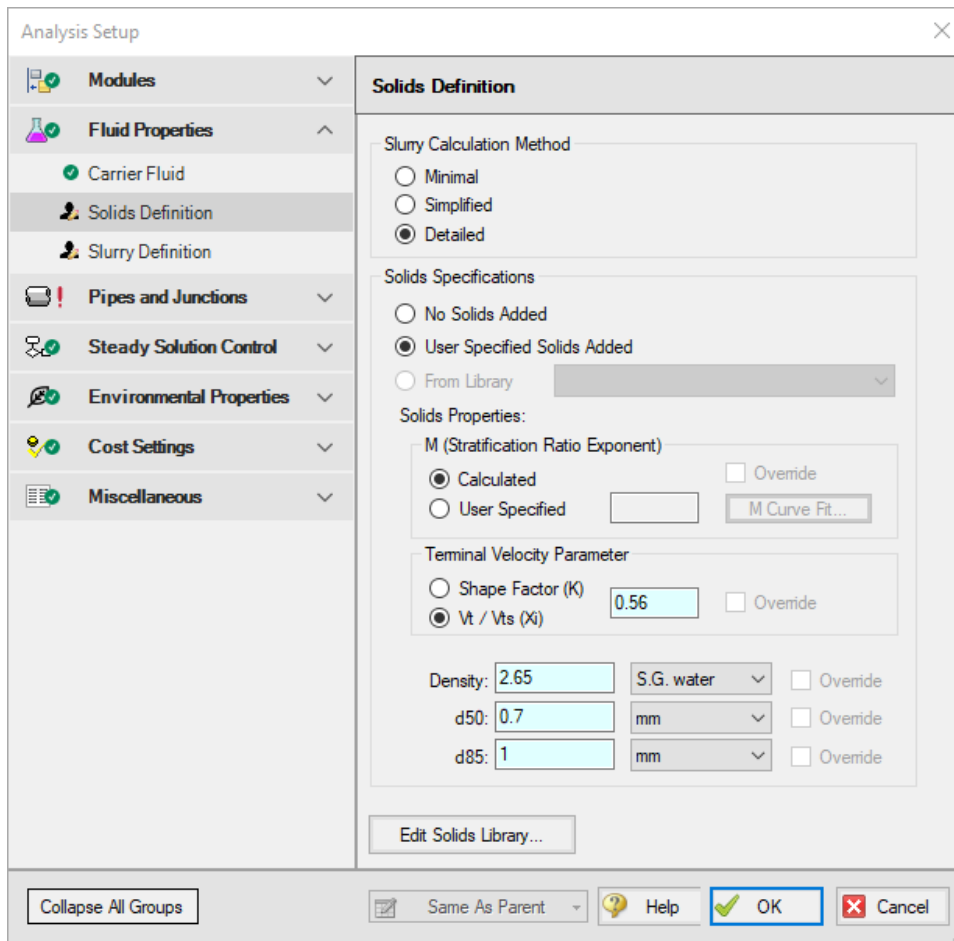*Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated*

Figure 2: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated
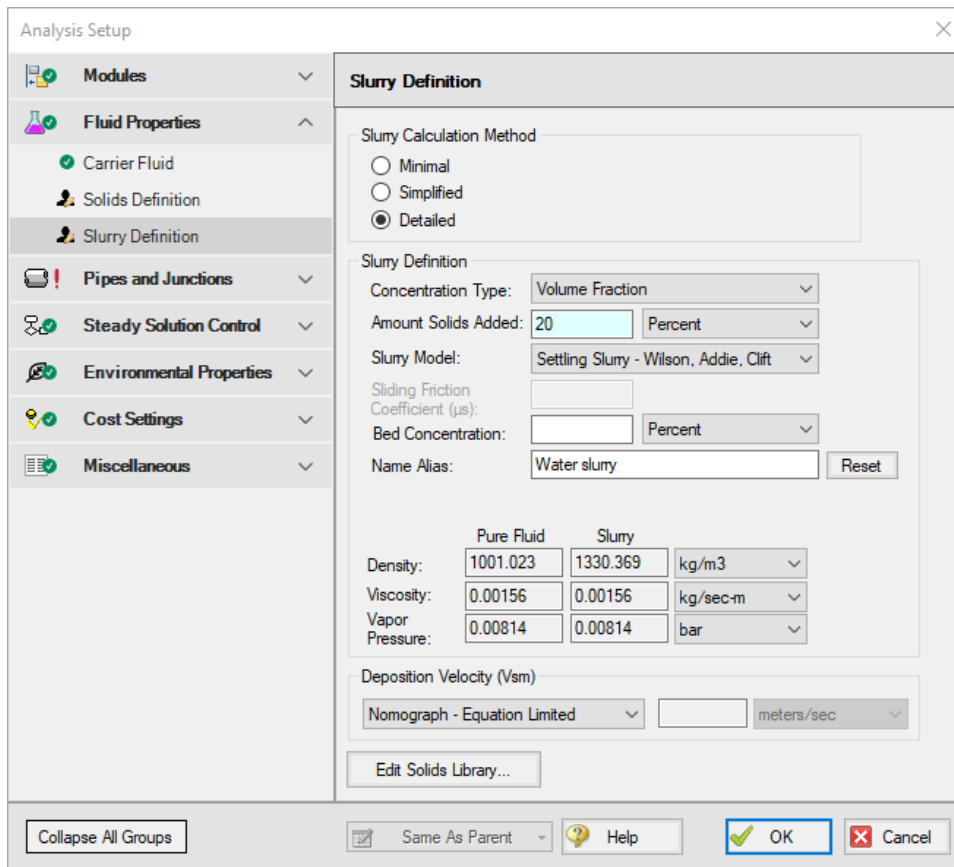
Figure 3: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
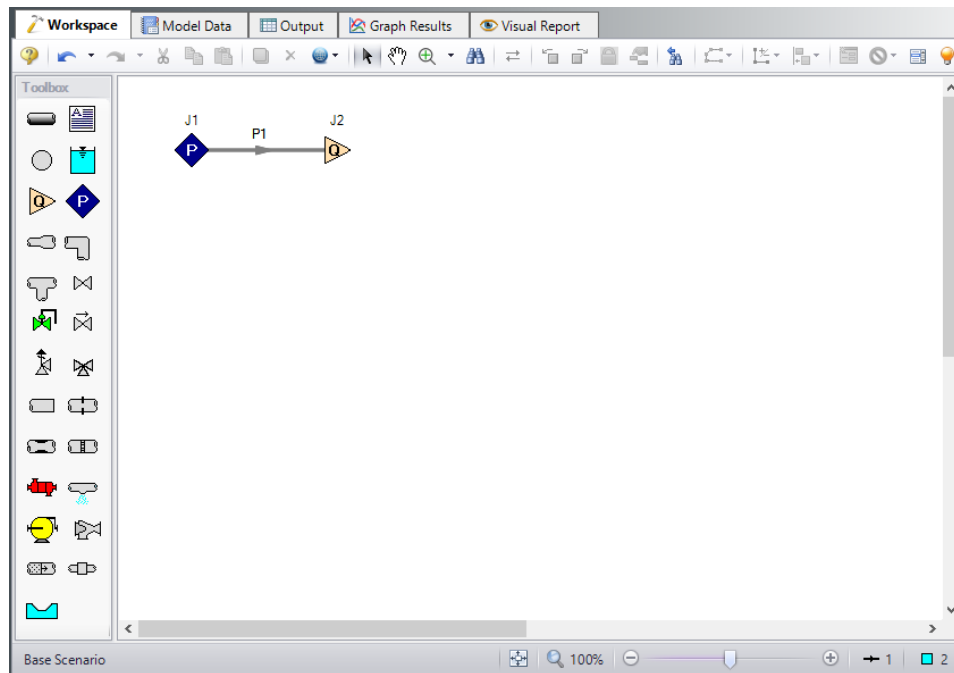
Figure 4: Layout of pipe system for Slurry System Feasibility Study example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
    a. Pipe Material = (User Specified)
    b. Pipe Geometry = Cylindrical Pipe
    c. Inner Diameter = 0.65 meters
    d. Friction Model Data Set = User Specified
    e. Friction Factor = Explicit Friction Factor
    f. Value = 0.012
    g. Length = 100 meters

Junction Properties

1. J1 Assigned Pressure
    a. Elevation = 0 meters
    b. Pressure = 10 bar
    c. Pressure Specification = Stagnation
2. J2 Assigned Flow

a.   Elevation = 0 meters

b.   Type = Outflow

c.   Flow Specification = Mass Flow Rate

d.   Flow Rate = 4000 m-ton/hr Solids (metric tons equivalent to 4.0 x 10^6 kg/hr)

    i.   (metric tons equivalent to 4.0 x 10^6 kg/hr)

➤**Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The maximum settling velocity for Condition A calculated by Fathom is 4.833 meters/sec, as shown in Figure 5. The value for the maximum settling velocity for Condition B can be reproduced in a similar fashion using Fathom.

Pipe	Name	Im (m/100 m)	Jm (m/100 m)	Velocity (meters/sec)	Settling Velocity Maximum (meters/sec)	Vm/Vsm	dH Slurry (meters)	dP (bar)	Mass Flow Rate (kg/sec)	Vol Flow Slurry (m3/hr)	(Im - Iw) / (Sm - Sw)
1	Pipe	6.120	4.596	6.323	4.833	1.308	4.596	0.6010	2,791	7,554	0.07152

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	9.734	9.734	10.000	10.000	7,554	2,791	0
2	Assigned Flow	9.133	9.133	9.399	9.399	7,554	2,791	0

Figure 5: Output for Condition A calculation showing maximum settling velocity

Step 7. Evaluate System Operability

To determine the feasibility of the selected operating conditions, it is helpful to examine how the operating points relate to the system operability. One method of evaluating the system operability is to plot the system curves in terms of head of slurry (Jm vs. velocity). It is useful to plot the system curves as lines of constant concentration, as well as lines of constant slurry throughput.

The system curves for a given slurry throughput and concentration can be plotted directly by using the graphing capabilities of Fathom. However, to examine the system operability over a range of flows and concentrations, it may be more convenient to use a spreadsheet application, such as Microsoft Excel, to plot the system data all together. Fathom's scenario manager and multi-scenario output features can be utilized to facilitate this process.

Evaluating the system in this manner may require a large number of scenarios to sufficiently capture the system operability. For your convenience, all of the necessary cases have been set up in the Fathom model file ***Metric - Slurry System Feasibility Study - SSL.fth***.

Steps 8 through 12 describe how the cases were set up in Fathom, and then copied into a spreadsheet for evaluation, should you wish to build the evaluation cases for yourself. The final evaluation of the operating points is presented in Step 13.

Step 8. Creating Scenarios to Evaluate System Operability

For this example, scenarios can be used to calculate Jm and velocity data for three different lines of constant concentration, as well as for three different lines of constant throughput.

Setup the scenarios for constant concentrations as follows:

1. From the Scenario Manager on the Quick Access Panel, create a scenario under the Base Scenario called *0.65 meters pipe*.
2. Under the *0.65 meters pipe* scenario, create a scenario called *Cvd = 0.15*. Load this scenario, and open the Analysis Setup window. Set the solids concentration to 15% in the Slurry Definition panel.
3. Under the *Cvd = 0.15* scenario, create scenarios for a range of mass flow rates from 1,000 m-ton per hr to 10,000 m-ton per hr in increments of 500 m-ton per hr, as shown in Figure 6. Load each of these scenarios and set the flow value in each Assigned Flow junction to the corresponding throughput value for each scenario.
4. Right-click scenario *Cvd = 0.15* and select Clone With Children. Name the new scenario *Cvd = 0.20*. In this scenario, open Analysis Setup and change the solids concentration to 20%. Repeat this process, and create a set of scenarios for *Cvd = 0.34*.
5. There are now three groups of scenarios, each for a different solids concentration. Each of the three concentrations will have a set of scenarios for a range of different through puts, as shown in Figure 7.
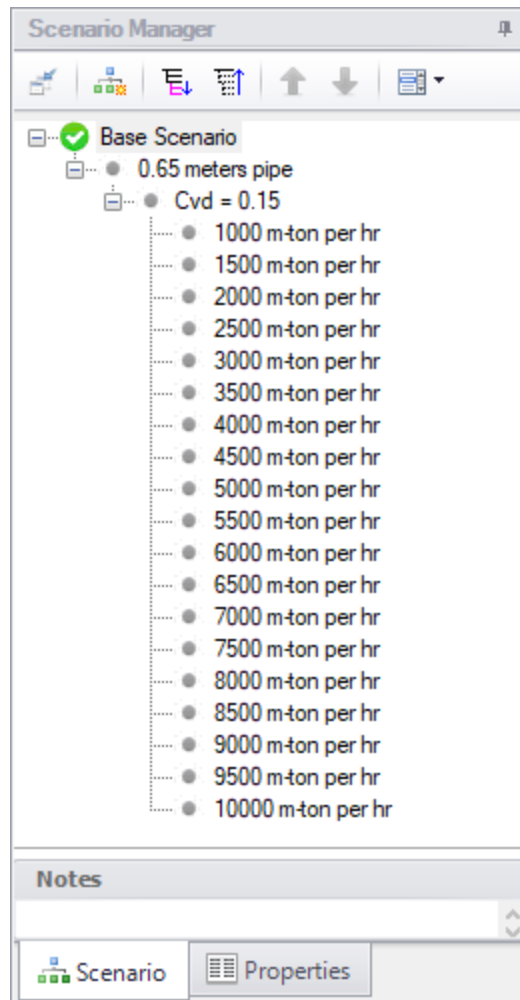
Figure 6: The Scenario Manager on the Quick Access Panel can be used to set up multiple throughput cases for a given solids concentration
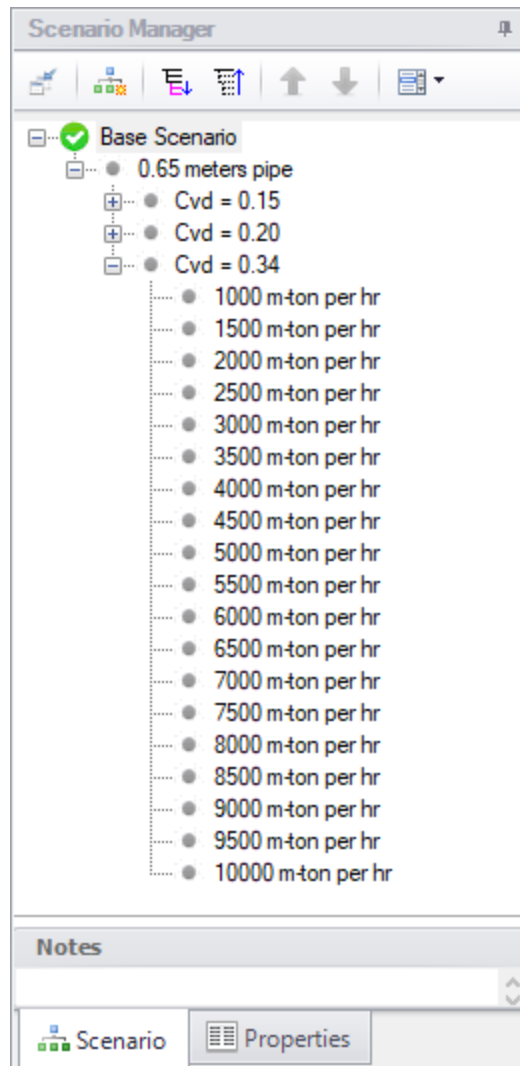
Figure 7: The Scenario Manager on the Quick Access Panel can be used to clone scenarios to easily create additional constant solids concentration cases

Setup the scenarios for constant throughput as follows:

1. Under the *0.65 meters pipe* scenario, create a scenario called *Flow = 3500 m-ton per hr*. Load this scenario, and set the flow value in the Assigned Flow junction 3500 m-ton/hr.

2. Under the *Flow = 3500 m-ton per hr* scenario, create scenarios for a range of solids concentrations, as shown in Figure 8. Load each of these scenarios and set the solids concentration value in the Slurry Definition panel in Analysis Setup to the corresponding solids concentration value for each scenario.

3. Right-click scenario *Flow = 3500 m-ton per hr* and select Clone With Children. Name the new scenario *Flow = 4000 m-ton per hr*. In this scenario, set the flow value in the Assigned Flow junction 4000 m-ton/hr. Repeat this process, and create a set of scenarios for *Flow = 4500 m-ton per hr*.

4. There are now three additional sets of scenarios, each for a different throughput value. Each of the three flows will have a set of scenarios for a range of different solids concentrations, as shown in Figure 9.
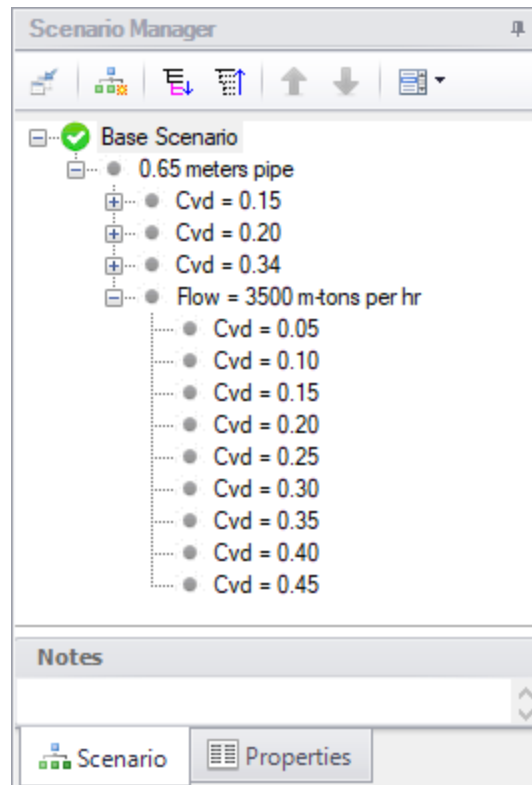

Figure 8: The Scenario Manager on the Quick Access Panel can be used to set up multiple solid concentration cases for a given throughput
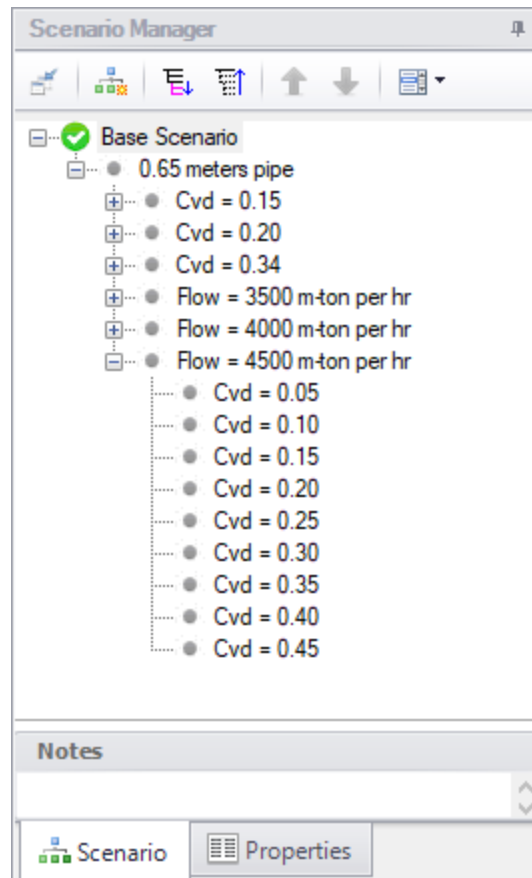
Figure 9: The Scenario Manager on the Quick Access Panel can be used to clone scenarios to easily create additional constant throughput cases

Step 9. Specify Output

Make sure that the Base Scenario is active, the open Output Control by selecting Output Control from the toolbar or Tools menu.

Knowing that for this example, the output data will be exported to a spreadsheet application, use the output control to set up the slurry output in a way that will make it simpler to plot the data in the spreadsheet. Open Output Control and on the Display Parameters tab, click the Slurry button. Then select the specific data to be displayed on the Slurry tab in the Output Window, as well as the order in which the data appears.

Figure 10 illustrates how the slurry output data could be organized in the Output Window. Because we will be plotting Jm vs. velocity in the spreadsheet, velocity and Jm are displayed next to each other in the output. One might also wish to plot Im vs. velocity, so velocity can be displayed again adjacent to the output data for Im.
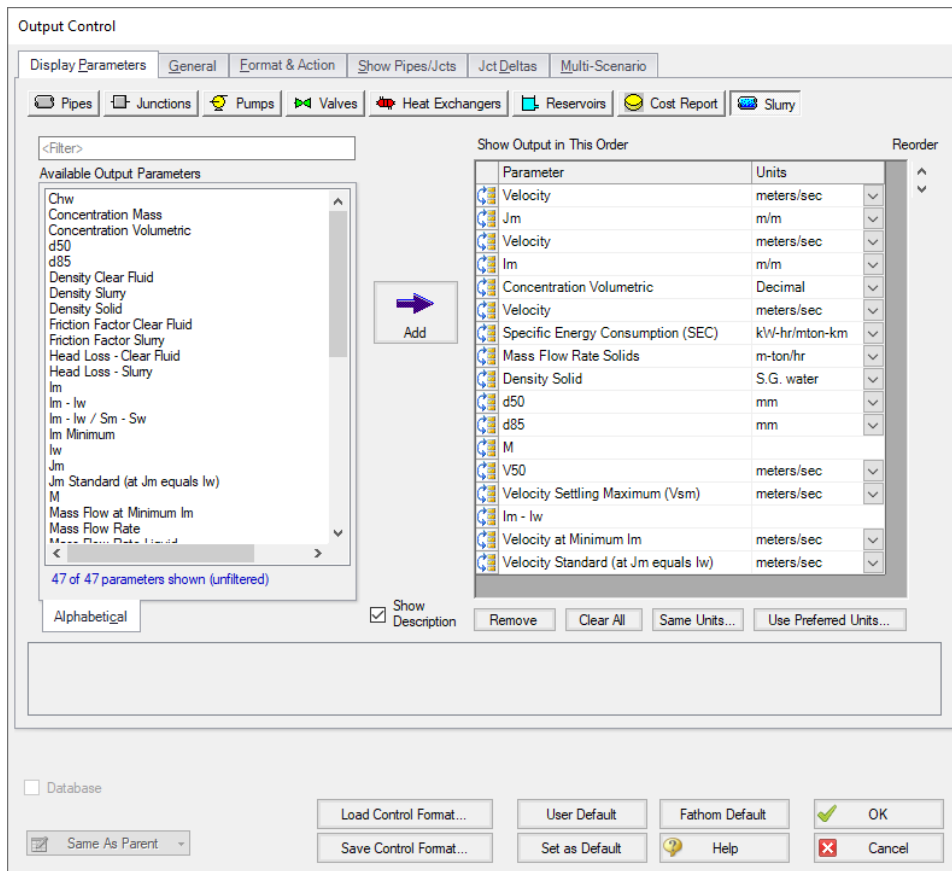
Figure 10: The Output Control window can be used to configure the slurry output into a format convenient for importing into a spreadsheet application

For this example, the multi-scenario output feature can also be used to simplify exporting the output data to a spreadsheet application. This feature allows the output from multiple scenarios to be viewed simultaneously. Load the *1000 m-ton per hr* scenario that is under the *Cvd = 0.15* scenario. Open the **Output Control** window and select the **Multi-Scenario** tab. Under Display Type, click **Selected Scenarios** and the Select Scenarios window appears. Check the box beside each scenario for which data is to be displayed in the Output, as shown in Figure 11. 84 of 92 scenarios should be selected. Click **OK** when finished.

Under Scenario Name Format, select **Show Scenario Ancestral Name**. Under Display Current Scenario, select **In the Order Listed**, if not selected already. Click the **Clear All Colors** button to make all scenario rows white.

This information should now appear, as shown in Figure 12. Close the Output Control window by clicking OK.
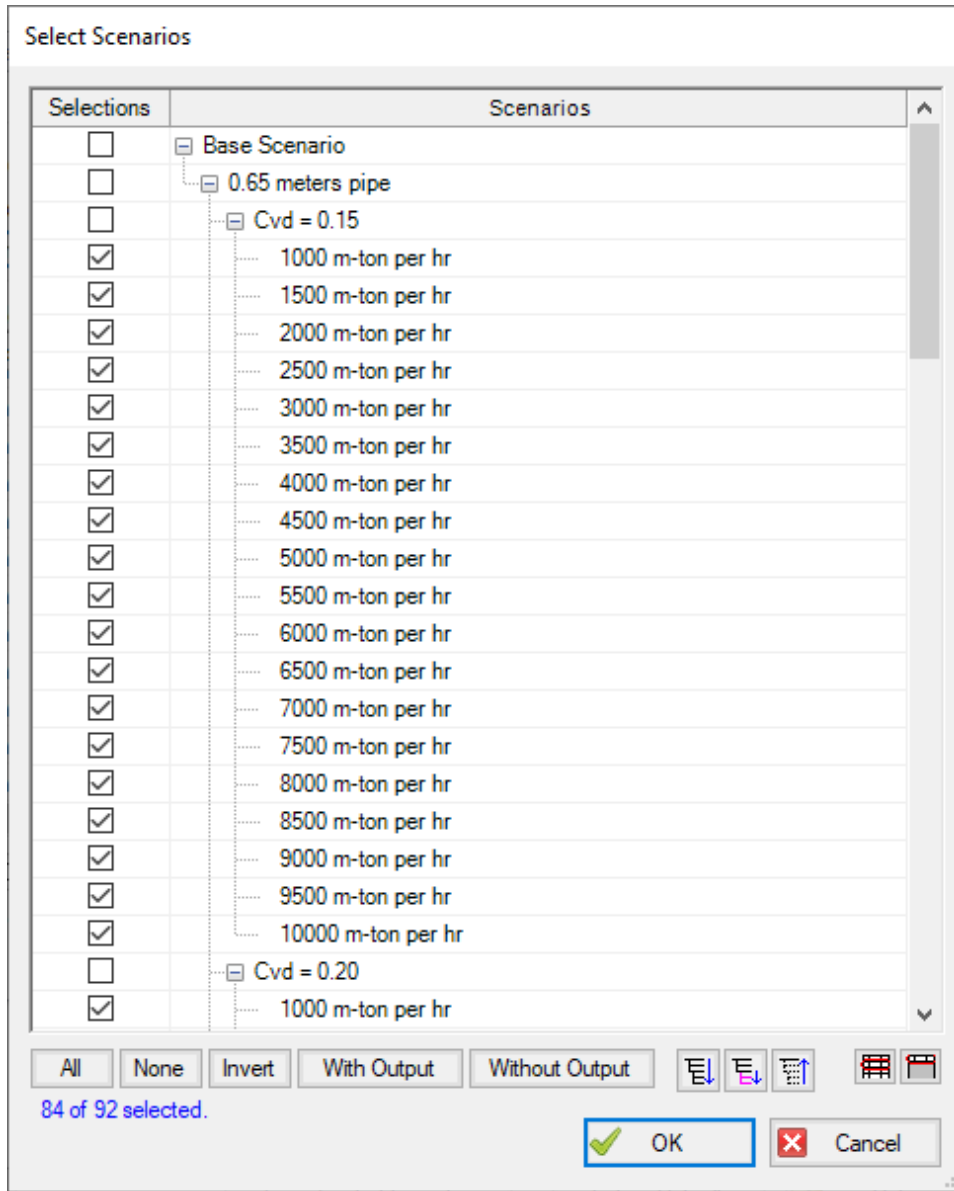
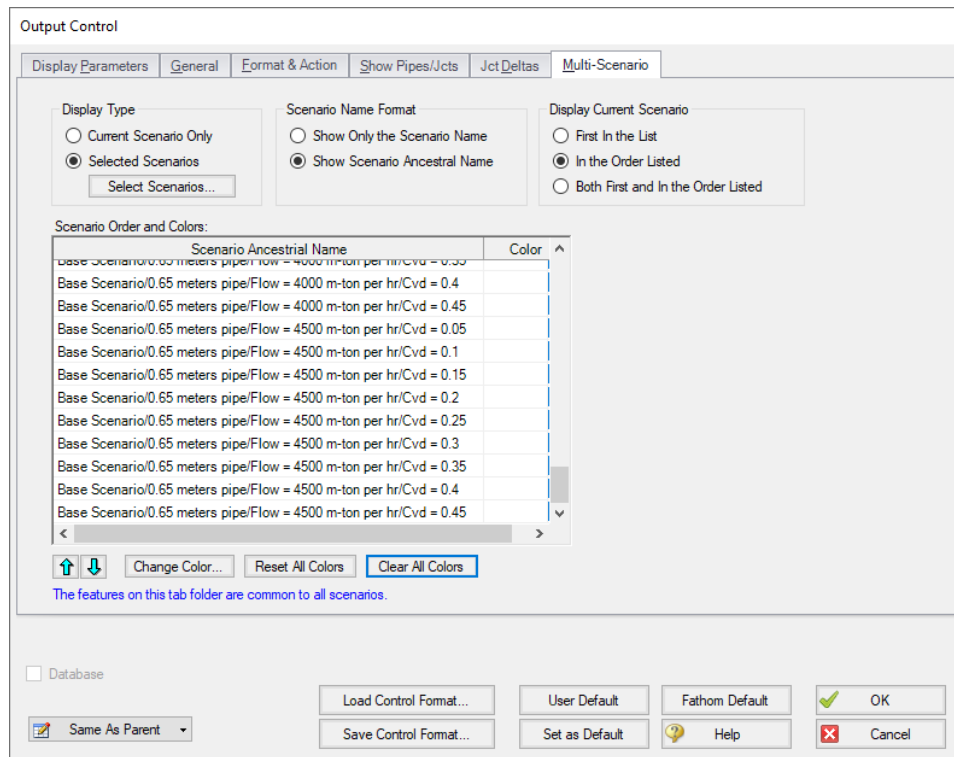*Figure 11: Multiple scenarios can be selected for display in the Output window*

Figure 12: The Output Control window can be used to display the output from multiple scenarios simultaneously

Step 10. Run the Model

All of the different cases that have been set up for this example in each of the scenarios must now be run so the results can be examined. Because there are so many scenarios, the Batch Run feature in the File menu can be used to make running all of the different scenarios much easier.

From the File menu, select Start Batch Run. Under Batch Run Type, Scenarios In Current Model should be selected. Click Add Scenarios, and in the Select Scenarios window, select all 84 of the scenarios to be run, then click OK. All of the selected scenarios will be displayed in the Scenarios to Run list, as shown in Figure 13.

Click Start Run to start the analysis. The batch run manager will load and run each of the listed scenarios sequentially.
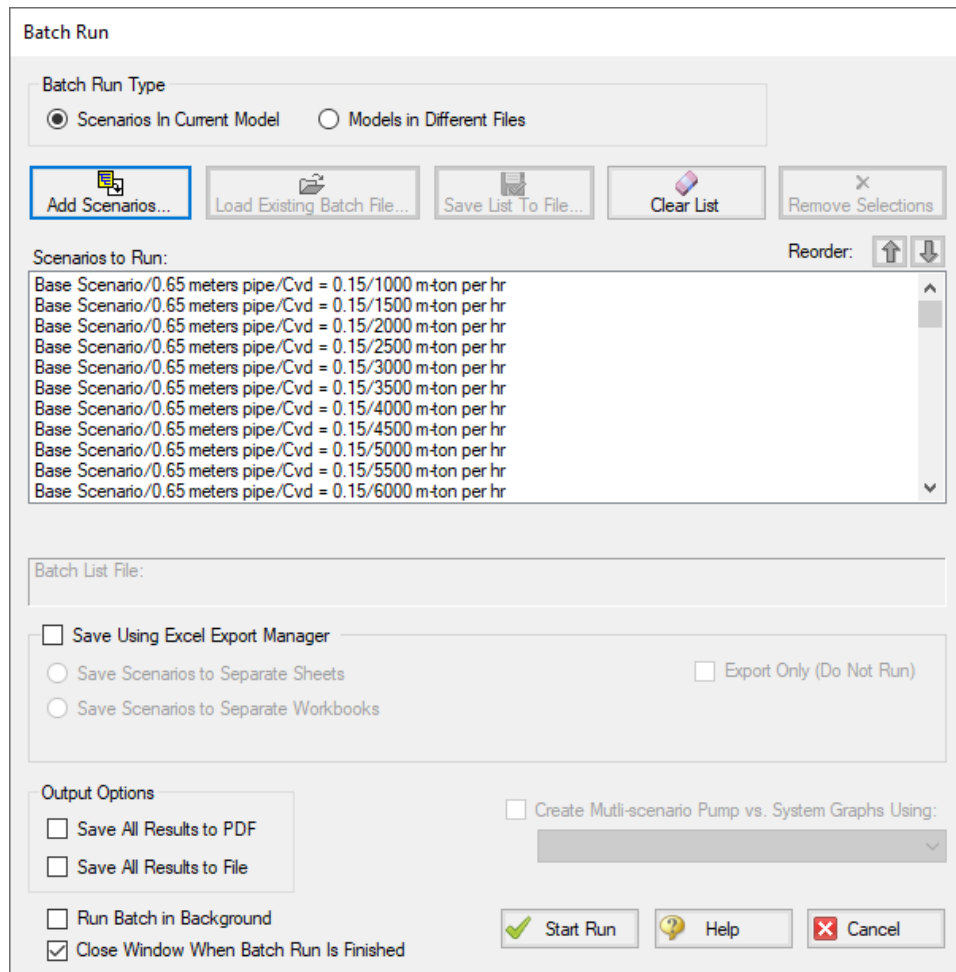
Figure 13: The Batch Run Manager is used to automatically run multiple scenarios

Step 11. Examine the Output

Once the batch manager has finished running all of the selected scenarios, load the scenario in which the multi-scenario output was set up (*1000 m-ton per hr* scenario that is under the *Cvd = 0.15* scenario).

The Output window contains all the data that was specified in the Output Control window. Because the multi-scenario output feature was selected, it will also display the calculated output for each of the selected scenarios simultaneously. Figure 14 shows the multi-scenario output on the Slurry tab.

Pipe	Velocity (meters/sec)	Jm (m/m)	Velocity (meters/sec)	Im (m/m)	Cv (Decimal)	Velocity (meters/sec)	SEC (kW-hr/mton-km)	Mass Flow Solids (m-ton/hr)	Density Solid (S.G. water)	d50 (mm)	d85 (mm)	M	V50 (meters/sec)	Settling Velocity Maximum (meters/sec)	Im - Iw	V at Min Im (meters/sec)	V Standard (meters/sec)
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1000 m-ton per hr	2.1077	0.09498	2.1077	0.11864	0.15000	2.1077	0.8125	1.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.1144550	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1500 m-ton per hr	3.1616	0.05352	3.1616	0.06685	0.15000	3.1616	0.4578	1.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0574486	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2000 m-ton per hr	4.2154	0.04159	4.2154	0.05195	0.15000	4.2154	0.3558	2.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0352277	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2500 m-ton per hr	5.2693	0.04022	5.2693	0.05023	0.15000	5.2693	0.3440	2.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0241066	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3000 m-ton per hr	6.3232	0.04427	6.3232	0.05530	0.15000	6.3232	0.3787	3.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0176819	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3500 m-ton per hr	7.3770	0.05189	7.3770	0.06481	0.15000	7.3770	0.4439	3.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0136096	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4000 m-ton per hr	8.4309	0.06223	8.4309	0.07773	0.15000	8.4309	0.5323	4.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0108426	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4500 m-ton per hr	9.4847	0.07487	9.4847	0.09352	0.15000	9.4847	0.6405	4.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0088751	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5000 m-ton per hr	10.5386	0.08960	10.5386	0.11192	0.15000	10.5386	0.7665	5.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0074197	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5500 m-ton per hr	11.5925	0.10629	11.5925	0.13276	0.15000	11.5925	0.9092	5.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0063098	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6000 m-ton per hr	12.6463	0.12483	12.6463	0.15593	0.15000	12.6463	1.0679	6.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0054422	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6500 m-ton per hr	13.7002	0.14520	13.7002	0.18136	0.15000	13.7002	1.2420	6.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0047499	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7000 m-ton per hr	14.7540	0.16733	14.7540	0.20902	0.15000	14.7540	1.4314	7.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0041876	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7500 m-ton per hr	15.8079	0.19123	15.8079	0.23886	0.15000	15.8079	1.6358	7.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0037242	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8000 m-ton per hr	16.8617	0.21685	16.8617	0.27087	0.15000	16.8617	1.8550	8.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0033372	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8500 m-ton per hr	17.9156	0.24420	17.9156	0.30503	0.15000	17.9156	2.0890	8.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0030104	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9000 m-ton per hr	18.9695	0.27326	18.9695	0.34133	0.15000	18.9695	2.3375	9.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0027316	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9500 m-ton per hr	20.0233	0.30402	20.0233	0.37975	0.15000	20.0233	2.6007	9.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0024917	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/10000 m-ton per hr	21.0772	0.33648	21.0772	0.42030	0.15000	21.0772	2.8784	10.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0022837	4.934	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1000 m-ton per hr	1.5808	0.18867	1.5808	0.25122	0.20000	1.5808	1.2903	1.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.2488684	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1500 m-ton per hr	2.3712	0.09779	2.3712	0.13021	0.20000	2.3712	0.6688	1.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.1249150	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2000 m-ton per hr	3.1616	0.06459	3.1616	0.08600	0.20000	3.1616	0.4417	2.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0765982	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2500 m-ton per hr	3.9520	0.05040	3.9520	0.06711	0.20000	3.9520	0.3447	2.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0524170	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3000 m-ton per hr	4.7424	0.04477	4.7424	0.05961	0.20000	4.7424	0.3062	3.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0384471	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3500 m-ton per hr	5.5328	0.04385	5.5328	0.05839	0.20000	5.5328	0.2999	3.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0295838	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4000 m-ton per hr	6.3232	0.04596	6.3232	0.06120	0.20000	6.3232	0.3143	4.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0235759	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4500 m-ton per hr	7.1135	0.05025	7.1135	0.06691	0.20000	7.1135	0.3437	4.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0192978	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5000 m-ton per hr	7.9039	0.05627	7.9039	0.07492	0.20000	7.9039	0.3848	5.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0161332	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5500 m-ton per hr	8.6943	0.06372	8.6943	0.08485	0.20000	8.6943	0.4358	5.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0137200	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6000 m-ton per hr	9.4847	0.07246	9.4847	0.09648	0.20000	9.4847	0.4956	6.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0118335	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6500 m-ton per hr	10.2751	0.08237	10.2751	0.10967	0.20000	10.2751	0.5633	6.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0103280	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7000 m-ton per hr	11.0655	0.09337	11.0655	0.12432	0.20000	11.0655	0.6386	7.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0091055	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7500 m-ton per hr	11.8559	0.10542	11.8559	0.14036	0.20000	11.8559	0.7209	7.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0080978	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8000 m-ton per hr	12.6463	0.11847	12.6463	0.15774	0.20000	12.6463	0.8102	8.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0072563	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8500 m-ton per hr	13.4367	0.13251	13.4367	0.17643	0.20000	13.4367	0.9062	8.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0065457	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/9000 m-ton per hr	14.2271	0.14750	14.2271	0.19640	0.20000	14.2271	1.0088	9.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0059396	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/9500 m-ton per hr	15.0175	0.16344	15.0175	0.21763	0.20000	15.0175	1.1178	9.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0054180	5.333	7.522
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/10000 m-ton per hr	15.8079	0.18032	15.8079	0.24010	0.20000	15.8079	1.2332	10.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0049656	5.333	7.522

Figure 14: The Output window shows the Slurry results table in the center Pipes section for multiple scenarios

Step 12. Transfer the Output Data to a Spreadsheet

The information displayed on the Slurry tab in the Output window can be copied and pasted directly into a spreadsheet application, such as Excel, as shown in Figure 15. If the scenarios and the output were set up carefully, the data will already be formatted in such a way that graphing the information using the spreadsheet graphing features can be extremely straightforward.

Using the data in the spreadsheet, plot the Jm vs. velocity data for the three lines of constant solids concentration, and the three lines of constant solids throughput, as shown in Figure 16.

Pipe	Velocity (meters/sec)	Jm (m/m)	Velocity (meters/sec)	Im (m/m)	Cv (Decimal)	Velocity (meters/sec)
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1000 m-ton per hr	2.1077	0.095	2.1077	0.1186	0.15	2.1077
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1500 m-ton per hr	3.1616	0.0535	3.1616	0.0669	0.15	3.1616
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2000 m-ton per hr	4.2154	0.0416	4.2154	0.052	0.15	4.2154
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2500 m-ton per hr	5.2693	0.0402	5.2693	0.0502	0.15	5.2693
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3000 m-ton per hr	6.3232	0.0443	6.3232	0.0553	0.15	6.3232
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3500 m-ton per hr	7.377	0.0519	7.377	0.0648	0.15	7.377
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4000 m-ton per hr	8.4309	0.0622	8.4309	0.0777	0.15	8.4309
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4500 m-ton per hr	9.4847	0.0749	9.4847	0.0935	0.15	9.4847
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5000 m-ton per hr	10.5386	0.0896	10.5386	0.1119	0.15	10.5386
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5500 m-ton per hr	11.5925	0.1063	11.5925	0.1328	0.15	11.5925
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6000 m-ton per hr	12.6463	0.1248	12.6463	0.1559	0.15	12.6463
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6500 m-ton per hr	13.7002	0.1452	13.7002	0.1814	0.15	13.7002
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7000 m-ton per hr	14.754	0.1673	14.754	0.209	0.15	14.754
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7500 m-ton per hr	15.8079	0.1912	15.8079	0.2389	0.15	15.8079
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8000 m-ton per hr	16.8617	0.2169	16.8617	0.2709	0.15	16.8617
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8500 m-ton per hr	17.9156	0.2442	17.9156	0.305	0.15	17.9156
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9000 m-ton per hr	18.9695	0.2733	18.9695	0.3413	0.15	18.9695
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9500 m-ton per hr	20.0233	0.304	20.0233	0.3798	0.15	20.0233
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/10000 m-ton per hr	21.0772	0.3365	21.0772	0.4203	0.15	21.0772
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1000 m-ton per hr	1.5808	0.1887	1.5808	0.2512	0.2	1.5808
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1500 m-ton per hr	2.3712	0.0978	2.3712	0.1302	0.2	2.3712
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2000 m-ton per hr	3.1616	0.0646	3.1616	0.086	0.2	3.1616
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2500 m-ton per hr	3.952	0.0504	3.952	0.0671	0.2	3.952
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3000 m-ton per hr	4.7424	0.0448	4.7424	0.0596	0.2	4.7424
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3500 m-ton per hr	5.5328	0.0439	5.5328	0.0584	0.2	5.5328
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4000 m-ton per hr	6.3232	0.046	6.3232	0.0612	0.2	6.3232
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4500 m-ton per hr	7.1135	0.0503	7.1135	0.0669	0.2	7.1135
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5000 m-ton per hr	7.9039	0.0563	7.9039	0.0749	0.2	7.9039
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5500 m-ton per hr	8.6943	0.0637	8.6943	0.0849	0.2	8.6943
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6000 m-ton per hr	9.4847	0.0725	9.4847	0.0965	0.2	9.4847
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6500 m-ton per hr	10.2751	0.0824	10.2751	0.1097	0.2	10.2751
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7000 m-ton per hr	11.0655	0.0934	11.0655	0.1243	0.2	11.0655
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7500 m-ton per hr	11.8559	0.1054	11.8559	0.1404	0.2	11.8559
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8000 m-ton per hr	12.6463	0.1185	12.6463	0.1577	0.2	12.6463
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8500 m-ton per hr	13.4367	0.1325	13.4367	0.1764	0.2	13.4367

Figure 15: The Slurry results table in the Output window can be copied and pasted into a spreadsheet application in order to utilize the graphing features of the spreadsheet to generate the slurry system curves

Step 13. Analyze the Results

The results are shown in Figure 16. The solid lines refer to the constant solids concentration cases, and the broken lines refer to the constant throughput cases.
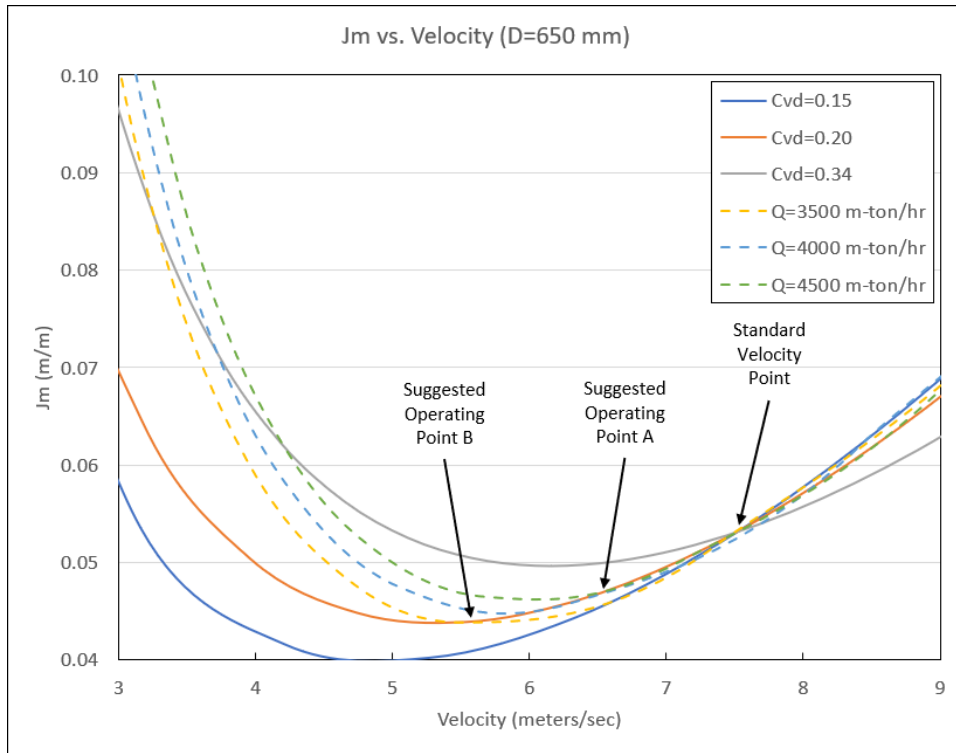
Figure 16: Slurry system curves for Slurry System Feasibility Study example

The arrows on the graph indicate the proposed operating conditions for the system on the 4000 m-ton/hr constant throughput line.

It is evident from the system characteristics that the proposed operating point Condition B, with the lower specific energy consumption, does not represent a feasible operating point for this slurry system. The system characteristics show the Jm decreasing as the mean velocity increases, so the operation would be unstable.

Even the first proposed operating condition, Condition A, is marginal. If the pump has a variable speed drive, and the solids concentration of the system does not fluctuate very much, this operating point could be used. However, as Figure 16 shows, if the concentrations vary from 3500 m-ton/hr to 4500 m-ton/hr, then the system operation is close to unstable for Condition A as well.

In order to allow for fluctuations in solids concentrations during system operations, and still maintain stable operation, it would be better to operate closer to the standard velocity. The standard velocity point is also shown in Figure 14 at about 7.5 m/s.

For this case, if operability is also considered, lower slurry concentrations, and higher energy consumption should be used to ensure system stability. Further, as discussed in the original reference, a different diameter pipe can be considered. The original reference suggests 0.55 m pipe as a better option than 0.65 m pipe because of operation closer to the standard velocity and lower capital cost.

Automated Network Sizing (ANS) Examples

The examples below utilize the Automated Network Sizing module.

Fathom ANS Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Three-Reservoir Model	Beginner	Water	3	0	Learn the basics of the ANS module such as defining all panels in the Sizing window to achieve a minimum flow volume objective
Control Valve	Beginner	Water	2	0	Learn how to use the ANS module and how Common Size Groups work to minimize the pipe weight of a system.
Cooling System, Creating Libraries	Intermediate	Water	21	2	Learn how to build cost libraries and engineering libraries for use with the ANS module. See how to use Scale Tables for defining pump or bend costs, how to specify material versus installation costs, and how to define costs for pipes.
Cooling System	Intermediate	Water	21	2	Learn how to use the ANS module to minimize monetary cost based on initial cost and recurring costs. See how to use the Assign Cost Libraries panel and the Sizing Summary panel. Explore how the system life and the sizing objective impacts the optimal pipe sizes.
Cooling System, Multiple Design Cases	Intermediate	Water	21	2	Learn how to use the ANS module to define and size a system based on a dependent design configuration, called a Dependent Design Case.
Hot Water System	Intermediate	Water	24	3	Learn how to use the ANS module to define a system with multiple specific design

Example	Complexity	Fluid	Pipes	Pumps	Description
					requirements and minimize initial and recurring costs. See how to use a Dependent Design Case to see how a system configuration affects the system sizing.
Variable Drive	Intermediate	Lube Oil	23	2	Learn how to use the ANS module to see the life cycle cost savings in a sized system using a Variable Frequency Drive instead of a Control Valve.
Housing Project	Advanced	Water	25	0	Learn how to use the ANS module to size a system with more than one Dependent Design Case with unique design requirements for each dependent case.

Beginner - Three Reservoir Problem - ANS

Summary

The objective of this example is to familiarize the user with the panels in the ANS module. We will apply automated sizing to determine the ideal pipe sizes from our previously built three-reservoir model to reduce costs while meeting certain requirements.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Minimizing the flow volume to minimize cost
- Specifying Pipe Design Requirements
- Choosing Candidate Sets

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also

watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Beginner - Three Reservoir.fth***
- ***US - Beginner - Three Reservoir - ANS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the ***US - Beginner - Three Reservoir Problem.fth*** example file and save it as a new file.

We will be using the Base Scenario to compare the original results without Sizing to the results with Sizing. Create a child scenario named *Sized*.

The workspace should look like Figure 1 below:
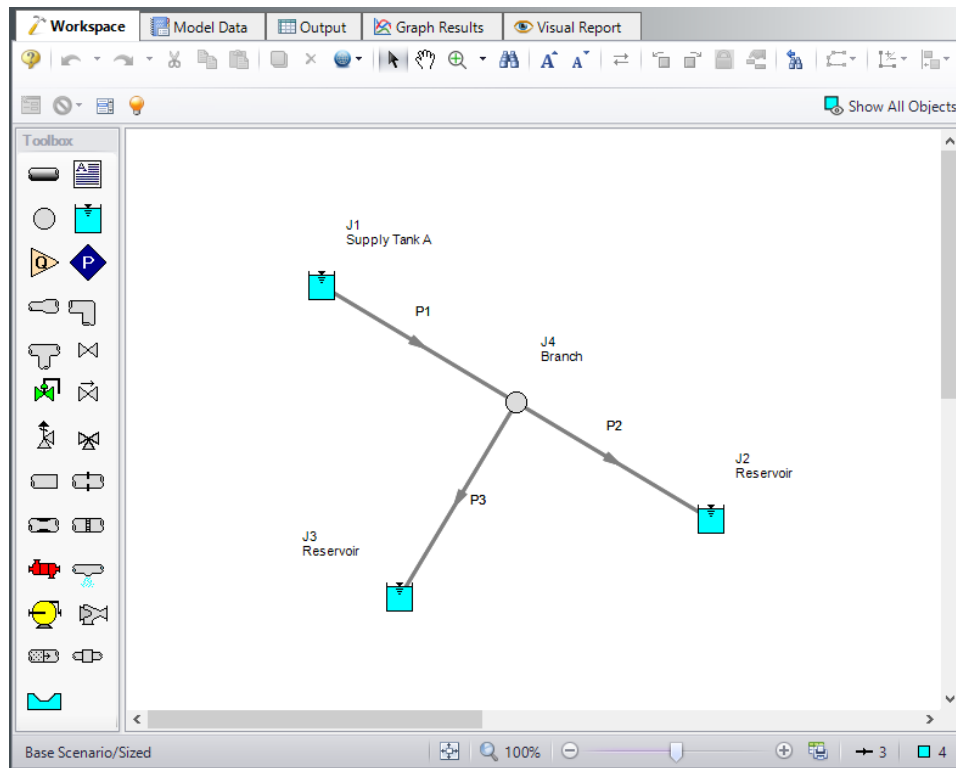
Figure 1: Layout of Three Reservoir Model in Workspace

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Figure 2 below shows the Analysis menu options when ANS is activated.
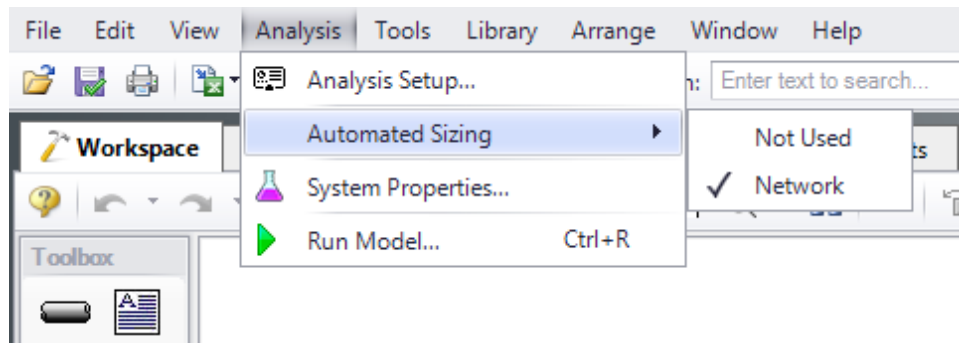
Figure 2: Activating Network Sizing from the Analysis menu

Step 4. Configure Sizing Settings

Here we will pursue a brief introduction to some of the sizing capabilities in the ANS module. A more complete discussion is given in the AFT Fathom Help file.

The Sizing window is comprised of multiple panels which can be accessed using the buttons on the Sizing Navigation panel along the bottom of the window, as shown in Figure 3. The sizing panels can be accessed in any order, though it is easiest to enter the information by navigating the panels from left to right, since the input on panels such as the Sizing Objective and Sizing Assignments panels will affect the options available on later panels. Depending on the type of sizing being done, some panels may be disabled or unused.

Each panel button contains either a green checkmark or red circle which denotes the completion status of that panel. If the minimum required information is present to run the model, the symbol will be green, whereas the red symbol represents incomplete input. The amount of information required will vary based on whether the Sizing Level Status (located on the Sizing Objective panel) is set to Perform Sizing, Calculate Costs, or Do Not Size. A detailed summary of the items which have been completed and the items which are still incomplete can be seen in the Sizing Status panel, opened from the Sizing Toolbar.

Note that some panels will always be shown as complete since the model can be run without any additional information entered on them, such as with the Design Requirements panel. However, in order to find the best system design it will often be necessary to enter more than the minimum information required by the solver.

The Lock Panel toggle located on the Sizing Toolbar prevents changes to the current panel when it is enabled. This is primarily useful to prevent editing once a scenario has been run, since any changes that are made to a scenario which has output will cause all output to be erased. By default, the ANS module will lock all panels after a sizing run is completed, requiring panels to be unlocked before any changes can be made. This setting can be changed on the Sizing Objective panel. All panels can be locked or unlocked simultaneously by using the Lock/Unlock All Panels buttons on the Sizing Toolbar.
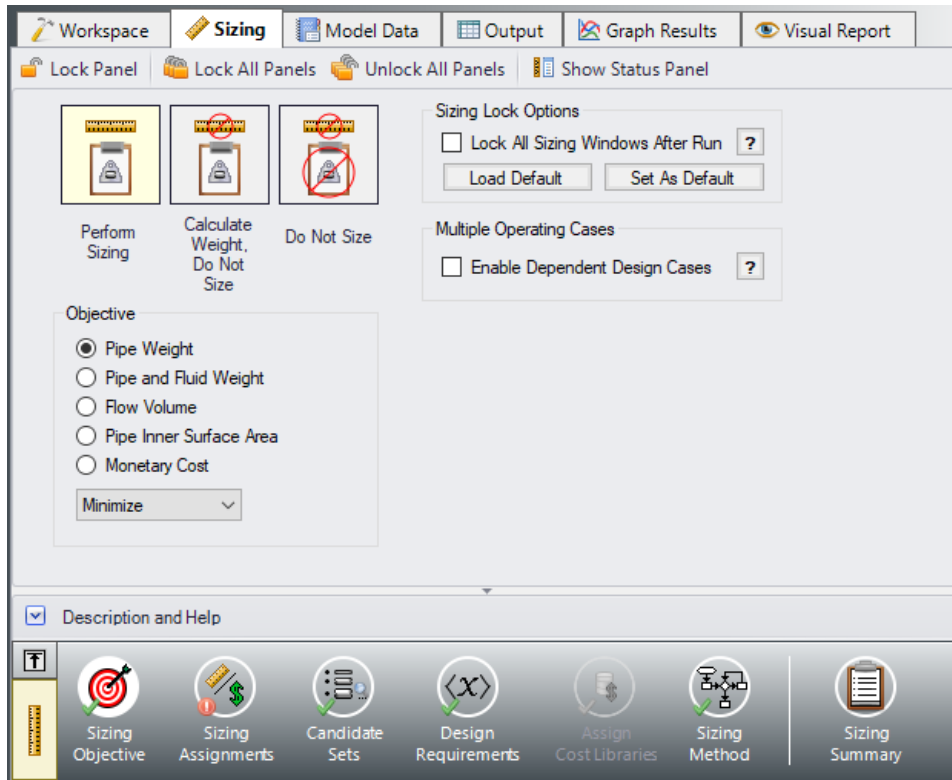
Figure 3: Sizing Window showing common sizing panel features

A. Sizing Objective

The Sizing Objective panel is used to set the Sizing Option, which is the type of calculation that will be performed, and to select an Objective which will be used in the sizing process.

For the Sizing Option, Perform Sizing should generally be selected, since this is the primary function of the ANS module. However, the other two options may be used for troubleshooting or informational purposes. For example, the Calculate Cost, Do Not Size option may be useful to calculate an initial cost for the system, or to verify your cost libraries. The Do Not Size option allows the model to be run normally in AFT Fathom without requiring any cost or sizing information. This is useful to preserve the information already entered in the Sizing window.

➢**Select Perform Sizing** (if not already selected) to begin configuring the sizing settings.

Note: The Sizing Option can always be seen on the box on the left of the Sizing Navigation panel. Clicking this box will bring you to the Sizing Objective panel, where this status can be changed.

The easiest way to size the system is to set the objective to a non-monetary value such as pipe weight or flow volume rather than directly analyzing monetary cost. Here we will minimize flow volume. The flow volume is merely the sum of the internal volumes of all sized pipes. As the flow volume is minimized, the pipe material, and therefore cost, is minimized along with it. Typically it is recommended to use the pipe

weight option, as this will have the best correlation to cost. However, certain systems, such as ducts may be better sized using flow volume.

Although we will not do so here, the ANS module can also minimize the monetary cost of the system. This is more powerful than sizing for flow volume, but also requires more time investment in setting up cost libraries.

➢**For the Objective, choose Flow Volume**, and make sure Minimize is selected from the drop-down list. The Sizing Objective panel should now appear as shown in Figure 4.
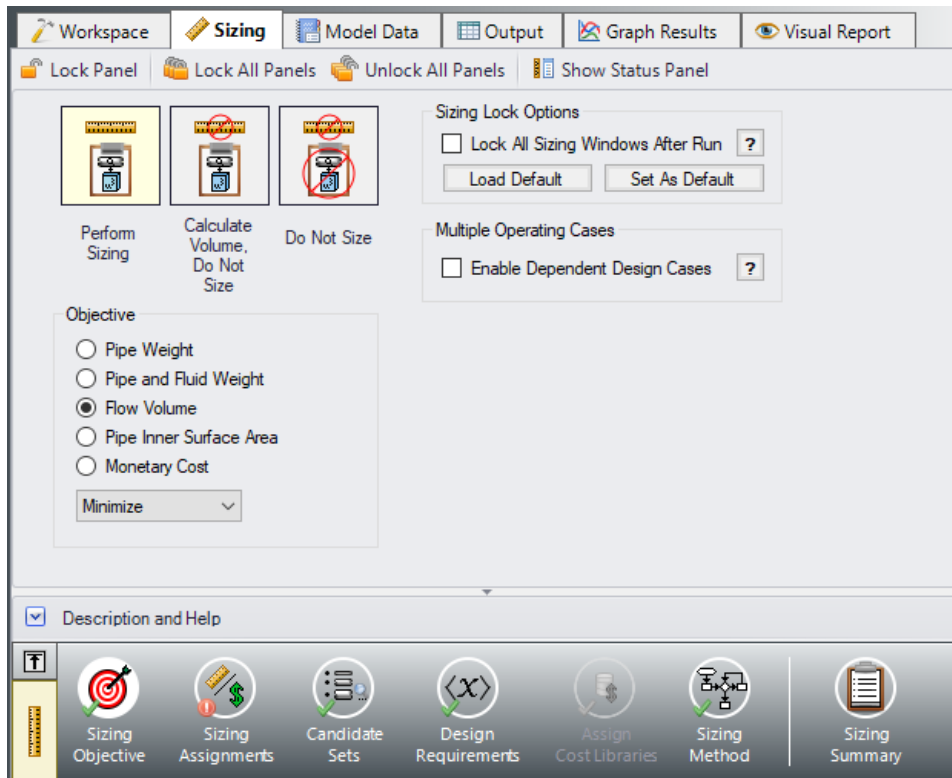
Figure 4: Sizing Objective set to minimize flow volume

B. Sizing Assignments

On the Sizing Navigation panel at the bottom of the window select the Sizing Assignments button. The Sizing Assignments panel allows the user to define what objects will be sized in the model, and what will be included in the cost calculation without being sized. Common Size Groups for pipes, or Maximum Cost Groups for pumps/control valves can be created on this panel as well when appropriate.

Let's consider the Sizing Assignments for the pipes, which should be displayed by default when we open this panel.

In the Sizing and Volume Options table there are several options under the categories to Automatically Size, or Do Not Size the pipe.

If an Automatically Size option is selected, the ANS module will treat the pipe diameter as a variable and vary it according to certain criteria that will be discussed shortly.

For a new system, typically all pipes will be desired to be sized and included in the cost, which is being calculated in terms of volume in this case. If you are instead analyzing the possible replacement of existing pipes, it may be better to size the pipes, but only include the cost if the size changes from that which exists.

If Do Not Size is selected, the pipe will retain the settings currently set in the Workspace. Why would one choose to not size a pipe? There could be a number of reasons, but one good reason is that the pipe represents a pipe in an existing system and the design does not allow the replacement of that pipe with a new one. Therefore its diameter is fixed, and sizing the pipe would serve no purpose.

Another reason may be if a certain size is necessary for the design due to certain requirements, in which case the pipe cost can be included without sizing the pipe by choosing Include in Cost.

This model is a new system, so all pipes will be sized and included in the cost.

➢**Select Always Include in Volume** for all three pipes (Figure 5).

Since we want to size the pipes independently in this model, we do not need to make any changes to the Pipe Grouping section of the table, so this panel is complete.

Note: For models which have junctions that can be sized, a Junctions button at the top of this panel will be available to set the Junctions Sizing Assignments. In this case we have chosen to analyze flow volume. While this value can easily be calculated for the pipes, Fathom does not have enough information to calculate a flow volume for the branch or the reservoirs, which means that they cannot be sized using this method, and the Junctions tab is unavailable. This same limitation exists when using other non-monetary objectives as well, such as pipe weight. If the size of a junction will have a large impact on the system cost, such as a pump, Monetary Cost should be used as the Objective.
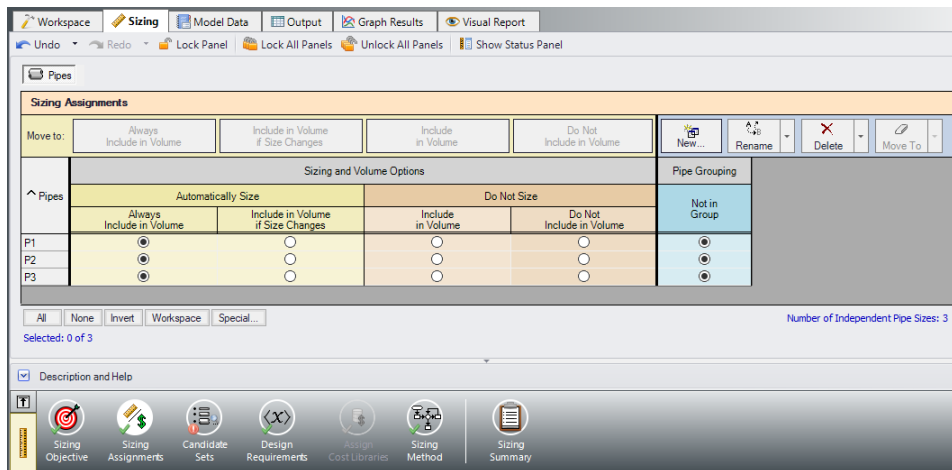
Figure 5: Sizing Assignments panel for Three Reservoir model

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Since commercial piping is limited to certain sizes, the ANS module needs a list of possible sizes from which to choose. This list is called a *Candidate Set*.

So as not to limit the sizing unnecessarily, the candidate sets should include both smaller and larger pipes than your anticipated final size. If you make the candidate set too small, you may limit the ability of the ANS module to find the best sizing. It is better to make the candidate set too large than too small. Experience applying the ANS module to actual systems will help you choose appropriate candidate sets. If after obtaining a solution you find that one or more of the sized pipes is at the extreme of the candidate set a warning will appear, and it is recommended to expand the defined candidate set.

To create a Candidate Set, do the following:

1. Under Define Candidate Sets, click **New**.
2. Give the set the name *All STD* and click **OK**.
3. From the drop down list choose **Steel - ANSI** if it is not already selected.
4. At the bottom of the Select Pipe Sizes window, make sure the Sort option selection is **Type, Schedule, Class**.
5. In the Available Material Sizes and Types list on the left, **select (not expand) STD** so that the schedule name is now highlighted.
6. Click **Add** to add all STD pipe sizes to the list on the right.
7. Click **OK**.

The All STD set will now appear.

We will apply the All STD set shortly. First, let's take a moment to understand what we have just done. We have created a candidate set that includes all pipe sizes in the library that are Steel - ANSI STD. At the low end this includes 1/8 inch pipe, and at the high end 48 inch pipe. This candidate set contains pipe sizes from a library of steel ANSI pipe, and not just specific pipes from the model. After we apply this candidate set to a specific pipe in the model and the sizing is run, the ANS module will select the best pipe size from this set of pipe sizes to achieve the objective.

We now need to define which pipes will use this Candidate Set during the sizing calculation. Each pipe that is being sized must have a Candidate Set assigned to it. Under Assign Candidate Sets to Pipes, **set each pipe in the model to use All STD** by selecting the radio button under this candidate set for each of the pipes. The Candidate Sets should now be fully defined and assigned to the appropriate pipes, as can be seen in Figure 6.
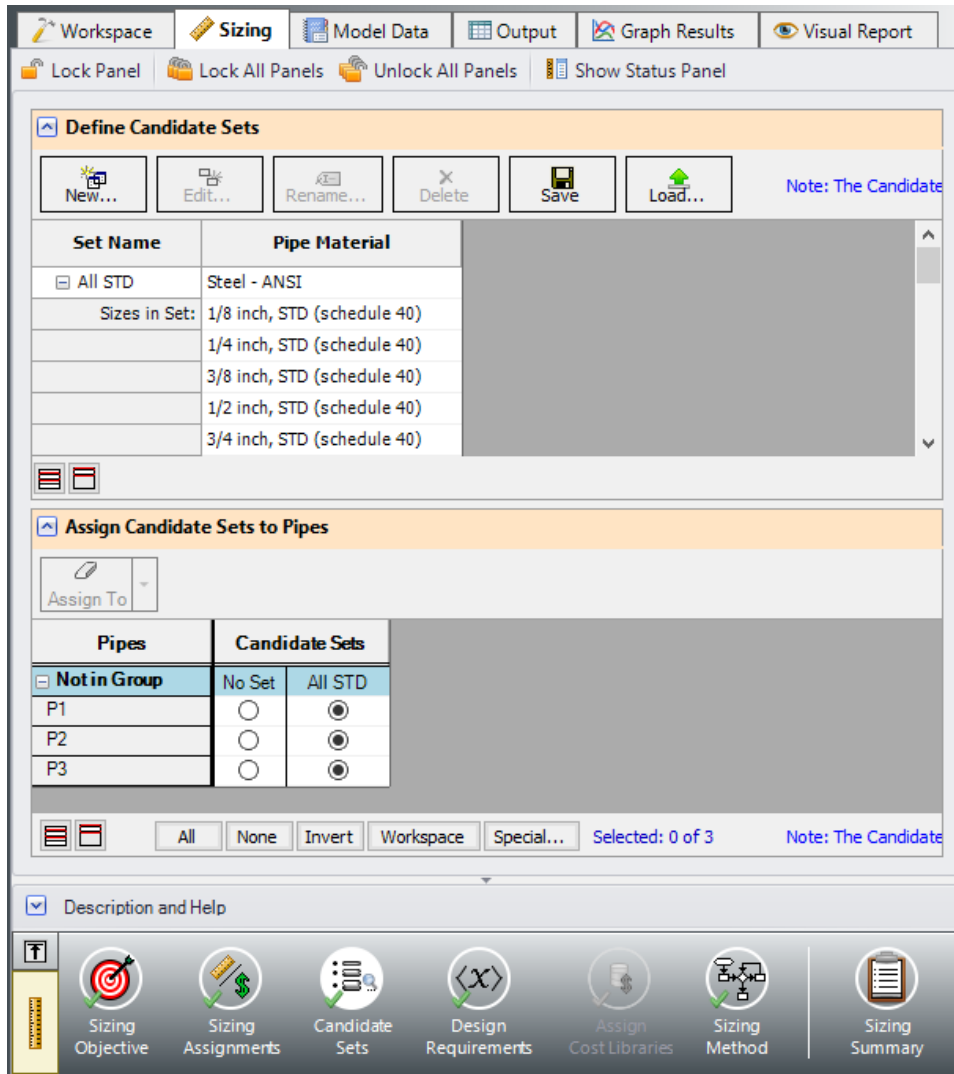
Figure 6: Candidate Sets window fully defined for the Three Reservoir model

D. Design Requirements

Select the Design Requirements button. If we do not define any design requirements for the model, the ANS module will automatically choose the smallest pipe size in the Candidate Set, since it has the smallest flow volume. Along with the smallest pipes we may get unacceptably low flow rates, or unacceptably high velocities and pressure drops. To maintain acceptable system operating conditions, we need to set design requirements.

We are going to define two design requirements for this case. Before adding any requirements, let's look more closely at the results of the model we are starting with. All pipe diameters are nominal 8 inch, and the results show that the water flows from reservoir J1 to reservoirs J2 and J3. The flowrate out of J1 is 2,388.0 gal/min, and the flowrates to J2 and J3 are 754.2 and 1,633.8 gal/min, respectively. Let's assume that there is a requirement to draw a minimum of 2,500 gal/min from J1, and to deliver at least

750 gal/min each to J2 and J3. What is the combination of pipe sizes which achieve the desired objective while still meeting these demands? In our case, we are defining the objective as minimizing the internal volume of the pipes (i.e., length times internal cross-sectional area).

We need to create two Design Requirements that represent the flow requirements at the reservoirs in order to answer this question. We will do this by defining Design Requirements for the inlet/outlet flowrates of the pipes where they connect to each of the Reservoirs. To do this:

1. Make sure that the **Pipes** button is selected.
2. Click **New** under Define Design Requirements.
3. Enter a name when prompted: *Min Flow Supply*. Click **OK**.
4. A new row will appear in the Pipe Design Requirements table. In this row, select **Volumetric Flow Rate** as the Parameter.
5. Choose **Minimum** for Max/Min.
6. Enter 2500 gal/min.
7. Now repeat the above process to define *Min Flow Delivery* with a **Volumetric Flow Rate Minimum** of **750 gal/min**.

To be functional, we need to apply these Design Requirements to the correct pipes. There are no limits to the number of Design Requirements that can be defined and applied.

In the Assign Design Requirements to Pipes section, expand the pipe list. Select the check box for Min Flow Supply next to pipe P1, and Min Flow Delivery next to pipes P2 and P3. This will set the minimum flow rate coming out of Reservoir J1 and going into Reservoirs J2 and J3 as described above. The fully defined and assigned Design Requirements are shown in Figure 7.
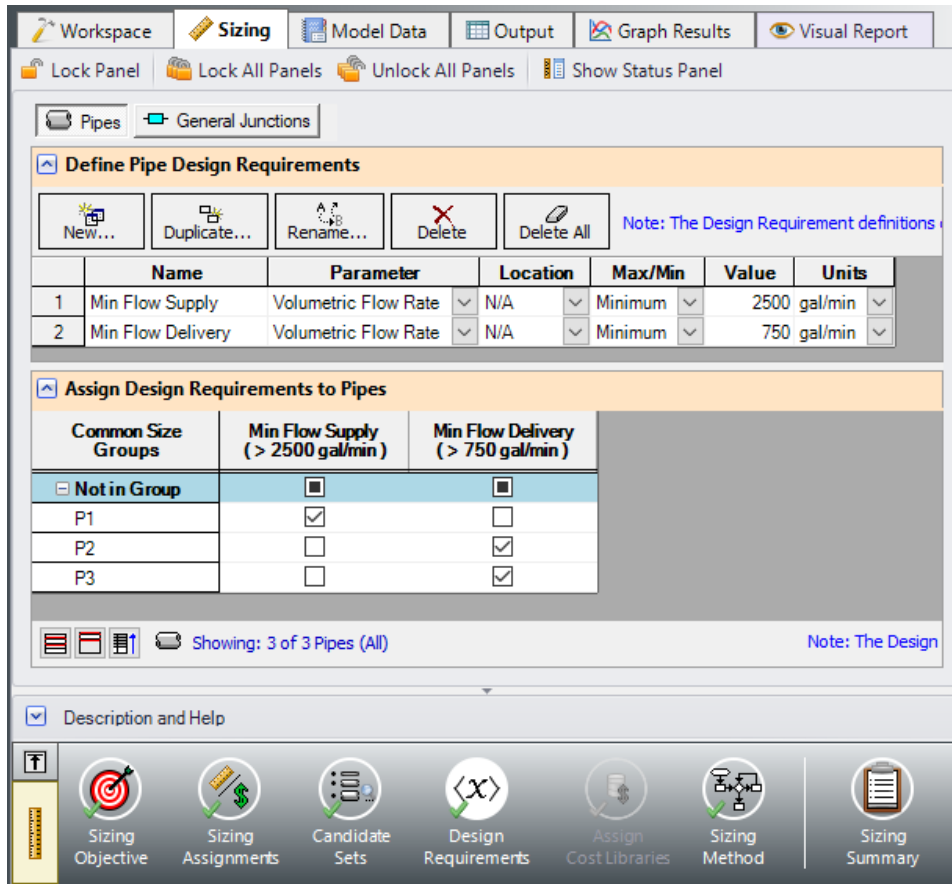
Figure 7: Design Requirements window with the minimum flow requirements defined

E. Assign Cost Libraries

When the Sizing Objective has been defined as Monetary Cost, it is necessary to create and assign cost libraries for the automated sizing, which can be done in the Assign Cost Libraries panel. Since we have defined the objective as Pipe Flow Volume, we will not need to assign any cost libraries, and this button is grayed out.

F. Sizing Method

Select the Sizing Method button to go to the Sizing Method panel.

The Sizing Method panel is used to set up the calculation methods for sizing the system. You can select whether discrete or continuous sizing will be used, and which method will be applied.

If Continuous Sizing is selected, the ANS module will ignore the defined Candidate Sets and report the ideal hydraulic diameter for the pipes being sized, which will likely not match any of the possible chosen commercial sizes. While this is not useful as a final solution, this may be helpful as a baseline to check the final solution using the provided discrete methods. Discrete Sizing will typically perform a continuous

sizing calculation as a basis, after which multiple discrete sizes above and below this solution will be evaluated to find the ideal sizing based on the provided Candidate Set.

➢**For this model, use Discrete Sizing.**

For the Search Method, the ideal method will often change based on the number of independent pipe sizes (shown in the Sizing Assignments panel on the bottom right), number of Design Requirements, and feasibility of the initial system design. The Help file provides more information on the strengths and weaknesses of each method. It is generally recommended to run the sizing with more than one method, as it is often not obvious which method will be most effective for each system. The suggest Method button can be used as a guide for which method to start with.

For a simple model such as this one, the MMFD or SQP method should be appropriate since there are only a few Design Requirements, and only three pipes are being varied.

➢**Choose the default Modified Method of Feasible Directions (MMFD).**

Step 5. Run the Model

Select Run from the Analysis menu. While the model is running, the Solution Progress window shows the Sizing Calls to Solver. This is how many times a complete hydraulic analysis was run.

The solver also displays the Current Cost and Best Feasible Cost, which will display the last calculated value for the cost, as well as the Best Feasible Cost which has been found so far (Figure 8). The solver will continue to iterate using the defined method until it finds the ideal sizing. For the selected method, the solver will first perform a continuous sizing to find a starting point, then test discrete solutions close to this continuous solution to find the final, discrete solution.

Note: At any time during the run, the user can pause the solution and use the Other Actions button to accept the currently displayed Best Feasible Cost as the final solution. For this example the solution is found and displayed quickly, but for larger systems with many pipes being sized, it may be more productive to pause the solution and accept the current solution in order to save time, especially if the displayed cost does not appear to be decreasing much further.
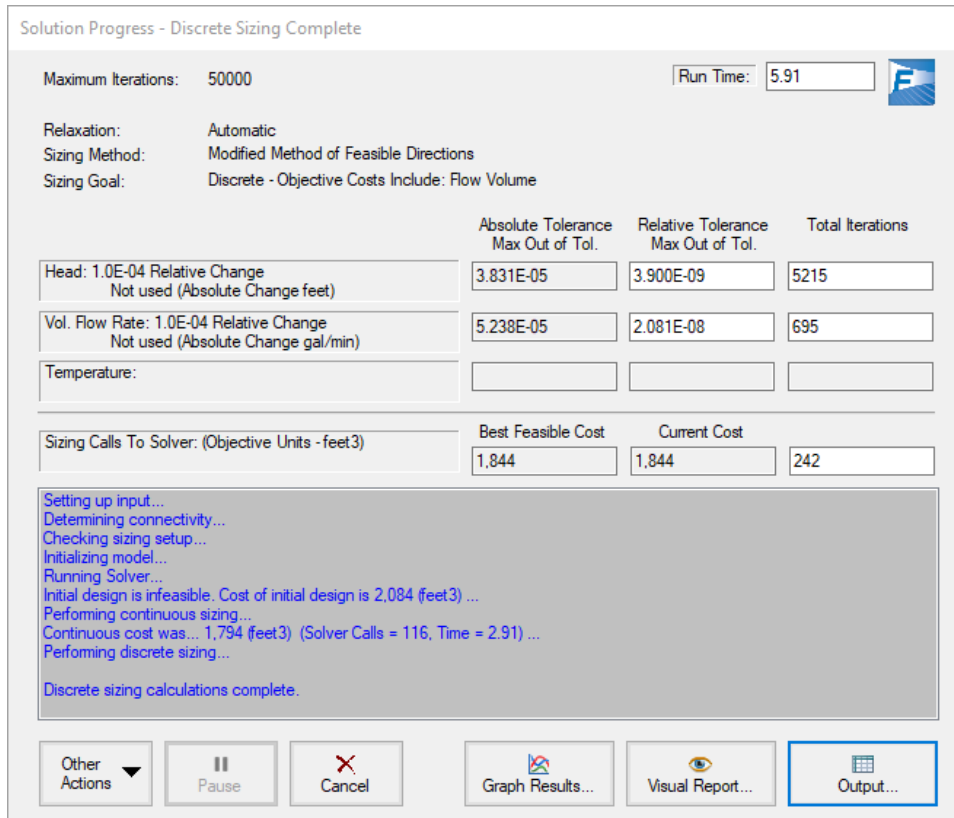
Figure 8: Solution Progress Window showing the progress of the sizing calculation

Step 6. Review the Sized Results

Click the View Output button to see the results. The General section shows the Cost Report, which indicates the volumes for the final solution. As shown in Figure 9, the minimum total volume identified by the ANS module was 1,844 feet3. In the Pipes section of the output on the Sizing tab, the final pipe sizes of 10, 8, and 6 inches are shown (Figure 9).

Now switch to the Pipe Design Requirements tab in the Pipe section (Figure 10). We can confirm that the calculated design is feasible, since all of the Design Requirements were met. That is, the flow from J1 (as shown in pipe P1) exceeded 2,500 gal/min, and the flows to J2 and J3 exceeded 750 gal/min. Notice that some of the Design Requirements are highlighted, indicating that they were active. An active design requirement is defined as a requirement that had an impact on the solution. If an active design requirement were disabled, a different solution would be found. Yellow highlights indicate that the design requirement was active in the final solution, while blue highlights indicate that the requirement was active when the continuous sizing was performed, but became inactive when the solver switched to discrete sizing.

You can also display the adjusted pipe sizes and resulting flow rates in the Visual Report.

Figure 9: Output Window with Cost Report and Pipe Sizes for Three Reservoir Model

Figure 10: Design Requirements results for Three Reservoir Model

Transferring Sized Results

When performing sizing, the Transfer Results to Initial Guesses feature on the Output window Edit menu takes on new meaning. When used in a non-sizing context, this feature takes the solved hydraulic results (i.e., pressures, flows rates, etc.) and assigns them to the initial guess values for the pipes and junctions.

When you select Transfer Results to Initial Guesses after automated sizing, you will see a dialog window appear like that shown in Figure 11. The ANS module allows you to transfer the hydraulic results, the

sized pipe diameters, or both. If you transfer the sized pipe diameters, each pipe's input diameter (back on the Workspace) will become the sized diameter.

This feature allows the sizing results to be saved to the model for use in further analysis as needed.
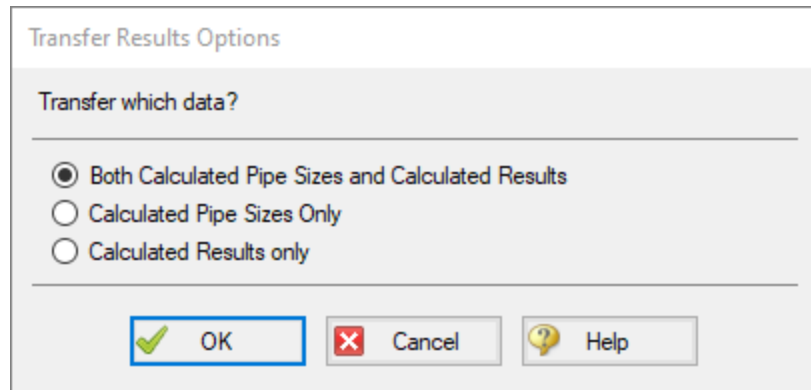
Figure 11: After sizing, the Transfer Results to Initial Guesses feature transfers both the hydraulic results and sized pipe size results

Conclusion

You have now used performed a network sizing to minimize flow volume with the ANS module.

Control Valve - ANS

Summary

This example will show how to use the ANS module to size a system based on pipe weight. For many systems the necessary cost data is not available to directly minimize cost, so minimizing system parameters such as weight or flow volume is useful. By minimizing parameters such as these we minimize pipe diameter, which directly correlates to minimizing pipe costs.

We will create a system that feeds water from an elevated reservoir to a lower reservoir at a specified flow rate. A control valve will be used to control the flow rate and can be placed anywhere within the line. We will need to determine what size of piping to use in order to achieve a minimum pressure drop of 12.5 psid across the control valve.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Sizing using pipe weight
- Creating a Common Size Group
- Defining Control Valve Design Requirements

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Control Valve.fth***
- ***US - Control Valve - ANS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Control Valve.fth** example file and save it as a new file. Create a child scenario from the Base Scenario named *Sized*. The workspace should look like Figure 1 below:
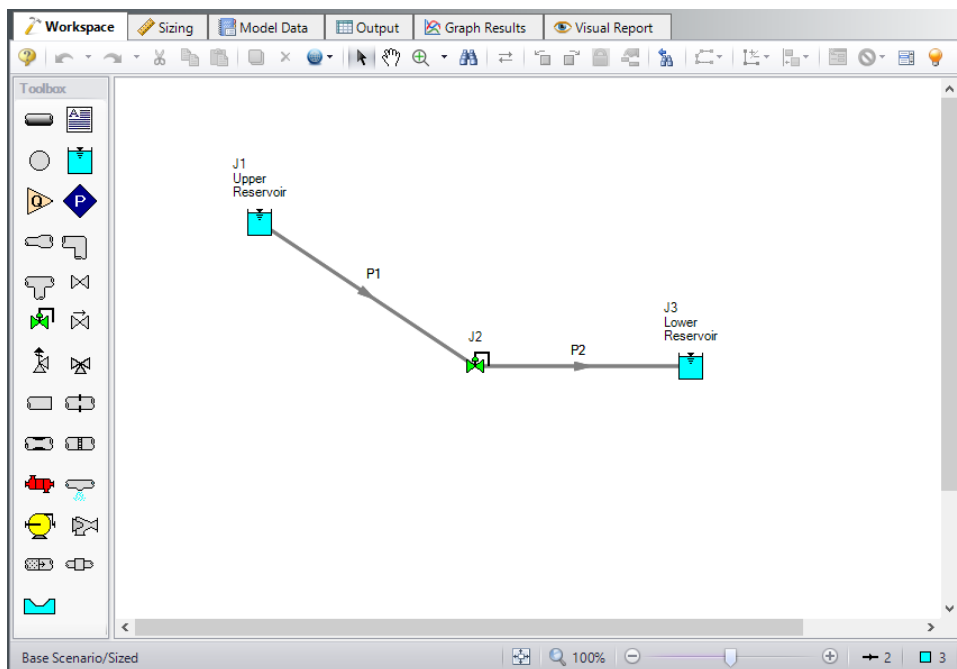
Figure 1: Control Valve System Layout

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Configure Sizing Settings

In this model we are going to perform sizing to minimize the pipe weight (therefore minimizing cost), while maintaining a minimum pressure drop of 12.5 psid across the control valve.

To do this, first go to the Sizing window by clicking on the Sizing tab, or from the Windows menu.

A. Sizing Objective

The Sizing Objective panel should be selected by default from the Sizing Navigation Panel along the bottom of the window. Input the following:

1. Sizing Option = Perform Sizing
2. Objective = Pipe Weight
3. Option = Minimize

B. Sizing Assignments

On the Sizing Navigation Panel at the bottom of the window select the Sizing Assignments button.

In this case we are sizing a new pipeline, so we want to size all pipes in the model.

➤**Under Automatically Size select Always Include in Weight** next to both pipes.

For this system it is required that both pipes be the same size. We will need to add both pipes to a Common Size Group to specify this. Do the following:

1. Click **New** above Pipe Grouping
2. Enter the a name *Main Pipes* and click **OK**. A new section will now appear with the Main Pipes group listed under Common Size Groups.
3. **Add both pipes** to this group by clicking the corresponding radio buttons (Figure 2).

Note that as many Common Size Groups as necessary can be created. Selection tools are available at the bottom of the window to help assign multiple pipes to a Common Size Group for larger models. Additionally, pipes can only be assigned to a group once they are set to be included in sizing.
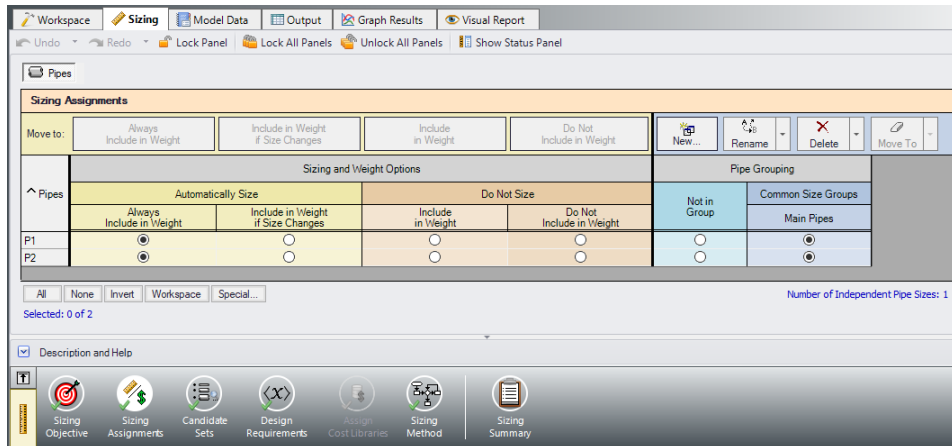
Figure 2: Pipes P1 and P2 settings in the Sizing Assignments window

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

1. Under Define Candidate Sets, click **New**.
2. Give the set the name *STD Pipes* and click **OK**.
3. In the Select Pipe Sizes window, choose **Steel - ANSI** from the drop-down menu if not already selected.
4. Make sure the Sort selection is for **Type, Schedule, Class** at the bottom of the window.
5. In the Available Material Sizes and Types on the left, **expand** the STD pipe sizes list.
6. Double-click each of the sizes **from 1 inch to 12 inch** to add them to the list on the right.
7. Now click **OK**.

The STD Pipes set will now appear as a Candidate Set at the top.

We now need to define which pipes will use this Candidate Set during the sizing calculation.

➢**Under Assign Candidate Sets to Pipes, set the Main Pipes Common Size Group** to use the STD Pipes Candidate Set. The Candidate Sets are now fully defined and assigned to the appropriate pipes, as can be seen in Figure 3.

Figure 3: Candidate Sets panel fully defined for the Control Valve model

D. Design Requirements

Select the Design Requirements button.

For this model we only have one Design Requirement, which is for the minimum pressure drop across the Control Valve. To define this requirement:

1. Select the **Control Valves** button at the top of the window.
2. Click **New** under Define Control Valve Design Requirements.
3. When prompted, enter the name *Min Pressure Drop*.
4. Next to Min Pressure Drop in the table, select **Pressure Drop Static** as the Parameter.
5. Choose **Minimum** as the Max/Min option.
6. Enter 12.5 psid.

Now we need to apply the defined Design Requirement to the Junction.

➢**Check the box next to the Control Valve J2** in the Assign Design Requirements to Control Valves section.

Note: Control valves must always have a design requirement specified for the minimum pressure drop/head, or the maximum Cv/percent open at the valve to ensure that reasonable operating conditions exist at the control valve. If no Design Requirements are applied, the ANS module will allow the control valve to control to any pressure required to meet the control setpoint, even if this requires the control valve to add pressure to the system. To avoid this in the final solution, a Design Requirement should be applied.

E. Assign Cost Libraries

When the Sizing Objective has been defined as Monetary Cost, it is necessary to create and assign cost libraries for the automated sizing, which can be done in the Assign Cost Libraries panel. Since we have defined the objective as Pipe Weight, we will not need to assign any cost libraries, and this button is grayed out.

F. Sizing Method

Select the Sizing Method button to go to the corresponding panel.

Make sure **Discrete Sizing** is selected, since we are sizing based on commercial pipe sizes.

For the Search Method in a simple model such as this one, the MMFD or SQP method should be appropriate since there is only one Design Requirement, and only one pipe size is being varied. Choose the default **Modified Method of Feasible Directions (MMFD)**.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Review the Sized Results

The output is shown below in Figure 4. We have shown all three sections of the output window with the General, Pipes, and Junctions sections. In the General Section, the Cost Report tab shows the minimized pipe weight for each of the pipes that were sized based on the design requirements.

In the pipe section, the Sizing tab is selected to show the final pipe sizes that were chosen. The ideal pipe size was found to be 6 inch pipe for this case. In the Junction section, the CV Design Requirements tab is shown. It can be seen that the Control Valve Design Requirement was met. The Design Requirement is colored blue, indicating that it was active in the final solution.

	Table Units: lbm	Type	Name	Pipe Weight
	Total of All Shown Costs			**3,795**
	Items In Sizing			**3,795**
	Items Not In Sizing			**0**
	Pipe Subtotal			**3,795**
o	P1	Pipe	Pipe	1,897
o	P2	Pipe	Pipe	1,897

Pipes | Sizing

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065

All Junctions | CV Design Requirements | Control Valve | Reservoir

Jct	Pressure Drop Static Minimum > 12.5 psid [Min Pressure Drop]
2	12.61

Figure 4: Output window for Control Valve Sizing

Step 7. Change the Control Valve Location

Now we would like to try something to further improve the system. For the first cut, we assumed that the control valve was in the middle of the pipe and that the pipes had to be the same size. If the pipes do not have to be the same length or size, we can further minimize the pipe weight and theoretically the cost of the piping.

To do this we will create another scenario.

1. In the Scenario Manager, **create a child** from the Sized scenario and name it *Unequal Pipes*. There should now be a check mark icon next to the scenario name to show that it is active.
2. On the Workspace open the Pipe Properties window for pipe P1 and change the length to **150 feet**.
3. Open the Pipe Properties window for pipe P2 and change the length to **50 feet**.
4. Go to the **Sizing** tab and click the **Sizing Assignments** button in the Sizing Navigation panel.
5. In the Pipe Grouping section of the table set **both** pipes to **Not in Group**. This will allow their sizes to be varied separately.

The scenario is now complete.

Step 8. Run New Scenario and Review Results

Now, we will run this scenario and see if we have minimized the pipe weight further. Select Run from the Analysis menu and allow the ANS module to size the system. When the analysis is complete, select Output to view the results.

The output window is shown below in Figure 5. We can see that pipe 2 was now chosen as 5 inch but pipe 1 was still chosen as 6 inch. Since pipe 1 was smaller the pressure drop across the pipe was greater, so the pressure drop across the valve was reduced from 12.61 to 12.52 psid. The pipe weight was reduced from 3,795 to 3,577 lbm.

Figure 5: Output window for Unequal Pipes Scenario

Conclusion

This example has demonstrated how simple it is to perform an automated sizing to design a system for a set of requirements. Pipe weight may not be directly proportional to pipe cost but it can be implemented very quickly when cost data is not available.

Cooling System - Creating Libraries - ANS

Summary

This example focuses on creating Engineering and Cost Libraries for a system, then connecting the libraries to the model in order to obtain accurate results based on the cost data.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Using external libraries
- Building engineering and cost libraries
- Using Scale Tables for cost
- Connecting and using existing engineering and cost libraries

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

This example is an addendum to the Cooling System - ANS example, but can be worked separately from the example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Cooling System.fth*
- *US - Cooling System Libraries - ANS.fth*
- *Creating Libraries - Cost Templates.xlsx* - template file for building cost libraries
- *Creating Libraries.dat* - completed engineering library
- *Creating Libraries Costs.cst* - completed cost library for *Creating Libraries.dat*
- *Steel - ANSI Pipe Costs.cst* - completed cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Cooling System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select Delete All Children.

For the analysis we will be creating the library files for a completed model file to focus solely on using libraries as part of the costing and/or sizing process.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Creating an Engineering Library

If it is desired to size any of the junctions in the system, such as the pumps or elbows, the first step is to create an engineering library containing information for each of the junctions that will be sized. This engineering library can later be used to attach a cost library for the junctions. Engineering libraries can also be connected to future models to reuse any of the junctions in future systems.

Note: This step is not required if it is only desired to include pipe costs for the pipe material information included in Fathom, since the cost libraries for the pipes can be built off of the default Fathom Pipe Material Libraries.

To add a junction to an engineering library:

1. Define the junction in the workspace.
2. **Select** the junction, go to the **Library menu**, and choose **Add Junction to Library**.
3. Give the junction a meaningful name and add it to the desired library by selecting it from the list.

For the purpose of this example the model has already been fully defined, so we will just need to add the junctions to the library which we will be calculating the cost for. Follow the steps above to add Pump J4 and Bend J3 to the LOCAL USER LIBRARY. Give the pump the name *Sizing 3000 gal/min* and the bend

the name *STD Bend*. In this case only one of each junction needs to be added to the library, since each of the pumps and bends in the model are identical. If we had multiple pump models, bend types, etc. then we might need to add multiple pumps, bends, etc.

The local user library is the default location where user junctions are placed for the local machine. If the junctions are stored in the local user library, it is possible to run all of the calculations, and attach a cost library for the junctions. However, this restricts the use of the saved data, as the local user library cannot be shared with other users or be moved to different machines. This is why it is often preferred to place the junctions into an external library. Follow the steps below to move the junctions:

1. Go to the **Library menu** and open the **Library Manager**.
2. At the bottom, select **Create New Library**, and give it the filename *Creating Libraries*. It is important to use a descriptive filename to avoid confusion later if the library needs to be reconnected, or sent to someone else.
3. Enter a Library Description. This is the name that will be displayed within the Library Manager itself. The description may be different from the file name, but it is typically recommended to make these the same to avoid confusion later. Use the name *Creating Libraries* for the description.
4. In the LOCAL USER LIBRARY, expand the Junctions option. The window should appear similar to Figure 1.
5. For both junctions that we added earlier, right-click the name, then click **Move Content**. In the Select Library window, click Creating Libraries and then click **Select**. This moves the junction from the local user library to the new external library. The pump and bend should now be visible within their respective sections on the right side of the window.
6. Select **Close** to complete the process. If you open the folder where the library was created in Step 4, the library file should now be available.

We are now ready to enter the cost data for the junctions in the model.
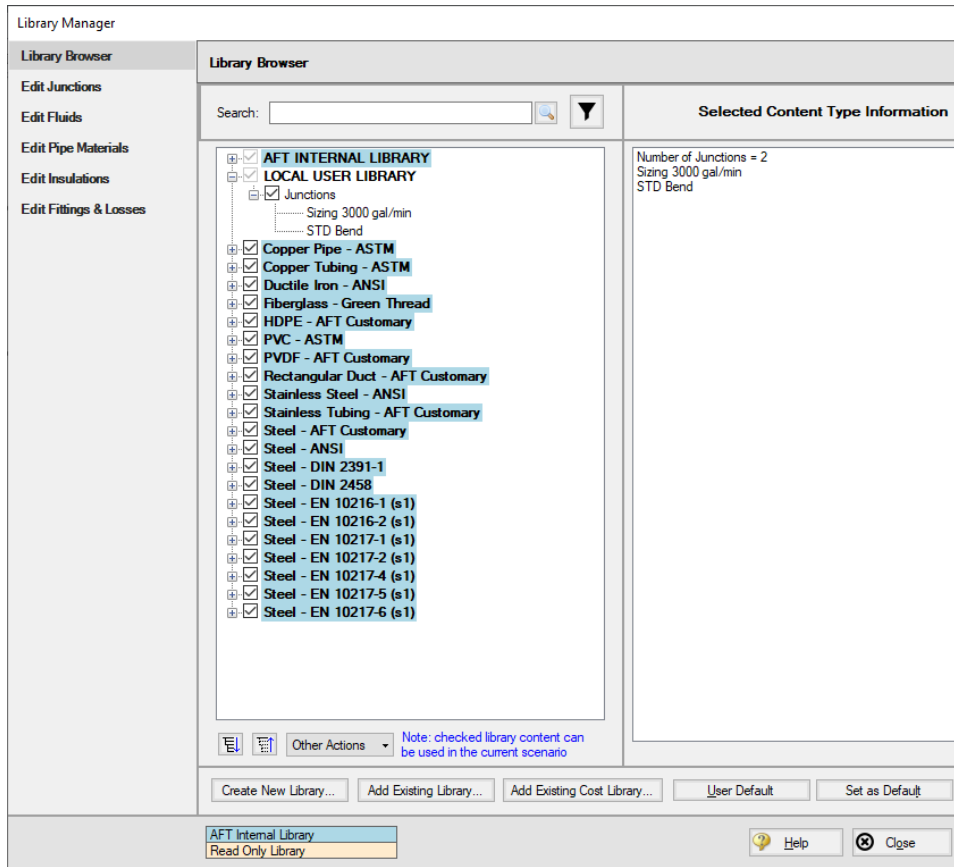
Figure 1: Edit Library tab in the Library Manager with a new external library created for the junctions to be added to
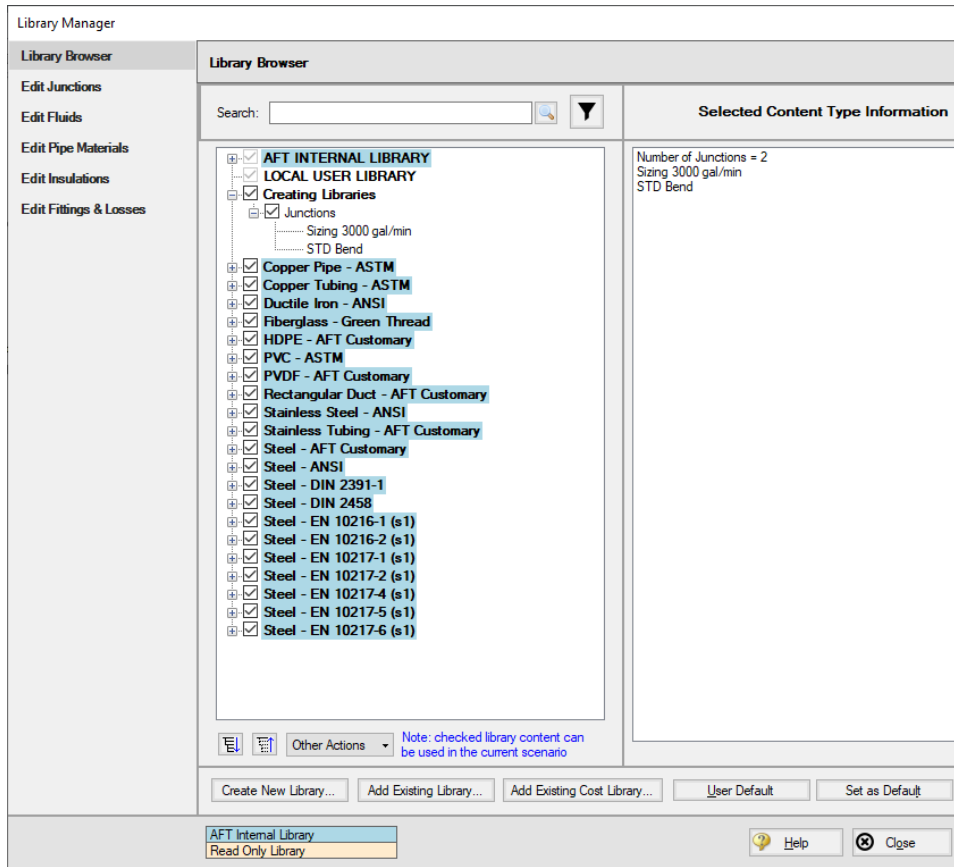
Figure 2: Library Manager showing the junctions moved to the new external library from the local user library

Step 5. Creating a Cost Library for the Equipment

Go to the **Library menu** and select **Cost Library** to open the Edit Cost Library window, which is used to create and view cost libraries. This window can also be accessed from the Sizing window on the Assign Cost Libraries panel. In this case we need to enter new cost data that we have obtained, so click **New**.

The list of Available Libraries should now be displayed as shown in Figure 3. Every cost library must be built off of an existing engineering or pipe material library. Browse to the newly created library, **Creating Libraries**. You may need to use the Browse button to find it if the library is not shown in the list. After you click the library to highlight it, click **Select**.
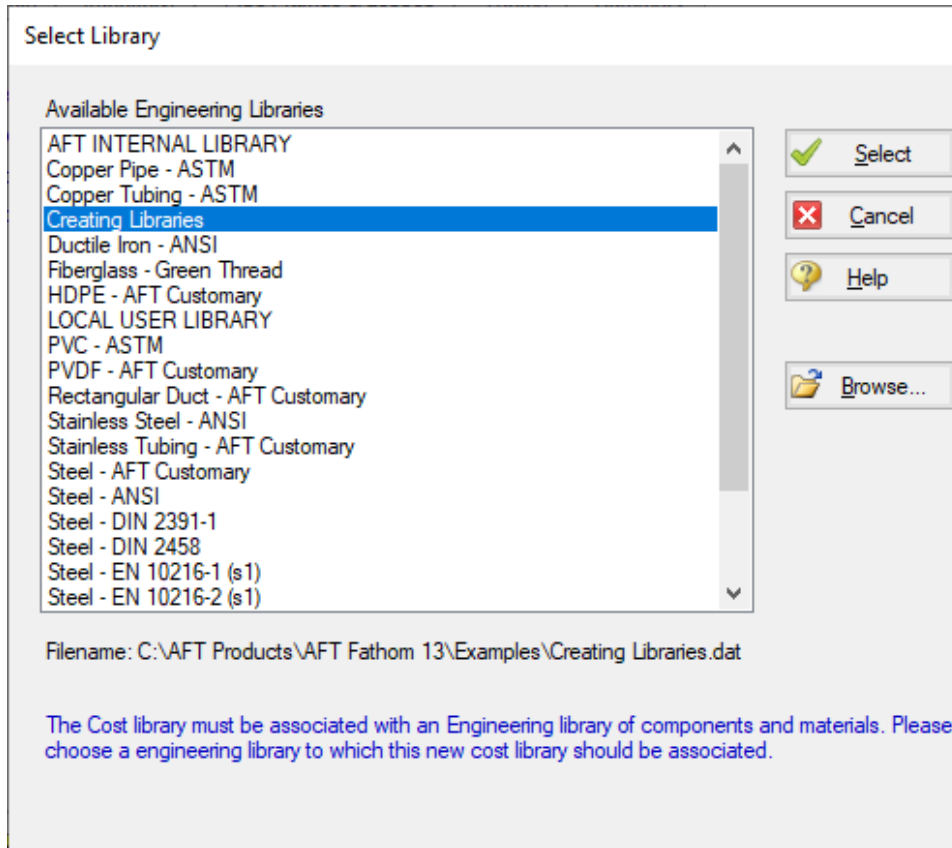
Figure 3: The Available Engineering Libraries list which can be used to build a cost library

From here we will be brought to the General tab. The description field is used as the name displayed within the Library Manager, similar to with the engineering library.

The Notes field can be used to provide information on references, or other details which may be useful for tracking purposes. Notes can be seen when reviewing library contents later using Browse Library Contents.

The other item of note is the Multipliers table. This can be useful to apply multipliers to account for inflation or uncertainty, without needing to calculate the costs directly. Note that multipliers can also be applied within the Assign Cost Libraries panel, so be cautious to avoid duplicate multipliers.

➢**In the Description field enter Creating Libraries Costs**, as seen in Figure 4. Now let's move on to entering cost data. Click the Junctions tab to view the Junctions overview.
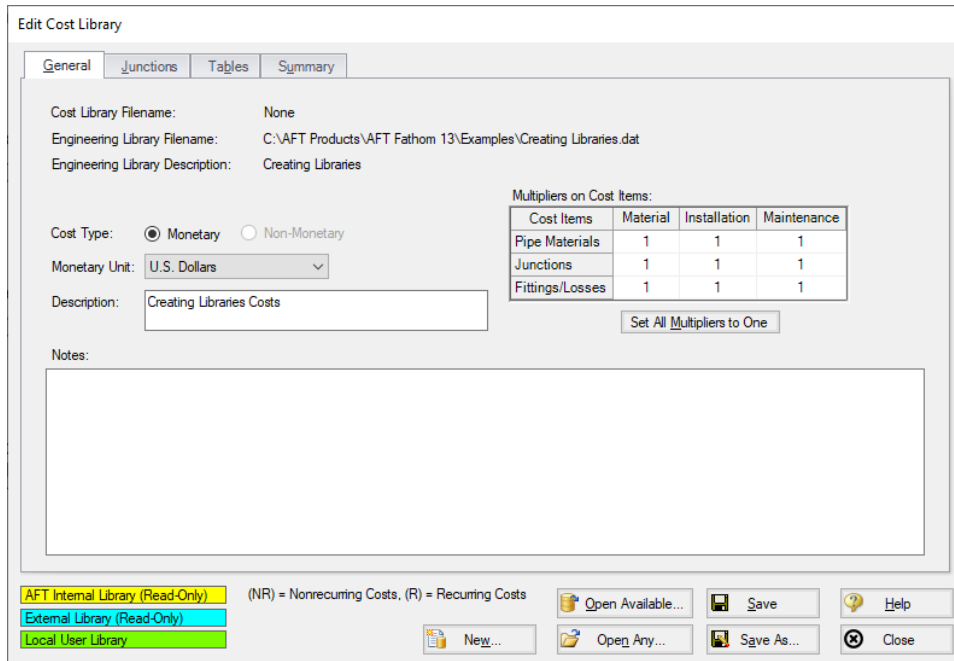
Figure 4: The General tab within the Edit Cost Library window

Within the Junctions tab each individual library item can be viewed under each Junction type. First expand the Pump section by clicking the +, and select the **Sizing 3000 gal/min** junction that was added earlier.

➤**Click New Cost** to add an item to the table. By default, a cost item will be added that is configured for cost entry per item. Costs can be categorized as Material, Installation or Maintenance Cost Type. This categorization becomes useful if it is desired to exclude certain costs from the calculation or the sizing, but otherwise does not impact how the costs are calculated.

Separately open up the Excel spreadsheet titled *Creating Libraries - Cost Templates.xlsx* found in the Examples folder, and go to the **Junction Costs (English)** tab. The cost data for the pumps and bends can be seen here, categorized by cost type.

In this case we are sizing the pump, so rather than having a cost for a specific pump model, we have a range of costs specified based on the power requirement for the pump. We will need to enter this data into the Tables tab first before we can define the cost item on the Junctions tab. Back in AFT Fathom in the Edit Cost Library window, click on the Tables tab at the top of the window.

➤**Click New Table** to begin defining the data table, and name it *US - Pump Material*. Multiple Table Types are offered as the dependent variables for the table, which can either be plotted against a set of multipliers which will be applied to a specific data point, or directly against cost, as can be seen in Figure 5.

The data we have available is the cost for pumps over a range of different power requirements, so choose **Power** and **Cost** as shown below, then click **OK**.
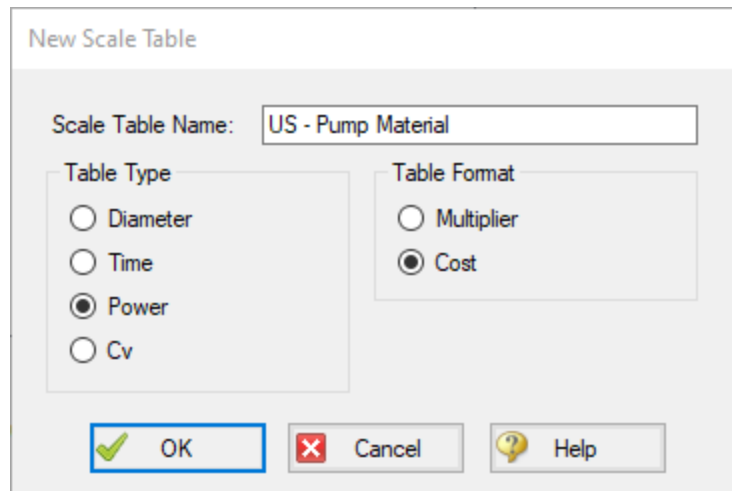
Figure 5: Table type and formatting options when creating a cost library

The new table name will now appear in the Tables list, and an entry section for Power vs Cost will become available. **Copy** the data in the Excel spreadsheet for Pump Material, then click the **Edit Table** button and choose **Paste**. Make sure that the table Units have been set to **hp**. Repeat this process of creating a new table for the pump installation costs using the data from the spreadsheet.

Now that the data tables have been created, we must assign them to the pump junction in the Junctions tab. Click on the **Junctions** tab and make sure that the **Sizing 3000 gal/min** pump model has been selected, then enter the cost information as follows:

1. Description = Purchase
   - This row is optional, but can be useful to provide clarification on the source of the entered cost.
2. Cost Type = Material (NR)
   - Note that NR denotes a non-recurring cost type, while R denotes a recurring cost type.
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = US - Pump Material

Click **New Cost** to create a second cost item for the pump and enter the cost information as follows

1. Description = Installation
2. Cost Type = Installation (NR)
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = US - Pump Installation

We have specified the cost types for each of the pump data tables, and have assigned them to our Sizing 3000 gal/min model contained in the engineering library. However, we still need to input the data for the Bends.

On the Tables tab, follow the same process as outlined above to **create two more tables** containing the cost information for the bends contained in the spreadsheet. Both tables will use cost for the Table Format, with Diameter as the Table Type. Be careful when filling in the table data to be sure that the correct units are applied for each table. When you are finished there should be four tables total, as shown below in Figure 7.

To assign the tables, return to the **Junctions** tab. Select the **Bend** and **add two cost items**, one applying the material costs table, and one for the installation costs table. When you are done the Bend should have two costs, as shown in Figure 8.
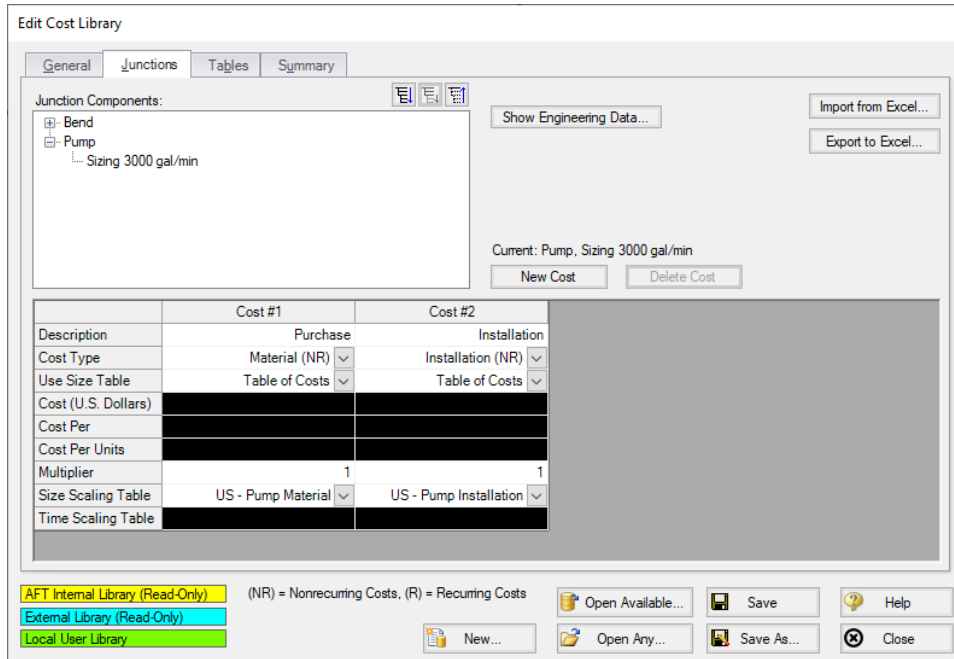
Figure 6: Pump Material and Installation costs assigned using cost tables
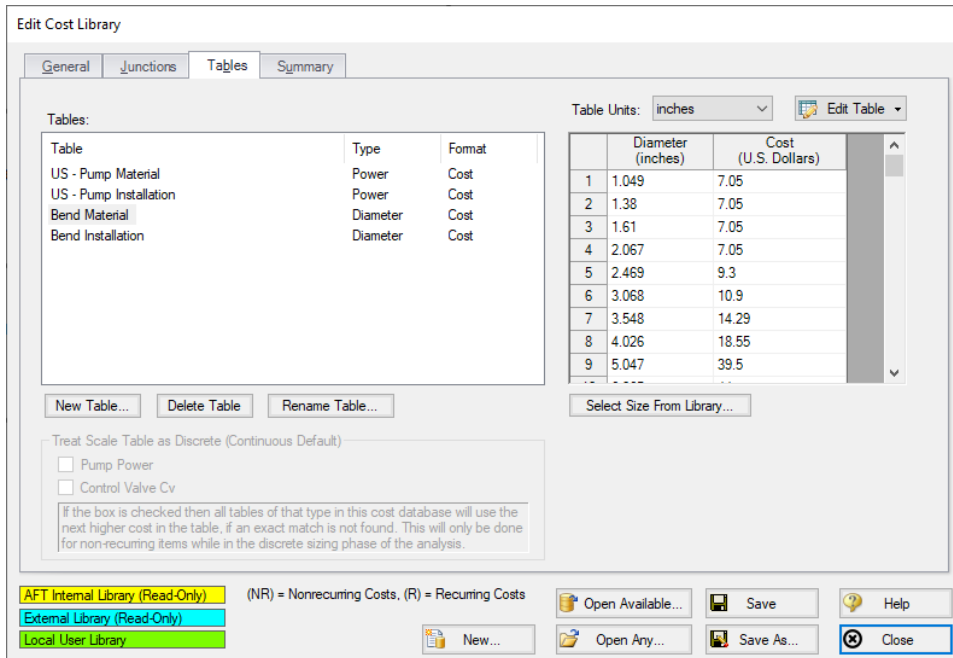
Figure 7: Complete tables list with material and installation cost tables for each of the junctions being analyzed
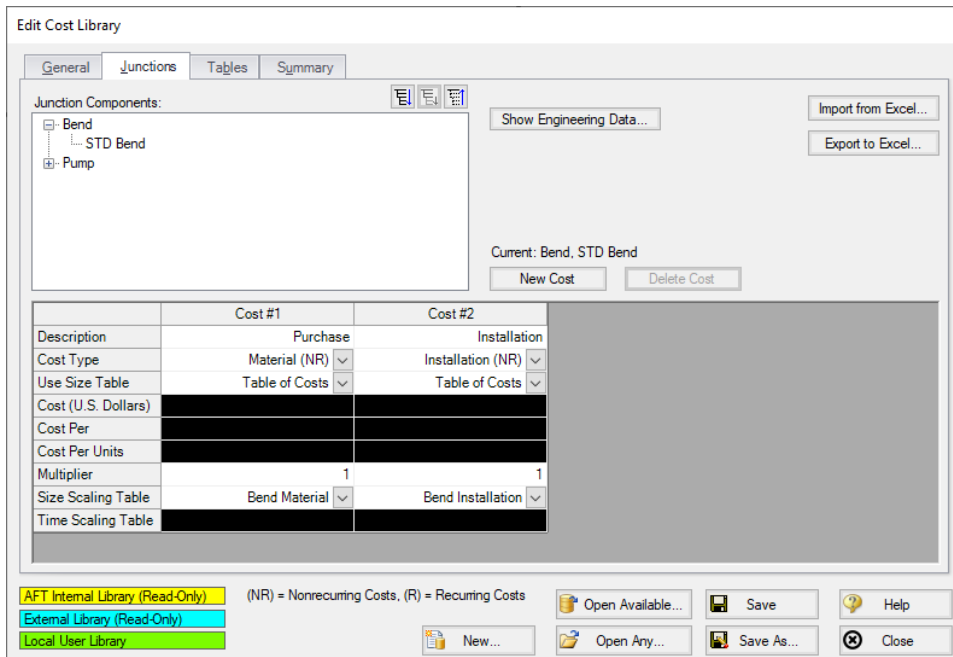
Figure 8: Completed cost items for the bend material and installation costs

Click **Save**, and give the cost library a name. We will use ***Creating Libraries Costs.cst*** in this case. It is often useful to give the cost library a similar name to the engineering library to make transferring and reconnecting the library faster. Once you have finished saving the library, click Close.

Step 6. Create Cost Library for Pipes

We now have costs for the equipment, but we still need costs for the pipes. In the Sizing tab in the model, go to the Candidate Sets panel by clicking the button. By expanding the existing Candidate set, it can be seen (as stated in the name) that we will need cost information for all STD Steel – ANSI pipe sizes from 1 inch to 36 inches.

1. Return to the **Edit Cost Library** window by going to the Library menu and selecting Cost Library, then click **New**.
2. From the Available Engineering Libraries list, click **Steel - ANSI**, which is the default AFT Steel-ANSI pipe materials library.
3. Click **Select** to choose this library. You should now be on the General tab of the Edit Cost Library window.
4. In Description, give the new pipe cost library the name *Steel - ANSI Pipe Costs*.
5. Go to the **Pipe Materials** tab. In the list section, we will be able to view all of the sizes and types for the Steel - ANSI material. Under Sort choose **Material, Type or Schedule, Nominal Size** so that they will be sorted by type rather than size.
6. Expand the **Steel - ANSI** list, then expand the **STD** list (Figure 9).
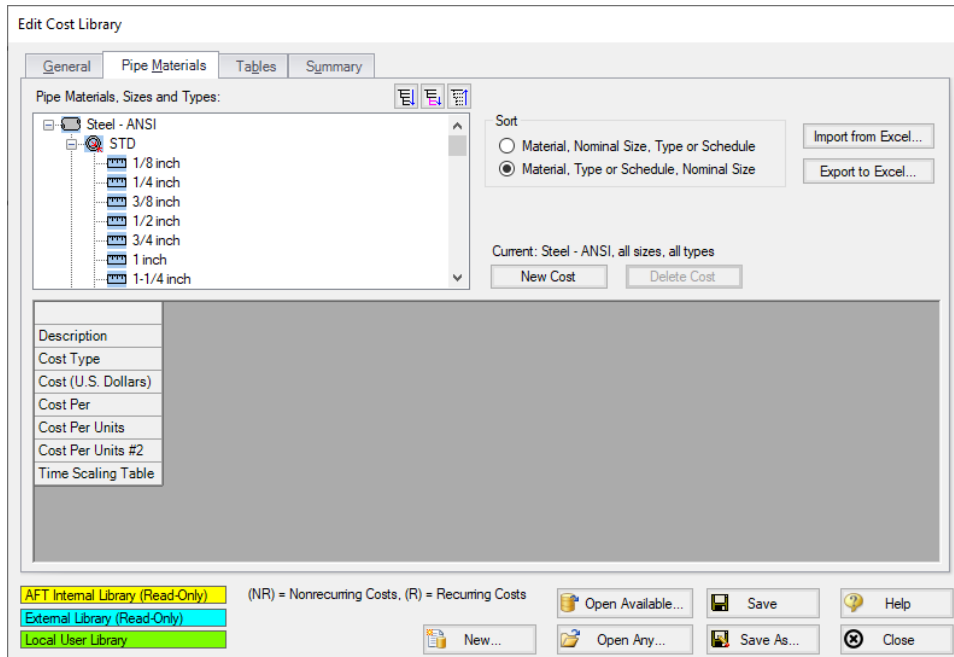
Figure 9: Pipe Materials tab in the Cost Library window showing the available STD sizes

If a size from the list is selected, the hydraulic diameter is displayed, along with a summary of the material, size, and type. The New Cost button will also become active (Figure 9). There are a few methods available when adding the pipe costs. We could individually add the Material and Installation costs for each pipe size by selecting each size from the pipe materials list, then using the New Cost button to add these costs to the table individually. Alternatively, the pipe costs can be entered into a template in Excel, then imported to the Cost Library all at once.

In this case, we have the Pipe costs available in a spreadsheet, so we will use the Excel import/export option. To do this we will first export a template for the desired pipe sizes, then we will enter the cost information in the template and import the updated file as detailed below.

1. Make sure that the STD pipe material type is highlighted in the list, then click **Export to Excel** on the right side of the window. The Excel Export window should be shown as in Figure 10, with all Steel-ANSI STD pipe sizes displayed.
2. Click **Export**, name the sheet, and save it to an easy to find location. The name does not matter, but we will need to be able to find this spreadsheet later to import back into the cost library.
3. When Fathom has finished the export a message will be displayed notifying the user of a successful transfer. Click **OK**, then browse to the exported worksheet and open it. The exported worksheet should contain a list of the selected pipe sizes, with empty columns where the cost per unit length can be entered for each cost type, as shown in Figure 11.
4. Open the ***Creating Libraries - Cost Template.xlsx*** spreadsheet with the costs for this example. Navigate to the Pipe Costs tab, and copy the Material and Installation costs for the 1 inch to 36 inch pipe sizes into the worksheet that we just exported. Ensure that data is entered for material and installation for each of the sizes from 1 to 36 inch, and that the length units are listed as feet in the worksheet, since these were the length units used by the manufacturer for their pricing.
5. Save and close the worksheet.

Note: The length units used for the spreadsheet are taken from the default model units, which can be changed from User Options in the Tools menu.
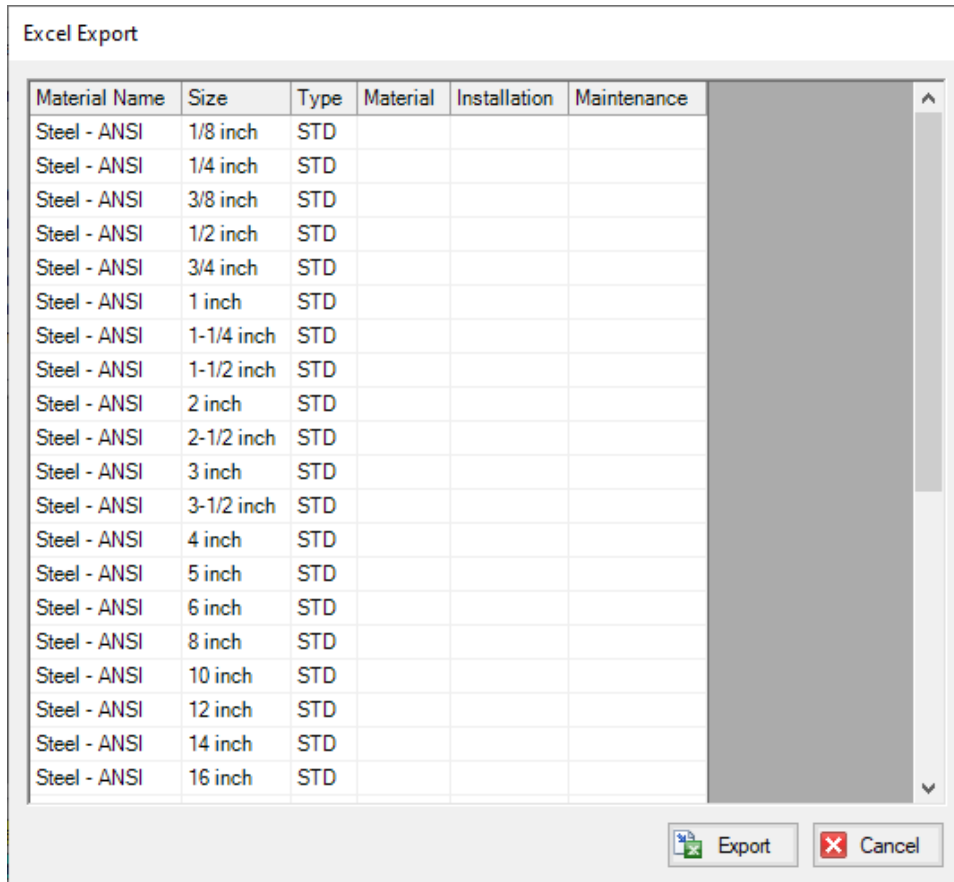
Figure 10: Excel Export window showing pipe items that will be included in the Excel sheet

Figure 11: Initial Excel sheet after exporting template for the STD Steel-ANSI pipe sizes

Back in Fathom in the Cost Library window, click the Import from Excel button, then browse to the work-sheet we just updated and open it. When the Import has finished click **OK** on the notification from Fathom, then click the 1 inch pipe size under the Steel - ANSI STD pipe type. The Material and Install-ation costs should now be filled in, as shown in Figure 12. Click Save to save the cost library with a

unique file name **Steel - ANSI Pipe Costs.cst**, select Yes to make the library connected, then close the window.

Note: When using the Export/Import features for the cost library, it is not required to enter information for every size that was exported in order to import the worksheet. It is important to not delete any headings or descriptive information for the pipe data that was included in the template. Worksheets can be exported /imported as many times as desired, but whatever data is in the spreadsheet will overwrite anything existing in the library, meaning that if the costs are left blank for a size, any cost information will be deleted for that pipe size in the library. Due to this fact it is recommended to export the current information in the library first before importing any change information so that an accidental overwrite does not occur.

In the Library Manager, the new cost library should now be shown for the Steel - ANSI pipes, as shown in Figure 13, indicating that the pipe cost library was automatically connected.
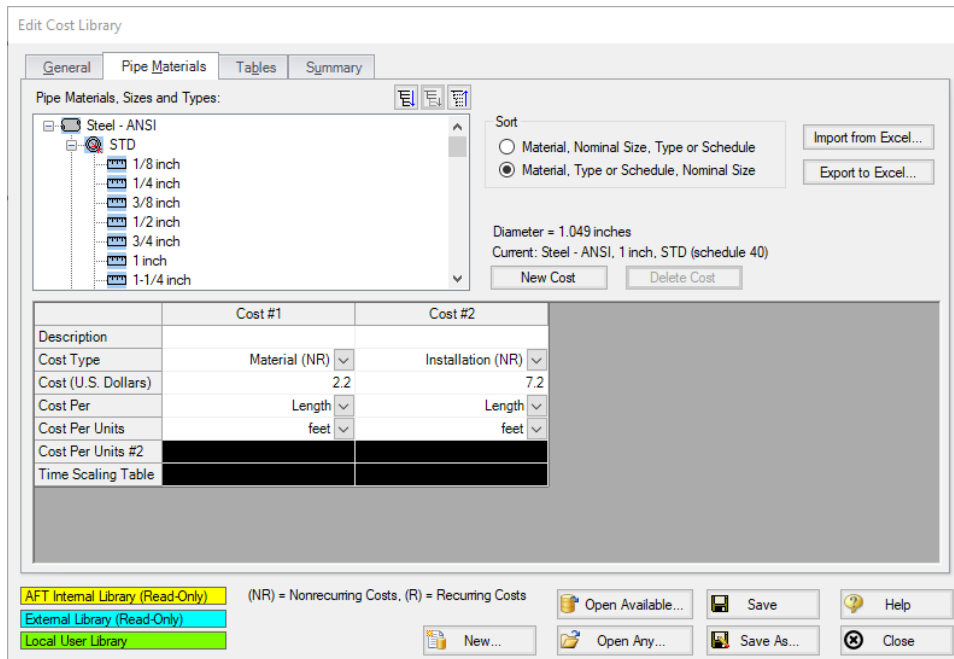
Figure 12: The Pipe Materials tab in the Cost Library window, showing the data recently imported
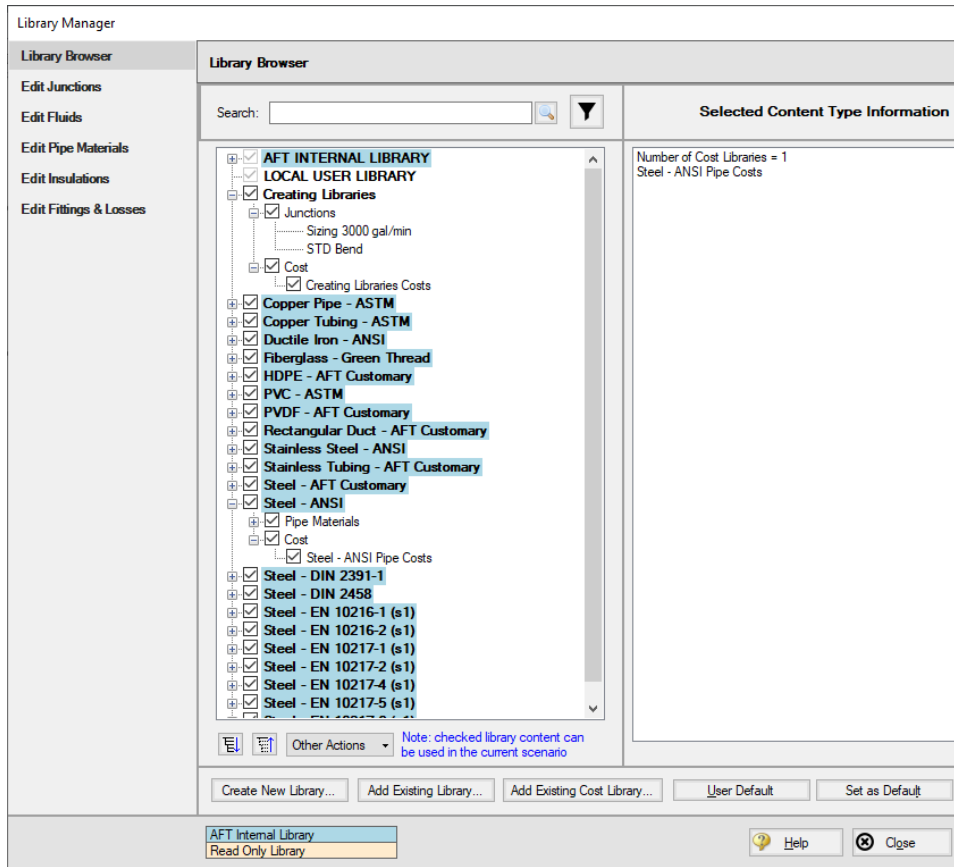
Figure 13: Library Manager after creating a engineering and cost libraries

Step 7. Define the Sizing window

To make use of the newly creating engineering and cost libraries, a few inputs are required in the **Sizing** window. Go to this window now.

A. Sizing Objective

By default, you should be on the **Sizing Objective** panel. For the Objective, select **Monetary Cost**. Then select **Calculate Cost, Do Not Size**.

B. Size/Cost Assignments

On the Size/Cost Assignments panel, make sure the **Pipes** button is selected at the top. Click **All** on the bottom left then click **Always Include in Cost** at the top to add all pipes to the Include in Cost list.

Click the **Pumps** button at the top to go to the pump size/cost assignments page. Select the radio button for both Pump 4 and 5 to **Include in Cost Report Only**.

Finally, go to the **General Junctions** button and add all four bends to the **Include in Cost Report Only** list.

C. Assign Cost Libraries

Go to the Assign Cost Libraries panel. **All** 22 pipes should assigned to the **Steel - ANSI Pipe Costs** library, as shown in Figure 14. Check the **Pumps** and **General Junctions** pages as well to confirm that those junctions are connected to the **Creating Libraries Costs**, as shown in Figure 15
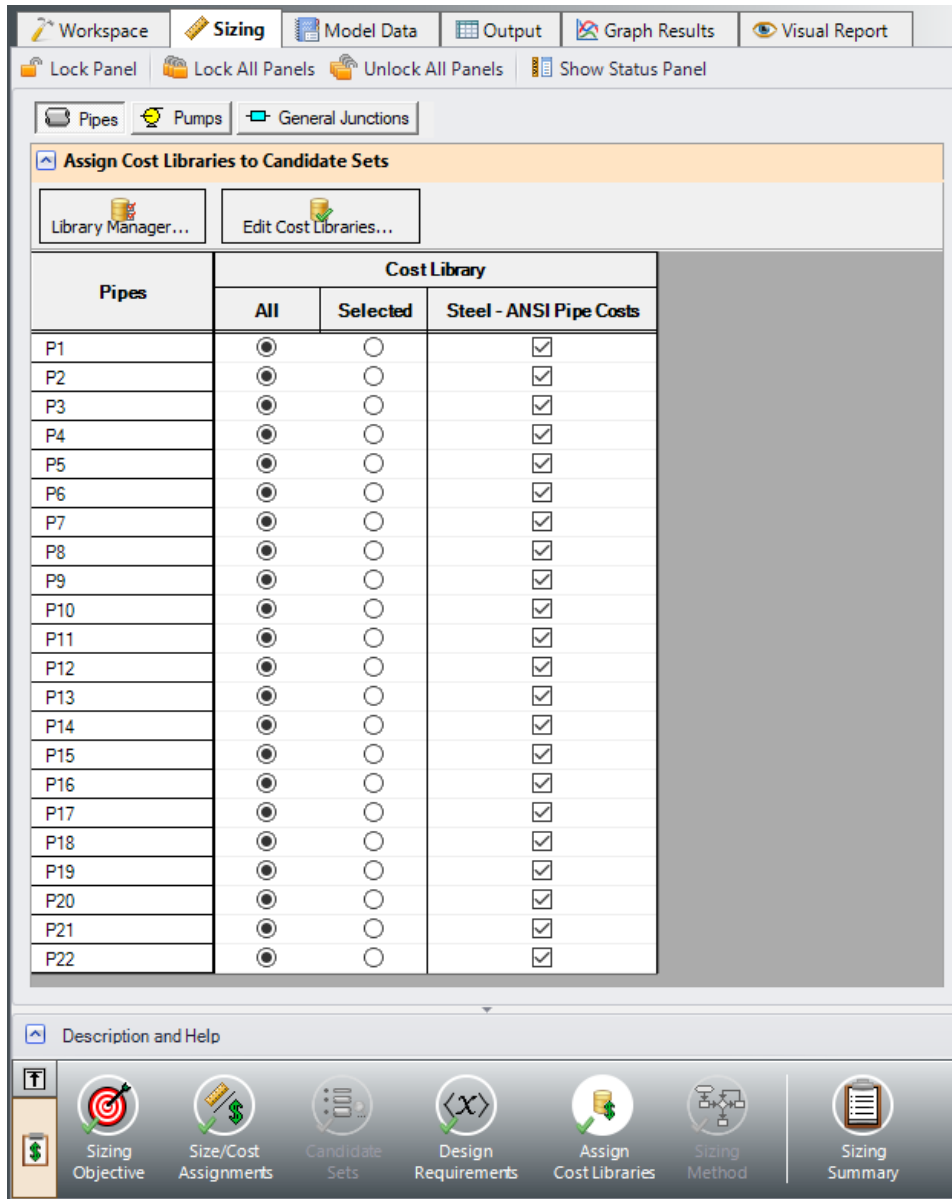
Figure 14: The Assign Cost Libraries panel with the new pipe cost library applied
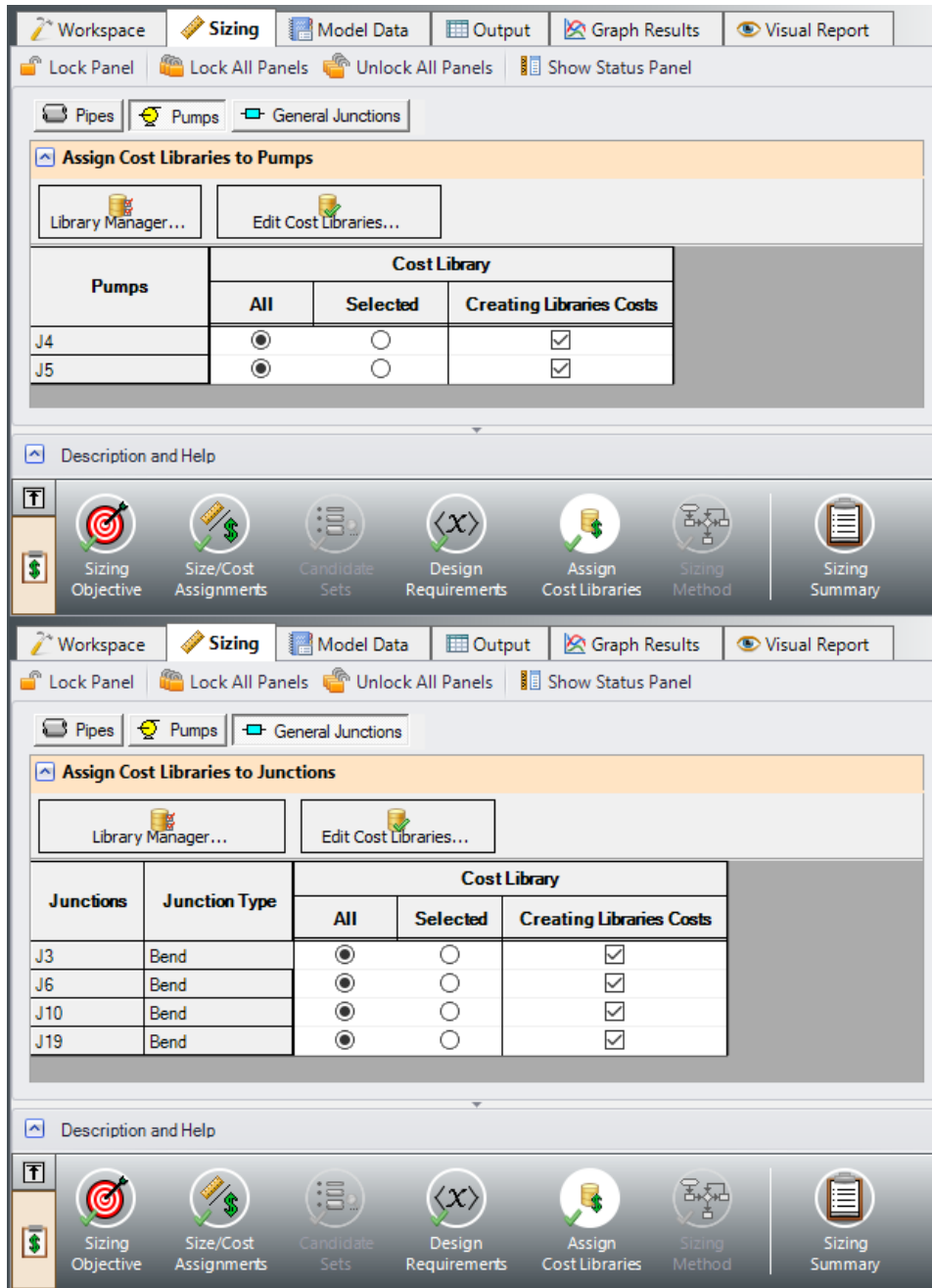
Figure 15: Applying the newly built cost library for the pumps and bends on the Assign Cost Libraries panel

In order to link our libraries to specific junctions in the model, we will need to apply the Engineering Library to the junctions in the Workspace. Open each of the **pump** junctions and select the **Sizing 3000 gal/min** item from the Library Jct list. Repeat this process for the four **bend** junctions in the model. Note that in this case the only thing that should change in the properties windows will be the name of the item from the Library list, since we created the library junctions based off of the junctions already defined in this model.

As a quick summary, we first connected the new libraries using the Library Manager. We then connected the junctions to the Engineering Library by using the Library Jct selection in each of the properties windows. In the Sizing tab we used the Assign Cost Libraries panel to select the corresponding Cost Library for each pipe and junction. The Assign Cost Libraries section is now complete.

Step 8. Run the Model and Check Output

Click Run from the toolbar or Analysis menu. Before the solver runs, the Model Diagnostics window will appear. Because there was no design requirements specified for the control valves, the control valves will add pressure if necessary. Typically, these messages should not be ignored. However, we will see from the output that the control valves will not add pressure. Therefore click Close on this message to dismiss it.

When the solver finishes, go to the Output window. On the Cost Report tab, the total cost should be $35,608. In the Library Sources tab in the Junction section, each of the Bends and Pumps should list the Creating Libraries Costs as their library source as shown in Figure 16. The Library Sources in the Pipes section should similarly show the pipe cost library Steel - ANSI Pipe Costs for both Material and Installation.

Figure 16: Output tab showing the current Library sources used in the model calculations

Cooling System - ANS

Summary

This example focuses on a closed loop cooling system that demonstrates some key features in using ANS to size a piping system. An existing model is used to investigate four potential sizing cases:

1. Size system for **initial cost** with a 10-year operating period
2. Size system for **life cycle** cost with a 10-year operating period
3. Size system for **life cycle cost** with an **initial cost limit** for a 10-year operating period
4. Size system for **life cycle** cost using a **manufacturer's pump curve** and with a 10-year operating period

The existing model was used to demonstrate AFT Fathom's capabilities for modeling closed-loop systems, pump curves, heat exchanger resistance curves, and fixed pressure drop control valves. This example will demonstrate how a system like the one built before can be seamlessly scaled up, meet design requirements, and minimize cost. For instance, the flow rate through the system will increase from 900 in the original model and system to 6,000 gal/min. The pipes must be sized and their costs minimized accordingly.

> **Note:** This example can only be run if you have a license for the ANS module.

Topics Covered

- Connecting and using existing engineering and cost libraries
- Sizing for initial or life cycle cost
- Using an initial cost limit with life cycle cost sizing
- Sizing a pump and sizing piping for the chosen pump
- Interpreting Cost Reports

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Cooling System.fth*
- *US - Cooling System - ANS.fth*
- *Cooling System.dat* - engineering library
- *Cooling System Costs.cst* - cost library for *Cooling System.dat*
- *Steel - ANSI Pipe Costs.cst* - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Cooling System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**.

Although the model is fully defined, a few alterations are necessary for this example to scale up the system. First make sure that the Workspace looks like Figure 1 below:
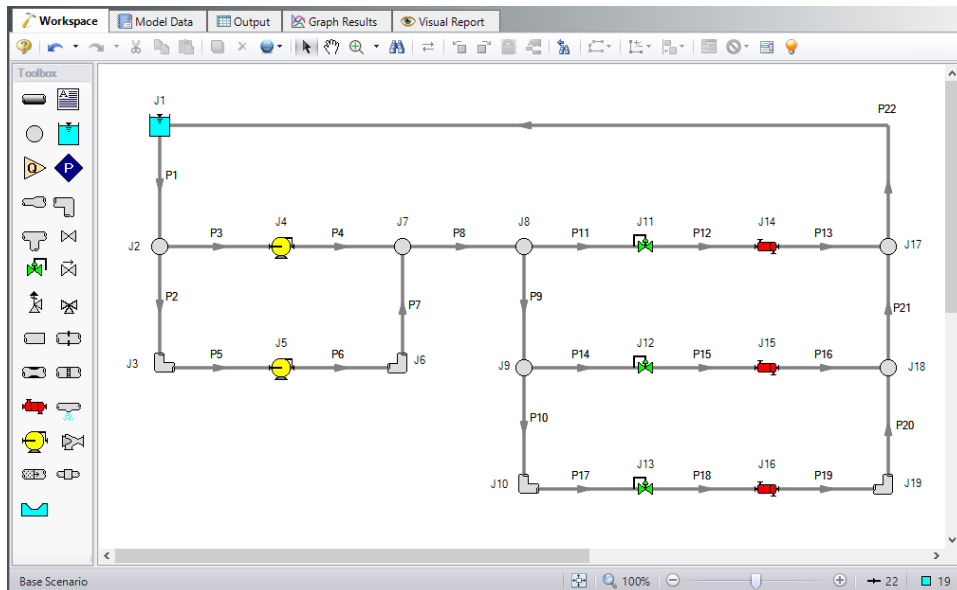
Figure 1: Workspace for Cooling System model

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 70 deg. F

Step 5. Define the Pipes and Junctions Group

As an initial guess to the scaled up system, we will change all pipe sizes to 36 inch. This should satisfy all Design Requirements and be a feasible solution, therefore it should be an acceptable initial guess for the system. Similar the junctions will be updated for the scaled up system.

Pipe Properties

Make the following changes to all pipes

1. Pipe Model tab
    a. Size = 36 inch
    b. Type = STD

Junction Properties

Make the following changes to these junctions.

1. J1 Reservoir
    a. Liquid Surface Elevation = 30 feet
    b. Liquid Surface Pressure = 0 psig
    c. Pipe Elevation = 0 feet
2. J11 & J12 Control Valves

a. Valve Type = Flow Control (FCV)

b. Control Setpoint = Volumetric Flow Rate

c. Flow Setpoint = 2000 gal/min

d. Always Control (Never Fail) = Unchecked

3. J13 Control Valve

a. Valve Type = Constant Pressure Drop (PDCV)

b. Control Setpoint = Pressure Loss

c. Pressure Drop = 5 psid

4. J14-J16 Heat Exchangers

a. Enter Curve Data =

Volumetric	Pressure
gal/min	psid
0	0
2000	50
4000	200

Step 6. Overview of Sizing Systems with Pumps

A centrifugal pump can be modeled with either the Pump Curve or Sizing Analysis Type. Generally if a specific pump has been identified the Pump Curve type would be chosen, whereas the Sizing option is useful during the selection stage in order to identify the pumping requirements for the system. In some cases even though the specific pump has not been chosen several candidate pumps are available. For this third case it would be best to model each candidate pump as a pump curve using separate scenarios for each pump.

Sizing a Pump

The pumps in our cooling water case are being sized, so the Sizing Analysis Type was used with assigned flows of 3000 gal/min. In order to allow the pumps to be defined as fixed flows the flow control valve at J17 has been defined at a fixed pressure drop of 5 psid to ensure that the minimum pressure drop requirement for this valve is met during the sizing stage. This model thus represents a first cut model where, based on the results, the two pumps will become actual pumps with pump curves, and control valve J17 will become a flow control valve controlling to 3000 gal/min, similar to control valves J9 and J13.

As already stated, when a pump is modeled as an assigned flow, it is not a specific pump from a specific manufacturer. Thus, the costs for the pump can only be approximated. To a first approximation, it should be possible to estimate the non-recurring cost (i.e., material and installation cost) for the pump as a function of power requirements. For instance, a typical one horsepower pump may cost $2000, and a ten horsepower may cost $4000. Other typical costs for different power requirements can be approximated. The actual cost for the pump will, of course, highly depend on the application. Pumps for special

applications such as corrosive fluids or high temperatures may have different cost structures, but a reasonable approximation of cost for various power usage levels should be possible. These costs are entered into a cost library, and are accessed by the solver. As ANS evaluates different combinations of pipe sizes, each combination will require a certain power from the pump. With a cost assigned to this power, ANS can obtain a cost to enter into the objective function that it minimizes. Later in this example, we will look at the costs for the pumps.

In addition to non-recurring costs, recurring costs can also be estimated. Specifically, the cost of the power used by the pump over a period of time can be determined. All ANS needs is an overall pump efficiency to determine an actual power from the ideal power. Again, since we do not have a specific pump selected yet, the efficiency can only be approximated. ANS calls this the Nominal Efficiency. In addition, users can provide a Nominal NPSHR.

One may ask how important it is to choose appropriate values at this level. It depends. If the model only sizes for non-recurring costs, then the nominal efficiency will affect the actual power requirement of the pump, and hence the cost obtained from the cost library. If the cost of the pump is large compared to the cost of the piping, the specified nominal efficiency will likely play a significant role in the system selection.

In addition, for models that include energy costs in the objective, the energy cost may be a minor or major portion of the overall system cost. If it is a major portion, the results of the sizing will likely be very sensitive to the specified nominal efficiency.

Therefore, it is recognized that at this first cut phase the values for nominal efficiency and nominal NPSHR are imprecise and users may need to go through a few iterations to arrive at satisfactory results.

Modeling Pumps with a Pump Curve

When one is modeling a pump curve, accurate NPSHR and efficiency (or power) requirements can be specified for the pump. The cost data for that pump can also be accurately specified. During the automated sizing run the impact of the pump on the system sizing will be in the area of energy costs and in the requirement to satisfy the pump's design requirements.

First, let's discuss the issue of recurring cost. The non-recurring costs for the pump will be fixed, and will therefore not impact the sizing process. However, energy costs will vary with the power required for the pump. Thus including energy cost in the sizing will influence the pipe size selection.

Second, the specific pump has certain design requirements. For example, the sized network will be of no value if the NPSHR is violated. Thus, the pipes have to be selected by ANS in such a way as to satisfy all pump design requirements.

Since a specific pump is not input for our cooling water model (yet), no specific pump selection can happen at this point. It may frequently be the case that, based on the results of modeling the pump as an assigned flow pump, several candidate pumps may exist. The best approach is to create a scenario for each of these pumps, and rerun the sizing for each candidate pump. This will yield an ideal candidate pipe system for each candidate pump, from which an educated choice can be made for the system design. All else being equal, the selection would be the candidate pipe system that offers the lowest cost. When approached in this manner, one is selecting the pump and the piping system as an entire system, which offers improved matching of the pump to the system, and also offers significant opportunity for cost reduction.

Summarizing the Pump Selection Process

Based on the preceding discussion, we can outline a process for selecting a pump that offers a matched pump and piping system at the lowest possible cost. This discussion is an extension of the discussion in the Fathom Help file regarding sizing pumps and pipes concurrently. Here, we will augment that discussion with the considerations that are relevant to sizing.

1. Phase 1: Size the pump
    a. Model the pump (or pumps) as assigned flow pumps using the Sizing Analysis Type. If flow control valves exist, select the one that offers the lowest pressure drop margin (as discussed in the AFT Fathom Help file), and set that one to a pressure drop control valve at the minimum required pressure drop.
    b. The largest uncertainty at this point is the cost of the pumps. Based on actual pump cost data, specify the approximate cost of pumps of different power levels for different flow rates. This is entered into a cost library.
    c. Along with the design flow rate, specify a nominal efficiency that is consistent with the application and your experience. If you are unsure, take a guess and, after completing Phase 1, return to this step and improve your guess.
    d. If you have a reasonable idea of what the NPSHR will be, specify that in the nominal NPSHR. If you are unsure, ignore nominal NPSHR and proceed. After completing Phase 1, check the NPSHA and make sure that pumps are available at the head requirements obtained from the ANS results that have sufficient NPSHR margin.
    e. Determine whether the design goal is to minimize initial cost or life cycle cost, and select the appropriate objective in the Sizing Objective panel.
    f. Run the model and review the results. Take note of the relative cost difference between the piping and the pump(s). If the pump costs are a significant percentage, then continued iteration on Phase 1 will likely be worthwhile. Continued iteration would involve obtaining more precise cost information for pump(s) at the flow, head and power usage levels obtained from the Phase 1 results. It would also involve obtaining more precise information on nominal efficiency and nominal NPSHR.
2. Phase 2: Select a pump
    a. Phase 1 worked with approximate cost and engineering data for your pump. With that information ANS sized the network for an optimal pipe and pump system combination. At this point, that optimal system should be viewed as preliminary. Based on the pump operating point from Phase 1, identify some candidate pumps that match the requirements.
    b. Change any flow control valves set to pressure drop control valves (as specified in Phase 1a) back to flow control valves.
    c. Identify the operational requirements for the flow control valves, such as minimum pressure drop, and enter those as Control Valve Design Requirements.
    d. For each candidate pump, create a child scenario. Each child scenario should be created below the scenario that has the flow control valves as discussed in Phase 2a and 2b.
    e. Enter the pump head, NPSHR and efficiency (or power) curve. Enter the actual pump cost as well into a cost library.
    f. Run each scenario. All else being equal, the scenario with lowest cost will be the preferred design. The pipe sizes obtained from this final solution may differ slightly from those obtained Phase 1. If there are significant pipe size differences between Phase 1 and Phase 2, it may indicate that the actual pump was not matched very well to the Phase 1 requirements. It could also mean that there was an error somewhere in the process.

g. In Phase 2f, it may not be true that all else is equal. For example, even if one design offers lower cost, you may prefer a higher cost system because your experience suggests the pump is more reliable. On another note, if it is possible to quantify the cost reliability, this can be included in the cost of the junction and the automated sizing process.

For this example, we will be completing the pump selection process above considering the objective of minimizing initial cost, then minimizing life cycle cost.

Step 7. Configure Sizing Settings

Load the Base Scenario. The hydraulic model is already defined for a regular Fathom run, but we need to complete the sizing settings before running the analysis. Navigate to the Sizing window by clicking the Sizing tab at the top.

A. Sizing Objective

The Sizing Objective window should be selected by default from the Sizing Navigation panel along the bottom. For this analysis, we are interested in sizing the system considering the monetary cost for the initial costs only.

1. Sizing Option = Perform Sizing
2. Objective = Monetary Cost
3. Option = Minimize
4. Options to Minimize Monetary Cost = Size for Initial Cost, Show Energy Costs
   - This will update the cost table to include both initial costs in the sizing calculation, and moves the Energy costs to be included in the Cost Report so that we can later compare the costs in this scenario to one where the Energy costs are included for the sizing. We do not have any information on Maintenance cost, so we will leave this cost in Ignore Cost.
5. System Life = 10 years
6. Energy Cost = 0.11 U.S. Dollars Per kW-hr
   - Note this assumes the pump will run all day, every day, for the entire 10 years. Alternatively, we could have created an Energy Cost Library, which would be connected in the Assign Cost Libraries section. An Energy Cost Library allows variable energy costs to be used to account for varying prices during different times of day, different seasons, etc. For now, we are simplifying the input by using a fixed cost.

The Sizing Objective window should now appear as shown in Figure 2.
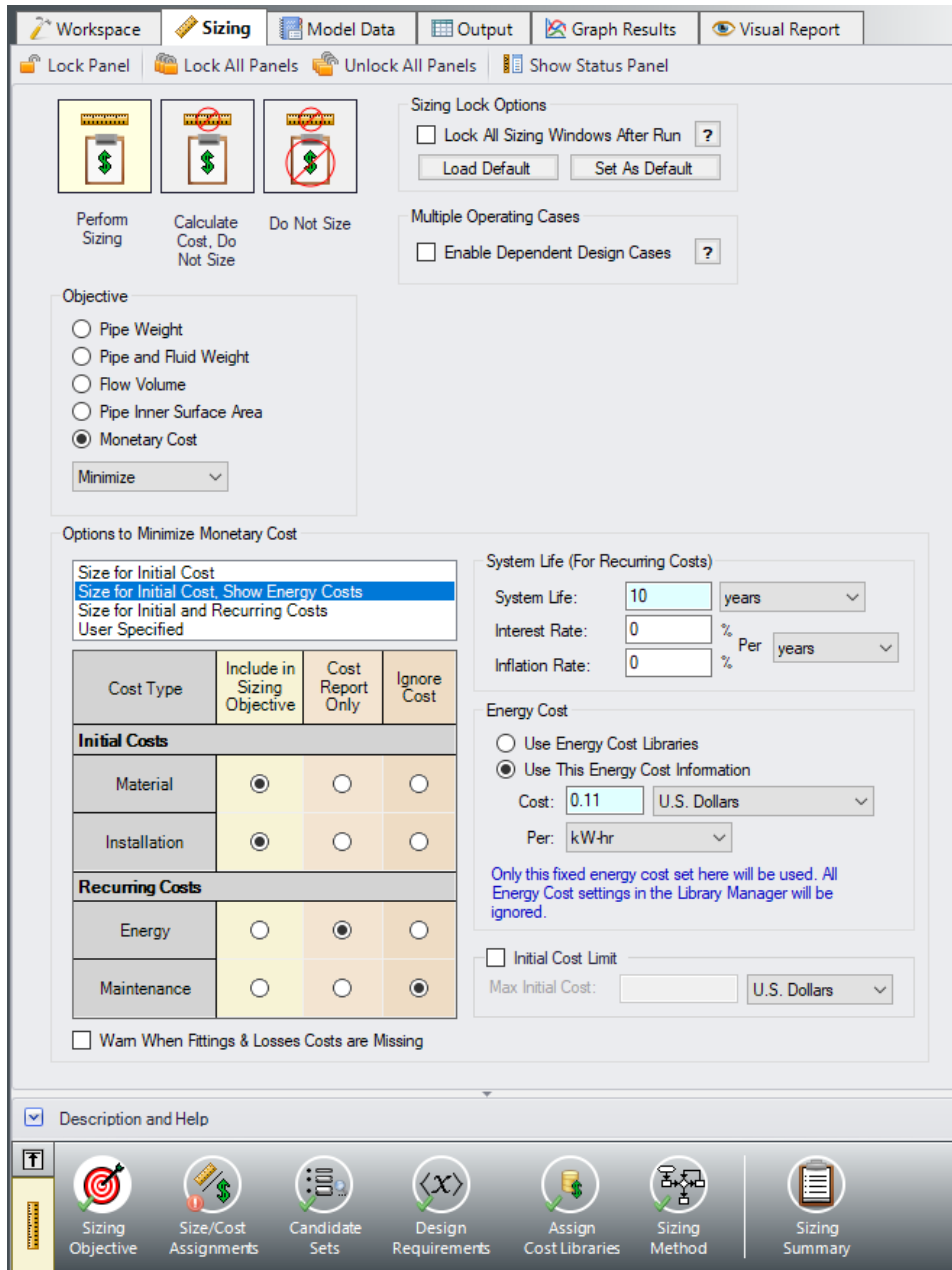
Figure 2: Sizing Objective panel setup for the initial cost analysis

B. Size/Cost Assignments

On the Sizing Navigation panel select the Size/Cost Assignments button. For this example we are sizing both the pipes and pumps for a new system, so we will need to calculate and minimize cost for all related objects in the model.

➢**Click All under the table** to select all of the pipes, then click the Always Include in Cost button next to Move to. This will move all of the pipe selections to the corresponding column to be sized.

To decrease the run time and reduce complexity, as well as to enforce some design uniformity, it is useful to create Common Size Groups to link the pipes in different sections of the model.

For example, if all the 22 pipes are selected to be Not In Group, then 22 unique pipe sizes will be selected by the automatic sizing engine. That could potentially mean, for example, different suction pipe diameters in pipes 1 and 2. A similar thing would happen for discharge pipe diameters. If one wanted the suction pipes 1 and 2 to be sized to the same diameter, one would group them into a Common Size Group. To see how this works, in this model all suction and discharge piping for both pumps will be put into a Common Size Group. What this means is that there will be one unique diameter chosen for this group of pipes, rather than potentially six different diameters.

For this system we will define five groups to be sized in this model. To set the groups, it is standard to select pipes which must have the same size due to their placement relative to other pipes and junctions, such as pipes at the entrance and exit of valves. In Figure 3 we have identified the five sections of the model to group the pipes.
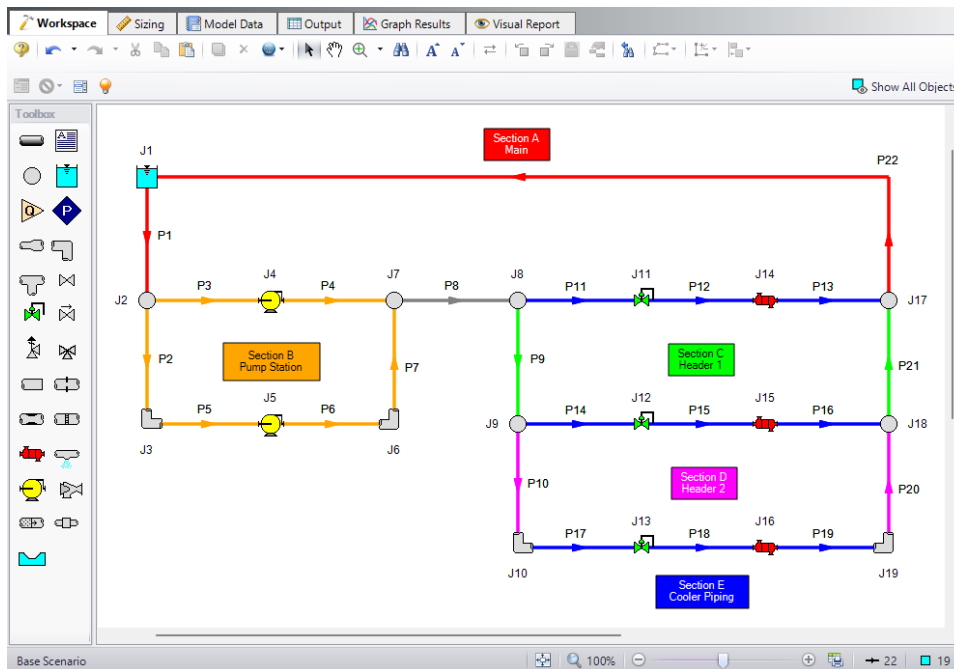
Figure 3: Workspace with separate sections identified to create Common Size Groups

First let's create a *Main* size group for pipes 1 and 22, which are red in Section A. We would expect these lines to be sized the same, as the inlet and outlet for the cooling reservoir.

> **Note:** If desired, the coloring and annotations in Figure 3 and in the completed model file can be recreated using Workspace Layers.

➢**Click New above the Pipe Grouping section** to create the group, and name it *Main*. Select the radio buttons under *Main* next to 1 and 22 to add them to the group. Alternatively, both pipes could be selected

in the Workspace, then be added to a new group by right-clicking the Workspace and choosing Add Pipe (s) to Common Size Group.

Now let's repeat this process for the piping in Section B around the pumps and control valves. Since this section contains identical pumps operating in parallel, we would expect the pipes in this section to have a common size. Similar to above, create a new group called *Pump Station*, and add pipes 2-7 as shown in orange in Figure 3 in section B to the new group by selecting them in the Workspace, or using the radio buttons.

Next, let's consider Section C-E, the piping near the coolers. In this case we have three coolers in parallel, with several segments of pipe connecting the cooler piping segments. We would expect the pipes directly connected to the coolers to have a common size, but the header piping (9, 10, 20, 21) would likely need to be larger, so we will group them separately. Create three Common Size Groups as follows:

1. *Header 1* with 9 and 21 (Section C, lime green)
2. *Header 2* with 10 and 20 (Section D, magenta)
3. *Cooler Piping* with 11-19 (Section E, blue)

The completed Size/Cost Assignments can be seen in Figure 4. Note that pipe 8 should be the only pipe not included in a group. This leaves six different pipe sizes that the ANS module will independently solve for during the automated sizing, one for each of the groups, and one for pipe 8. This is indicated in the bottom right of the Size/Cost Assignments panel, which shows that six independent pipe sizes exist for this configuration.

Note: Pipes in the Size/Cost Assignments table can be sorted by Common Size Group instead of sorting by number by clicking on the Pipe Grouping header. Clicking on the header for a particular group will sort the pipes in that specific group to the top of the window.

Figure 4: Size/Cost Assignments panel with Common Size Groups defined as outlined above

➤**Click the Pumps button** to switch to view the Pump Size and Cost Assignments. Select the option to Include in Cost Report and Sizing for both pumps so that the ANS module will size the pumps along with the pipes as shown in Figure 5.

Similar to the Common Size Groups for pipes, Pumps can be placed into Maximum Cost Groups based on monetary cost. If pump costs are added to a Maximum Cost Group, the ANS module will determine the pump in the group with the maximum costs, and set all pumps in the group to use the largest cost. In this case we will allow the pumps to be sized independently, so no groups need to be created.

We also want to include the Bends in the sizing for this model.

➤**Click the General Junctions button** and change each of the Bends (3, 6, 10, and 19) to Include in Cost Report and Sizing (Figure 6)
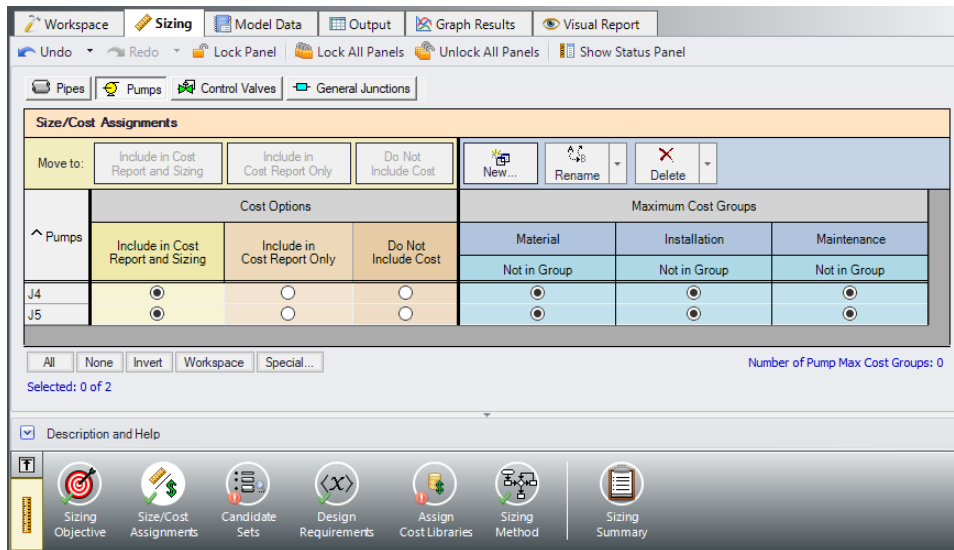
Figure 5: Size/Cost Assignments panel defined for the pumps
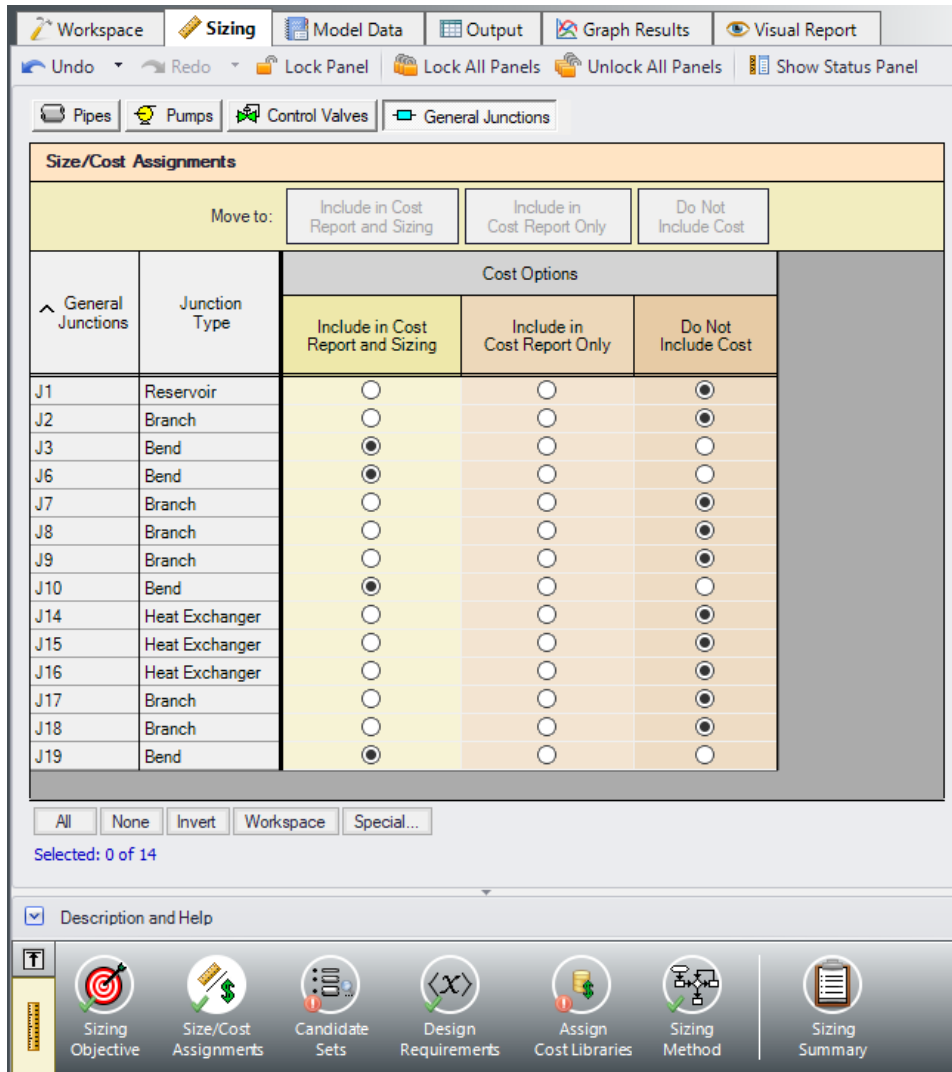
Figure 6: Size/Cost Assignments panel defined for the General Junctions

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Click New under Define Candidate Sets and name it *STD Steel 1"-36"*. In the Select Pipe Sizes window, select Steel - ANSI from the dropdown menu, expand the STD group (make sure the Sort selection on the bottom is Type, Schedule, Class), then add all pipes from 1 inch to 36 inch to the Candidate Set list on the right. Click OK to close the Select Pipe Sizes window.

In the bottom section of the window, assign each of the Common Size Groups and pipe P8 to the STD Steel 1"-36" Candidate Set. The window should appear as shown in the figure below.
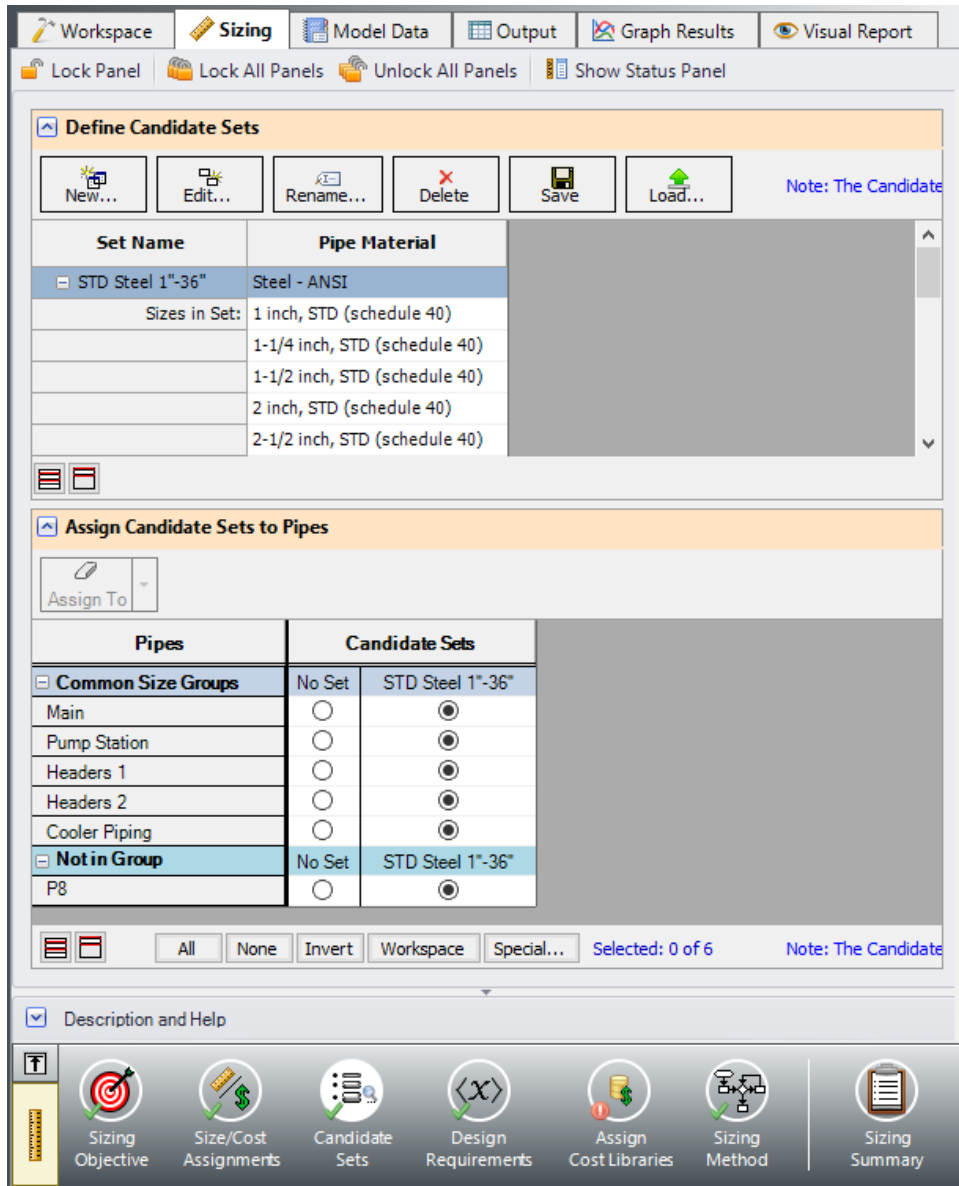
Figure 7: Candidate Sets panel fully defined for the model

D. Design Requirements

Select the Design Requirements button from the Sizing Navigation Panel.

For this model we have multiple requirements for the system:

1. *Max Velocity*: All pipes have a maximum velocity of 12 feet/sec
2. *Min Pressure*: All pipes have a minimum static pressure requirement of 15 psia

3. *NPSH Margin*: All pumps must have a minimum NPSH Margin of 10 feet (for this system the nominal NPSHR is 50 feet)

4. *Design DP*: All control valves must have a minimum static pressure drop of 5 psid

5. All heat exchangers must receive a minimum flow of 1900 gal/min

Define and assign the first two requirements listed above as Pipe Design Requirements as shown in Figure 8, and item 3 as a Pump Design Requirement as shown in Figure 9. Notice that we have accounted for the minimum flow requirement in this system by using flow control valves at each of the heat exchangers. By using assigned flow pumps and flow control valves at the heat exchangers (we will do this later) we will incorporate the necessary flow rate as a boundary condition, so we do not need to apply a Design Requirement to account for this item.

For Control Valves 11 and 12 define and apply the minimum pressure drop requirement in the Control Valves section as shown in Figure 10. Note that for pump sizing purposes as discussed earlier Control Valve 13 has been defined as a Constant Pressure Drop valve, so the pressure drop Design Requirement does not need to be applied to this valve.
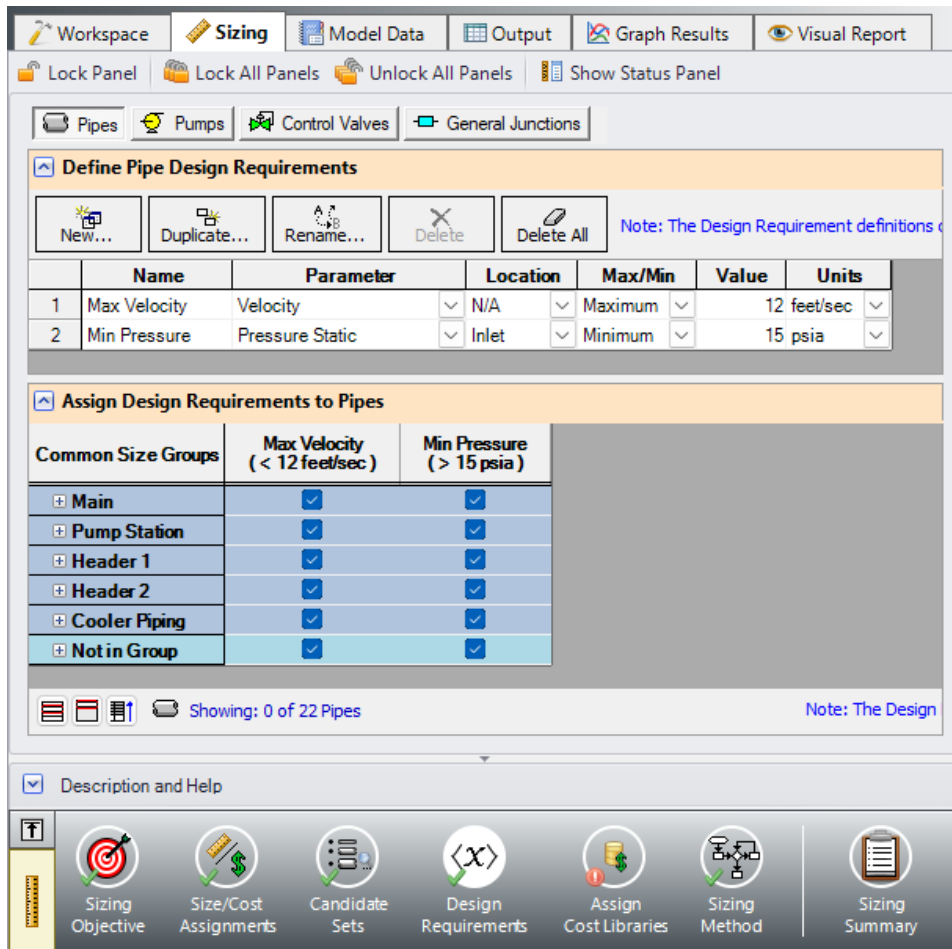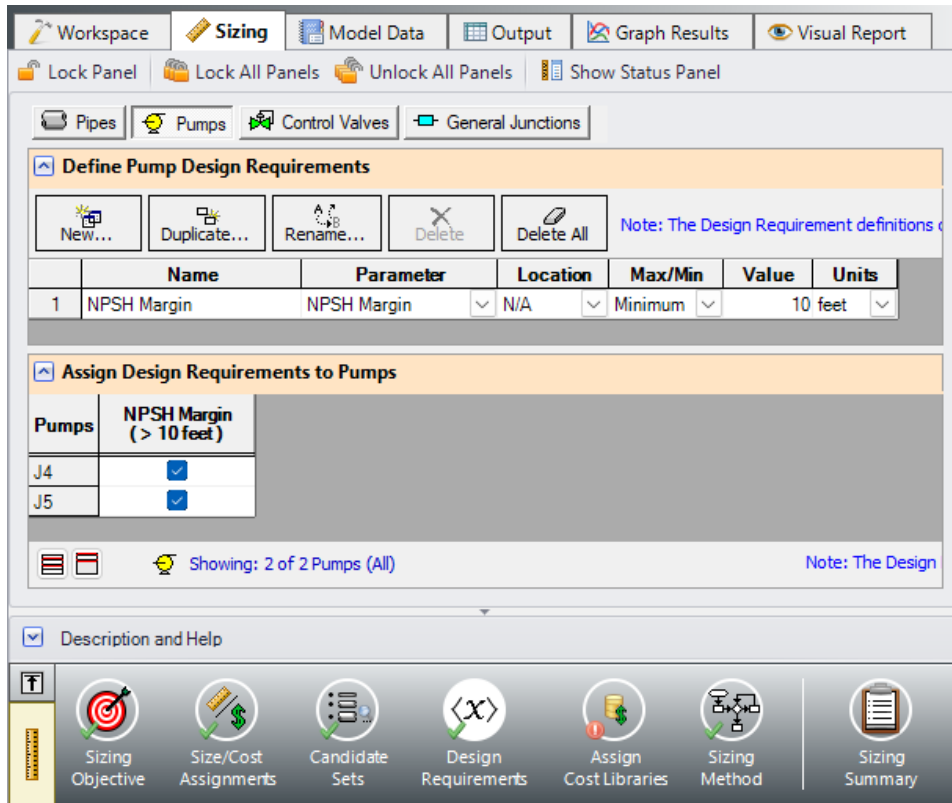

Figure 8: Design Requirements panel for pipes
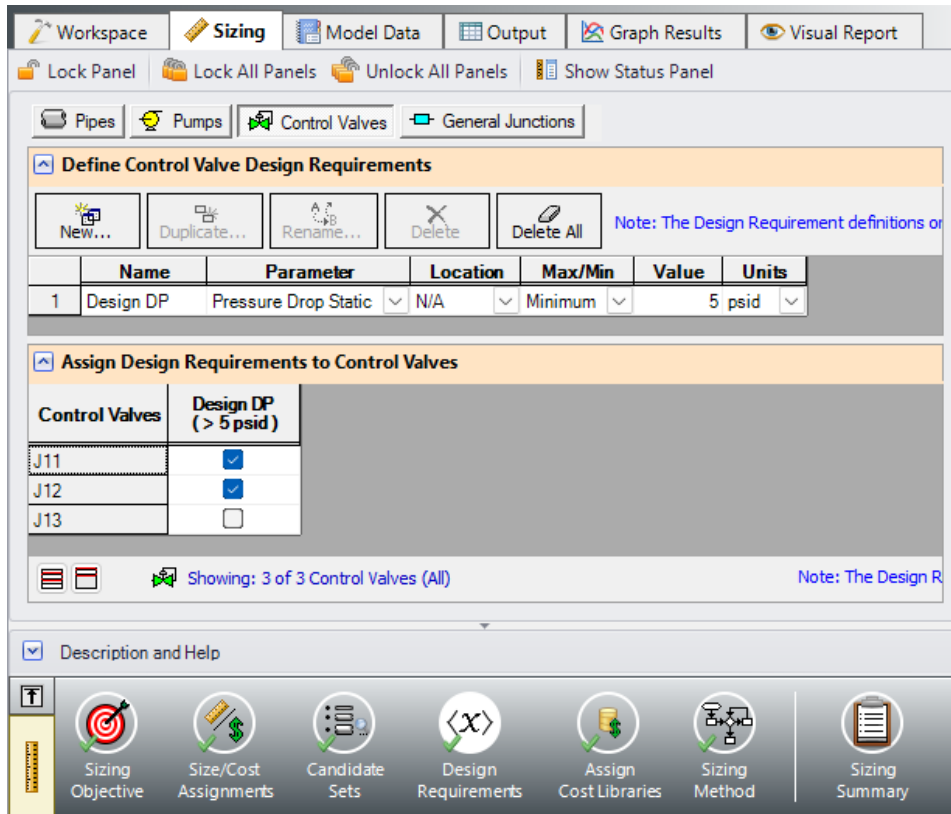
Figure 9: Design Requirements panel defined for the pumps

Figure 10: Design Requirements panel defined for the control valves

E. Assign Cost Libraries

Select the Assign Cost Libraries button. For this model the engineering and cost libraries have already been created, but we will need to connect and apply them.

Note: The Cooling System, Creating Libraries - ANS example provides instructions on how to build the libraries for the first part of this example, and may be completed later for practice with creating libraries. The provided pre-built libraries will be needed to obtain the pump curve data for Step 16 of this example.

➢**Open the Library Manager** by opening the Library menu and selecting Library Manager. The AFT INTERNAL LIBRARY, LOCAL USER LIBRARY, and the pipe material libraries should be available and connected by default. Available libraries will be shown based on libraries that Fathom has identified on your machine. The default connected libraries will be visible here, and additional libraries may be visible from other example files or from libraries previously built and used on your machine.

This example uses one user-defined engineering library *Cooling System.dat*, which contains the data for the pumps and bends. There are also two cost libraries, *Cooling System Costs.cst* and *Steel - ANSI Pipe Costs.cst*, which contain the cost data for the junctions and the pipe materials, respectively. *Cooling System Costs.cst* was built off of the engineering library *Cooling System.dat*, and *Steel -*

ANSI Pipe Costs.cst was built off of the Steel - ANSI Pipe Material Library. To connect these libraries, do the following:

1. Click Add Existing Library.

2. Browse to the Fathom 13 Examples folder (located by default in C:\AFT Products\AFT Fathom 13\Examples\), and open the file titled ***Cooling System.dat***.

3. Repeat the above steps, but choose Add Existing Cost Library to browse for ***Cooling System Costs.cst***, then ***Steel - ANSI Pipe Costs.cst***.

Note that it may be possible that the model file already has the cooling system libraries connected, in which case you will be notified that the library you are trying to add is already available. If the libraries are available, you will simply need to make sure they are connected by ensuring a checkmark is next to the name. There should be a library titled **Cooling System** with two sub libraries: **Junctions** and **Cost** (Cost should have a cost file titled **Cooling System Costs** beneath it. Additionally a Cost sub library should appear subordinate to the Steel - ANSI, the library **Steel - ANSI Pipe Costs** should appear beneath it.

Note: When a library is selected in the Library Manager, the filename and location can be viewed in the Selected Content Information section.

Once you have confirmed that the necessary libraries are connected, click Close to exit the Library Manager.
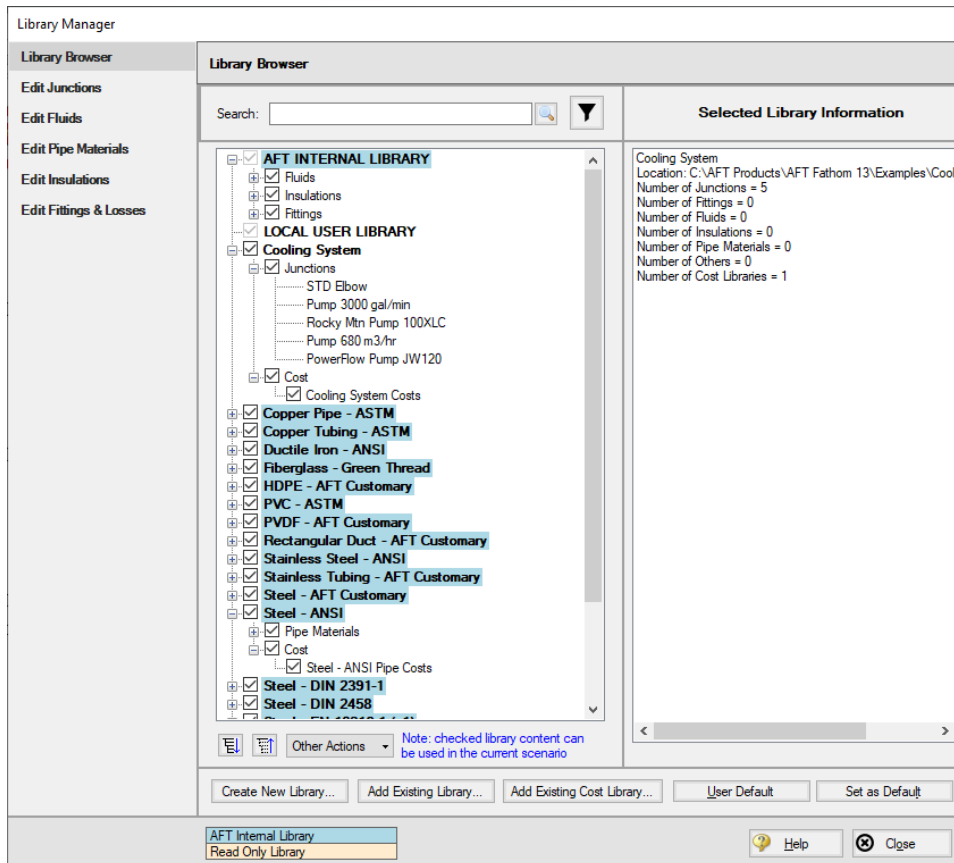

Figure 11: Library Manager with Cooling System junction libraries connected

Since we are connecting a pre-built engineering library, we will need to take an additional step to confirm that the junctions in the Workspace are linked to the Cooling System engineering library that we added.

1. Go to the Workspace, and open Pump J4
2. From the Library Jct list, select **Pump 3000 gal/min**. The pump input should change, and Fathom will recognize that this junction is connected to the engineering and cost library for this pump. Click OK.
3. Repeat step 2 for Pump J5.
4. Open each of the Bends (junctions 3, 6, 10, and 19), and select **STD Elbow** for each from the Library Jct list, and click **OK**.

We have now re-connected the junctions that will be considered for the Cost Report and Sizing Calculations to the engineering library. This is important, since a junction must be connected to an engineering library before a cost can be assigned for it. Let's now return to the Sizing tab.

Back in the Assign Cost Libraries panel for the Pipes, All should be selected for the STD Steel 1"-36" inch Candidate Set with only the Steel - ANSI Pipe Costs library shown. The user can create and connect as many cost libraries as desired, which can be useful to easily update the cost of individual objects, rather than having to keep all of the costs in one large library. If multiple cost libraries are assigned to a Candidate Set, then the costs will be stacked to find a cumulative cost.

➤**Click on the Pumps button** to view the cost library assignments for the Pumps. The only visible library should be the Cooling System Costs library, as this was the only cost library connected for the associated engineering library. Make sure that the checkboxes are selected under the library name so that the cost data will be applied for each of the pumps. To review the cost data for the Bends click the General Junctions button to verify that the Cooling System Costs are being applied to each of the bends.

F. Sizing Method

Select the Sizing Method button to go to the Sizing Method panel.

➤**Choose Discrete Sizing**, if not already selected, since it is desired to select discrete sizes for each of the pipes in the model.

For this model we have six independent pipe sizes as mentioned earlier (five Common Size Groups and pipe 8), and 48 design requirements (2 pipe design requirements applied to all of the pipes, the pump NPSH requirement, and the minimum pressure drop for control valves). Due to the number of independently sized pipes, it would be recommended to try the MMFD or SQP search method.

Select **Sequential Quadratic Programming (SQP)** for the sizing method.

Note: Search Methods can give different results in certain cases. It may be prudent to try several methods and to proceed with the one that yields optimal results.

G. Sizing Summary

From the Sizing Navigation Panel select the Sizing Summary button.

The Sizing Summary panel allows the user to view all of the sizing input for the model in a tree view layout, which is shown in Figure 12. Items can be organized by sizing parameters (Design Requirements,

Candidate Sets, etc.), or information can be viewed based off an individual pipe. For clarity these categories can be shown/hidden using the check boxes at the top of the panel.

Each tree node can be expanded/collapsed individually by using the '+' and '-' buttons on the node. Alternatively, the buttons on the bottom of the panel can be used to collapse/expand all items. Right-clicking on the nodes provides additional options to collapse/expand items, view related nodes, or to copy information to the clipboard. The Sizing Summary panel can be printed from the printer button at the bottom of the panel.
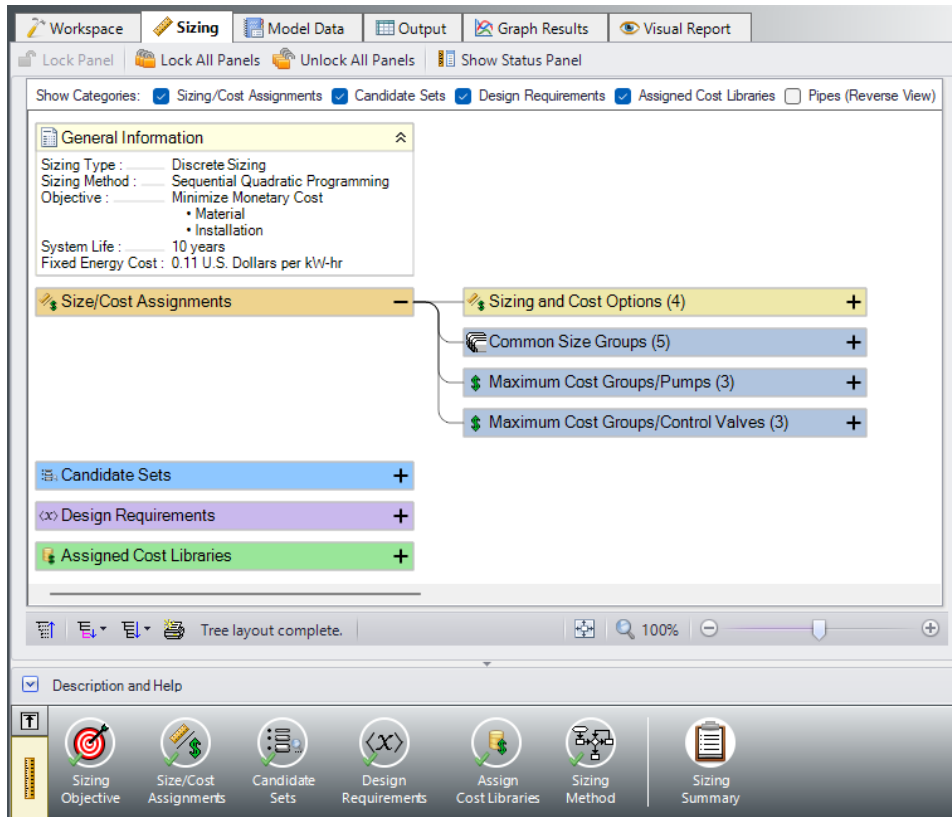
Figure 12: Sizing Summary panel layout with the Size/Cost Assignments expanded. The number of children nodes for each item can be seen in parentheses.

Step 8. Set up the Scenario Manager

To compare the four potential sizing cases, child scenarios will be set up in the Scenario Manager. Go to the Scenario Manager now and create the following child scenarios based on Figure 13 below. These will include the Not Sized scenario and the four potential sizing cases: Initial Cost, Life Cycle Cost, Initial Cost Limit, and Actual Pump Curve.
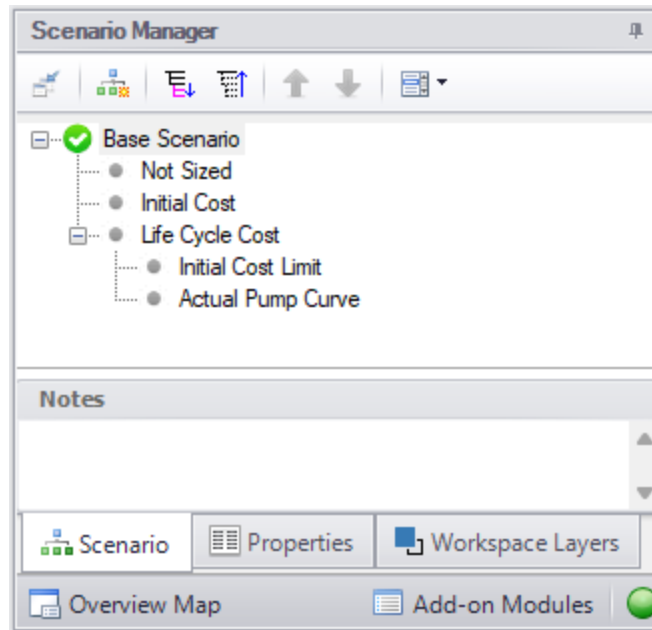
Figure 13: Scenario Manager showing the child cases for the Cooling System ANS example

Step 9. Set up the Not Sized scenario

Load the **Not Sized** scenario by double-clicking on it in the Scenario Manager.

Navigate to the **Sizing** window and go to the **Sizing Objective** panel. Select **Calculate Cost, Do Not Size**. In the Options to Minimize Monetary Cost, the selection should be User Specified. Make sure that the radio buttons for **Material**, **Installation**, and **Energy** are selected for **Cost Report Only**, while **Maintenance** should have **Ignore Cost** selected.

This will calculate the life cycle cost of the system before any sizing is performed. This cost will be used as a comparison to the sized scenarios.

Step 10. Run the Model

Select Run Model in the Analysis menu or click Run Model in the Common Toolbar. This will open the Solution Progress window. This window allows you to watch as the Fathom Solver and ANS module converge on a solution.

After the run has been completed, the results can be reviewed clicking the Output button at the bottom of the solution progress window.

Step 11. Review the Not Sized results

The Cost Report is shown in the General Section of the Output window (see Figure 14). No sizing was performed, so all cost values will appear in the Items Not In Sizing row. The total cost for this system is $2,414,737. This includes all costs over 10 years. The initial cost is the total of the material and install-ation cost: $437,988. The energy cost for the system is $1,976,749.

	Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
	TOTAL OF ALL MODEL COSTS									**2,414,737**
	Total of All Shown Costs			**335,997**	**101,991**	**437,988**	**0**	**1,976,749**	**1,976,749**	**2,414,738**
	Items In Sizing			**0**	**0**	**0**	**0**	**0**	**0**	**0**
	Items Not In Sizing			**335,997**	**101,991**	**437,988**	**0**	**1,976,749**	**1,976,749**	**2,414,738**
	Pipe Subtotal			**284,056**	**80,818**	**364,874**	**0**	**0**	**0**	**364,874**
o	P1	Pipe	Pipe	10,967	3,120	14,088	0	0	0	14,088
o	P2	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P3	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P4	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P5	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P6	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P7	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P8	Pipe	Pipe	109,674	31,204	140,878	0	0	0	140,878
o	P9	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P10	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P11	Pipe	Pipe	5,484	1,560	7,044	0	0	0	7,044
o	P12	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P13	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P14	Pipe	Pipe	5,484	1,560	7,044	0	0	0	7,044
o	P15	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P16	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P17	Pipe	Pipe	5,484	1,560	7,044	0	0	0	7,044
o	P18	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P19	Pipe	Pipe	2,742	780	3,522	0	0	0	3,522
o	P20	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P21	Pipe	Pipe	1,645	468	2,113	0	0	0	2,113
o	P22	Pipe	Pipe	109,674	31,204	140,878	0	0	0	140,878
	Bend Subtotal			**9,884**	**7,589**	**17,473**	**0**	**0**	**0**	**17,473**
o	J3	Bend	Bend	2,471	1,897	4,368	0	0	0	4,368
o	J6	Bend	Bend	2,471	1,897	4,368	0	0	0	4,368
o	J10	Bend	Bend	2,471	1,897	4,368	0	0	0	4,368
o	J19	Bend	Bend	2,471	1,897	4,368	0	0	0	4,368
	Pump Subtotal			**42,058**	**13,584**	**55,641**	**0**	**1,976,749**	**1,976,749**	**2,032,390**
o	J4	Pump	Pump	21,027	6,791	27,819	0	988,330	988,330	1,016,149
o	J5	Pump	Pump	21,030	6,792	27,823	0	988,419	988,419	1,016,241

Figure 14: The Cost Report in the Output window shows the total and individual costs (in U.S. Dollars) for the sized system

With this basis, we will move on to the Initial Cost scenario.

Step 12. Run the Initial Cost scenario and review the results

Load the Initial Cost scenario using the Scenario Manager. This scenario should have all the inputs already specified, so no action is required besides running the model.

Click Run Model on the toolbar or from the Analysis menu. When finished, view the results by clicking the Output button at the bottom of the Solution Progress window.

Go to the Cost Report, as shown in Figure 15. ANS shows all costs in the Cost Report, including those that were not used in the automated sizing. The total cost for this system is $2,287,193. This includes all

costs over 10 years. The initial cost, which was the basis for the automated sizing, was the total of the material and installation ($159,649). This total is listed as the total for the Items in Sizing row, since it was specified to only include the Initial Cost for material and installation on the Sizing Objective section. Individual items that were included in the automated sizing have their costs highlighted in green.

Other costs that are displayed in the Cost Report are Items Not in Sizing. These are items that have costs associated with them, but were not included in the automated sizing.

Note that the Items Not in Sizing total to $2,127,544. Looking across at the subtotal one can see that all of this cost is in Operation/Energy (i.e., pump power costs). The TOTAL cost is the sum of the Non-Recurring Sub Total and the Recurring Sub Total. For the next scenario this operation/energy cost will be included in the automated sizing.

Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									2,287,193
Total of All Shown Costs			105,846	53,803	159,649	0	2,127,544	2,127,544	2,287,193
Items In Sizing			105,846	53,803	159,649	0	0	0	159,649
Items Not In Sizing			0	0	0	0	2,127,544	2,127,544	2,127,544
Pipe Subtotal			58,013	36,820	94,833	0	0	0	94,833
P1	Pipe	Pipe	2,476	1,518	3,994	0	0	0	3,994
P2	Pipe	Pipe	227	178	405	0	0	0	405
P3	Pipe	Pipe	378	297	675	0	0	0	675
P4	Pipe	Pipe	378	297	675	0	0	0	675
P5	Pipe	Pipe	378	297	675	0	0	0	675
P6	Pipe	Pipe	378	297	675	0	0	0	675
P7	Pipe	Pipe	227	178	405	0	0	0	405
P8	Pipe	Pipe	24,762	15,176	39,938	0	0	0	39,938
P9	Pipe	Pipe	227	178	405	0	0	0	405
P10	Pipe	Pipe	164	139	302	0	0	0	302
P11	Pipe	Pipe	545	462	1,007	0	0	0	1,007
P12	Pipe	Pipe	273	231	504	0	0	0	504
P13	Pipe	Pipe	273	231	504	0	0	0	504
P14	Pipe	Pipe	545	462	1,007	0	0	0	1,007
P15	Pipe	Pipe	273	231	504	0	0	0	504
P16	Pipe	Pipe	273	231	504	0	0	0	504
P17	Pipe	Pipe	545	462	1,007	0	0	0	1,007
P18	Pipe	Pipe	273	231	504	0	0	0	504
P19	Pipe	Pipe	273	231	504	0	0	0	504
P20	Pipe	Pipe	164	139	302	0	0	0	302
P21	Pipe	Pipe	227	178	405	0	0	0	405
P22	Pipe	Pipe	24,762	15,176	39,938	0	0	0	39,938
Bend Subtotal			736	1,562	2,298	0	0	0	2,298
J3	Bend	Bend	217	426	643	0	0	0	643
J6	Bend	Bend	217	426	643	0	0	0	643
J10	Bend	Bend	151	355	506	0	0	0	506
J19	Bend	Bend	151	355	506	0	0	0	506
Pump Subtotal			47,097	15,421	62,518	0	2,127,544	2,127,544	2,190,062
J4	Pump	Pump	23,422	7,664	31,086	0	1,059,984	1,059,984	1,091,070
J5	Pump	Pump	23,675	7,757	31,432	0	1,067,560	1,067,560	1,098,991

Figure 15: The Cost Report in the Output window shows the total and individual costs (in U.S. Dollars) for the sized system

In the Pipes section of the Output window the final pipe sizes from the automated sizing can be seen, along with the Design Requirements status. For this model, the final system used pipe sizes varying from 10 to 16 inches, as seen in Figure 16. The information for the pumps can be seen in the Pump Summary tab in the General Section, which is also shown in Figure 16.

When a monetary cost objective is selected, additional tabs are available that display the Library Sources used for the Cost Report for each of the pipes/junctions. In this case both the Pipe Material and Pipe Installation costs for all of the pipes came from the Steel - ANSI Pipe Costs library which was connected earlier. Information for the Pumps and Bends came from the Cooling System Costs library. If multiple

cost libraries are used for a category they will all be listed separated by commas. The Library Sources for the Pipes can be seen in Figure 17.

Figure 16: Pump and Pipe Sizes after automated sizing is performed

Figure 17: Library Sources for the Pipes in the Cost Report

Within the General tab of the Output window at the bottom of the report, the initial cost of the items included in the sizing can be seen, which was $437,988 for this design, showing that the initial costs changed to $159,649, a 63% reduction! We can also see that the initial design was feasible, meaning all of the design requirements were met (Figure 18). The ANS module both kept all design requirements satisfied, as well as minimized the initial cost, all in under 3 seconds.

Although there was a 63% reduction in initial costs, the total cost decreased by 5%. Next, we will see how much more the total cost can be reduced by doing a life cycle system sizing.
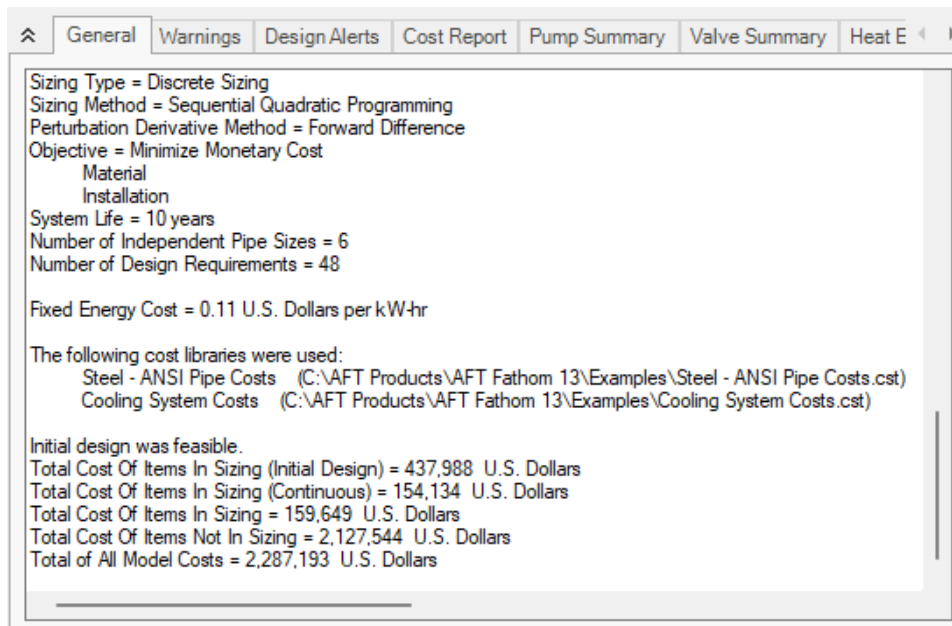
Figure 18: General section of Output window showing initial status of Items in Sizing

Step 13. Size System for Life Cycle Cost over 10 Years

Load the Life Cycle Cost scenario from the Scenario Manager. Go to the Sizing window, and make sure that the Sizing Objective panel is opened. Under Options to Minimize Monetary Cost, choose **Size for Initial and Recurring Costs**. This will move the Energy costs selection to Include in Sizing Objective. The scenario is now complete.

➤**Click Run from the Analysis menu** to run the automated sizing. When the calculations have finished, click Output to view the results.

Step 14. Compare the Sizing Results

After viewing the Cost Report, it can be seen that the total cost is now $2,216,402. The Items Not in Sizing row now has zero values, and the Energy columns are highlighted green, reflecting the changes made to include these costs in the objective to minimize monetary cost. The Cost Report is shown in Figure 19.

Taken on its own, this new cost represents a savings of $198,335 (8%) compared to the Not Sized scenario. Compared to the Initial Cost scenario, which was sized to minimize initial cost only, this new scenario represents a savings of $70,791. This represents a total 3% cost reduction. In the Initial Cost scenario, the non-recurring cost was $159,649, while the overall cost was $2,287,193. Now the non-recurring cost is $196,186 while the overall cost is $2,216,402. The initial cost thus increased by $36,537 in order to reduce the operating cost from $2,127,544 to $2,020,216 (a reduction of $107,328). The source of the operating cost is the cost of power for the pumps. To reduce pump power usage, it makes sense to increase the pipe size and thus reduce frictional losses. The larger pipe sizes can be reviewed

by looking at the Sizing tab in the Pipe Output section. From the Pump Summary tab it can be seen that the pump power usage decreased by 7.5 hp at each of the pumps, which is approximately a 5% power requirement decrease at each pump. The Pump Summary and final pipe sizes can be seen in Figure 20.

Considering this cost breakdown between the scenarios highlights the importance of choosing to include the energy costs in the initial cost scenario. If they were not in the Cost Report of the initial cost scenario, the total cost would be $159,649. Since the life cycle cost scenario total cost is nearly ten times larger, it would appear (at first glance) to be a significant cost increase. However, this would be misleading, since the energy costs still exist in both scenarios, though they are only being minimized in one case. To get a clear comparison between designing for initial cost and life cycle cost one needs to make an apples-to-apples comparison, including all relevant costs in each case, as is done for this example.

By performing both analyses the designer has quantitative data on the impact of initial cost design on operational costs of the system, and can make an informed design choice.

| General | Warnings | Design Alerts | Cost Report | Pump Summary | Valve Summary | Heat Exchanger Summary | Reservoir Summary |

Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									**2,216,402**
Total of All Shown Costs			133,229	62,957	196,186	0	2,020,216	2,020,216	2,216,402
Items In Sizing			133,229	62,957	196,186	0	2,020,216	2,020,216	2,216,402
Items Not In Sizing			0	0	0	0	0	0	0
Pipe Subtotal			87,926	45,862	133,788	0		0	133,788
P1	Pipe	Pipe	3,695	1,861	5,556	0	0	0	5,556
P2	Pipe	Pipe	458	254	712	0	0	0	712
P3	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P4	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P5	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P6	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P7	Pipe	Pipe	458	254	712	0	0	0	712
P8	Pipe	Pipe	36,948	18,612	55,560	0	0	0	55,560
P9	Pipe	Pipe	458	254	712	0	0	0	712
P10	Pipe	Pipe	458	254	712	0	0	0	712
P11	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P12	Pipe	Pipe	378	297	675	0	0	0	675
P13	Pipe	Pipe	378	297	675	0	0	0	675
P14	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P15	Pipe	Pipe	378	297	675	0	0	0	675
P16	Pipe	Pipe	378	297	675	0	0	0	675
P17	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P18	Pipe	Pipe	378	297	675	0	0	0	675
P19	Pipe	Pipe	378	297	675	0	0	0	675
P20	Pipe	Pipe	458	254	712	0	0	0	712
P21	Pipe	Pipe	458	254	712	0	0	0	712
P22	Pipe	Pipe	36,948	18,612	55,560	0	0	0	55,560
Bend Subtotal			1,792	2,982	4,774	0	0	0	4,774
J3	Bend	Bend	525	852	1,377	0	0	0	1,377
J6	Bend	Bend	525	852	1,377	0	0	0	1,377
J10	Bend	Bend	525	852	1,377	0	0	0	1,377
J19	Bend	Bend	217	426	643	0	0	0	643
Pump Subtotal			43,510	14,113	57,623	0	2,020,216	2,020,216	2,077,839
J4	Pump	Pump	21,728	7,047	28,774	0	1,009,286	1,009,286	1,038,061
J5	Pump	Pump	21,783	7,067	28,849	0	1,010,930	1,010,930	1,039,779

Figure 19: Cost Report for the network after sizing for life cycle cost

Figure 20: Pump and Pipe Sizes after automated sizing is performed for life cycle cost

Step 15. Size System for Life Cycle Cost with Initial Cost Limit

While the energy cost savings from life cycle cost sizing are desirable, the budget for the project may not support the higher initial cost that is required. In that case, we can apply an initial cost limit to the life cycle cost sizing.

Load the Initial Cost Limit scenario from the Scenario Manager.

➢**Return to the Sizing Objective panel** and select the option for Size for Initial and Recurring Costs and check the box next to Initial Cost Limit. We will set the limit as $175,000, which is slightly more than the initial cost of the scenario that minimized initial cost at the expense of an increased life cycle cost. Run the model and go to the Output tab.

A summary of the results with the initial cost limit compared to each of the initial and life cycle cost sizing scenarios for a 10 year life cycle can be seen in Table 1. Notice that while the material and installation costs were limited to $175,000, the total costs over a 10 year lifetime are decreased by about $53,280 compared to the Initial Cost scenario. The initial cost increased from $159,649 to $175,233: a $15,584 increase. Therefore, a $15,584 initial cost increase leads to a life cycle decrease of $53,280. This savings comes from the reduction pump energy costs. These reduced energy costs were achieved by

increasing the Main pipes (P1 & P22), pipe P8, and the Header pipes while decreasing the pipes in Cooler Piping.

Figure 21 shows a comparison of the pipe sizes throughout the system for the Initial Cost scenario, the Life Cycle Cost scenario, and the Life Cycle Cost with Initial Cost Limit scenario. Note that this diagram was generated outside of AFT Fathom, but sized results can be shown for each scenario individually using Workspace Layers.

Table 1: Cost Summary of Sizing Runs for Cooling System

Scenario	Material	Installation	Non-Recur-ring Sub Total	Operation/Energy	TOTAL	Reduction
Not Sized	335,997	101,991	437,988	1,976,749	2,414,737	-
Initial Cost	105,846	53,803	159,649	2,127,544	2,287,193	**127,545**
Life Cycle Cost	133,229	62,957	196,186	2,020,216	2,216,402	**198,336**
Initial Cost Limit	117,520	57,713	175,233	2,058,681	2,233,913	**180,825**
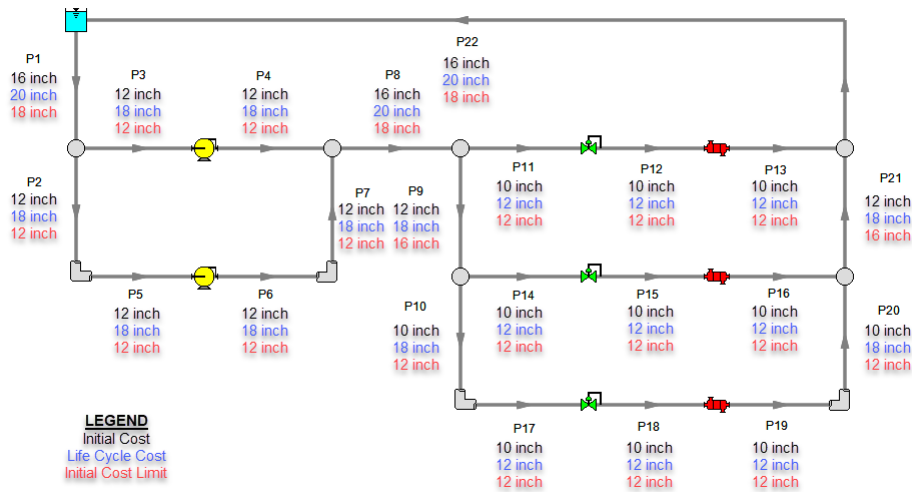
Figure 21: Pipe sizes selected by the ANS module for the life cycle cost and life cycle cost with an initial cost limit compared to sizing for initial cost only

Step 16. Automated Sizing with Pump Curve Data

As discussed in the section *Summarizing the pump selection process*, once the pump is sized then actual pumps can be modeled. The actual pump should closely match the sizing results in the following areas: generated head at the design flow, efficiency at the design flow, and cost. We will apply a pump curve for the case that was sized for life cycle cost to complete the pump selection process.

Reviewing the results for the life cycle cost scenario one can see that the sized system calls for a pump of about 141 hp that generates about 130 feet of head at 3,000 gal/min. The nominal efficiency used in the sizing part of the analysis was 70%. The material cost for such a pump was about $21,800, and the installation cost was about $7,000. Note that if no actual pump can be found that reflects these requirements, then the phase 1 of the pump selection process described above should be repeated with better performance and/or cost data for the pump.

Based on the sizing analysis, we will choose a pump from the (fictitious) Rocky Mountain Pump Company. Its model 100XLC is a close match for the requirements of a 10-year life cycle design. The model 100XLC has the following characteristics: at 3,000 gal/min its head is 135 feet and efficiency is 70%. Its material cost is $50,000 and installation cost is $22,500. To set up a new scenario for the system with the candidate pump data:

1. Load the Actual Pump Curve scenario and go to the Output window. Currently it should be identical to the parent scenario Life Cycle Cost. Run the scenario.

2. Open the Edit menu and select **Transfer Results to Initial Guesses**. When the Transfer Results Options window appears, select **Calculated Pipe Sizes Only** and hit **OK**.

    > **Note:** The pipe sizes in Model Data are now equal to the pipe sizes selected by ANS for the Life Cycle Cost scenario. Starting from a more optimal solution can aid convergence and in this case will improve solver execution time.

3. Go back to the Workspace. In the properties window for each of the **pumps** select **Rocky Mtn Pump 100XLC** from the Library Jct list to import the pump data from the library as shown in Figure 22, then click OK.

4. In the Control Valve properties window for **J13**, change the Valve Type to **Flow Control (FCV)**, and enter a Flow Setpoint of **2000 gal/min**. Uncheck Always Control (Never Fail), Now that we have changed the control valve model, we will need to add a new Design Requirement in order to account for the minimum pressure drop.

5. Go to the **Sizing** window, navigate to the **Design Requirements** panel, select the **Control Valves** button at the top of the window, and **apply** the Design DP design requirement to J13.
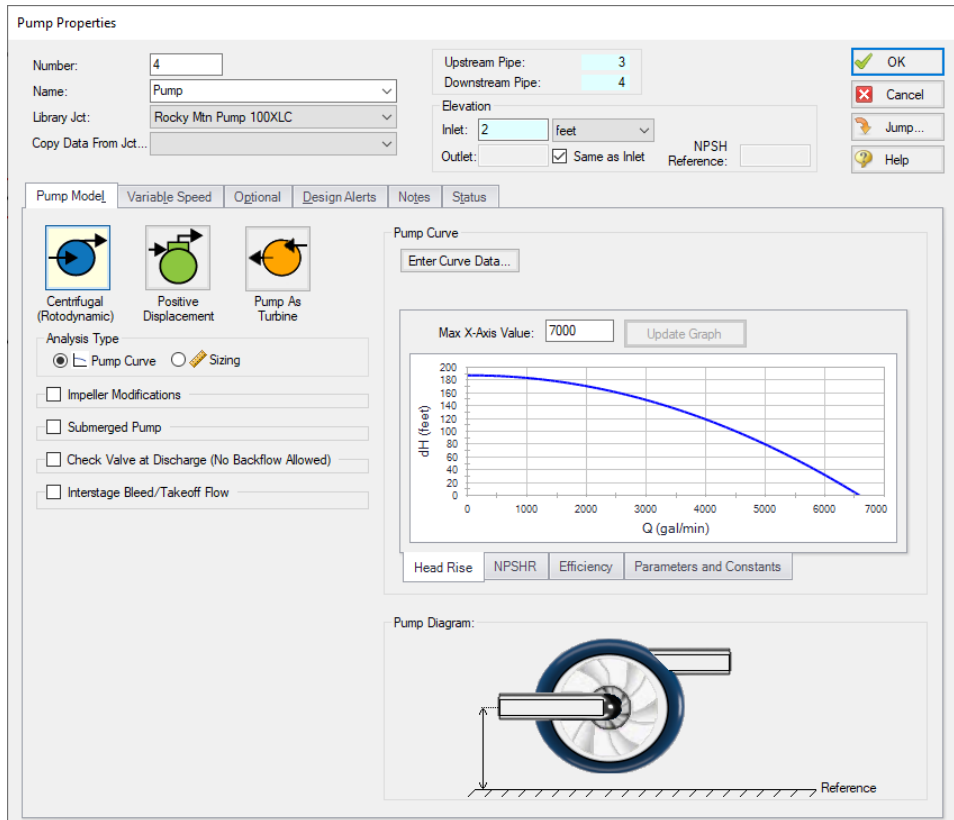
Figure 22: Pump Properties window with the Rocky Mtn Pump 100XLC selected

The necessary changes are now complete to size the system with the updated pump and flow control valve data.

➤**Select Run from the Analysis menu** to perform the automated sizing - this may take some time.

When the solver has finished, click Output to review the results. The overall cost for the system is about $2,558,744. Clicking the Pump Summary tab shows that the operating point with the actual pump curve is similar to the sizing scenario.

Now we have sized the system for use with an actual pump.

| General | Warnings | Design Alerts | Cost Report | Pump Summary | Valve Summary | Heat Exchanger Summary | Reservoir Summary |

	Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
	TOTAL OF ALL MODEL COSTS									**2,558,744**
	Total of All Shown Costs			**158,749**	**83,382**	**242,131**	**0**	**2,316,613**	**2,316,613**	**2,558,744**
	Items In Sizing			**158,749**	**83,382**	**242,131**	**0**	**2,316,613**	**2,316,613**	**2,558,744**
	Items Not In Sizing			**0**	**0**	**0**	**0**	**0**	**0**	**0**
	Pipe Subtotal			**58,013**	**36,820**	**94,833**	**0**	**0**	**0**	**94,833**
o	P1	Pipe	Pipe	2,476	1,518	3,994	0	0	0	3,994
o	P2	Pipe	Pipe	227	178	405	0	0	0	405
o	P3	Pipe	Pipe	378	297	675	0	0	0	675
o	P4	Pipe	Pipe	378	297	675	0	0	0	675
o	P5	Pipe	Pipe	378	297	675	0	0	0	675
o	P6	Pipe	Pipe	378	297	675	0	0	0	675
o	P7	Pipe	Pipe	227	178	405	0	0	0	405
o	P8	Pipe	Pipe	24,762	15,176	39,938	0	0	0	39,938
o	P9	Pipe	Pipe	227	178	405	0	0	0	405
o	P10	Pipe	Pipe	164	139	302	0	0	0	302
o	P11	Pipe	Pipe	545	462	1,007	0	0	0	1,007
o	P12	Pipe	Pipe	273	231	504	0	0	0	504
o	P13	Pipe	Pipe	273	231	504	0	0	0	504
o	P14	Pipe	Pipe	545	462	1,007	0	0	0	1,007
o	P15	Pipe	Pipe	273	231	504	0	0	0	504
o	P16	Pipe	Pipe	273	231	504	0	0	0	504
o	P17	Pipe	Pipe	545	462	1,007	0	0	0	1,007
o	P18	Pipe	Pipe	273	231	504	0	0	0	504
o	P19	Pipe	Pipe	273	231	504	0	0	0	504
o	P20	Pipe	Pipe	164	139	302	0	0	0	302
o	P21	Pipe	Pipe	227	178	405	0	0	0	405
o	P22	Pipe	Pipe	24,762	15,176	39,938	0	0	0	39,938
	Bend Subtotal			**736**	**1,562**	**2,298**	**0**	**0**	**0**	**2,298**
o	J3	Bend	Bend	217	426	643	0	0	0	643
o	J6	Bend	Bend	217	426	643	0	0	0	643
o	J10	Bend	Bend	151	355	506	0	0	0	506
o	J19	Bend	Bend	151	355	506	0	0	0	506
	Pump Subtotal			**100,000**	**45,000**	**145,000**	**0**	**2,316,613**	**2,316,613**	**2,461,613**
o	J4	Pump	Pump	50,000	22,500	72,500	0	1,160,829	1,160,829	1,233,329
o	J5	Pump	Pump	50,000	22,500	72,500	0	1,155,784	1,155,784	1,228,284

Figure 23: Cost Report for system with actual Pump and FCV data sized for 10 year life cycle cost

Cooling System, Multiple Design Cases - ANS

Summary

Some pipe system designs have multiple operating conditions to satisfy. For instance, the cooling water example discussed in the previous example may have a requirement that it operate with one of the pumps turned off. This can be modeled in the ANS module by taking advantage of its ability to model multiple systems concurrently.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Defining dependent Design Cases
- Using Duplicate Special to create Dependent Designs
- Specifying fractional pump costs with Dependent Design Cases
- Using multiple Dependent Design Cases

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Since this example is a continuation of the Cooling System - ANS example, it is recommended to complete that example first before beginning this example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Cooling System - ANS.fth***
- ***US - Cooling System Multiple Design Cases - ANS.fth***
- ***Cooling System.dat*** - engineering library
- ***Cooling System Costs.cst*** - cost library for ***Cooling System.dat***
- ***Steel - ANSI Pipe Costs.cst*** - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Cooling System - ANS.fth** example file and save it as a new file. Delete the annotations and hide the layers for Section A through E. The Workspace should look like Figure 1 below.

Browse to the Life Cycle Cost scenario in Scenario Manager. Create a child scenario from this called *Dependent Design Cases* to work in for this example.
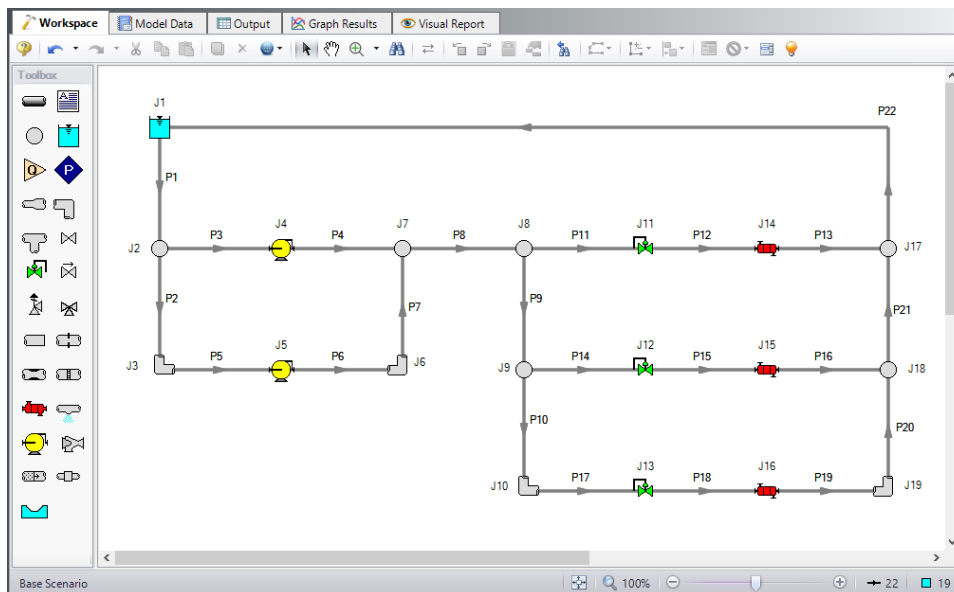
Figure 1: Workspace for Cooling System model

Step 3. Enable Dependent Design

When modeling multiple design cases, the different cases need to be broken up into a primary design case, of which there is one, and dependent design cases, of which there can be any number.

The primary design case can be any of the design cases that are being addressed in your model. Typically, it will represent the primary operating condition of the pipe system. The dependent design cases represent the same physical operating system, but are used to represent differing operating conditions. If it is not clear which case should be the primary case, just pick any of the cases and call that the primary one. Design cases other than the primary case are referred to as dependent design cases. The reason

why we choose a primary case and refer to the other cases as dependent cases will become clearer as we progress through this topic.

For this example the primary design case is the system running with all pumps turned on. We want to also consider the case where only one pump is running. The one pump running case will be created as a dependent design case.

Dependent design case modeling is enabled in the Sizing Objective panel.

➢**In the Dependent Design Case scenario go to the Sizing window and to the Sizing Objective panel.** Check the box next to **Enable Dependent Design Cases**. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation Panel as shown in Figure 2.
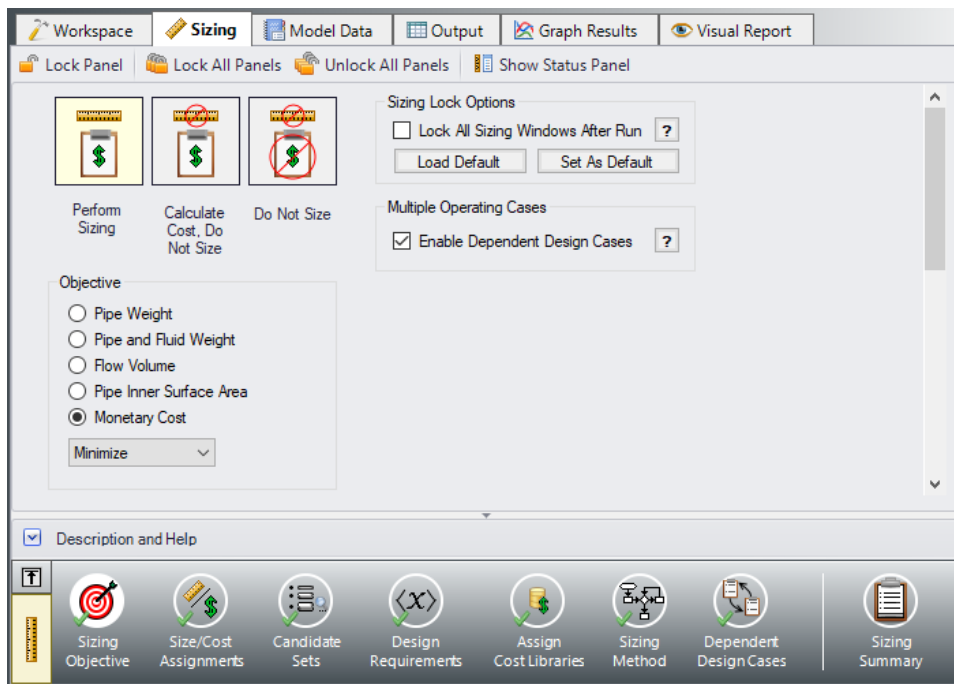
Figure 2: The Dependent Design Cases panel is enabled from the Sizing Objective panel

Step 4. Setting up the Dependent Design Case

A. Using Duplicate Special

The easiest way to set up a dependent design case once it has been enabled is to use Duplicate Special, which will allow us to make an exact duplicate of the model in order to setup the additional operating cases. Note that you should fully define the model and all the sizing settings for the primary case before creating the dependent cases, as has been done in this model.

To create the dependent cases:

1. Make sure that the scenario where the dependent cases will be created is loaded. In this case, you should have the Dependent Design Cases scenario loaded.
2. Go to the **Workspace**.
3. Go to the **Edit menu** and choose **Select All**.
4. Go to the **Edit menu** again and choose **Duplicate Special** (see Figure 3).
5. Choose an increment for the pipe and junction numbers. The number should be large enough to avoid conflicts with numbers used for other pipes and junctions in this and all other scenarios. In this case use **100**.
6. Select the **Make Dependent Design Case** check box.
7. Click **OK** and move the duplicated version of the primary design case to an open area in the Workspace. The model should now appear similar to Figure 4.

Note that Duplicate Special also allows you to do the following:

*   Create a group for the dependent design case. This can simplify managing the pipe and junction objects in the design case.

*   Hide the pipe and junction in the output. You may want to do this for reasons to be discussed later in this topic.

*   Copy Layer Style Settings From Original Objects. If the visibility of the Section A through E layers is toggled back on, they will apply the same colors to both the duplicate and original model if this option was checked while creating the duplicate.

*   Keep Common Size Groups. This feature is useful when using Duplicate Special for reasons other than dependent case creation, and can't be used when making a dependent design case.

*   Clear the Design Requirements. For reasons to be discussed, frequently dependent design cases will employ different design requirements than the primary design case. This option allows you to clear all the current requirements, so new ones can be assigned.
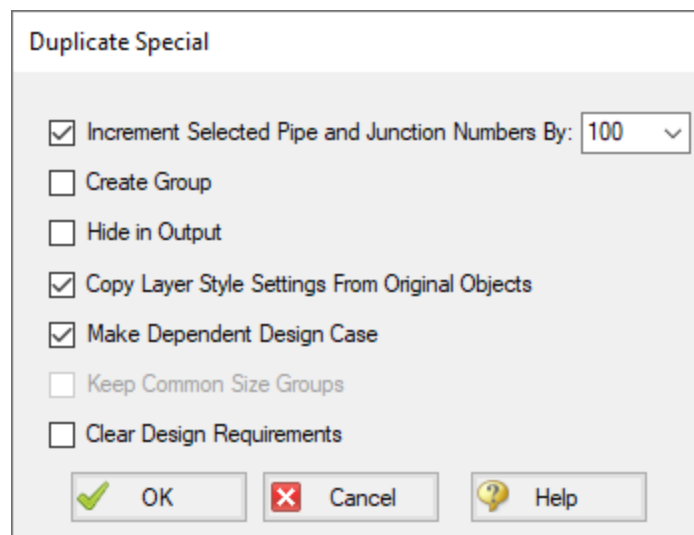
Figure 3: Duplicate Special simplifies creation of dependent design cases
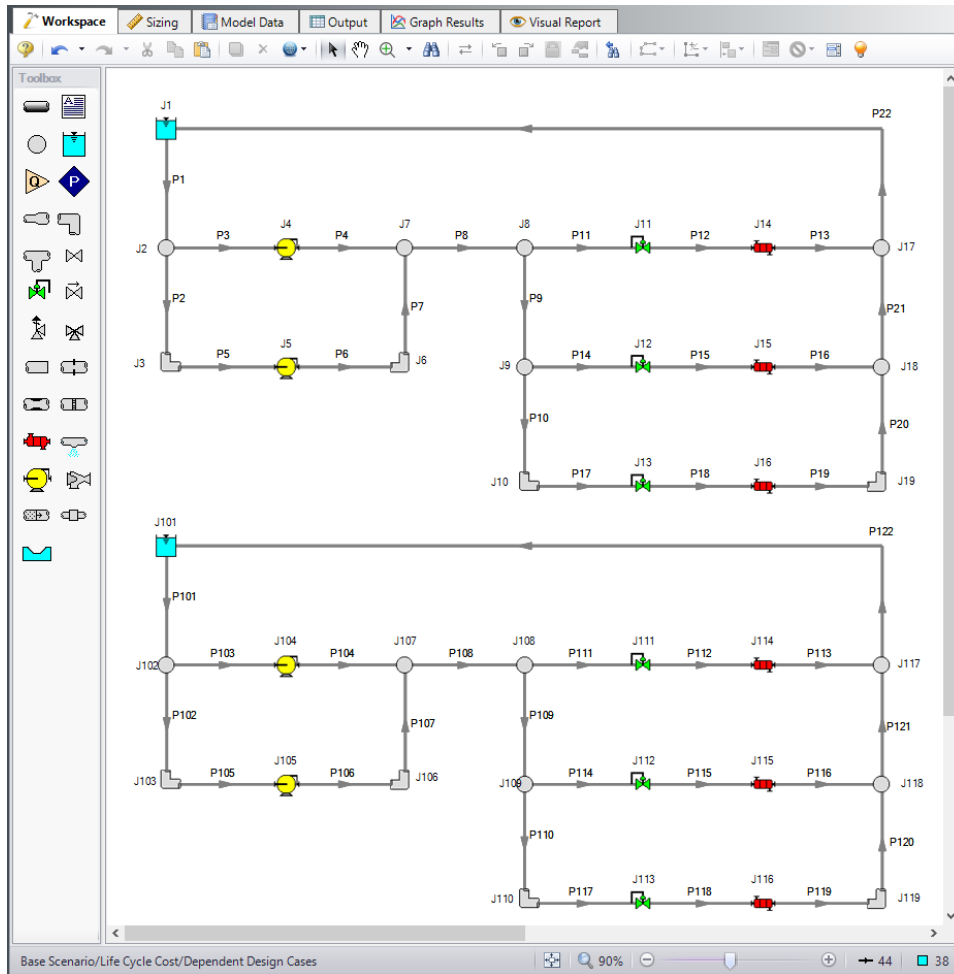
Figure 4: Workspace with the Dependent Design Case created for the Cooling System model

B. Apply the Dependent Design Operating Conditions

The dependent design case capability takes advantage of the ANS module's ability to run multiple models in the same Workspace. The pipes and junctions in Figure 4 that are numbered greater than 100 (i.e., the dependent design case) represent the same pipes and junctions as those numbered less than 100 (i.e., the primary design case). It is therefore critically important that only certain data in the dependent case is changed.

Frequently the Design Requirements or Special Conditions on the pipes or junctions will be changed. However, data such as a pipe's length or pump's performance should never change. The reason is that once this kind of data is changed, the pipe or junction is no longer the same pipe or junction, and thus the purpose of the dependent link is invalidated. The ANS module does not prevent changing data that should not be changed, but when the model is run it will check to make sure all data that should be the same actually is the same.

In this example we are analyzing the case where one pump in the system is shut off, so we will use Special Conditions on the pump. With one pump shutoff we will not have enough flow to supply all three heat exchangers, so we will also close one of the heat exchangers as well as a control valve.

➢**Change the Special Conditions of J105 to Pump Off No Flow.**

➢**Change the Special Conditions of J111 to Closed.**

Selecting these junctions on the Workspace and using the F4 keyboard shortcut is an option for applying these special conditions quickly. The dependent case should now appear as shown in Figure 5.
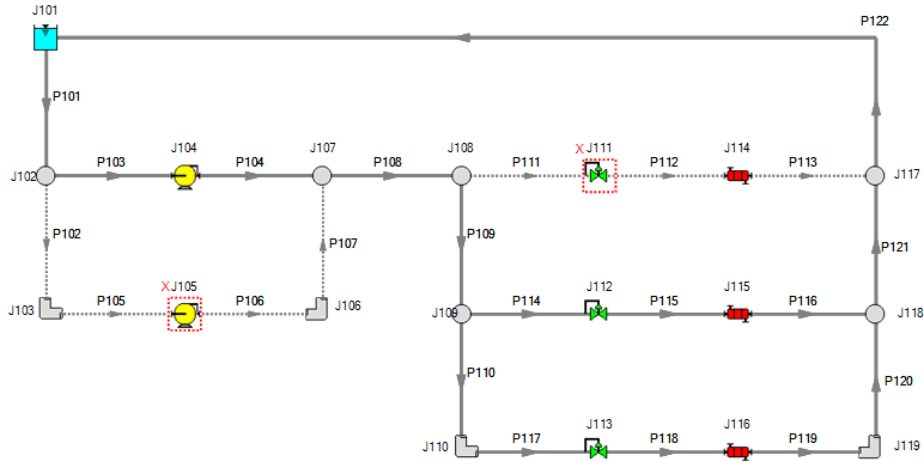
Figure 5: Flow off for Pump 105 and Control Valve 111 in the Dependent Design Case

We will now need to adjust the Design Requirements for the model. Go to the Design Requirements panel on the Sizing window.

Currently it can be seen that Duplicate Special maintained all of the same design requirement applications from the primary case in the dependent case. While most of the design requirements used in the primary case still apply, we will need to remove the Design Requirements from the closed sections of the model. If the model were run without making any changes, the ANS module would not be able to find a feasible system, since it is impossible to achieve a minimum flow of 2000 gal/min in Heat Exchanger 114 now that it has been forced closed. A similar problem exists with the minimum NPSH margin being applied to Pump 105, which is also closed.

The flow requirements for Heat Exchanger 115 and 116 will also need to be reconsidered. Since there is only one pump running a lower supply flow rate is expected, so the minimum flow rate required by the coolers should be lowered to 1500 gal/min.

The pressure drop across the control valves still needs to be a minimum of 5 psid. For J112, this is accomplished by using a Design Requirement whereas J113 is defined as a constant pressure drop control valve (PDCV) with the required pressure drop.

Apply the changes as follows:

1. Click the Pumps button to go to the pump Design Requirements
2. Uncheck the box next to J105 to remove the NPSH requirement from Pump 105 in the dependent case (Figure 6).

3. Click the Control Valves button to go to the control valve Design Requirements.

4. Uncheck the box next to J111 to remove the pressure drop requirement from the closed control valve (Figure 7).

5. Return to the Workspace and open the Control Valve Properties window for J112.

6. Change the Flow Setpoint to 1500 gal/min(Figure 8).

No other changes are needed, so the Dependent Design Case scenario is now complete.

➢**Select Run from the Analysis menu** to perform the automated sizing, and go to the Output window once the run is complete to view the results.
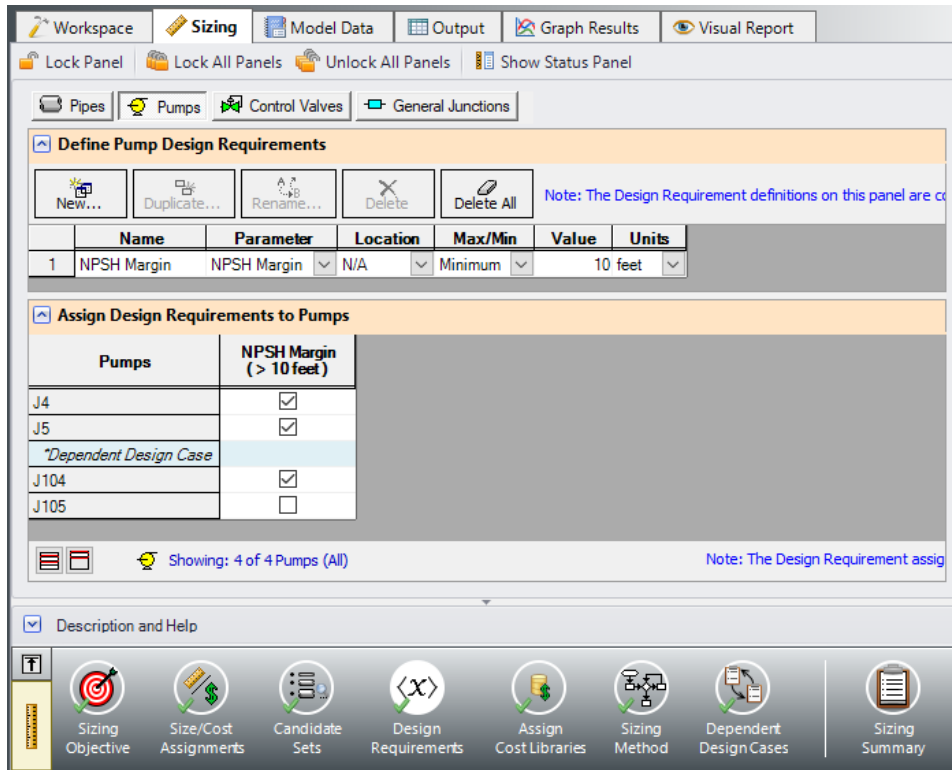
Figure 6: Pump Design Requirements for the dependent design case
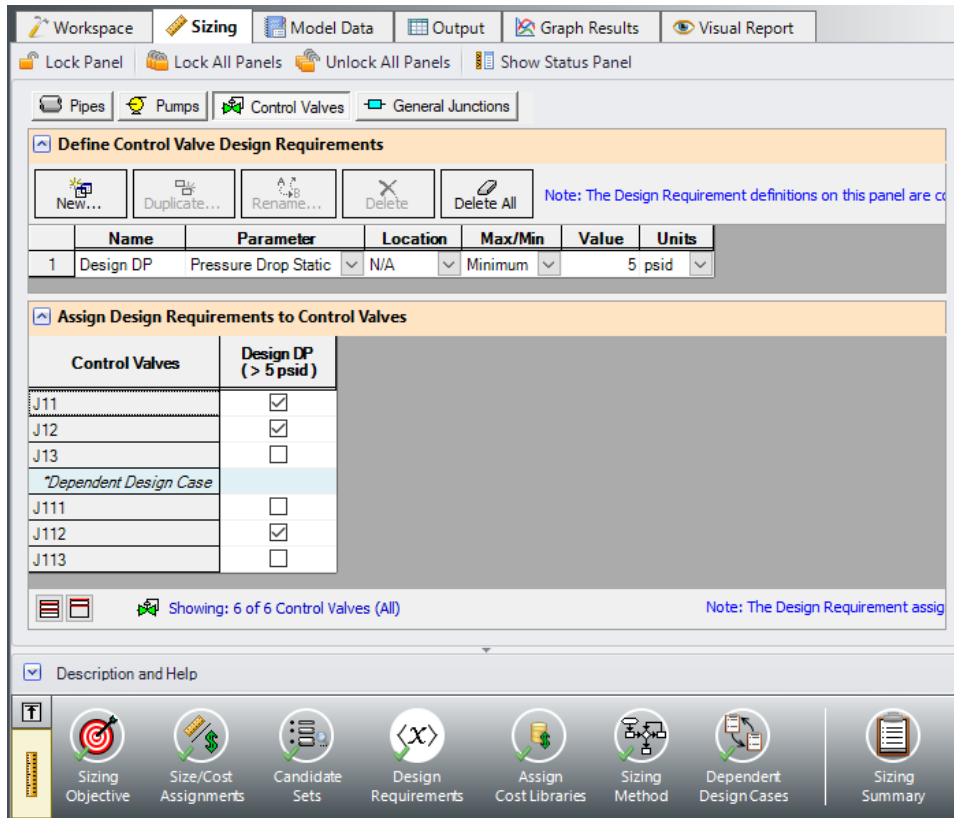
Figure 7: Control Valve Design Requirements for the dependent design case
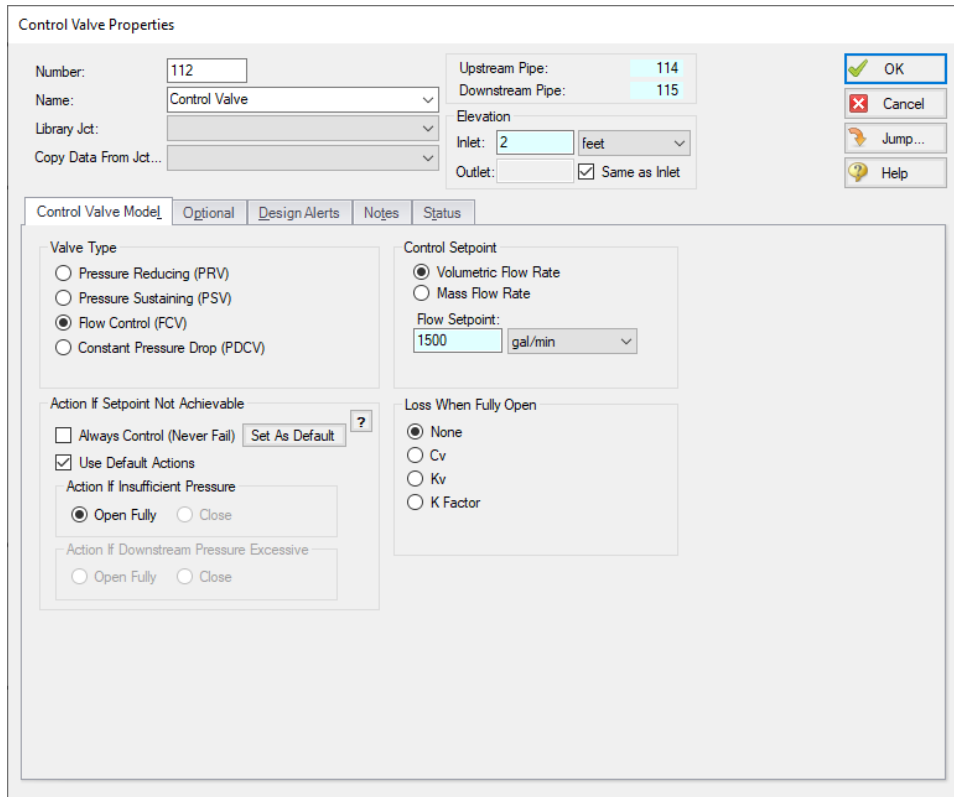
Figure 8: Flow Setpoint adjusted in the dependent design case for FCV J112

Summary

The cost report for the Dependent Design Case scenario can be seen in Figure 9 below.

Note that the pipes and junctions in the dependent case are separate entities for modeling purposes, but represent the same pipes and junctions as the primary case. Thus when performing automated sizing, we only count the cost of the pipes and junctions once. This occurs in the primary design case, which in our model is the two pumps running case. This is why only the primary design case pipes are shown in the cost report.

With one exception, it is not possible for a model that includes a dependent design case to yield a better design than the primary case alone. At best, it can yield the same design, such as in this scenario. This is due to the fact that the dependent design case imposes additional restrictions on the solution by adding additional design requirements. If the design for the primary design case is not sufficient for the dependent design case, then the design must be changed, which will increase cost in some manner.

Turning off Pump 105 in this scenario required no adjustments in the pipe sizes, so the initial sizing results remained unchanged.

Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									**2,216,402**
Total of All Shown Costs			**133,229**	**62,957**	**196,186**	**0**	**2,020,216**	**2,020,216**	**2,216,402**
Items In Sizing			**133,229**	**62,957**	**196,186**	**0**	**2,020,216**	**2,020,216**	**2,216,402**
Items Not In Sizing			**0**	**0**	**0**	**0**	**0**	**0**	**0**
Pipe Subtotal			**87,926**	**45,862**	**133,788**	**0**	**0**	**0**	**133,788**
P1	Pipe	Pipe	3,695	1,861	5,556	0	0	0	5,556
P2	Pipe	Pipe	458	254	712	0	0	0	712
P3	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P4	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P5	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P6	Pipe	Pipe	764	423	1,187	0	0	0	1,187
P7	Pipe	Pipe	458	254	712	0	0	0	712
P8	Pipe	Pipe	36,948	18,612	55,560	0	0	0	55,560
P9	Pipe	Pipe	458	254	712	0	0	0	712
P10	Pipe	Pipe	458	254	712	0	0	0	712
P11	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P12	Pipe	Pipe	378	297	675	0	0	0	675
P13	Pipe	Pipe	378	297	675	0	0	0	675
P14	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P15	Pipe	Pipe	378	297	675	0	0	0	675
P16	Pipe	Pipe	378	297	675	0	0	0	675
P17	Pipe	Pipe	755	594	1,349	0	0	0	1,349
P18	Pipe	Pipe	378	297	675	0	0	0	675
P19	Pipe	Pipe	378	297	675	0	0	0	675
P20	Pipe	Pipe	458	254	712	0	0	0	712
P21	Pipe	Pipe	458	254	712	0	0	0	712
P22	Pipe	Pipe	36,948	18,612	55,560	0	0	0	55,560
Bend Subtotal			**1,792**	**2,982**	**4,774**	**0**	**0**	**0**	**4,774**
J3	Bend	Bend	525	852	1,377	0	0	0	1,377
J6	Bend	Bend	525	852	1,377	0	0	0	1,377
J10	Bend	Bend	525	852	1,377	0	0	0	1,377
J19	Bend	Bend	217	426	643	0	0	0	643
Pump Subtotal			**43,510**	**14,113**	**57,623**	**0**	**2,020,216**	**2,020,216**	**2,077,839**
J4	Pump	Pump	21,728	7,047	28,774	0	1,009,286	1,009,286	1,038,061
J5	Pump	Pump	21,783	7,067	28,849	0	1,010,930	1,010,930	1,039,779

Figure 9: Cost Report for the Dependent Design Case scenario

Hot Water System - ANS

Summary

This example will take an existing hot water system design, scale it up, and size the pipes to meet a set of system requirements at a minimal cost. The system cost is included to demonstrate the need for sizing. The system must meet the following requirements:

- Minimum 10 psid drop across flow control valves in all cases
- Maximum 8 feet/sec velocity in all pipes in all cases
- No NPSH violations
- All pumps must operate between 80% to 100% of the BEP (Best Efficiency Point)
- System must also function with Pump A turned off with no BEP restrictions
- The flow rate through the heat exchangers is controlled by FCVs set to 3,000 gal/min.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Defining pump Design Requirements
- Sizing the system to minimize initial costs
- Using Dependent Design Cases

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Hot Water System.fth*
- *US - Hot Water System - ANS.fth*

- *Hot Water System.dat* - engineering library
- *Hot Water System Costs.cst* - cost library for *Hot Water System.dat*
- *Steel - ANSI Pipe Costs.cst* - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **US - Hot Water System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**.

Although the model is fully defined, a few alterations are necessary for this example to scale up the system. Open the Workspace Layers tab in the Quick Access Panel and load the Layer Preset **ANS Sizing Group View** to display the sizing group colors. The Workspace should now appear as shown in Figure 1 below. Colors for each sizing group can be toggled individually by toggling their respective layer.
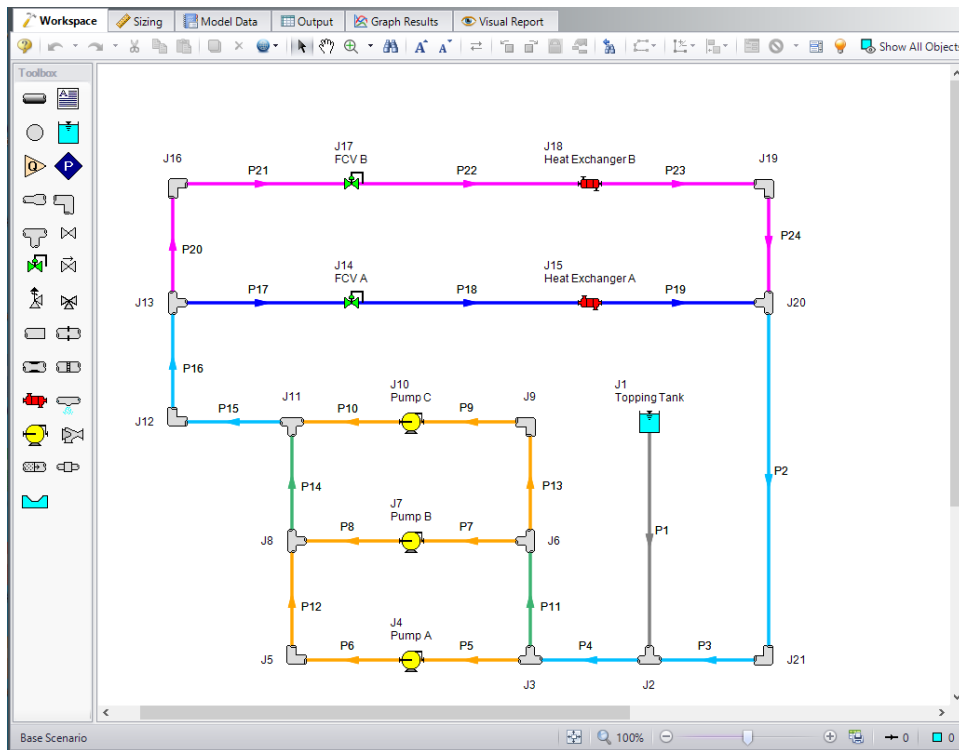
Figure 1: Workspace for the hot water system model with Common Size Groups indicated by color

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Pipes and Junctions Group

For this example, the existing system is being scaled up. As an initial design, set all pipe diameters to 20 inches and increase the flow setpoints of the control valves to 3000 gal/min.

Make the following changes to these pipes junctions

1. All Pipes
    a. Size = 20 inch
    b. Type = STD (schedule 20)
2. J14 & J17 Control Valves
    a. Flow Setpoint = 3000 gal/min

Step 5. Configure Sizing Settings

We will need to define the sizing settings to account for the three given Design Requirements while minimizing the initial costs for the system. To do this, we will follow the Sizing steps as outlined below.

A. Sizing Objective

The Sizing Objective panel should be selected by default in the Sizing Navigation panel.

Select Perform Sizing.

For the Objective, make sure that **Monetary Cost** is selected, and **Minimize** is chosen from the drop-down.

We will also need to specify which types of monetary cost we want to size for. Choose **Size for Initial and Recurring Costs**, if not selected already, to move the Material and Installation costs to the column to be sized.

Set the System Life to 10 years.

Under Energy Cost, select Use This Energy Cost Information and enter 0.12 U.S. Dollars Per kW-hr.

B. Size/Cost Assignments

On the toolbar at the bottom of the window select the Size/Cost Assignments button. The Size/Cost Assignments window allows the user to define what will be sized in the model.

To increase the analysis speed and reduce complexity, it is useful to create Common Size Groups to group the pipes in different sections of the model. For this system we have defined 5 groups to be sized, as can be seen in Figure 2. We do not want to size pipe P1, but we will include it in the cost calculation for the sake of comparison later.

➢**Create the Common Size Groups** shown in Figure 1 and Figure 2 in the model by either using the radio buttons in the table, or by selecting the pipes in the Workspace and adding them to groups from the right click menu.
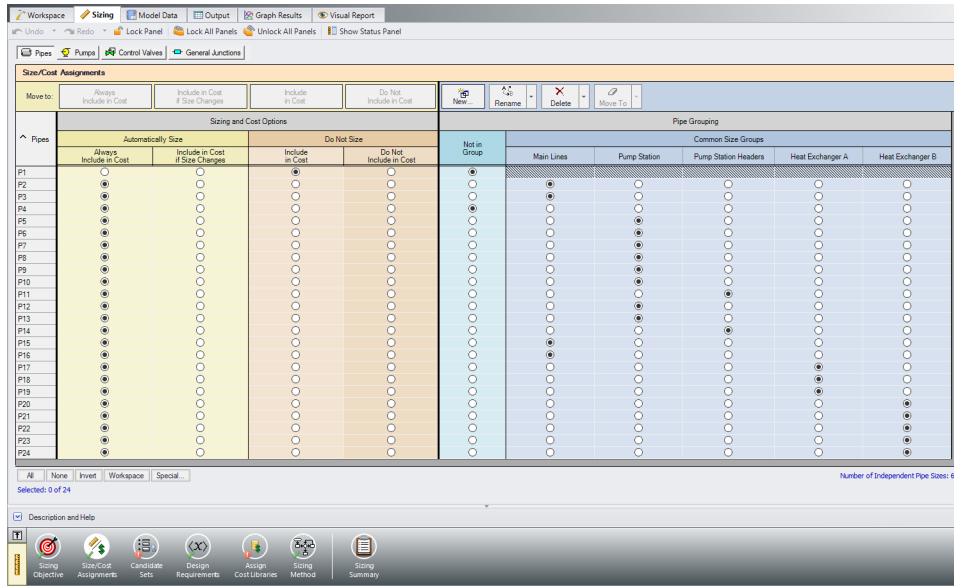
Figure 2: Size/Cost Assignments using Common Size Groups to simplify calculations

Besides the pipes, it is also desired to include the pumps in the sizing.

➢**Select the Pumps button** then set each of the pumps to be included in the Cost Report and Sizing.

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Click New under Define Candidate Sets and name it *STD Steel 1"-36"*. In the Select Pipe Sizes window, select Steel - ANSI from the dropdown menu, expand the STD group (make sure the Sort selection on the bottom is Type, Schedule, Class), then add all pipes from 1 inch to 36 inch to the Candidate Set list on the right. Click OK to close the Select Pipe Sizes window.

In the bottom section of the window, check that each of the Common Size Groups have been assigned to use the existing Candidate Set.

D. Design Requirements

Select the Design Requirements button.

For this model we only have three given Design Requirements which will need to be applied for the pipes, control valves, and pumps. We will start with the pipes:

1.   Ensure the Pipes button at the top of the window is selected.
2.   Click the New button under Define Pipe Design Requirements.
3.   Enter a name when prompted: *Max Pipe Velocity*.
4.   In the table, select Velocity as the Parameter.
5.   Choose Maximum for Max/Min, and enter 8 feet/sec.

Now that the requirement is defined, make sure that it has been applied to all pipes in the model by checking the box next to each of the Common Size Groups in the bottom section of the window, as shown in Figure 3.

Repeat the same process as above by clicking the Pumps, then Control Valves button to apply the Design Requirements for the Pumps and Control Valves as follows, and as is shown in Figure 5 and Figure 4 respectively.

- All of the Pumps must have a minimum % Flow of BEP of 80% and a maximum % Flow of BEP of 100%
- Both Control Valves must have a minimum stagnation pressure drop of 10 psid

Note: The Design Requirements are not enforced when the model is not being sized. Instead, the design requirements are treated as Design Alerts, which is useful to assess the feasibility of the initial design.
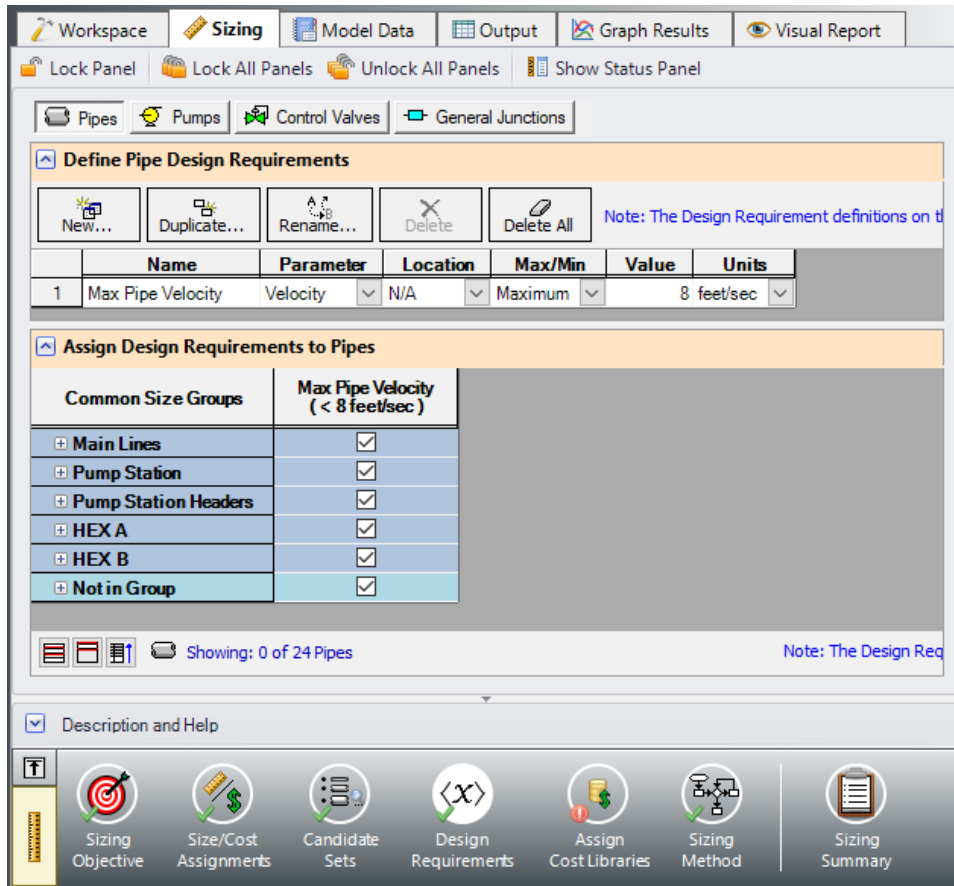
Figure 3: Pipe Design Requirements

Figure 4: Pump Design Requirements
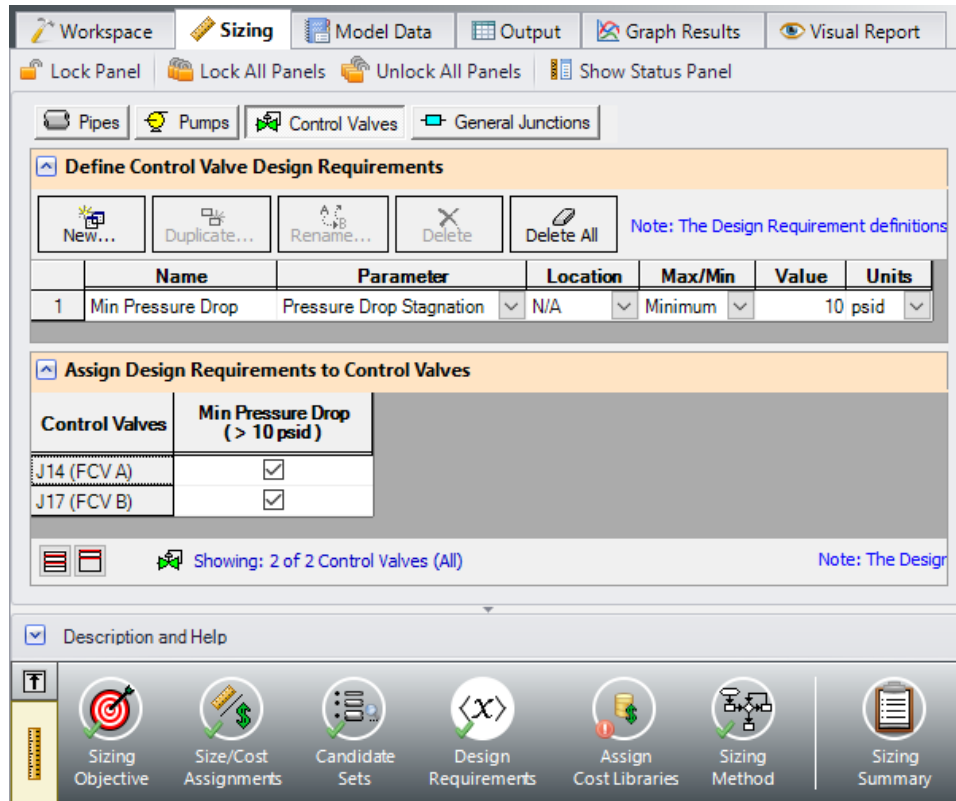
Figure 5: Control Valve Design Requirements

E. Assign Cost Libraries

Select the Assign Cost Libraries button. For this model the engineering and cost libraries have already been created, but we will need to check that they have been connected and applied properly.

➢**Open the Library Manager** by clicking the Library Manager button to see which libraries are connected. The AFT INTERNAL LIBRARY and LOCAL USER LIBRARY should be available and connected. The AFT internal library also has a cost library associated with it. This example uses the engineering library *Hot Water System.dat* and two cost libraries, *Hot Water System Costs.cst* and *Steel - ANSI Pipe Costs.cst* which are linked with the Hot Water System engineering library and the pipe material library for Steel - ANSI, respectively. If the libraries are connected, there will be a check in the box next to their name, as shown in Figure 6.

If they are not in the available library list, add them from the Examples folder in the AFT Fathom 13 folder. Click Add Existing Library to navigate to and select the library *Hot Water System.dat*. Next, click Add Existing Cost Library and add *Hot Water System Costs.cst* and *Steel - ANSI Pipe Costs.cst*.

The US - ANS Hot Water Heat Exchanger junction will only be used for the heat exchanger resistance curve. The US - ANS Hot Water Pump junction will be used for the three pumps. These are the only junctions in this library that will be used in this example.

Once they have been added to the list, check to see that they are also connected. If they are not automatically connected, click the check box next to their name to connect them as seen in Figure 6. Click Close.
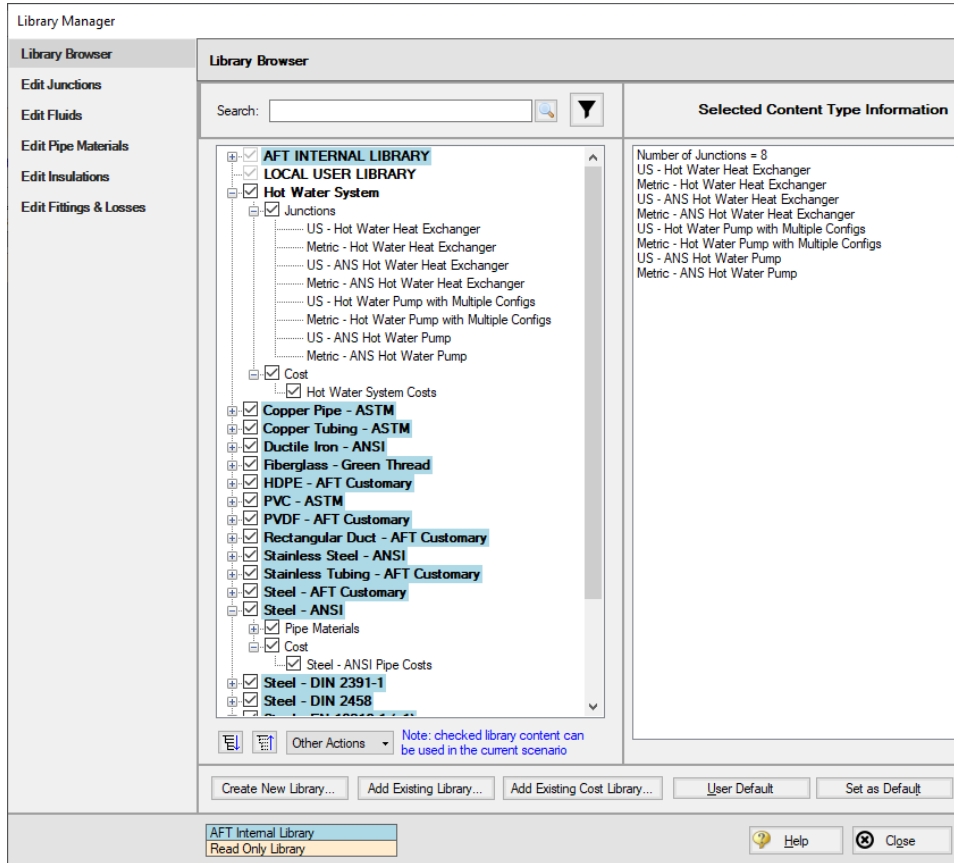
Figure 6: Library Manager with Hot Water Pumps connected

Back in the Assign Cost Libraries window for the Pipes, the **STD Steel Pipe 1"-36"** Candidate Set should now show the **Steel - ANSI Pipe Costs** library applied to it.

Navigate to the Workspace and connect all three pump junctions to the **US - ANS Hot Water Pump** library junction by opening the pump properties window and selecting **US - ANS Hot Water Pump** from the Library Jct list. Do the same for both Heat Exchanger junctions, selecting **US - ANS Hot Water Heat Exchanger**.

Back in the Sizing window on the Assign Cost Libraries panel, click on the Pumps button to view the cost library assignments for the Pumps. Similar to the Pipes, the **Hot Water System Costs** library should be selected for Pumps 4, 7, and 10, showing that it is being applied.

F. Sizing Method

Select the Sizing Method button from the Sizing Navigation panel.

Make sure that Discrete Sizing is selected for the Sizing Type.

For this model both the **Modified Method of Feasible Directions (MMFD)** and the Sequential Quadratic Programming (SQP) methods will produce the same results, but MMFD is much faster. Select MMFD for the Search Method.
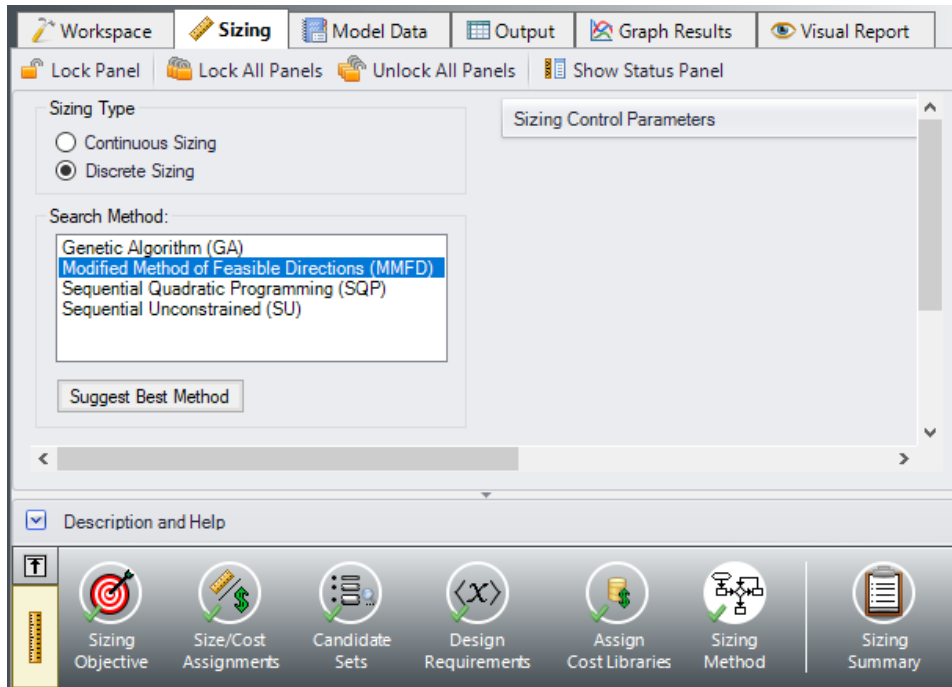
Figure 7: Sizing Method Settings to discretely size the system

Step 6. Create Scenarios

For this system we would like to get a baseline cost for the current design to compare against the sizing results.

In the Scenario Manager in the Quick Access Panel create two scenarios and name them *Before Sizing* and *After Sizing*.

Load the Before Sizing scenario if it is not already active by double-clicking the scenario name in the Scenario Manager. We need to set this scenario to calculate costs without sizing the system.

In the Sizing window go to the Sizing Objective panel, then select **Calculate Cost, Do Not Size**.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Review the Results

The Cost Report for the initial design can be seen in Figure 8. The total cost comes out to $2,162,905. In the pipe output section there is a Pipe Design Requirements tab and in the junction output section there is a Pump Design Requirements and Control Valve Design Requirements tab. These tabs will show if the current model fulfills the Design Requirements. We can investigate the design requirement tabs and see that the initial system meets all of the Design Requirements, **meaning that it is a feasible solution**; however, a lower cost system can likely be found by sizing the system.

	Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
	TOTAL OF ALL MODEL COSTS									2,162,905
	Total of All Shown Costs			391,204	191,099	582,302	0	1,580,602	1,580,602	2,162,905
	Items In Sizing			0	0	0	0	0	0	0
	Items Not In Sizing			391,204	191,099	582,302	0	1,580,602	1,580,602	2,162,905
	Pipe Subtotal			363,938	183,328	547,266	0	0	0	547,266
	P1	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P2	Pipe	Pipe	55,422	27,918	83,340	0	0	0	83,340
o	P3	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P4	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P5	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P6	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P7	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P8	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P9	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P10	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P11	Pipe	Pipe	5,542	2,792	8,334	0	0	0	8,334
o	P12	Pipe	Pipe	5,542	2,792	8,334	0	0	0	8,334
o	P13	Pipe	Pipe	5,542	2,792	8,334	0	0	0	8,334
o	P14	Pipe	Pipe	5,542	2,792	8,334	0	0	0	8,334
o	P15	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P16	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P17	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P18	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P19	Pipe	Pipe	27,711	13,959	41,670	0	0	0	41,670
o	P20	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P21	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o	P22	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o	P23	Pipe	Pipe	27,711	13,959	41,670	0	0	0	41,670
o	P24	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
	Pump Subtotal			27,266	7,771	35,036	0	1,580,602	1,580,602	1,615,639
o	J4	Pump	Pump A	9,094	2,592	11,686	0	527,348	527,348	539,034
o	J7	Pump	Pump B	9,078	2,587	11,665	0	525,907	525,907	537,571
o	J10	Pump	Pump C	9,094	2,592	11,686	0	527,348	527,348	539,034

Figure 8: Cost Report for the initial system design before sizing

Step 9. Run the Model and View Results

Switch to the After Sizing scenario in the Scenario Manager. This scenario should already be fully defined for sizing.

Select Run from the Analysis menu to perform the automated sizing calculations. Using the MMFD method, the automated sizing takes about 2 minutes, depending on your machine. When the run has completed click Output to view the results, as shown in Figure 9.

The automated sizing has given us savings of about $156,500 overall on material, installation, and operation costs over the lifespan of the system. This is a 7% reduction.

		Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
		TOTAL OF ALL MODEL COSTS									2,006,366
		Total of All Shown Costs			271,013	154,709	425,722	0	1,580,644	1,580,644	2,006,365
		Items In Sizing			252,539	145,403	397,942	0	1,580,644	1,580,644	1,978,586
		Items Not In Sizing			18,474	9,306	27,780	0	0	0	27,780
		Pipe Subtotal			243,747	146,938	390,684	0	0	0	390,684
		P1	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o		P2	Pipe	Pipe	55,422	27,918	83,340	0	0	0	83,340
o		P3	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o		P4	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o		P5	Pipe	Pipe	7,550	5,940	13,490	0	0	0	13,490
o		P6	Pipe	Pipe	3,775	2,970	6,745	0	0	0	6,745
o		P7	Pipe	Pipe	7,550	5,940	13,490	0	0	0	13,490
o		P8	Pipe	Pipe	3,775	2,970	6,745	0	0	0	6,745
o		P9	Pipe	Pipe	7,550	5,940	13,490	0	0	0	13,490
o		P10	Pipe	Pipe	3,775	2,970	6,745	0	0	0	6,745
o		P11	Pipe	Pipe	3,714	2,276	5,991	0	0	0	5,991
o		P12	Pipe	Pipe	2,265	1,782	4,047	0	0	0	4,047
o		P13	Pipe	Pipe	2,265	1,782	4,047	0	0	0	4,047
o		P14	Pipe	Pipe	3,714	2,276	5,991	0	0	0	5,991
o		P15	Pipe	Pipe	9,237	4,653	13,890	0	0	0	13,890
o		P16	Pipe	Pipe	18,474	9,306	27,780	0	0	0	27,780
o		P17	Pipe	Pipe	9,785	6,707	16,492	0	0	0	16,492
o		P18	Pipe	Pipe	4,893	3,354	8,246	0	0	0	8,246
o		P19	Pipe	Pipe	14,678	10,061	24,738	0	0	0	24,738
o		P20	Pipe	Pipe	4,893	3,354	8,246	0	0	0	8,246
o		P21	Pipe	Pipe	9,785	6,707	16,492	0	0	0	16,492
o		P22	Pipe	Pipe	4,893	3,354	8,246	0	0	0	8,246
o		P23	Pipe	Pipe	14,678	10,061	24,738	0	0	0	24,738
o		P24	Pipe	Pipe	4,893	3,354	8,246	0	0	0	8,246
		Pump Subtotal			27,266	7,771	35,037	0	1,580,644	1,580,644	1,615,681
o		J4	Pump	Pump A	9,061	2,582	11,643	0	524,468	524,468	536,111
o		J7	Pump	Pump B	9,144	2,607	11,751	0	531,709	531,709	543,460
o		J10	Pump	Pump C	9,061	2,582	11,643	0	524,468	524,468	536,111

Figure 9: Sizing output based on the initial cost of the system with all pumps operating

Step 10. Create Dependent Design Case

We also want the system to function with one pump turned off. To size a system with two different operating cases, we can create a Dependent Design Case that has the exact same pipes and junctions as the primary design but has a different set of requirements.

Create a new child of the After Sizing scenario in the Scenario Manager, and name it *Sizing with Pump A off*.

Here are the steps to create this Dependent Design Case:

1. Go to the **Sizing Objective** panel and check the box next to **Enable Dependent Design Cases** option.
2. Go to the Dependent Design Cases panel. You should now see instructions displayed to create the Dependent Design Case, along with a summary table of dependent design settings. Now go to the **Workspace** and choose **Select All** from the **Edit menu**.
3. Open **Duplicate Special** from **Edit menu**, enter an increment of **100** and select **Make Dependent Design Case**.
4. Click **OK**, then **paste** the duplicated model so that it does not overlap the original.
5. When the Workspace has been duplicated, select the **Pump A** from the Dependent Design Case

(J104). In the Pump Properties window select the **Optional tab** and check the **Pump Off No Flow** radio button under Special Conditions. This will turn Pump A off.

We have now made all of the changes required in the Workspace. We will also need to make some adjustments to the Design Requirements for our Dependent Design Case. With Pump A turned off we no longer require any of the pumps to operate between 80% and 100% of BEP. We will therefore need to remove the Pump Design Requirements.

➢**Go to the Design Requirements panel** and select the **Pumps** button. In the Assign Design Requirements to Pumps section, **remove** the Design Requirements for all of the dependent design pumps by unchecking the boxes as is shown in Figure 10.

Once this is done run the model again and go to the Output window.
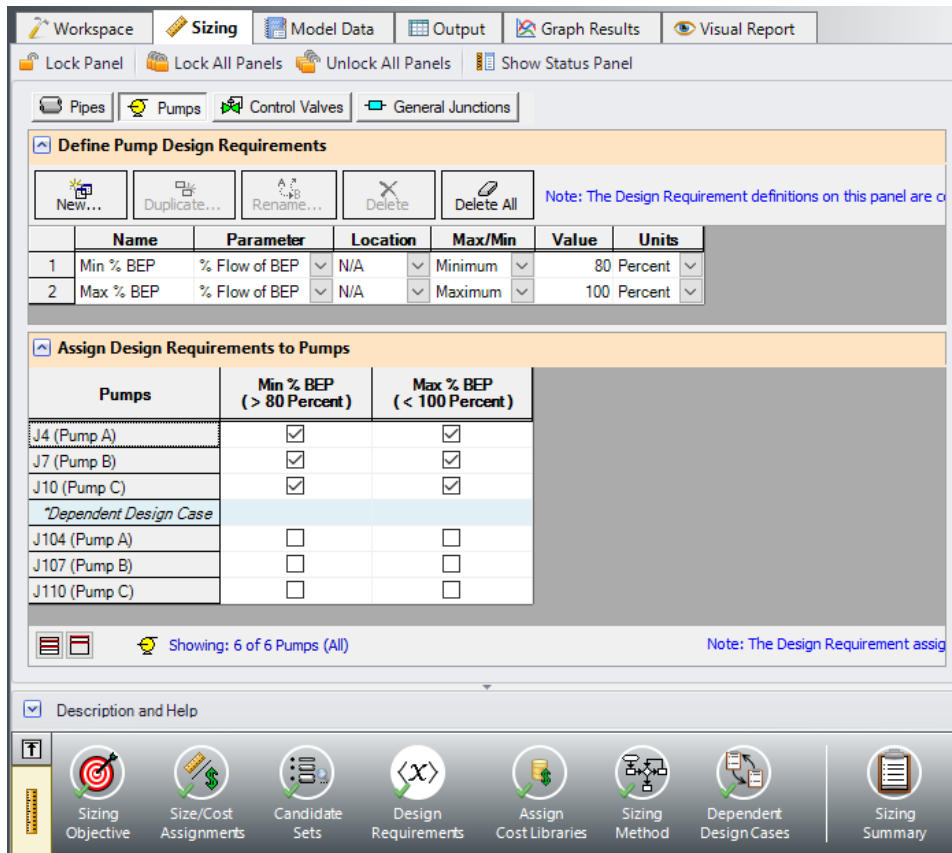
Figure 10: Pump Design Requirements defined for the Dependent Design Case pumps

Summary

Table 1 compares the results for each of the scenarios. The final design is the lowest cost that meets all the requirements. It can be seen that in order to meet the system requirements for the case where Pump A is turned off, slightly larger pipe sizes are used, resulting in a higher system cost.

The Final Design provides a $136,517 (6%) cost reduction from the Initial Design while still meeting all design requirements.

Table 1: Summary of costs for Hot Water System scenarios

Design Stage	Initial Design	Design	Final Design
Scenario	Before Sizing	After Sizing	Sizing with Pump A off
Meets all Requirements	Yes	Yes	Yes
Material	391,204	271,013	286,068
Installation	191,099	154,709	159,651
Non-Recurring Sub Total	582,302	425,722	445,719
Operation/Energy	1,580,602	1,580,644	1,580,669
TOTAL	2,162,905	2,006,366	2,026,388

Variable Drive - ANS

Summary

As seen in other examples, the ANS module can be used to size a system for life cycle cost. The ANS module will size the complete system, pumps and pipes to minimize the total cost of operating the system for the life of the system. Often sizing the pumps and pipes is not enough because the control system is often a large portion of the initial cost and the method of control, flow control valves (FCV) or variable frequency drives (VFD), can have a large impact on the system energy usage. This example will demonstrate how to size pumps and control valves in parallel and then will investigate the savings of using a VFD to control flow instead of a FCV.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Using Scenarios to size a system for two different sets of system requirements
- Using Common Size Groups and observing their effect on how well the ANS module can size a system

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Since this example employs many concepts from the Cooling System - ANS example, it is recommended to complete that example first before beginning this example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *US - Variable Drive - ANS.fth*
- *Variable Drive.dat* - engineering library
- *Variable Drive - Pump Costs.cst* - cost library for *Variable Drive.dat*

- *Variable Drive - VFD Costs.cst* - cost library for *Variable Drive.dat*
- *Variable Drive - Pipe Costs.cst* - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Set up Library Manager

This example uses engineering and cost libraries. It is important that these libraries are added and connected to Fathom for defining the pipes and junction.

Go to the Library menu and open Library Manager. Click Add Existing Library and navigate to the AFT Fathom 13 Examples folder. Select the library *Variable Drive.dat* and click Open. Make sure that the Variable Drive engineering library in Library Manager has a check mark next to it. The cost libraries will be added later during the Sizing steps.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = User Specified Fluid
   b. Name = Lube Oil
   c. Density = 0.7 S.G. water
   d. Dynamic Viscosity = 0.29 kg/sec-m

Step 5. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions

fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
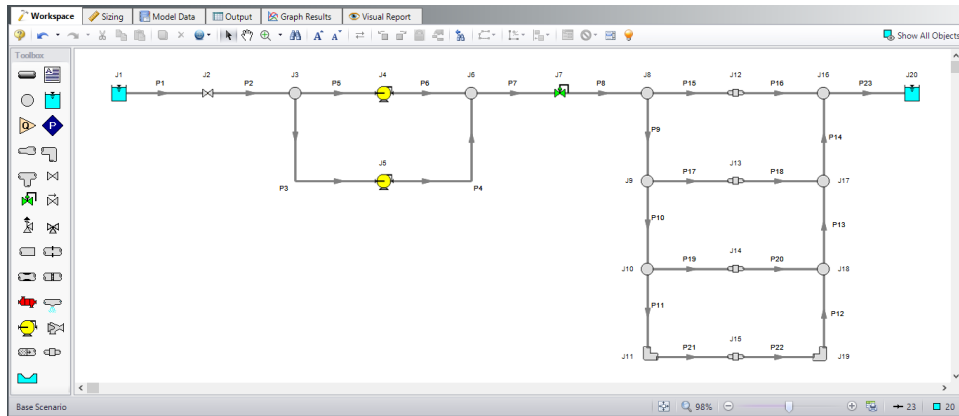
Figure 1: Layout of Generator Lube Oil System

Pipe Properties

1. Pipe Model tab
   a. Pipe Material = Steel - ANSI
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = 6 inch
   d. Type = schedule 40
   e. Friction Model Data Set = Standard
   f. Lengths =

Pipe	Length (feet)
1	20
2-4	10
5-6	6
7	4
8	15
9-14	10
15-22	20
23	50

Junction Properties

1. J1 Reservoir
   a. Name = Lube Oil Supply Tank
   b. Liquid Surface Elevation = 20 feet
   c. Liquid Surface Pressure = 1 atm
   d. Pipe Depth = 10 feet
2. J2 Valve
   a. Name = Isolation Valve
   b. Inlet Elevation = 0 feet
   c. Valve Data Source = User Specified
   d. Loss Model = K Factor
   e. K = 0.15
3. All Branches
   a. Elevation = 0 feet
4. J4 & J5 Pumps
   a. J4 Name = Primary Pump
   b. J5 Name = Auxiliary Pump
   c. Library Jct = Pump 2000 gal/min Sizing
   d. Inlet Elevation = 0 feet
   e. J5 Special Condition = Pump Off No Flow
5. J7 Control Valve
   a. Name = 2000 gal/min Control
   b. Inlet Elevation = 0 feet
   c. Valve Type = Constant Pressure Drop (PDCV)
   d. Control Setpoint = Pressure Loss
   e. Pressure Drop = 10 psid
6. J11 & J19 Bend
   1. Inlet Elevation = 0 feet
   2. Type = Standard Elbow (knee, threaded)
   3. Angle = 90 Degrees
7. J12-J15 General Components
   a. J12 Name = Generator #1
   b. J13 Name = Generator #2
   c. J14 Name = Generator #3
   d. J15 Name = Generator #4
   e. Library Jct = US - Generator
   f. Inlet Elevation = 0 feet
8. J20 Reservoir
   a. Name = Return Tank
   b. Liquid Surface Elevation = 15 feet
   c. Liquid Surface Pressure = 1 atm
   d. Pipe Depth = 15 feet

The system supplies lube oil to a bank of four generators. Each generator requires a flow rate of 500 gal/min. The auxiliary pump will not be turned on but the material and installation pump costs will be included in the cost report and automated sizing.

The system uses the following specifications:

1. Minimum 10 psid drop across the flow control valve
2. Flow rate at each generator must be between 450 and 550 gal/min
3. The cost of the auxiliary pump and piping must be included in the sizing.
4. The pump is expected to be about 80% efficient.
5. The system uses lube oil with a specific gravity of 0.7 and viscosity of 0.29 kg/s-m.

Step 6. Configure Sizing Settings

We want to first size our system to determine the required pump parameters, which will then be used to choose a pump model and perform a final sizing calculation.

To start, set up the model to size the pump system for life cycle cost. Select the Sizing tab.

A. Sizing Objective

The Sizing Objective panel should be selected by default from the navigation panel. Choose the settings as follows:

For the Sizing Option choose **Perform Sizing**.

For the Objective, choose **Monetary Cost**, and select the **Minimize** option from the drop-down list.

Under Options to Minimize Monetary Cost, choose **Size for Initial and Recurring Costs**. This should automatically configure the Cost Objective table to include the material, installation, and energy costs in the sizing.

Define a **10 year** System Life.

For the Energy Cost area choose **Use This Energy Cost Information**, with a cost of **0.12 U.S. Dollars Per kW-hr**.

B. Size/Cost Assignments

On the Sizing Navigation panel go to the **Size/Cost Assignments** panel.

We are designing a new system, so we want to include all of the pipes in the model for sizing.

➢**Select the All button** at the bottom of the panel to select all pipes, then click **Always Include in Cost** at the top of the table to move all pipes to be included in the sizing calculation.

To decrease the analysis speed and reduce complexity, it is useful to **create Common Size Groups** to group the pipes in different sections of the model. For this system we want to create six groups to be sized, as can be seen in Figure 2 based on the pipe coloring, and Figure 3 based on the table. The groups were chosen by considering sections of pipe which would logically have similar demands due to the flow capacity and connected junctions.

Create each of the groups by selecting the appropriate pipes in the Workspace, then right-clicking on the Workspace and choosing Add Pipes to Common Size Group -> New Group and giving the group a name. Alternatively the groups can be created on the Sizing window by clicking New to create and name the group, then adding pipes to the group using the radio buttons. The Size/Cost Assignments panel should appear as shown in Figure 3.
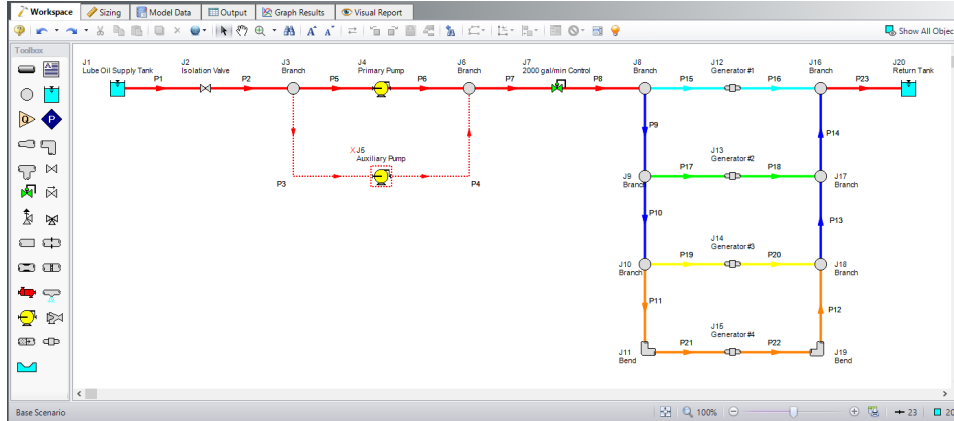
Figure 2: Workspace showing pipe groups by color

Figure 3: Size/Cost Assignments using Common Size Groups to simplify calculations

➢**Click the Pumps button** at the top of the panel to switch to the Pump Size and Cost Assignments. As stated earlier, we want to size both pumps, even though the Auxiliary Pump is not in use, so select **Include in Cost Report and Sizing** for each of the Pump junctions. Since the pumps are in parallel in this model, it is desired for the pumps to be the same size. This means that they will have the same material and installation costs, though the maintenance cost will be different since the Auxiliary Pump is not regularly used, unlike the Primary Pump. To account for this we will need to create a Maximum Cost Group, which will set both pumps to use the cost information for the most expensive pump in the group. If a Maximum Cost Group is not used, then the Auxiliary Pump will have its material and installation costs set to

the lowest possible cost. (The cost at the pump is estimated using power, and the power at the Auxiliary Pump is set at zero since it is not in use during normal operation.)

➢**Create a New Maximum Cost Group** named *Pumps*, and add the Material and Installation costs to it for each of the pumps, but under Maintenance, make sure to leave both pumps in the Not in Group column. The Pumps Size/Cost Assignments should appear as shown in Figure 4.
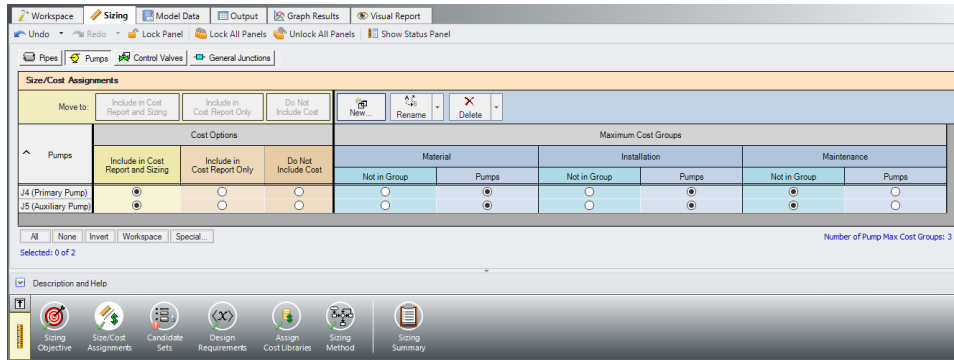
Figure 4: Pump Size/Cost Assignments with Maximum Cost Groups assigned

C. Candidate Sets

Go to the **Candidate Sets** panel.

For this system we will consider a range of Steel-ANSI schedule 40 pipes as candidates. To define this in Fathom, create a new Candidate set as follows:

1. Create a new set by clicking **New**, and name the set *Galvanized Steel Pipe 1-18 in*.
2. Select **Steel - ANSI** as the material from the drop-down list
3. In the Available Material Sizes and Types, expand the sizes for **schedule 40**, then add each of the sizes from **1 inch to 18 inch** by double clicking the names or selecting them and clicking Add>>.
4. Click **OK** to finish editing the set.

In the bottom section of the panel, assign each of the Common Size Groups to the Galvanized Steel Pipe 1-18 in Candidate Set. The panel should appear as shown in Figure 5.
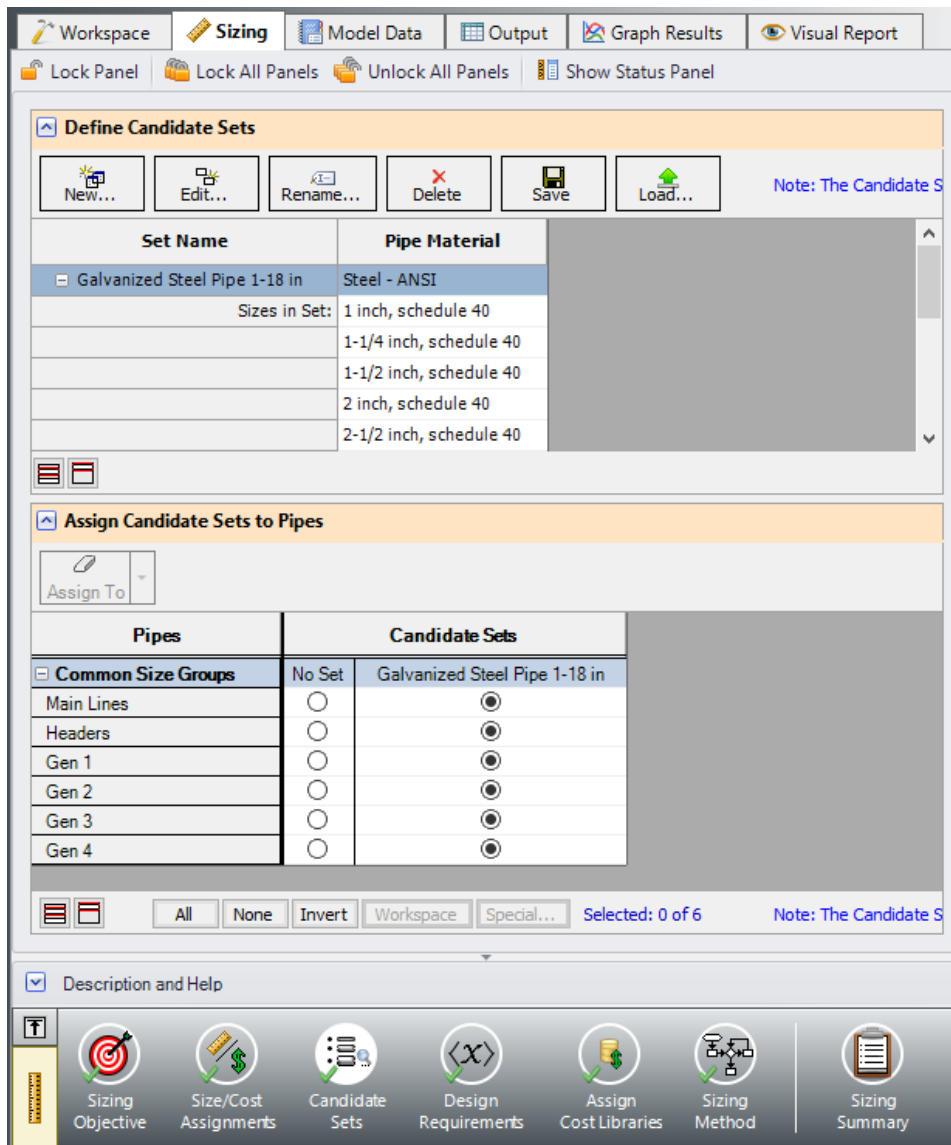
Figure 5: Candidate Sets panel fully defined for the model

D. Design Requirements

Go to the **Design Requirements** panel.

As can be seen above, this model has three given Design Requirements which need to be applied for the generators and control valve. Since we are not able to directly apply a flow rate design requirement on the generators, we will instead apply the flow rate requirements for the generators to the pipes adjacent to the generators. To do this:

1. Make sure the Pipes button at the top of the window is selected.
2. Click **New** under Define Pipe Design Requirements.

3. Enter the name *Maximum Generator Flow*.

4. In the table for Parameter, select **Volumetric Flow Rate**.

5. For Max/Min, choose **Maximum**, and enter **550 gal/min**.

6. Repeat the process from steps 1-5 to define a Design Requirement for a **Minimum** generator flow rate of **450 gal/min**.

Now that the generator flow rate requirements are defined, apply them to the common size groups for the generator piping (Gen 1, Gen 2, Gen 3 and Gen 4) by checking the appropriate boxes in the Assign Design Requirements to Pipes section.

Click on the **Control Valves** button and create a **New** requirement named *Minimum Pressure Drop* to set the **Pressure Drop Stagnation** parameter to a **Minimum** of **10 psid**. Apply the Design Requirement to Control Valve J7.

E. Assign Cost Libraries

Go to the Assign Cost Libraries panel. For this model, the engineering and cost libraries have already been created, but we will need to check that they have been connected and applied properly.

➤**Open the Library Manager**, and check that the engineering library **Variable Drive** is connected, indicated by a check mark next to its name. If it is not connected, then check the box next to its name.

Next, connect to the three cost libraries. Click Add Existing Cost Library and navigate to the AFT Fathom 13 Examples folder and add the cost libraries *Variable Drive - Pump Costs.cst*, *Variable Drive - VFD Costs.cst*, and *Variable Drive - Pipe Costs.cst*. The two cost libraries, Variable Drive - Pump Costs and Variable Drive - VFD Costs should be shown indented below the engineering library Variable Drive, and the Variable Drive - Pipe Costs library should be indented below Steel - ANSI. Once all of the necessary libraries have been connected, click **OK**.

In the Assign Cost Libraries panel for the Pipes, All should be selected for the Galvanized Steel Pipe 1-18 in Candidate Set, with the Variable Drive - Pipe Costs library being applied.

➤**Click on the Pumps button** to view the cost library assignments for the Pumps. We will only be using Variable Drive - Pump Costs for this scenario, so check the box for this cost library for both J4 and J5 to make sure that we are not applying any extra costs.

F. Sizing Method

Select the Sizing Method button to go to the corresponding panel.

➤**Choose Discrete Sizing**, since it is desired to select discrete sizes for each of the pipes in the model.

This model contains six design variables and 18 design requirements, so the best methods to choose from will be the MMFD or SQP method.

➤**Select the default method**, Modified Method of Feasible Directions (MMFD).

Step 7. Create Scenarios

Before running the model, create two child scenarios, one named *Calculate Costs, Do Not Size* and the other named *Flow Control Valve Initial Cost Sizing*. This will provide a comparison of the initial cost of the model without sizing, to the cost of the sized system.

Load the Calculate Costs, Do Not Size scenario and go to the Sizing window, then to the Sizing Objective panel. Select Calculate Costs, Do Not Size. We will run this scenario first.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Review the Not Sized and Initial Cost Sizing results

With all the pipes set to 6 inches the design requirements are met and the system is in a reasonable state. Navigate to the Cost Report tab and note that the total cost is $816,094.

Now load the **Flow Control Valve Initial Cost Sizing** scenario and go to the Sizing window to the **Sizing Objective** panel. Under Options to Minimize Monetary Cost, choose **Size for Initial Cost, Show Energy Costs**. The scenario is now ready to be run.

Run the model, then click the Output button once the automated sizing is completed.

After sizing the model for Initial Cost, the total cost over 10 years is now calculated as $752,292. The new pipe sizes and cost totals can be seen in Figure 6 below.

Although this sized system uses larger pipe sizes, and therefore increased material and installation cost, than the initial design (4-8 inch versus only 6 inch), the savings comes from a the sized pump requiring much less power to operate. In one short simulation, the system was sized to meet all design requirements and reduce the cost by close to $60,000.

Figure 6: Pipe Sizes based on automated sizing for Initial Costs

Step 10. Size the system based on lifetime cost

In the Scenario Manager on the Quick Access Panel, make a child scenario from the Base Scenario named *Flow Control Valve Life Cycle Sizing*. Load then run this scenario.

On the Output tab in the Cost Report, it can be seen that the total cost for this case is $632,734 using pipe sizes ranging from 8 inches to 12 inches, as can be seen in Figure 7. Though not shown, the Pump section of the Cost Report shows that the two pumps have been set to the same Material and Installation costs due to the Maximum Cost Group that was defined on the Size/Cost Assignments panel. Since the Initial Cost sizing results in a lifetime cost *increase* of about $50,000, the Life Cycle sizing results will be used for the pump configuration information to choose a pump from a manufacturer.

Figure 7: Pipe Sizes based on automated sizing for Life Cycle Costs

Step 11. Choose an Actual Pump for the System

Create a child scenario from the Flow Control Valve Life Cycle Sizing scenario, and name the new scenario Flow Control Valve *Real Pump*. We will now need to update the Pumps and Control Valve with the settings for the proposed system. Load the new Flow Control Valve Real Pump scenario using the Scenario Manager, then make the following changes:

1. For each of the pumps open the Pump Properties window and select **Pump 2000 gal/min Real** from the Library Jct list. This will copy the Pump Curve data into the pump. Click **OK**. Make sure to do the same for the second pump.

2. Open the Control Valve and change the Valve Type to **Flow Control (FCV)** with a Volumetric Flow Rate of 2000 gal/min. Click **OK**

The model should now be ready for analysis. Run the model once more and go to the Output window to review the results, as can be seen in Figure 8 and Figure 9. Note that this scenario does take a bit longer to run than the sizing scenario. With data for the actual pump, the automated sizing finds a slightly more efficient design for the Life Cycle cost analysis by using 6 inch lines at some of the generators rather than 8 inch lines, which provides approximately $20,000 reduction in cost from the sizing scenario. Note that the Overall Efficiency is 83% for the real pump compared to the assumed 80% of the original sizing pump.

Figure 8: Pump Summary for operation of actual pump

Figure 9: Output for automated sizing with actual pump curve entered

Step 12. Compare Cost Using a VFD Pump

As a final step we will compare an alternate VFD model for the pump to analyze the cost savings which can be achieved by using a VFD instead of using a flow control valve. To start, make an additional child scenario from the FCV Life Cycle Sizing scenario, and name it *VFD Pump*. Complete the following steps to update the model:

1. Go to the Workspace. For each of the pumps, open the Pump Properties window and choose the **Pump 2000 gal/min VFD Real** pump from the Library Jct list. Click **OK**

2. Open the Control Valve Properties window. On the Optional tab set the Special Condition to **Fully Open - No Control**. Click **OK**.

3. Go to the Sizing window, and go to the **Design Requirements** panel. For the Control Valve **uncheck** the box for the minimum pressure drop requirement, since the control valve is not active.

4. Navigate to the **Assign Cost Libraries** panel. We will need to represent the additional Material/Installation costs for the VFD, so change the Libraries to **All** so that all libraries will be applied for each of the pumps.

The changes for this scenario should now be complete. Run the model once more and go to the Output window to review the results. With the VFD Pump being used, the sizing for the pipes is actually the same as the FCV Life Cycle Sizing scenario. However, the cost is significantly decreased to $503,649. This is primarily due to the decreased speed of the pump, as can be seen in Figure 10.

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)
4	Show	Primary Pump	2,000	194.6	40.16	132.5	82.72	91.53	56.64	1,806	110.8	N/A	4.975
X5	Show	Auxiliary Pump	0	0.0	N/A	N/A	N/A	0.00	N/A	N/A	N/A	N/A	N/A

Figure 10: Pump summary for operation of VFD pump

Summary

A summary of the total costs for each of the scenarios can be seen in Table 1. It can be seen from the table that this particular system over its 10 year life span is heavily weighted towards operational costs; as the operational costs are further decreased the overall cost for the system decreases, even though the initial costs may be increased.

Table 1: Summary of Automated Sizing Results for Lube Oil System in thousands of dollars

Life Cycle Cost (U.S. Dollars)	Feasible?	Material	Installation	Total Non-Recurring	Operation/Energy	Total Life Cycle
Calculate Costs, Do Not Size	Yes	35,369	14,425	49,793	766,300	816,094
Flow Control Valve Initial Cost Sizing	Yes	33,819	12,932	46,751	705,541	752,292
Flow Control Valve Life Cycle Sizing	Yes	45,475	16,653	62,128	570,606	632,734
Flow Control Valve Real Pump	Yes	42,474	15,798	58,272	556,307	614,579
Variable Frequency Drive Pump	Yes	45,509	16,663	62,172	441,478	503,649

Housing Project - ANS

Summary

This example will size the pipes for a water supply system to a housing development. This example has three operating states. These states are the following:

1. No fire hydrants open, supply to each house is 10 gal/min with a minimum pressure of 60 psig.
2. North Hydrant open with 100 gal/min at 90 psig, supply to each house is 2 gal/min, no minimum pressure.
3. South Hydrant open with 100 gal/min at 90 psig, supply to each house is 2 gal/min, no minimum pressure.

For this example, we will evaluate all three conditions. First, we will use the ANS module to size the system assuming a requirement to use only one pipe size throughout. After this we will size the pipes assuming the pipes can be three different sizes.

> **Note:** This example can only be run if you have a license for the ANS module.

Topics Covered

- Using Dependent Design Cases to satisfy multiple operating modes for a system
- Using the Scenario Manager to size a system for multiple sets of system requirements
- Understanding the effects of Common Size groups on sizing

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner - Three Reservoir example. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the Beginner - Three-Reservoir - ANS example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Housing Project.fth***
- ***US - Housing Project - ANS.fth***

- *Housing Project Costs.cst* - cost library for **PVC - ASTM** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the *US - Housing Project.fth* example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**. Make sure that the Workspace looks like Figure 1 below:
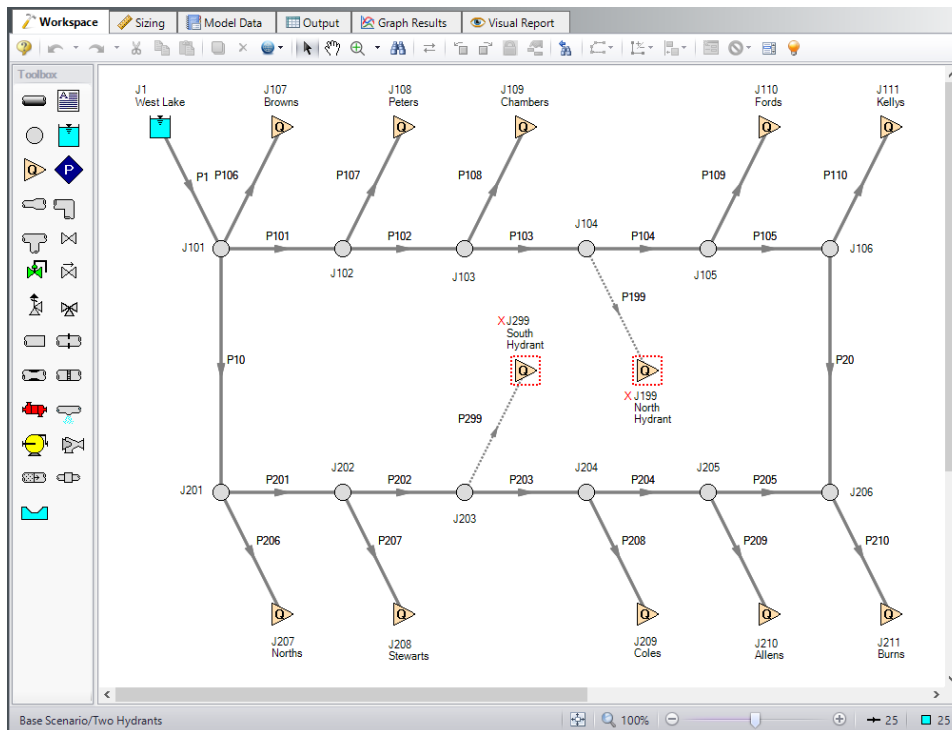
Figure 1: Model layout for normal system operation

This example consists of a reservoir that supplies water to 10 houses and two fire hydrants. All the pipes in the system are Schedule 40 PVC pipe. The supply line to each house is set to 1 inch PVC. The automated sizing of this system will only be used to size the supply line from the reservoir, the neighborhood mains and hydrants.

As mentioned in the introduction, the supply to each house with no fire hydrants open (i.e., the Normal Flow case) is 10 gal/min with a minimum pressure of 60 psig. The model can be set up in two ways;

1. Each house could be represented as an assigned flow with a minimum pressure constraint placed on the supply pipe to each house (as done here), or,

2. Each house can be represented as an assigned pressure with a minimum flow constraint placed on the pipe.

The first option was chosen because with either fire hydrant open, there is still an assigned flow on each house but there is no minimum pressure.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Configure Sizing Settings

For this system we will be sizing the neighborhood mains for three different operating cases, one for normal operation, and two different cases with hydrants in use. When modeling multiple design cases, the different cases need to be broken up into a primary design case, of which there is one, and dependent design cases, of which there can be any number.

The primary design case can be any of the design cases that are being addressed in your model. Typically, it will represent the primary operating condition of the pipe system. If it is not clear which case should be the primary case, just pick any of the cases and call that the primary one. Design cases other than the primary case are referred to as dependent design cases. This naming convention is used since the dependent design cases are built and defined by duplicating the primary design case, as will be discussed later. The normal flow case will be used as the primary design case, so we will start setting up the sizing settings based on the requirements for normal operation.

A. Sizing Objective

Go to the Sizing window and the Sizing Objective panel should open by default. For this analysis, we are interested in sizing the system considering the monetary cost for the initial costs only.

1. Set the Sizing Option to **Perform Sizing**, if not already.
2. For the Objective, choose **Monetary Cost**, and make sure **Minimize** is selected from the drop-down list.
3. Under Options to Minimize Monetary Cost, choose **Size for Initial Cost**. This will update the cost table to include both initial costs in the sizing calculation while ignoring the recurring costs.

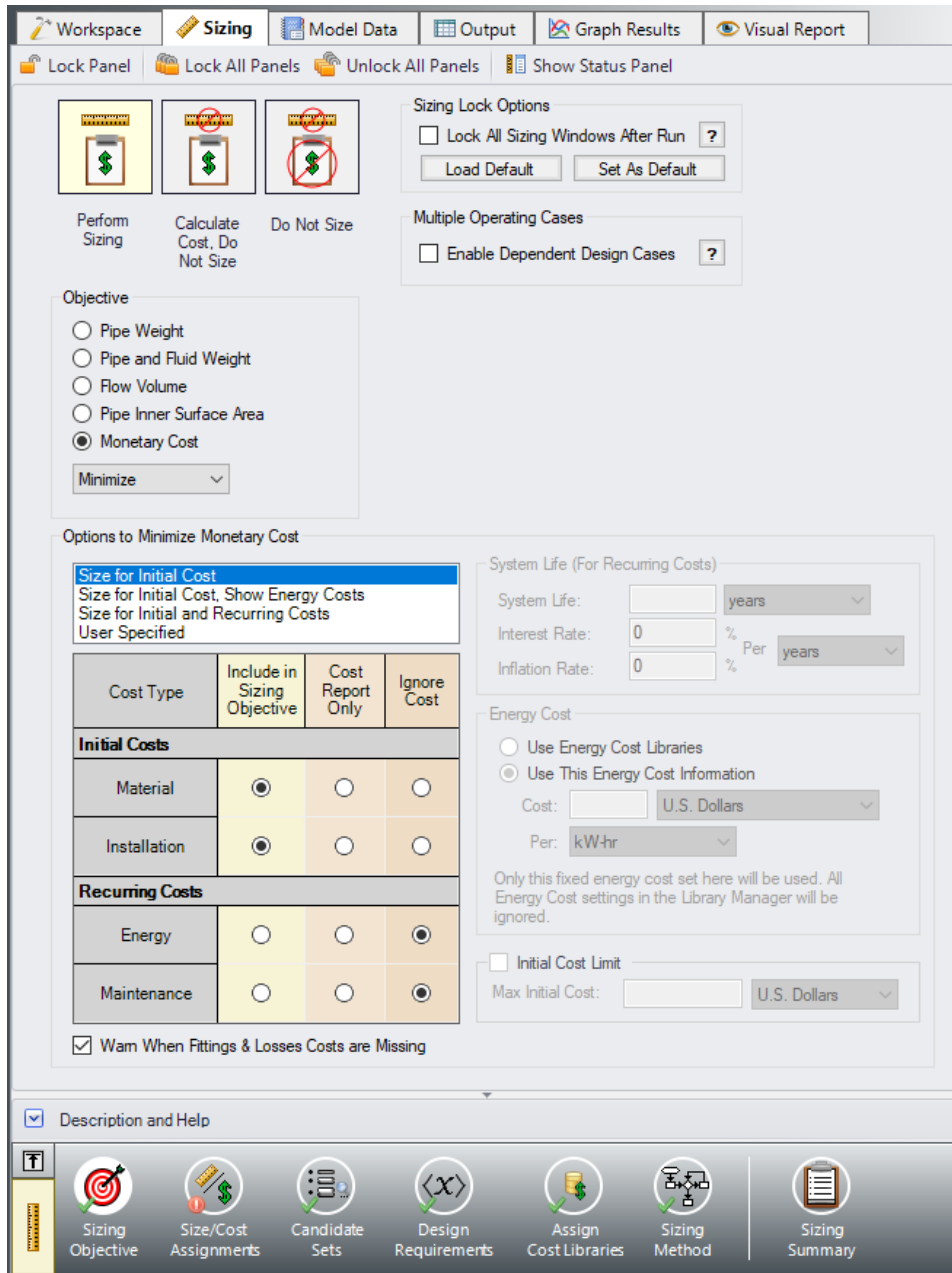The Sizing Objective window should now appear as shown in Figure 2.

Figure 2: Sizing Objective window setup for an Initial Cost analysis

B. Size/Cost Assignment

On the toolbar at the bottom of the window select the **Size/Cost Assignments**. For this scenario we will size the neighborhood mains, system supply line and hydrant supply lines assuming that they are all the same size. The supply lines to the houses must be 1" PVC so they will not be sized, but we will still calculate the costs, since we will need to purchase that piping as well.

1. Go to the Workspace, and select the lines that will be sized as shown in Figure 3.

2. Right click the Workspace and choose **Add Pipe(s) to Common Size Group**, and select **New Group**. Give the group the name *Neighborhood Mains*. Click OK on the Information window that appears.

3. Return to the Sizing window. The neighborhood mains, supply line, and the hydrant supply lines should now all be included in the new pipe group, and should be moved to Always Include in Cost as shown in Figure 4.

4. Move each of the house supply lines to **Do Not Size - Include in Cost**. All of the pipes can be moved at once by using the shift key to highlight all of the rows, then clicking the Include in Cost button.

We are not considering any of the junctions as part of the sizing, so the Size/Cost Assignments are complete for now.

Note: Though the hydrant supply lines are closed in the primary design case, we need to include them in the Common Size Group so that they will be sized when we create the dependent design cases for the open hydrant cases.
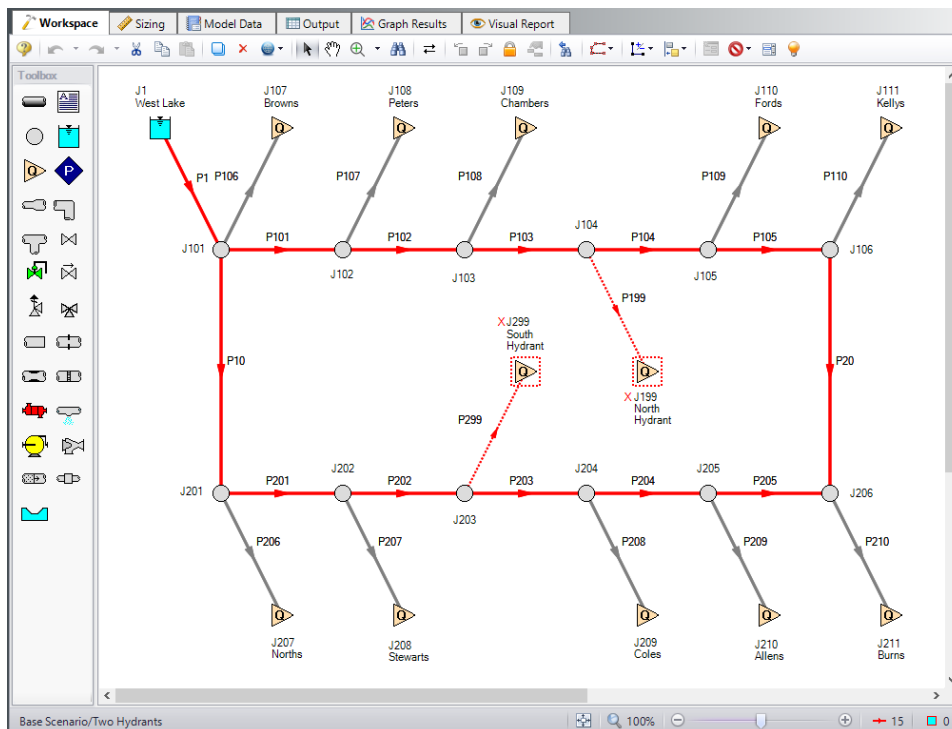

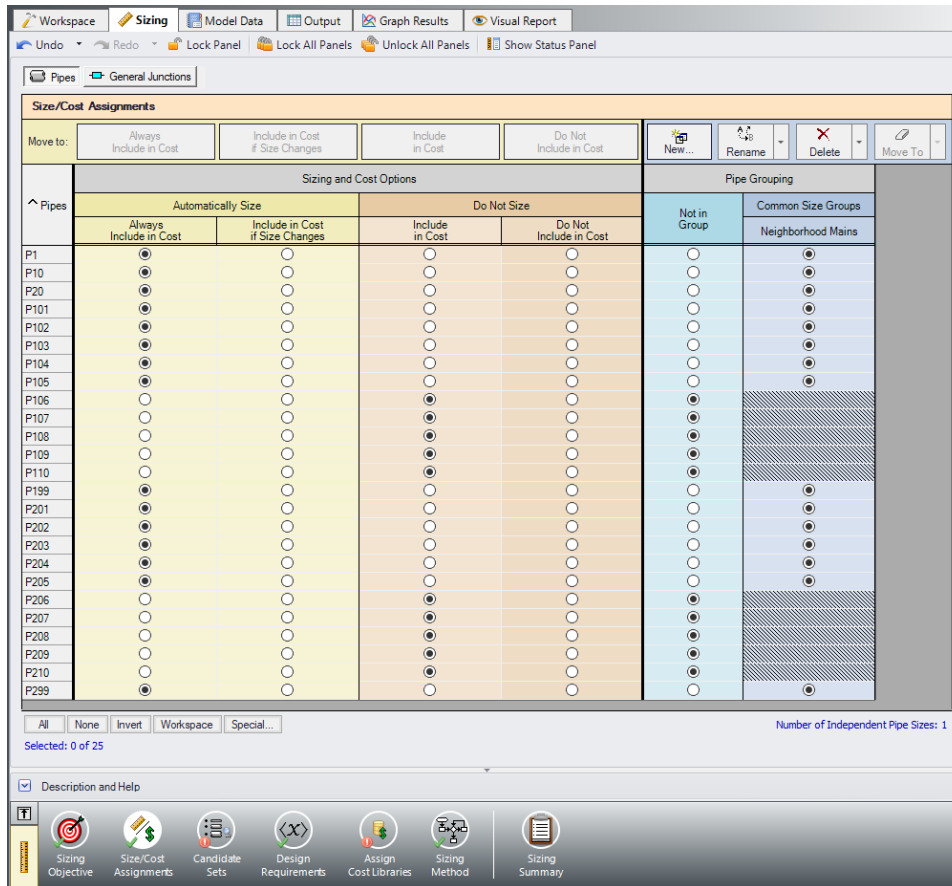*Figure 3: Workspace with pipes selected to add to the Common Size Group*

Figure 4: Size/Cost Assignments window with Common Size Groups defined as outlined above

C. Candidate Sets

Click on **Candidate Sets** to open the Candidate Sets panel. In this case we will be considering schedule 40 PVC pipe within the range of 1" to 10".

1. Select **New**, and name the candidate set *PVC sch 40*.
2. Choose **PVC - ASTM** from the material list.
3. At the bottom of the window select to sort by **Type, Schedule, Class** (if not already selected).
4. Expand the schedule 40 type, then double click each of the sizes **from 1 inch to 10 inch** to add them to the Candidate Set.
5. Click **OK** to accept the defined set.

In the bottom section of the window make sure that the Common Size Group is assigned to use the new PVC sch 40 Candidate Set. The window should appear as shown in Figure 5.
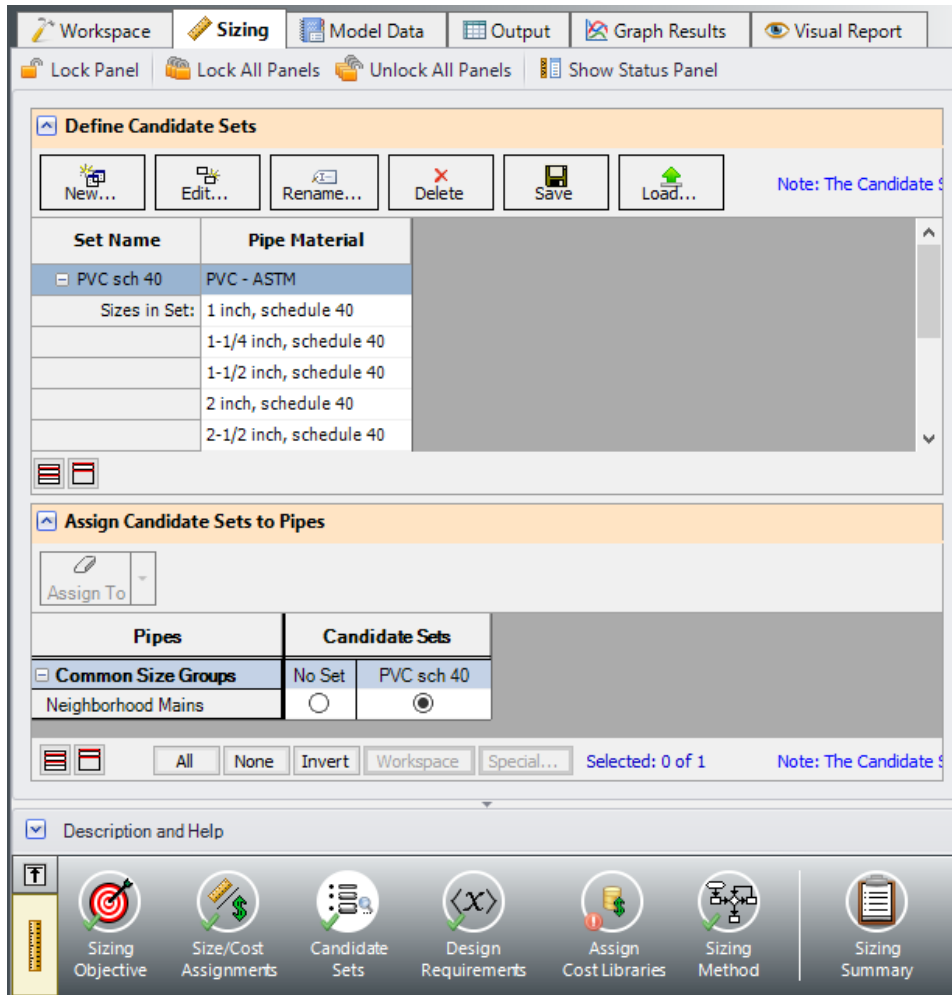
Figure 5: Candidate Sets window fully defined for the model

D. Define Design Requirements

Select **Design Requirements**.

For the standard operation scenario the only design requirement is for the minimum supply pressure of 60 psig to each of the houses. With the **Pipes** button selected click **New** to add a *House Supply* Design Requirement specifying the **Pressure Static** at the **Outlet** of the pipe to a **Minimum** value of **60 psig**.

Click **New** again to create a second Design Requirement named *Fire Supply* specifying the **Pressure Static** at the **Outlet** of the pipe to a **Minimum** value of **90 psig**. We will apply this requirement later when the dependent design cases are created.

In the bottom section of the window, make sure that the House Supply Design Requirement is applied to only the supply lines to the houses and not the Neighborhood Mains, as shown in Figure 6.
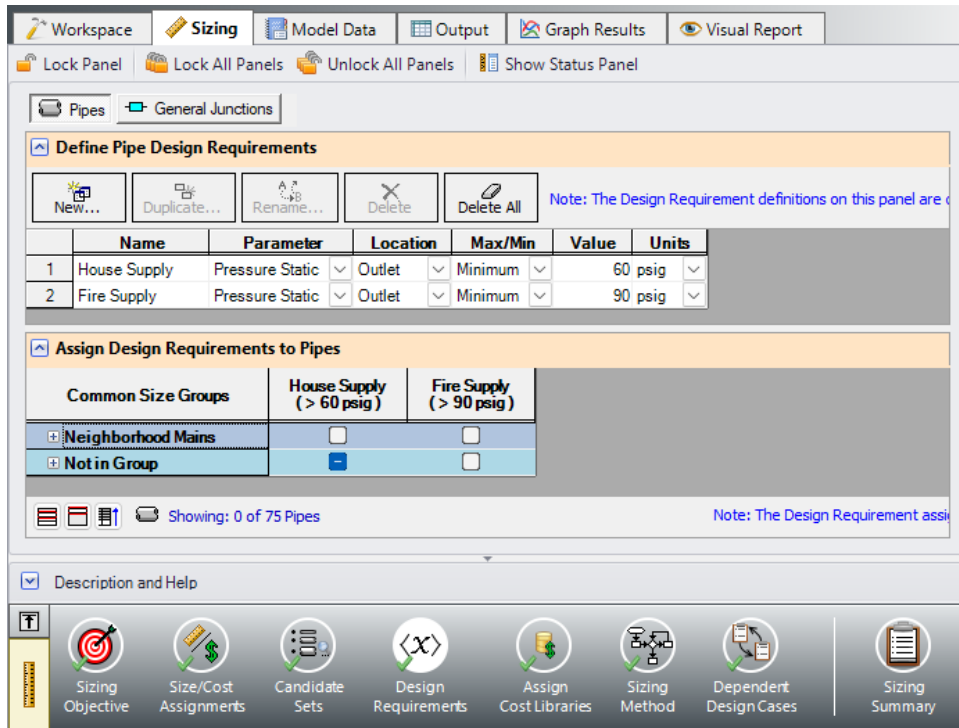
Figure 6: Design Requirements defined for the primary design case

E. Assign Cost Libraries

Select **Assign Cost Libraries**. For this model the pipe cost library has already been created, but we will need to connect and apply it.

➤**Open the Library Manager** by clicking the Library Manager button to see which libraries are connected.

This example uses a pre-built cost library for the pipes containing the PVC material costs. To connect the library do the following:

1. Click **Add Existing Cost Library**.
2. Browse to the Fathom 13 Examples folder (located by default in C:\AFT Products\AFT Fathom 13\Examples\), and open the file titled ***Housing Project Costs.cst***.

The new library should now be visible in the list and be connected to the model, indicated with a box checked next to the name, as shown in Figure 7. The cost library for the pipes should appear with the name ***Housing Project Costs.cst*** and be indented under the PVC - ASTM library within the standard pipe material library list.

Once you have confirmed that the library is connected, click Close to exit the Library Manager.
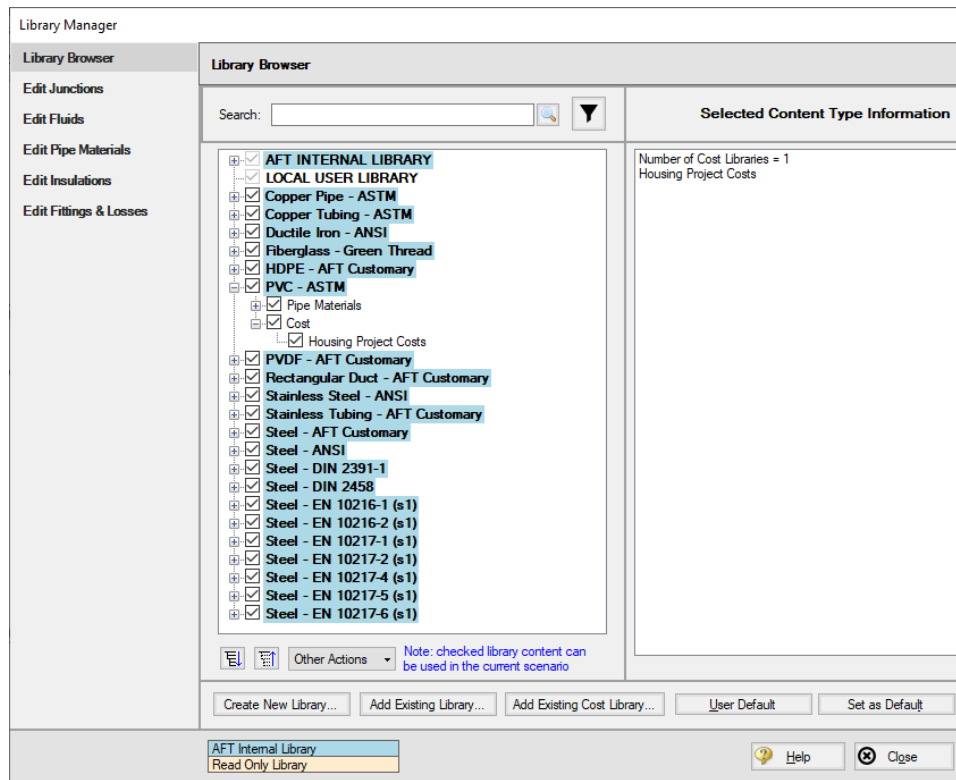
Figure 7: Library Manager with Housing Project Costs library connected

Back in the Assign Cost Libraries panel for the Pipes, the Housing Project Costs library should be the only available cost library, and should be selected to show that it is being applied to the candidate set and to each of the house supply lines.

F. Sizing Method

Select **Sizing Method** to go to the Sizing Method panel.

Ensure that **Discrete Sizing** is chosen with the **Modified Method of Feasible Directions (MMFD)** for the Search Method.

G. Dependent Design Cases

We have now set up the system to be sized for the primary design case, but we will also need to account for the cases where either the north or south hydrant are in use. There are a few changes that occur for each of these cases where a hydrant is in use:

1. The flow rate at each of the houses decreases to 2 gal/min
2. The minimum inlet pressure for each of the hydrants must be 90 psig
3. The minimum pressure for each of the houses is removed

We will create two dependent design cases to account for the scenarios where the North Hydrant or the South Hydrant is open. To make the dependent design cases complete the following:

1. Go to the **Sizing Objective** panel and click the box next to **Enable Dependent Design Case**. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation panel.

2. Navigate to the Dependent Design Case panel. You should now see instructions displayed to create Dependent Design Cases, along with a summary table of dependent design settings. We will now need to use the Duplicate Special feature to create the dependent design cases.

3. Go to the **Workspace** and go to the Edit menu and choose **Select All**.

4. Go to the Edit menu again and select **Duplicate Special**, enter an increment of **400** and click the boxes next to **Make Dependent Design Case** and **Clear Design Requirements** (Figure 8). Click **OK**.

5. Move the duplicated pipes and junctions below the original ones on the Workspace to distinguish them from the Primary Design Case. Click **Paste**.

6. Select only the primary design case model and repeat steps 4 and 5 to create a second dependent design case (increment the pipe numbers by **1000** to avoid conflicting pipe numbers). There should now be two copies of the original model in the Workspace which will be used to represent the alternate operating cases.

7. Open the **J599** and **J1299** hydrants by selecting them and toggling the **Special Condition to None** so that the North Hydrant is open in the second case and the South Hydrant is open in the third case.

8. Use **Global Junction Edit** to change all 20 of the Assigned Flow junctions at the Dependent Design Case <u>homes</u> (junctions 500's, 600's, 1100's, and 1200's) to an Outflow of **2 gal/min** (see Figure 9). Make sure to not change the Assigned Flow junctions that represent the hydrants.

9. In the Sizing window navigate to the **Design Requirements** panel. Assign the **Fire Supply** Design Requirement created earlier to pipe **P599** and **P1299** by checking the adjacent boxes.

Return to the **Dependent Design Cases** panel.

When Duplicate Special was performed with Dependent Design Case (DDC) selected, each of the duplicated pipes was created with a special type of grouping. For example, pipes 401 and 1001 were grouped with pipe 1 as DDC pipes (see Figure 10). This type of assignment allows the dependent design pipes to be sized, but to not be counted in the cost so that the cost will not be duplicated.
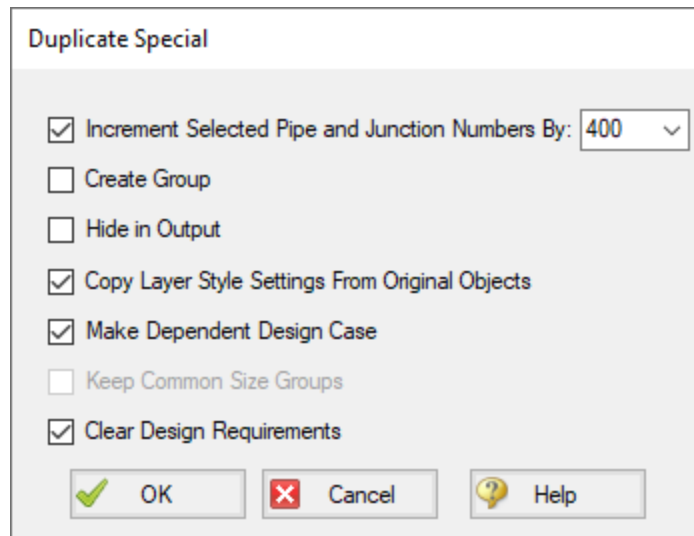
Figure 8: Duplicate Special settings to create a dependent design case
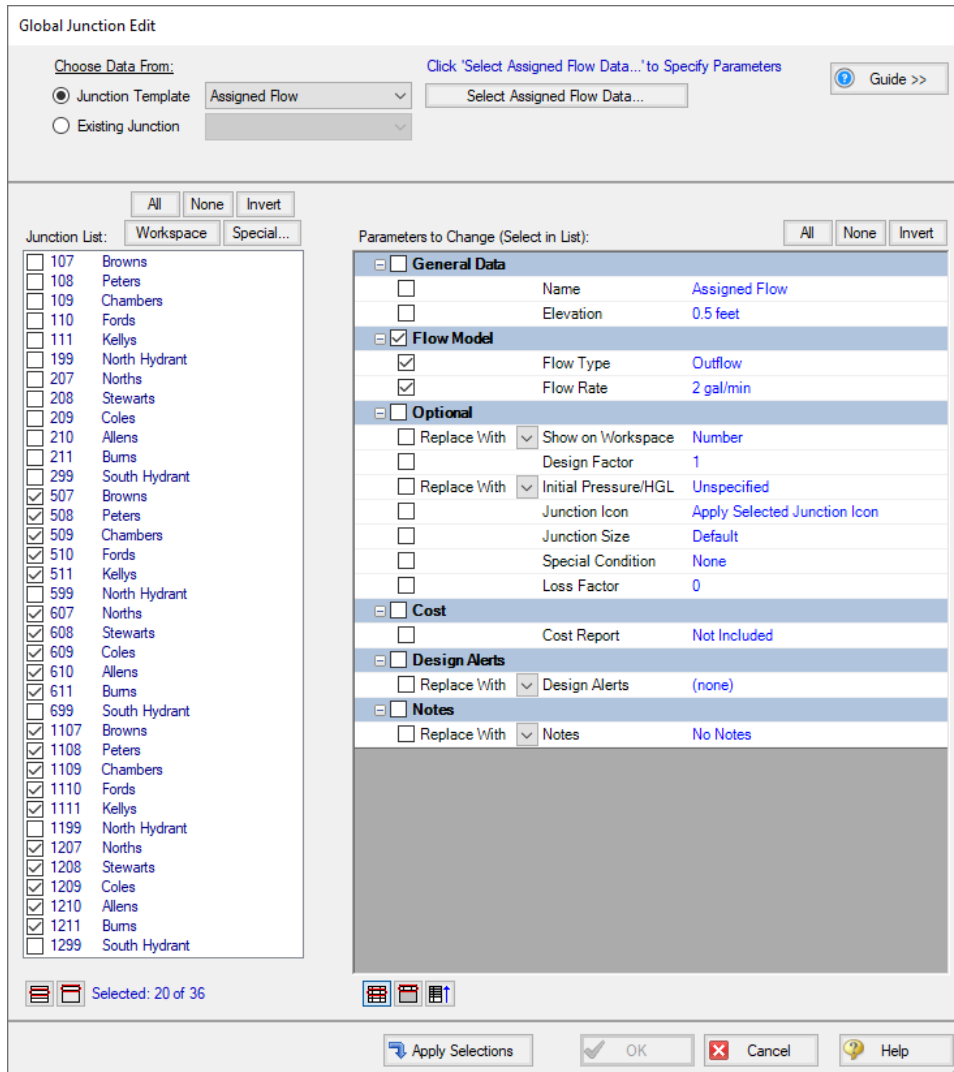
Figure 9: Global Junction Edit is used to change the flows in the homes for the dependent cases

Figure 10: Pipes in Dependent Design Cases have a special grouping relationship with pipes in the primary case

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Review the Sized Results

After the run finishes, examine the final pipe sizes calculated by the ANS module. The results for pipe size and overall cost are shown in Figure 11. One can see that the resultant size is 4-inch pipe. The lines to the houses are listed as N/A on the sizing output since they were not included in the sizing.

The cost for all sized pipes is $64,449. This is the cost for Items in Sizing. Since we did not size the pipes to the homes (because these were fixed at 1- inch), there is an additional cost of $7,288 for Items Not In Sizing. The pipes that were sized will be distinguished in the Cost Report by having a green background color. The total cost is merely the sum of the two: $71,737.

It should be noted that if we did not have monetary cost information for the pipes in this example, we could get nearly identical sizing results for this system by choosing a non-monetary objective. A monetary cost objective is primarily necessary in order to size equipment like pumps, control valves, etc. with the pipes.

| General | Warnings | Design Alerts | Cost Report | Reservoir Summary |

Table Units: U.S. Dollars	Material	Installation	TOTAL
TOTAL OF ALL MODEL COSTS			**71,737**
Total of All Shown Costs	24,776	46,961	71,737
Items In Sizing	22,869	41,580	64,449
Items Not In Sizing	1,907	5,381	7,288
Pipe Subtotal	24,776	46,961	71,737
o P1	9,900	18,000	27,900
o P10	1,485	2,700	4,185

| Pipes | Sizing | Pipe Design Requirements | Library Sources |

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
10	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
20	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
101	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
102	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
103	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
104	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
105	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
106	N/A	N/A	N/A	N/A	N/A
107	N/A	N/A	N/A	N/A	N/A
108	N/A	N/A	N/A	N/A	N/A
109	N/A	N/A	N/A	N/A	N/A
110	N/A	N/A	N/A	N/A	N/A
199	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
201	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
202	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
203	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
204	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
205	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
206	N/A	N/A	N/A	N/A	N/A
207	N/A	N/A	N/A	N/A	N/A
208	N/A	N/A	N/A	N/A	N/A
209	N/A	N/A	N/A	N/A	N/A
210	N/A	N/A	N/A	N/A	N/A
299	Pipe	PVC - ASTM	4 inch	schedule 40	4.026

Figure 11: Cost summary and final pipe sizes

Step 7. Increase the Independent Pipe Sizes

To increase savings in the system we can remove some of the pipes from the Common Size Group, which will allow them to be varied separately and increase the number of Independent Pipe Sizes for the model.

Let's create a child scenario to see how much savings can be found.

In the Scenario Manager, right-click the Two Hydrants scenario, click **Create Child**, and name it *3 Pipe Sizes*. Navigate to the **Size/Cost Assignments** panel in the Sizing window. Unlock the panel if it has been locked and move pipe **P1** to **Not in Group**. Lastly, create a new Common Size Group by clicking **New**, naming it *Hydrant Pipes*, and moving pipes **P199** and **P299** to the new group.

Run the scenario and go to the Output to view the results.

The Cost Report and pipe size results for the 3 Pipe Sizes scenario shown in Figure 12. The total cost is the cost per unit length for material and installation multiplied by the length of each section. The cost data does not include excavation costs. Even with these rough costs, significant savings can be achieved. If the developer is willing to use three pipe sizes, the savings will be approximately $2,500 or 3.5% of the total cost.

With this example, we can see that the limitations placed on the system by Common Size Groups can impact the sizing results.

In theory, to obtain the best possible sizing results, one can ungroup all the pipes in the neighborhood and allow each pipe to be sized independently. However, in practice this can result in increased fittings, installation, and maintenance costs, which is why common size groups are desirable.

	General	Warnings	Design Alerts	Cost Report	Reservoir Summary		

	Table Units: U.S. Dollars	Material	Installation	TOTAL
	TOTAL OF ALL MODEL COSTS			69,252
	Total of All Shown Costs	23,206	46,046	69,252
	Items In Sizing	21,300	40,665	61,965
	Items Not In Sizing	1,907	5,381	7,288
	Pipe Subtotal	23,206	46,046	69,252
o	P1	9,900	18,000	27,900

	Pipes	Sizing	Pipe Design Requirements	Library Sources	

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	PVC - ASTM	4 inch	schedule 40	4.026
10	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
20	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
101	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
102	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
103	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
104	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
105	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
106	N/A	N/A	N/A	N/A	N/A
107	N/A	N/A	N/A	N/A	N/A
108	N/A	N/A	N/A	N/A	N/A
109	N/A	N/A	N/A	N/A	N/A
110	N/A	N/A	N/A	N/A	N/A
199	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.548
201	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
202	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
203	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
204	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
205	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.549
206	N/A	N/A	N/A	N/A	N/A
207	N/A	N/A	N/A	N/A	N/A
208	N/A	N/A	N/A	N/A	N/A
209	N/A	N/A	N/A	N/A	N/A
210	N/A	N/A	N/A	N/A	N/A
299	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	3.548

Figure 12: Cost Report and pipe sizes for the system with three independently varied pipe sizes

Combined Modules Examples

The examples below utilize multiple AFT Fathom modules.

Combined Module Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Fixed Head Supply Tank	Advanced	Water	3	1	Learn how to use the GSC module, XTS module, and cost calculations together to model system performance over time.

Fixed Head Supply Tank - GSC XTS

Summary

Previous examples have illustrated how to use each of the AFT Fathom modules by themselves. However, AFT Fathom also allows you to use the modules together, in any combination you choose. This allows even greater flexibility in the types of analyses that can be accomplished using AFT Fathom.

This example shows how the GSC and XTS modules and cost calculations can be performed simultaneously by examining a fixed head supply tank system.

A process plant requires the delivery of water at a fixed head, regardless of the system demand. This is accomplished by pumping the water to a supply reservoir, and maintaining a constant fill-level by varying the supply pump speed as the process demands change. Throughout any 24-hour period the process demands vary from as little as 500 gal/min to peak demands of 2,500 gal/min.

Use the GSC and XTS modules to analyze the system performance over a 24-hour period. Perform cost calculations to determine the energy costs for the pump with this daily operation over one year.

Note: This example can only be run if you have licenses for the GSC and XTS modules.

Topics Covered

- Using the GSC and XTS modules and performing cost calculations simultaneously

Required Knowledge

This example assumes the user has already worked through the Beginner - Three Reservoir example, or has a level of knowledge consistent with that topic. You can also watch the AFT Fathom Quick Start Video Tutorial Series on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition, it is assumed that the user has worked through the Beginner -Filling a Tank - XTS example, the Beginner - Heat Transfer in a Pipe - GSC example, and the Beginner - Cost Calculation example and is familiar with the basics of GSC, XTS module, and cost calculation analyses.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***US - Fixed Head Supply Tank - GSC XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate GSC and Activate XTS. Make sure Use is selected for GSC and Transient is selected for XTS. Two new groups will appear in Analysis Setup titled Goal Seek and Control and Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the two new options called Goal Seek & Control and Time Simulation. From here you can quickly toggle between Ignore (normal AFT Fathom) and Use (GSC mode) for GSC and between Steady Only (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 70 deg. F

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.
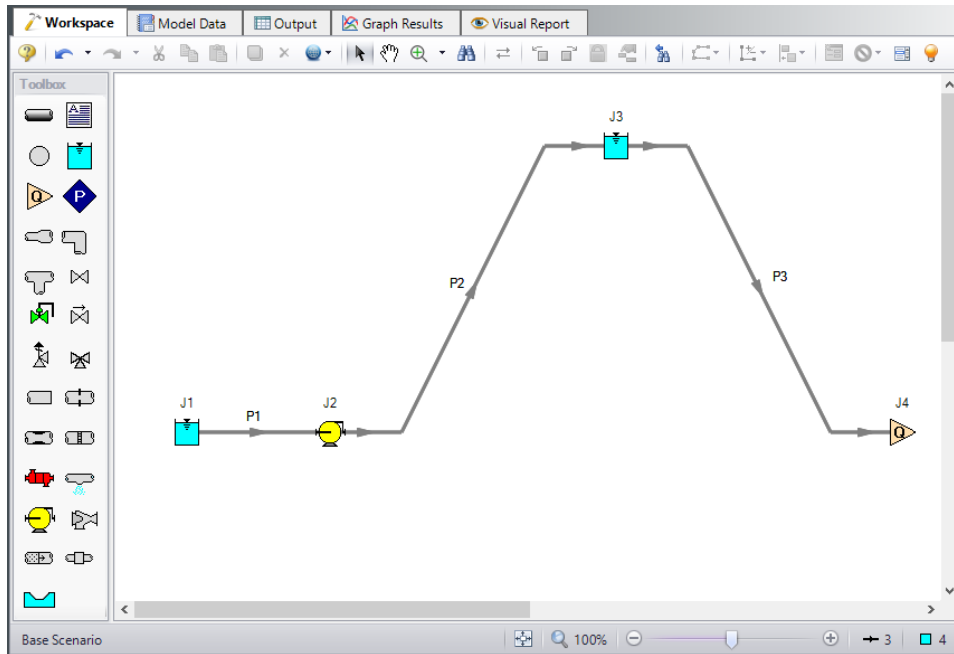
Figure 1: Fixed Head Supply Tank model layout

Pipe Properties

1. Pipe Model tab
   a. Pipe Material = Steel - ANSI
   b. Pipe Geometry = Cylindrical Pipe
   c. Size = 5 inch
   d. Type = STD (schedule 40)
   e. Friction Model Data Set = Standard
   f. Lengths =

Pipe	Length (feet)
1	30
2	500
3	100

Junction Properties

1. J1 Reservoir

    a. Tank Model = Infinite Reservoir
    b. Liquid Surface Elevation = 30 feet
    c. Liquid Surface Pressure = 0 psig
    d. Pipe Depth = 30 feet

2. J2 Pump

    a. Inlet Elevation = 0 feet
    b. Pump Model = Centrifugal (Rotodynamic)
    c. Analysis Type = Pump Curve
    d. Enter Curve Data =

Volumetric	Head	Efficiency
gal/min	feet	Percent
0	500	0
1250		60
2500	480	80
3750		70
5000	250	40

    e. Curve Fit Order = 2

3. J3 Reservoir

    a. Name = Supply Tank
    b. Tank Model = Finite Open Tank
    c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)
    d. Cross-Sectional Area = Constant, 314 feet2
    e. Liquid Level Specification = Height from Bottom
    f. Liquid Height = 15 feet
    g. Liquid Surface Pressure = 0 psig
    h. Tank Height = 20 feet
    i. Pipe Depth = 15 feet
    j. Tank Bottom Elevation = 20 feet

3. J4 Assigned Flow

    1. Elevation = 0 feet
    2. Type = Outflow
    3. Flow Specification = Volumetric Flow Rate
    4. Flow Rate = 500 gal/min

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. For this example, we will be adding one pump variable: pump speed.

Click **New Variable** and change the Junction Type to **Pump**. By default, J2 (Pump) and Variable Speed will be selected for Object Number and Name and Variable Parameter, respectively. The Variables panel should look now like Figure 2.
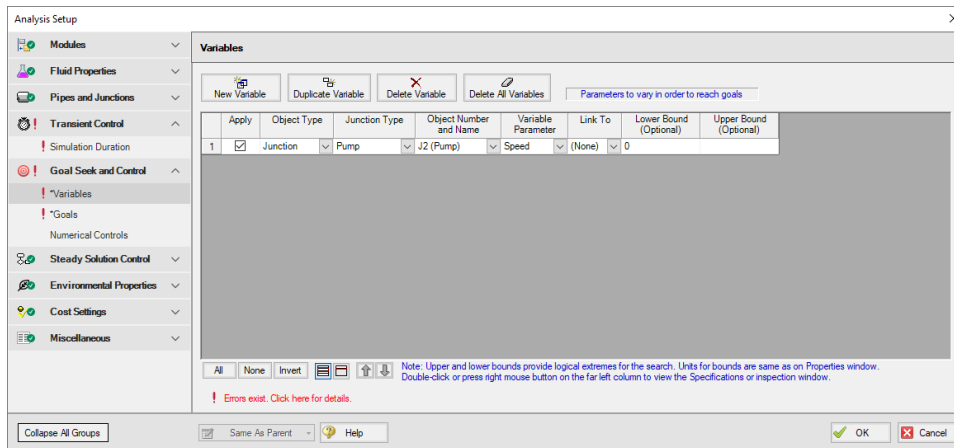
Figure 2: Defined Variables panel

Goals Panel

Navigate to the Goals panel. The desired goal is for the net volumetric flow rate into the junction to be zero, which will keep the water level in the tank constant.

Click **New Goal** and change the Object Type or **Reservoir**. By default, Vol. Flow Rate New Into Jct should be the selected Goal Parameter. Enter **0** as the Goal Value and select **J3 (Supply Tank)** for the Object ID. The Goals panel should now look like Figure 3.
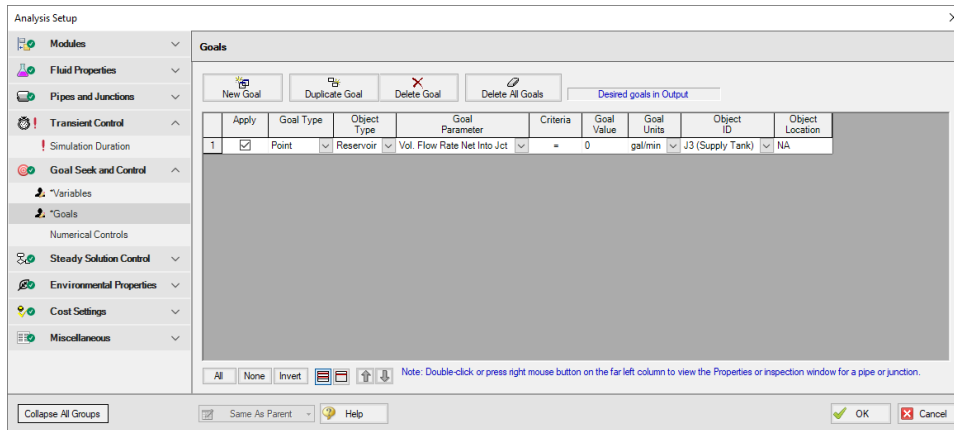
Figure 3: Defined Goals panel

Step 6. Define the Transient Control Group

The XTS module is used to model the process demand changes over the 24-hour analysis period. Navigate to the Simulation Duration panel in the Transient Control group to view the Transient Control settings. Change the Start Time units to **hours** and enter a Stop Time of **24**. For Time Step, changes the units to **minutes** and enter **10**. The Simulation Duration panel should now look like Figure 4. The transient analysis is now set to run for a period of 24-hours with a time step of 10 minutes. Click **OK** to save and close Analysis Setup.
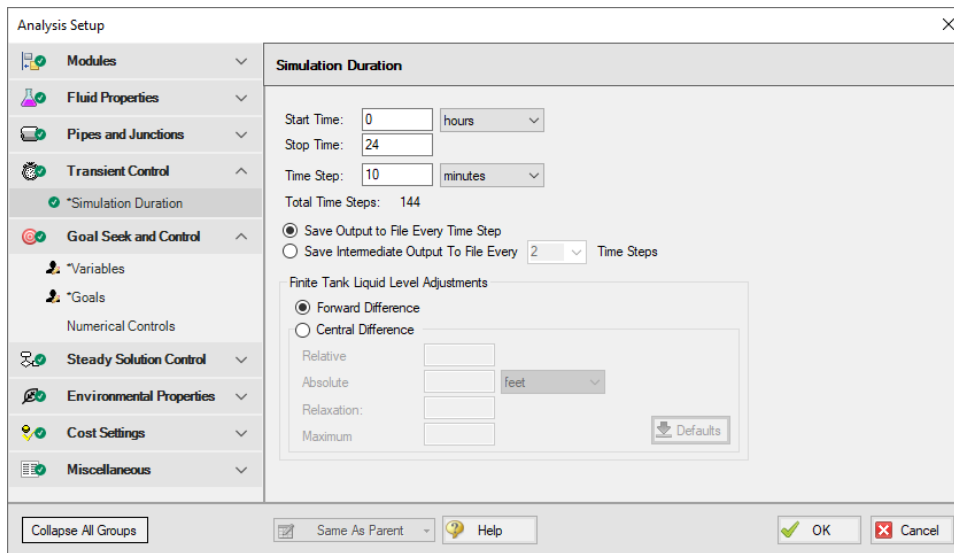
Figure 4: Transient Control settings for a 24-hour period

Step 7. Junction Transient Data

The flow variation in this model is driven by the J4 Assigned Flow junction. Open the J4 Assigned Flow Properties window and select the Transient tab. The process flow demands are modeled as time-varying flows. The flow profile over the 24-hour period is defined in the Transient Data table. Use Table 1 below to fill in the Transient Data table. Then, click Show Graph and make sure the Junction J4 Transient Data graph matches Figure 5.

Note that the time units are in seconds. Alternatively, you can change the preferred time units in User Options from seconds to hours to change the units of this table, then enter the values in this table in hours instead. However, to keep to the default units, this table is provided in time units of seconds.

Table 1: Transient Data table for J4 Assigned Flow

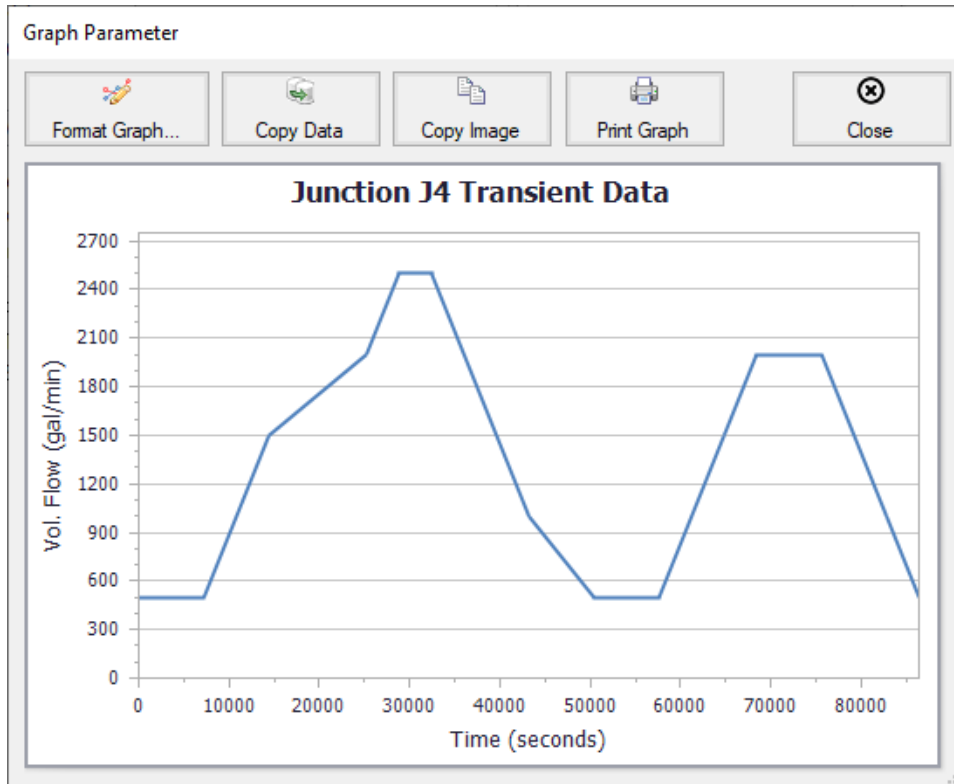| Data Point	Time (seconds)	Vol. Flow (gal/min)
1	0	500
2	7200	500
3	14400	1500
4	25200	2000
5	28800	2500
6	32400	2500
7	43200	1000
8	50400	500
9	57600	500
10	64800	1500
11	68400	2000
12	75600	2000
13	82800	1000
14	86400	500

Figure 5: Flow demand over 24 hours at the J4 junction

In addition, the J3 Reservoir junction is modeled as a finite reservoir (Figure 6). The J3 Reservoir can potentially drain and fill, and the varying pump speed (and, hence, flow) is what will keep its liquid level at 15 feet. No further input is required for the Supply Tank.
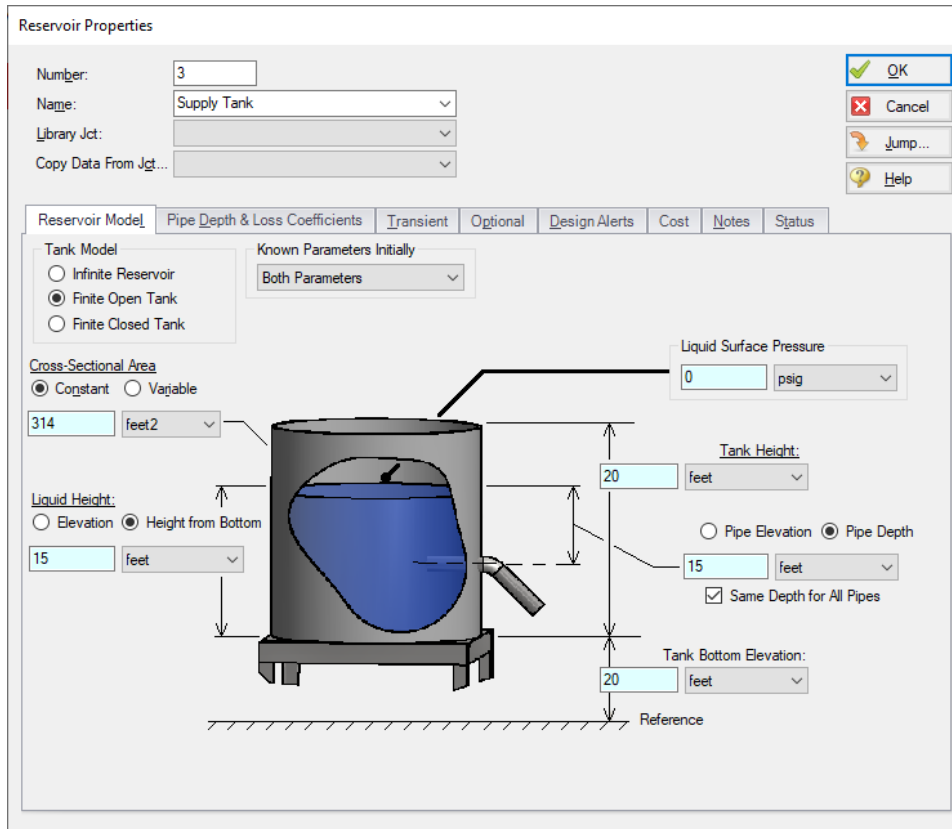
Figure 6: J3 Reservoir junction is finite, which means it can drain and fill

Step 8. Cost Settings

For this problem, the pump energy cost for the 24-hour analysis period is entered as a fixed energy cost on the Cost Settings panel. Open the Cost Settings panel from the Analysis Setup window to view the energy cost settings.

For Cost Calculation, select **Calculate**. Under Cost Definitions, check the box for **Operation/Energy** because that is the only cost of interest for this example.

Set the Cost Time Period System Life to **1 year**. The 24-hour operation is assumed to repeat every day for this one-year period, and the total energy cost will be calculated assuming the 24-hour operation repeats non-stop throughout the year. More complicated cost variation is possible when using energy cost libraries.

For Energy Cost, make sure **Use This Energy Cost Information** is selected and enter a Cost of **0.12 U.S. Dollars Per kW-hr**.

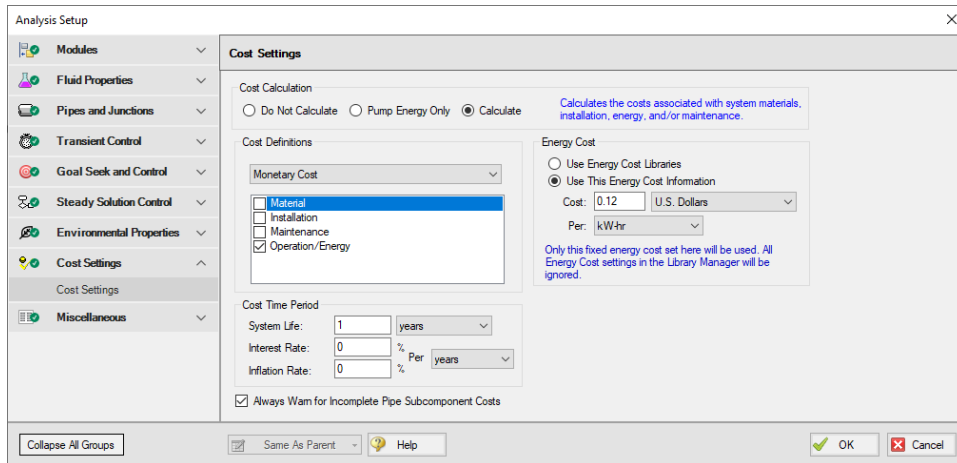The Cost Setting panel should now look like Figure 7 below.

Figure 7: Cost Settings window applies Operation/Energy cost, can set the energy cost, and specifies the cost time period

Return to the Workspace and open the properties window for **Pump J2**. Navigate to the **Cost** tab and under Cost Report, select **Include Cost in Report**. Click **OK**.

Step 9. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

> **Note:** When the GSC and XTS modules are used simultaneously, the GSC module is rerun for each time step. The GSC module variables are recalculated for the system conditions at each time step to ensure the specified goals are always met. This will typically result in longer run times, due to the increase in solution iterations that must be solved for each time step analysis.

Step 10. Examine the Output

When using multiple modules, the goal seek, transient and cost analysis results are shown in the Output window in the same manner as when they are used individually.

First open Output Control and go to the Format & Action tab. In the Time Simulation Formatting section, change the units to hours. Now all time units for XTS related output parameters will be shown using time units of hours.

The GSC variable and goal results in the General Output section can be viewed for each time step by using the slider bar at the bottom of the Output window, as shown in Figure 8.
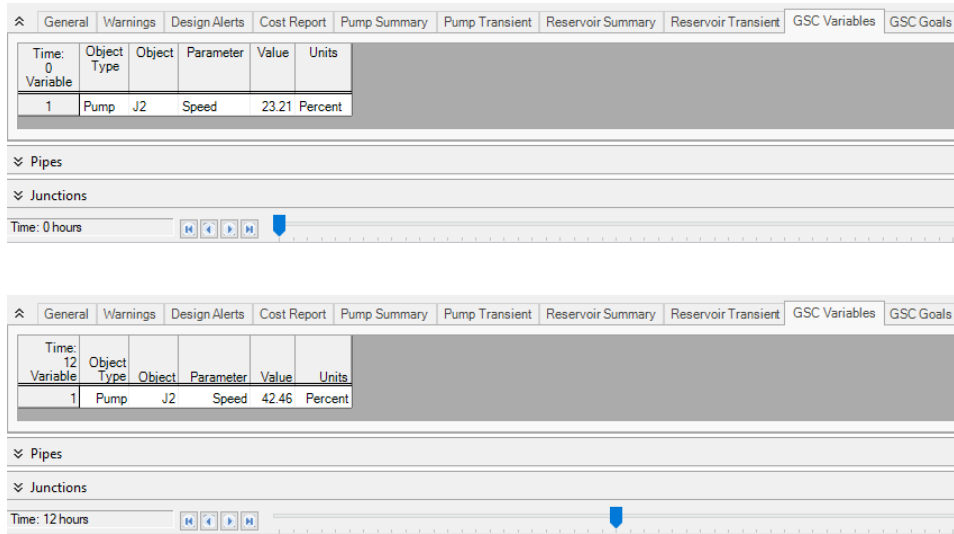
Figure 8: The slider bar at the bottom of the Output window can be used to view the transient results at any time step, including the GSC module variable and goal results. The GSC variable results for Time = 0 hours and Time = 12 hours are shown

Cost data is displayed in the Cost Report in the General Output section, as shown in Figure 9. The one-year energy cost for the supply pump in this example was $84,213.
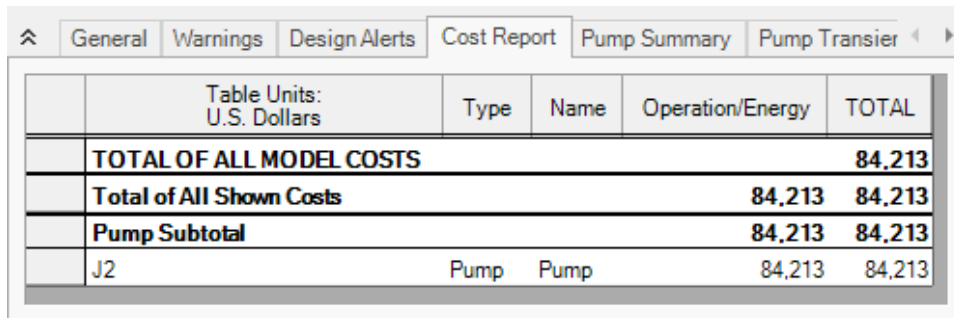

Figure 9: The cost data is shown in the Cost Report in the General Output section of the Output window

Figure 10 shows how the GSC module varied the pump's speed over time with the varying process flow rate.

Figure 11 shows that the water level in the process supply tank changes little over time because the net volumetric flow is held constant. The pump speed changes to increase and decrease the flow in order to keep the net flow into the Supply Tank zero. The minor changes in tank level are a result of the Forward Difference tank level calculation and can be reduced using the Central Difference method and a finer time step. These can be adjusted within the Transient Control section of Analysis Setup.
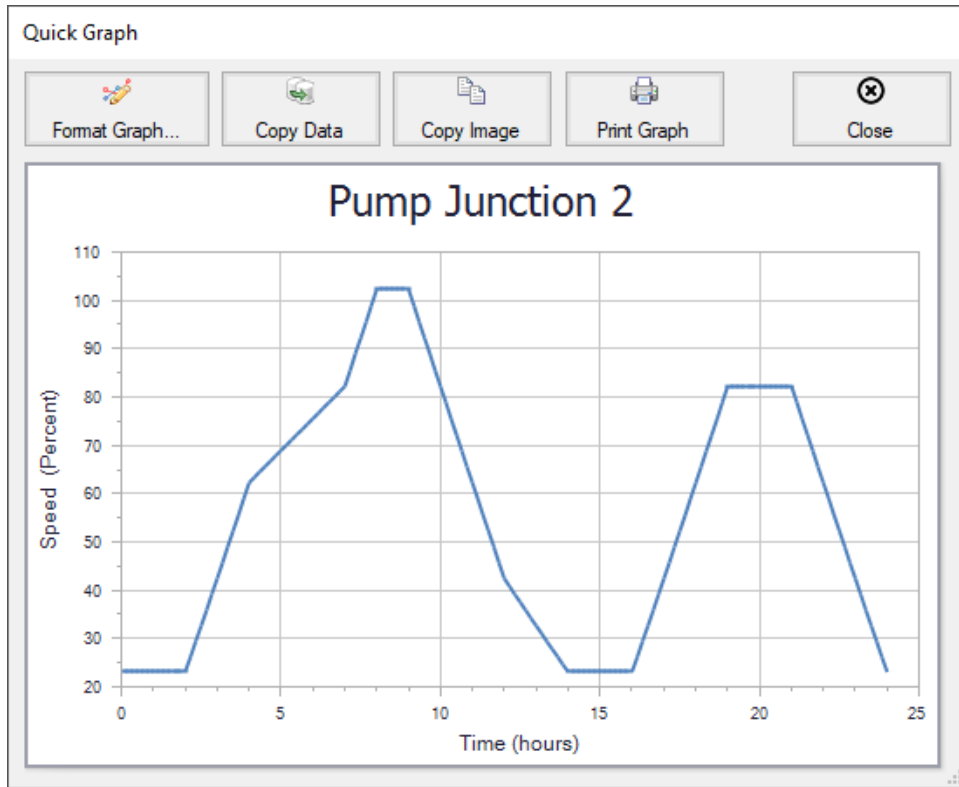
Figure 10: The GSC module varies the supply pump speed as the process flow rate changes over time (top)
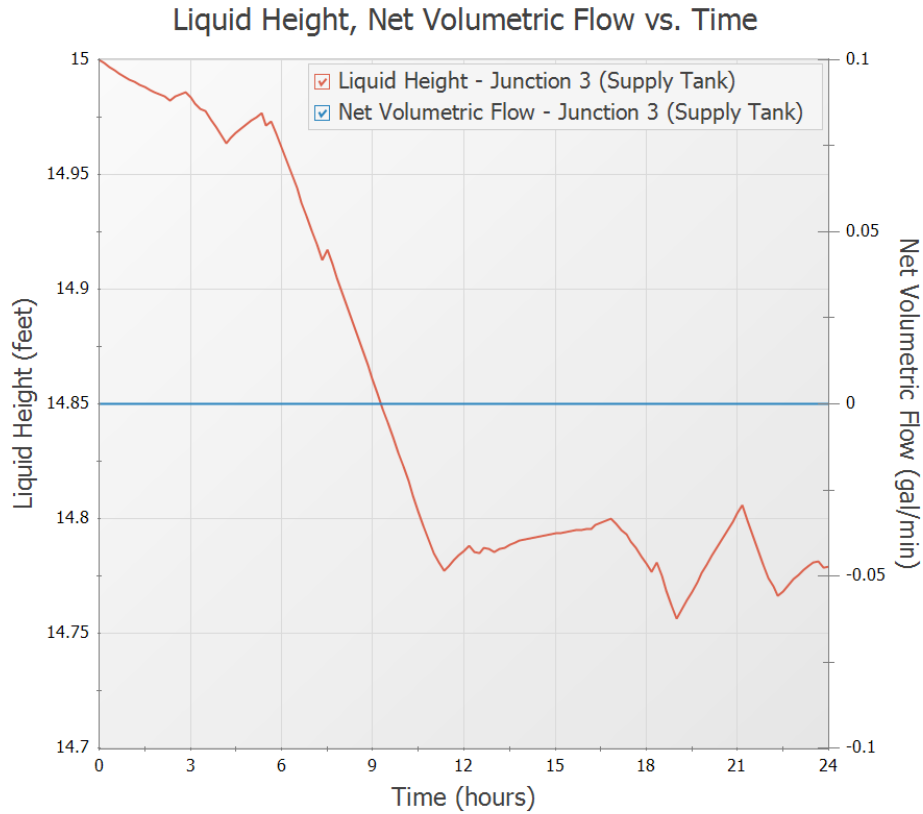
Figure 11: The water level in the process supply tank (left axis) remains relatively constant over time and the net volumetric flow rate (right axis) remains exactly at zero due to the variation in the pump speed

Summary

Individually, the AFT Fathom modules provide you with powerful analytical tools to assist you in the design of your pipe systems. The ability to combine the capabilities of these tools provides you with even greater and more powerful modeling capabilities.

Index

W

Workspace Layers