AFT Fathom™ 10

Quick Start Guide

English Units

Dynamic solutions for a fluid world ™

Applied Flow Technology

AFT Fathom™

Quick Start Guide

English Units

AFT Fathom Version 10

Incompressible Pipe Flow Modeling

Dynamic solutions for a fluid world ™

CAUTION!

AFT Fathom is a sophisticated pipe flow analysis program designed for qualified engineers with experience in pipe flow analysis and should not be used by untrained individuals. AFT Fathom is intended solely as an aide for pipe flow analysis engineers and not as a replacement for other design and analysis methods, including hand calculations and sound engineering judgment. All data generated by AFT Fathom should be independently verified with other engineering methods.

AFT Fathom is designed to be used only by persons who possess a level of knowledge consistent with that obtained in an undergraduate engineering course in the analysis of pipe system fluid mechanics and are familiar with standard industry practice in pipe flow analysis.

AFT Fathom is intended to be used only within the boundaries of its engineering assumptions. The user should consult the AFT Fathom Help system for a discussion of all engineering assumptions made by AFT Fathom.

Printed in the United States of America.

First Printing.

Contents

CHAPTER 1

Introducing AFT Fathom

Welcome to AFT Fathom™ 10, Applied Flow Technology's powerful pipe and duct flow modeling tool. With AFT Fathom you can model incompressible flow in complex pipe and duct networks, including coupled heat transfer in pipes and ducts, heat exchangers and pumps, and the resultant effect on fluid properties such as viscosity.

AFT Fathom 10 has four available add-on modules, which extend AFT Fathom's extensive modeling capabilities into new areas. The modules can be used individually or together.

- The Extended Time Simulation (XTS) module allows the engineer to model time varying system behavior.
- The Goal Seek & Control (GSC) module allows the engineer to perform multivariable goal seeking and simulate control system functions.
- The Settling Slurry (SSL) module allows the engineer to perform the complex property and system interaction calculations associated with settling slurry flows.
- The Automated Network Sizing (ANS) module automates the process of sizing pipes and ducts in a complex network based on required conditions, such as maximum pressure or minimum flow, to reduce cost or weight.

AFT Fathom is a proven tool which provides the engineer with an extremely broad and powerful feature set ready to tackle the most demanding analysis problems.

Modeling capabilities

AFT Fathom can be used to model a wide variety of engineering systems. AFT Fathom consists of the standard version plus four optional add-on modules.

Standard version capabilities

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Pressure fed systems and gravity fed systems
- Pumped systems, including multiple pumps in parallel or in series
- Pumps with variable speed, controlled pressure, controlled flow, viscosity corrections, and multiple stages with takeoff flow
- Systems with pressure and/or flow control valves
- Systems with valves closed and pumps turned off
- Heat transfer and system energy balance
- Systems with variable density and viscosity
- Multiple design cases in a single model file
- Non-Newtonian fluid behavior
- Cost calculation of pump energy usage

Add-on module capabilities (optional)

- Transient modeling (XTS)
- Pump starting and stopping over time (XTS)
- Valve opening and closing over time (XTS)
- Tank liquid level changes automatically calculated (XTS)
- Multi-variable goal seeking and control simulation (GSC)
- System performance for settling slurry calculations (SSL)
- Automated design of a piping network (ANS)

AFT Fathom provides hundreds of standard loss models for pipe system components and allows you to enter your own loss data. Variable loss models that depend on flow are supported.

AFT Fathom's powerful solution engine is based on standard techniques used for many years in industry. The Newton-Raphson method is used to solve the fundamental equations of pipe flow that govern mass and momentum balance. Solutions are obtained by iteration, and matrix methods optimized for speed are employed to obtain convergence.

Thermophysical property data

AFT Fathom derives physical properties from one of five sources. The first is the standard AFT Fathom set of incompressible fluids which contains data for about 70 common fluids (called AFT Standard).

The second is User Specified Fluid, for which data is provided by the user. The third is water (and steam) data from the ASME Steam tables.

The fourth is the NIST REFPROP database, first introduced in AFT Fathom 10. REFPROP is licensed from the National Institute of Standards and Technology and is included in AFT Fathom. REFPROP has a database of approximately 150 fluids and supports user specified fluid mixtures. AFT Fathom is restricted to non-reacting mixture calculations.

The fifth is the Chempak™ database, first introduced in AFT Fathom 4. Chempak is licensed from Madison Technical Software and is offered as an optional add-on to AFT Fathom. Chempak has a database of approximately 700 fluids and supports user specified fluid mixtures. AFT Fathom is restricted to non-reacting mixture calculations.

Engineering assumptions in AFT Fathom

AFT Fathom is based on the following fundamental fluid mechanics assumptions:

- Incompressible flow (single phase)
- One-dimensional flow
- No chemical reactions
- Steady-state conditions (the XTS module allows transient modeling)

AFT Fathom Primary Windows

The AFT Fathom window has five subordinate windows that work in an integrated fashion. Each is located on a separate, moveable tab. You work exclusively from one of these windows at all times. For this reason, they are referred to as *Primary Windows*.

Of the five Primary Windows, two are input windows, two are output windows, and one displays output and input information. Figure 1.1 shows the relationship between the Primary Windows.

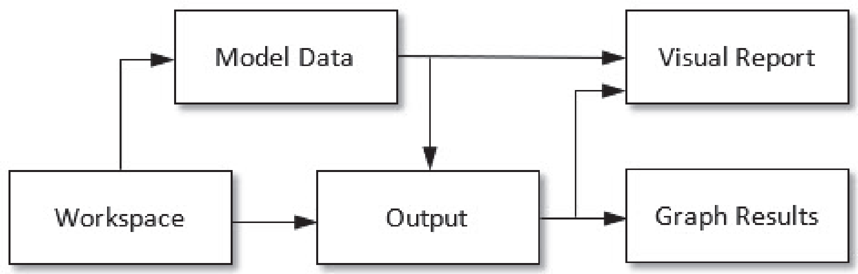


Figure 1.1 Primary Window workflow in AFT Fathom

Input windows

The two windows that function exclusively as input windows are the Workspace window and the Model Data window. These two windows, one graphical and the other text-based, work together to process model input data with immense flexibility. The tools provided in these two windows allow you to model a large variety of pipe networks.

The Visual Report window can function in support of both input and output data. As an input window, it allows you to see the input data superimposed on the pipe system schematic created on the Workspace.

Output windows

The two windows that function exclusively as output windows are the Output window and the Graph Results window. The Output window is text-based, while the Graph Results window is graphical. These two windows offer a powerful and diverse range of features for reviewing analysis results for modeling errors, gaining a deeper understanding of the pipe system's flow behavior, and preparing the results for documentation.

As an output window, Visual Report allows you to see the output results superimposed on the pipe system schematic created on the Workspace.

The five Primary Windows form a tightly integrated, highly efficient system for entering, processing, analyzing, and documenting incompressible flow analyses of pipe networks.

Note: AFT Fathom will support dual monitors. You can click and drag any of the five primary window tabs off of the main Fathom window. Once you drag one of the Primary Windows off of the 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.

CHAPTER 2

Sizing a Pump Example

This chapter is intended to give you the big picture of AFT Fathom's layout and structure. A number of other example model discussions are included in a Help file distributed with AFT Fathom called FathomExamples.chm. It can be opened from the Help menu by choosing “Show Examples”.

This example demonstrates a sample calculation to size a pump for a specified system.

Topics covered

- Model building basics
- Entering Pipe and Junction data
- Specifying System Properties
- Specifying Output Control
- Creating Visual Reports
- Graphing results
- Sizing a pump and entering pump curve data

Required knowledge

No prior knowledge is required for this example.

Model file

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

- *Sizing a Pump.fth* - AFT Fathom model file

This example is provided in the model file under the English Units scenario. It is also provided in Metric units under the Metric Units scenario.

This example will require you to build the model from scratch to help familiarize yourself with the steps required to build a complete model in AFT Fathom. Therefore, use this example model file as a reference only.

Problem statement

For this problem, 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 Steel - ANSI, 4 inch, STD (schedule 40). Constant fluid properties are assumed.

The supply reservoir has a surface pressure of 0 psig, and the liquid surface is at an elevation of 10 feet. The discharge reservoir has a 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?

Step 1. Start AFT Fathom

➢ **To start AFT Fathom**, click Start on the Windows taskbar, choose All Programs, then AFT Products, and then AFT Fathom. (This refers to the standard menu items created by a setup. You may have chosen to specify a different menu item).

As AFT Fathom is started, the Fathom Startup Window appears, as shown in Figure 2.1. This window provides you with several options you can choose before you start building a model.

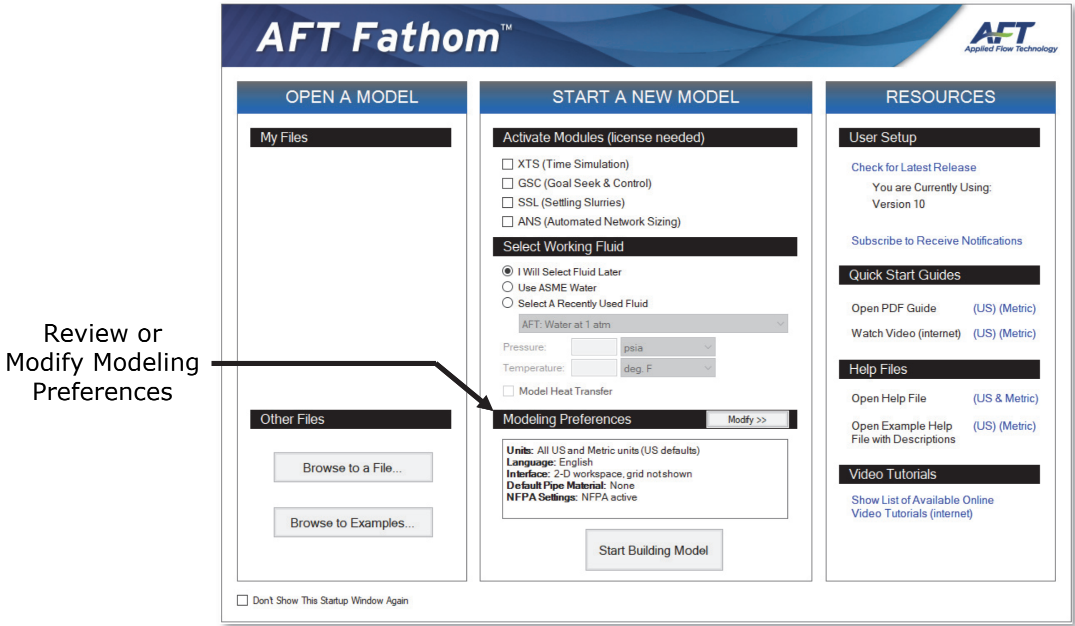


Figure 2.1 Startup Window with Modeling Preferences Collapsed

Some of the actions available are:

- Open a recent model, browse to a model, 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

If this is the first time that you have started AFT Fathom, Modeling Preferences will be expanded in the middle section of the Startup Window, as shown in Figure 2.2. If this is not the first time that you have started AFT Fathom, the Startup Window will appear with Modeling Preferences collapsed, as shown in Figure 2.1.

When collapsed, you can view your current Modeling Preferences at the bottom of Start a New Model. To further review or adjust your preferences, click the “Modify>>” button (see Figure 2.1).

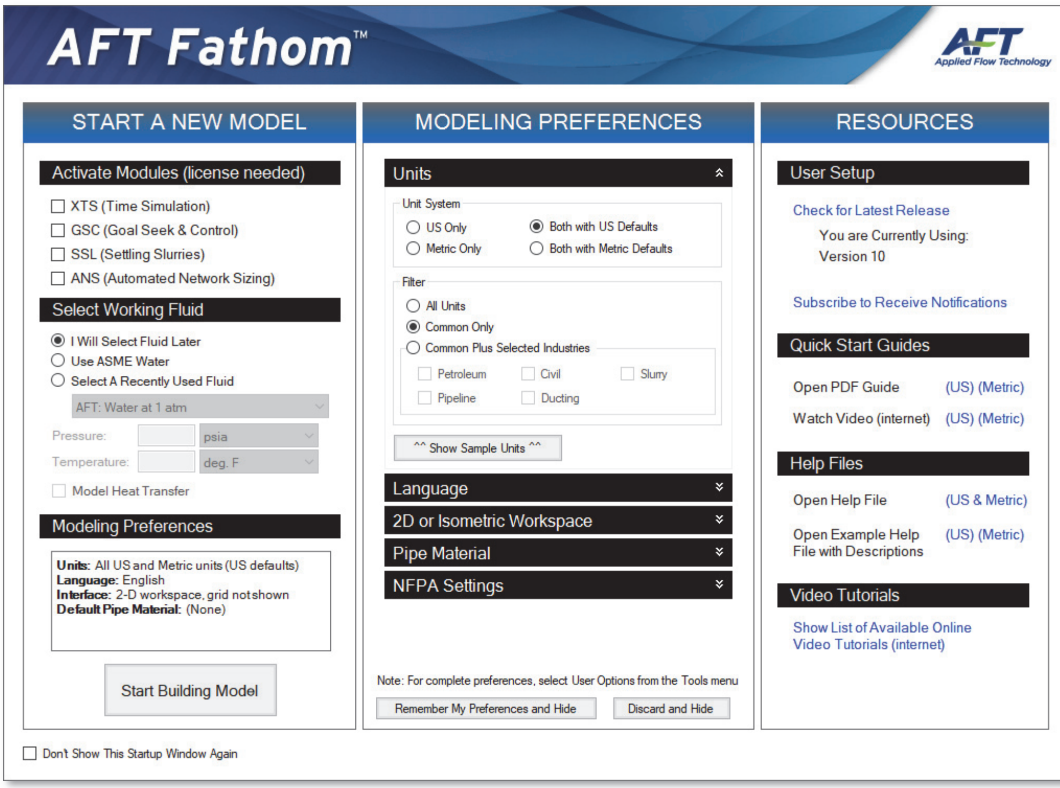


Figure 2.2 Startup Window with Modeling Preferences Expanded

With Modeling Preferences expanded, as in Figure 2.2, select “Both with US Defaults” under Unit System. Select “Common Only” under Filter to show only commonly used units, instead of “All Units.”

You can “Show Sample Units” to see which units will be included based on your selections, as shown in Figure 2.3.

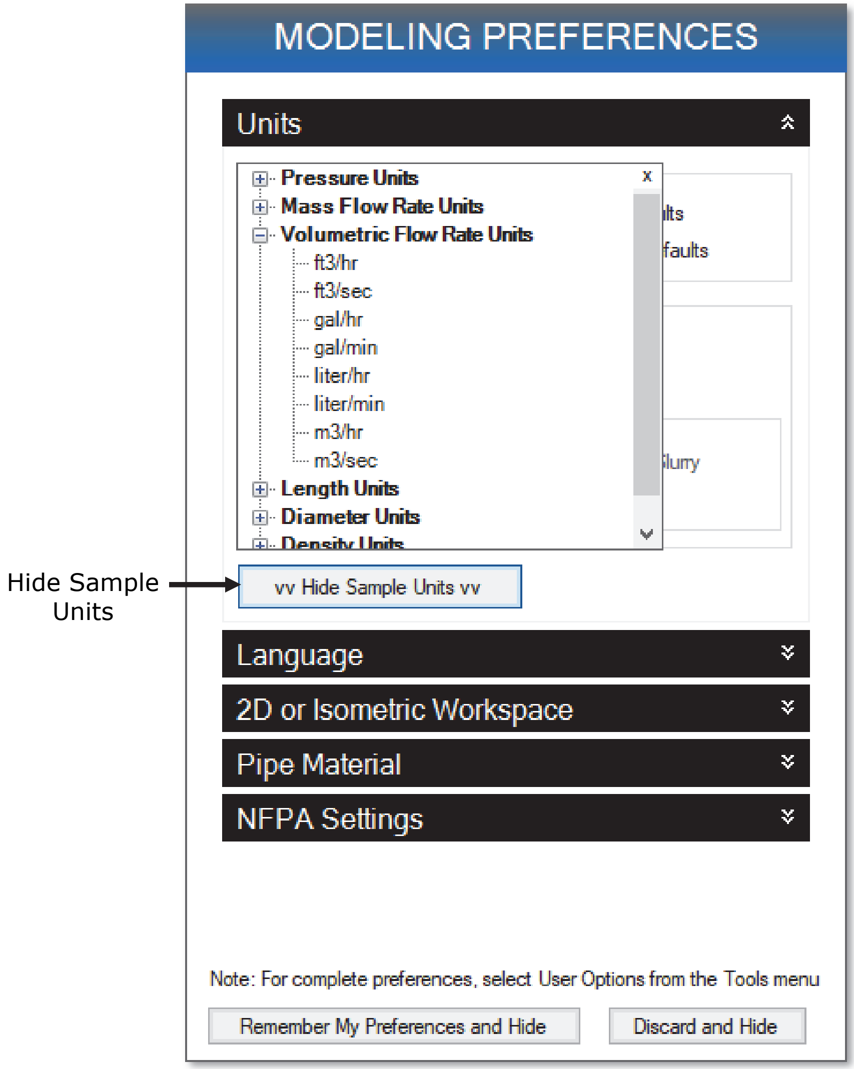


Figure 2.3 Show Sample Units with "Common Only" Selected

The other Filter option is "Common Plus Selected Industries," which will add units from the industries that you select. Once you have finished modifying your Modeling Preferences, click "Remember My Preferences and Hide." Now that your unit preferences are set, click "Start Building Model."

The Workspace window is the initial active (large) window, as seen in Figure 2.4. The five tabs in the AFT Fathom window represent the five primary windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

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 displayed on the far right. It gives easy access to a variety of features such as the Scenario Manager, the Checklist Panel, and viewing pipe and junction properties. The Workspace takes up the rest of the window.

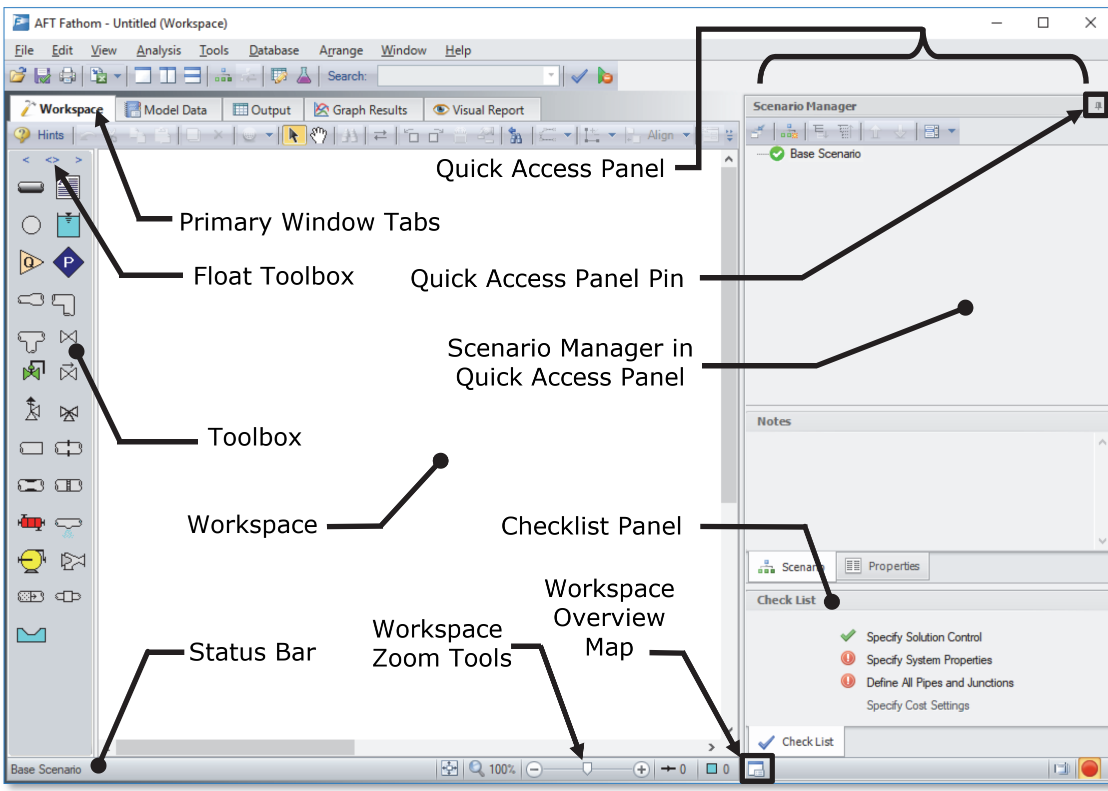


Figure 2.4 The Workspace window is where the model is built. The other four Primary Windows are found on the tabs along the top of the Workspace. The Status Bar shows the model status

You will build your pipe flow model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Float Toolbar. From here, you can choose the location of the Toolbox in reference to the Workspace by specifying Dock Left, Float, or Dock Right. Below the Float Toolbar are two drawing tools. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. Next to this tool is the Annotation tool. 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 components that connect pipes and 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 hold your mouse pointer over any of the Toolbox tools, a ToolTip identifies the tool's function.

Unpinning the Quick Access Panel

By default, the Quick Access Panel is pinned to the Workspace window so that it is constantly displayed. The Quick Access Panel can be unpinned so that it is only displayed when the mouse is moved over the tab displayed on the right edge of the Workspace window. Unpin the Quick Access Panel by clicking on the picture of the pin displayed in the top right corner of the Quick Access Panel (see Figure 2.4). For the remainder of the examples in this guide, the Quick Access Panel will be unpinned.

Step 2. Lay out the model

To lay out the pump sizing model, you will place three junctions on the Workspace. Then you will connect the junctions with pipes.

A. Place a reservoir

- **To start, drag a Reservoir junction from the Toolbox** and drop it on the Workspace. Figure 2.5a 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 have created on the Workspace will take on the default ID number of 1. You can change this to any desired integer greater than zero but less than 100,000.

Editing on the Workspace

Once on the Workspace, junction objects can be moved to new locations and edited using the features on the Edit menu. Cutting, copying, and pasting are all supported. A single level of undo is available for all editing operations.

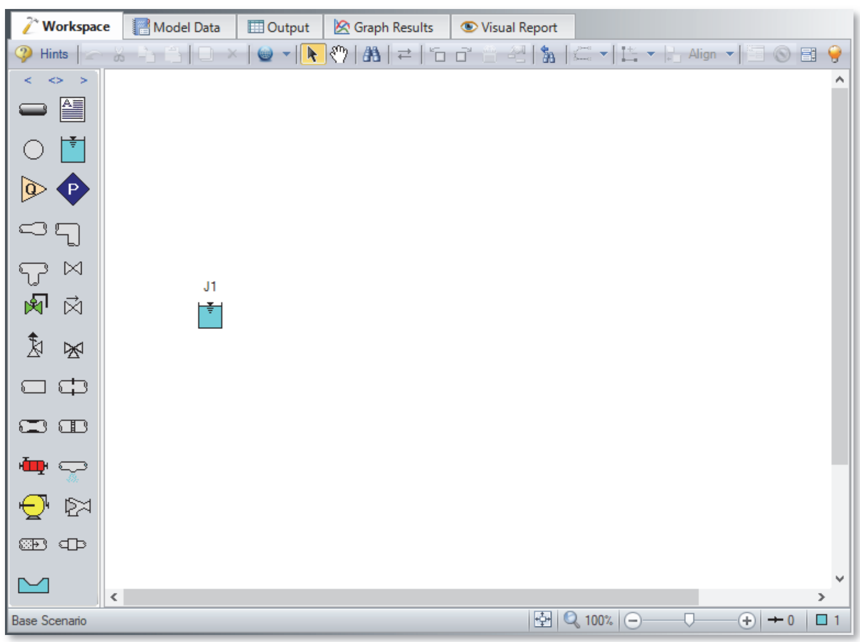


Figure 2.5a Walk through model with first junction placed

B. Place the other junctions

Drag a Pump junction from the Toolbox and drop it somewhere to the right of the Reservoir junction (see Figure 2.5b).

If a new junction type you want to add already exists on the Workspace, you have the option of duplicating that junction. You do this by choosing Duplicate from the Edit menu. Either duplicate the first reservoir or drag a new Reservoir junction onto the Workspace. Place the new reservoir somewhere to the right of the Pump junction (see Figure 2.5b).

Note: The relative location of objects in AFT Fathom is not important. Distances and heights are defined through dialog boxes. These relative object locations on the Workspace establish the connectivity of the objects, but have *no bearing on the actual length or elevation relationships*. The Isometric Pipe Drawing Mode (see Chapter 3) can be used to visually represent the three-dimensional nature of a system.

If the icons do not line up exactly, the calculations will not be affected. However, your model may have a nicer appearance if the icons line up. You can align the icons by using the align features selected from the Arrange menu.

Your model should now appear similar to that shown in Figure 2.5b.

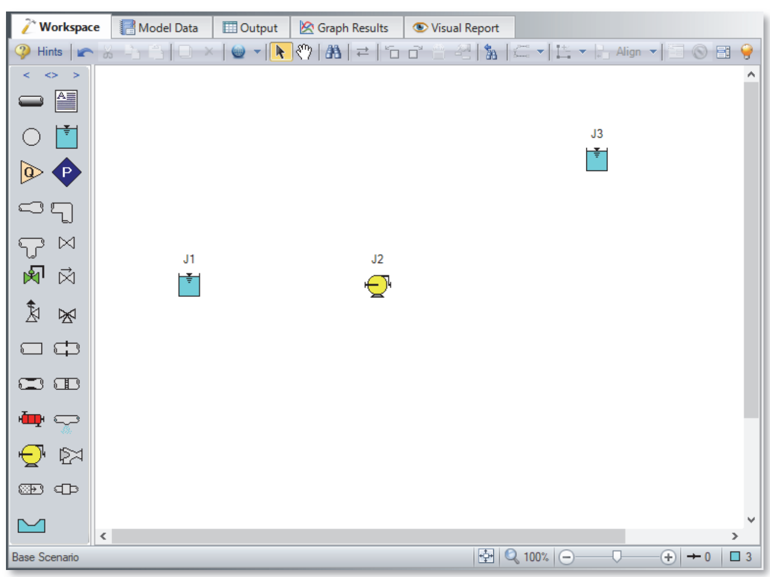


Figure 2.5b Walk through model with all junctions placed

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

C. Draw a pipe between J1 and J2

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

➢ **To create a pipe,** click the Pipe Drawing Tool icon on the Toolbox. The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe below the junctions, similar to that shown in Figure 2.5c.

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

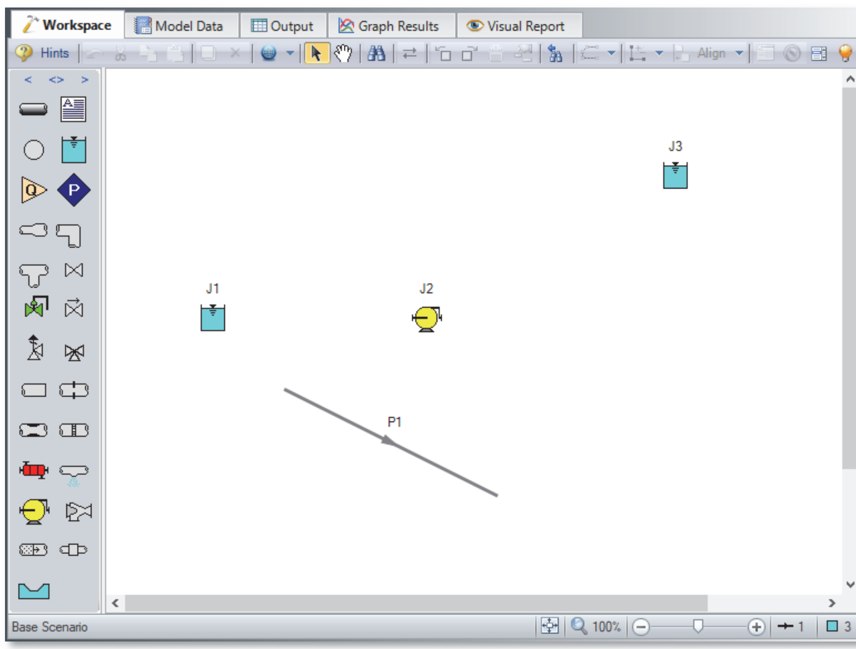


Figure 2.5c Walk through model with first pipe drawn

➢ **To place the pipe between J1 and J2,** 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 2.5d). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J2 Pump icon (see Figure 2.5e).

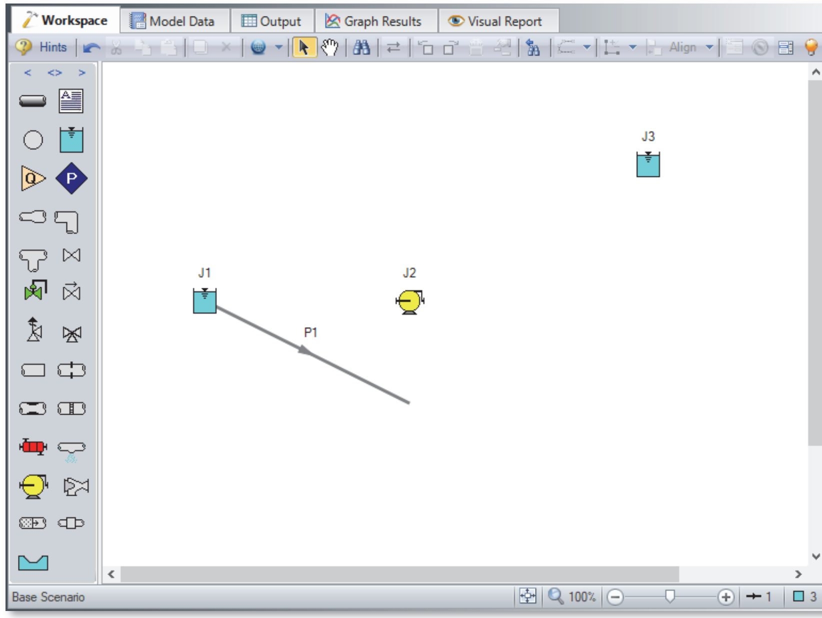


Figure 2.5d Walk through model with first pipe partially connected

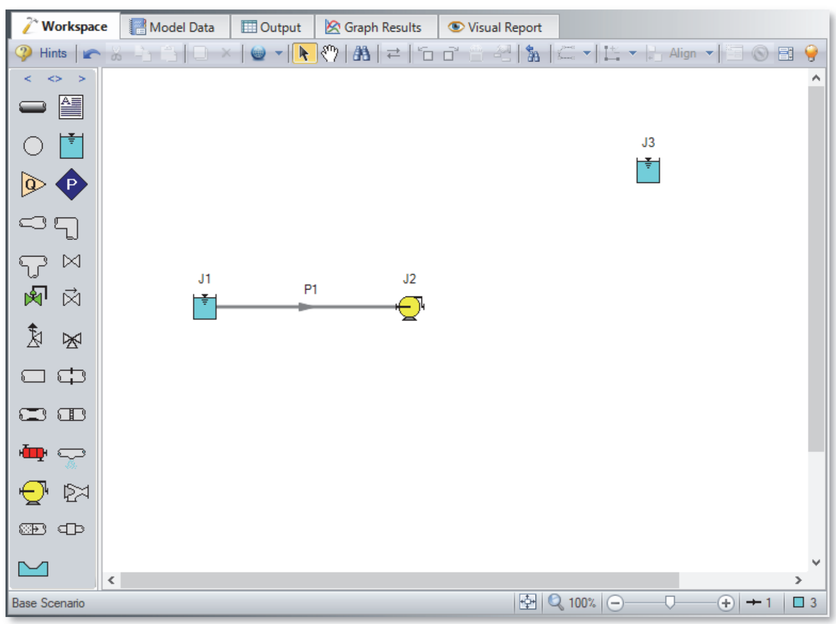


Figure 2.5e Walk through model with first pipe connected

Reference positive flow direction

There is an arrow on each pipe 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 Pipe Direction button on the Workspace Toolbar.

In general, the reference positive flow direction is used for reference purposes only and need not be the actual flow direction. 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 flow direction is the opposite of that obtained by the Solver, the output will show the flow rate as a *negative number*.

D. Add the remaining pipe

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

- **Activate the Pipe Drawing Tool again**, position the mouse pointer on the J2 Pump, then press and hold the left mouse button. Stretch the pipe across to the J3 Reservoir, and then release the mouse button. Your model should now look similar to Figure 2.5f.

At this point, all the objects in the model are graphically connected. Save the model by selecting Save in the File menu or by clicking on the diskette button on the Main Toolbar.

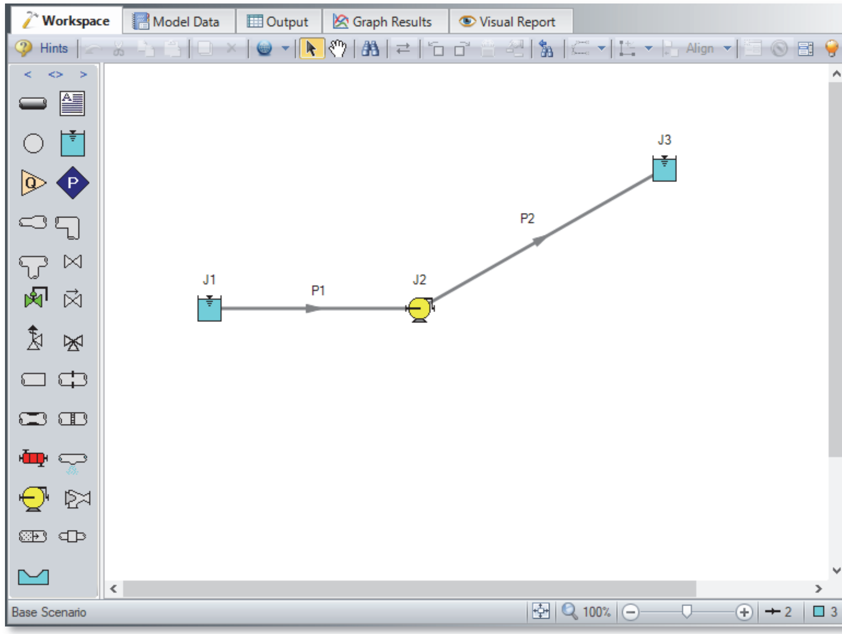


Figure 2.5f Walk through model with all pipes and junctions placed

---

Note: Some users find it desirable to lock objects to the Workspace once they have been placed. This prevents accidental movement and disruption of the connections. You can lock all the objects by choosing Select All from the Edit menu, then selecting Lock Object from the Arrange menu. The lock button on the Workspace Toolbar will appear depressed, indicating it is in an enabled state, and will remain so as long as any selected object is locked. Alternatively, you can use the grid feature and snap to grid. The grid options can be modified through the User Options window.

Step 3. Complete the checklist requirements

➢ **Next, click the checkmark on the Main Toolbar** at the top of the AFT Fathom window. This opens the Checklist window (see Figure 2.6). The Checklist window (also simply called the Checklist) contains four or more items depending on which modules you may be using. Each item needs to be completed before AFT Fathom allows you to run the Solver.

The state of each Checklist item can also be seen by clicking the Model Status light on the Status Bar at the bottom right corner of the AFT Fathom window (see Figure 2.4). This will cause the Checklist to be displayed in the Quick Access Panel window to the right of the Workspace. The Checklist can also be opened from the View menu. Once the Checklist is complete, the Model Status light in the lower right corner turns from red to green.

A. Specify solution control

The first Checklist item, Specify Solution Control, is always checked when you start AFT Fathom because AFT Fathom assigns default solution control parameters. In general, you do not need to adjust these values. If necessary, you can make adjustments by opening the Solution Control window from the Analysis menu.

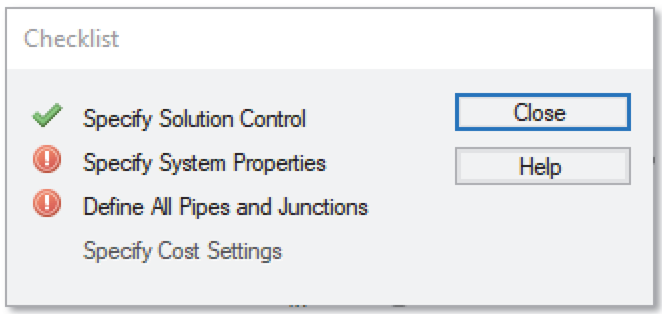


Figure 2.6 The Checklist tracks the model's status. You will see additional Checklist items if any of the add-on modules are in use

B. Specify system properties

The second item on the Checklist is Specify System Properties. To complete this item, you must open the System Properties window. This window allows you to select your fluid(s) as well as gravitational acceleration and atmospheric pressure.

You can directly enter density and viscosity data (using User Specified Fluid), select a fluid from the standard AFT Fathom fluid database (AFT Standard), or select Water Data from ASME Steam Tables. Additionally, you can select from fluids and/or create mixtures using the NIST REFPROP database (included in AFT Fathom) or the Chempak database (an optional add-on to AFT Fathom). Users can also add their own custom fluids to the AFT Standard database.

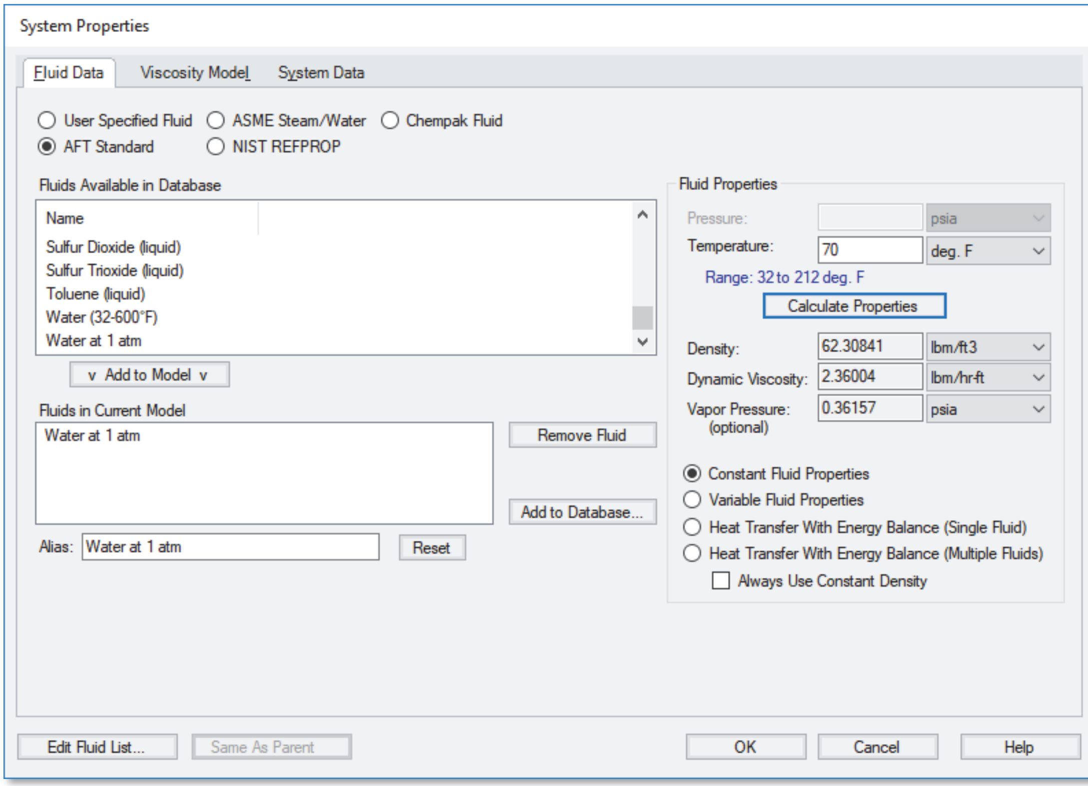


Figure 2.7 The System Properties window lets you select fluids for the model and create mixtures

- **Select System Properties from the Analysis menu** to open the System Properties window (see Figure 2.7). For this example, use the AFT Standard database and select “Water at 1 atm” in the upper list, then click the “Add to Model” button. Enter a temperature of 70 deg. F and click the Calculate Properties button.

- **Open the Checklist** by clicking the status light on the Status Bar and you should now see the second item checked off.

C. Specify cost settings

Skipping the third Checklist item for the moment, the fourth Checklist item, Cost Settings, is always deactivated when you start AFT Fathom. The Cost Settings window provides a way to define energy costs, and the types of costs to be included in the cost calculations. By default, the cost calculation is turned off, and hence nothing further is required for this Checklist item.

Step 4. Define the model components (checklist item #3)

The third item on the Checklist, “Define All Pipes and Junctions”, is not as straightforward to satisfy as the other three. This item includes the required input data and connectivity for all pipes and junctions.

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 light bulb icon on the Workspace Toolbar (alternatively, you could choose “Show Object Status” from the View menu). Each time you click the light bulb, “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”.

Showing undefined objects

Another useful feature is the Undefined Objects list (Figure 2.8). This can be opened from the View menu by clicking on “List Undefined Objects…” or by clicking on “Define All Pipes and Junctions” on the Checklist. Here all objects with incomplete information are listed. Clicking on an undefined pipe or junction will display the property data that is missing. Click the Close button to stop showing this window.

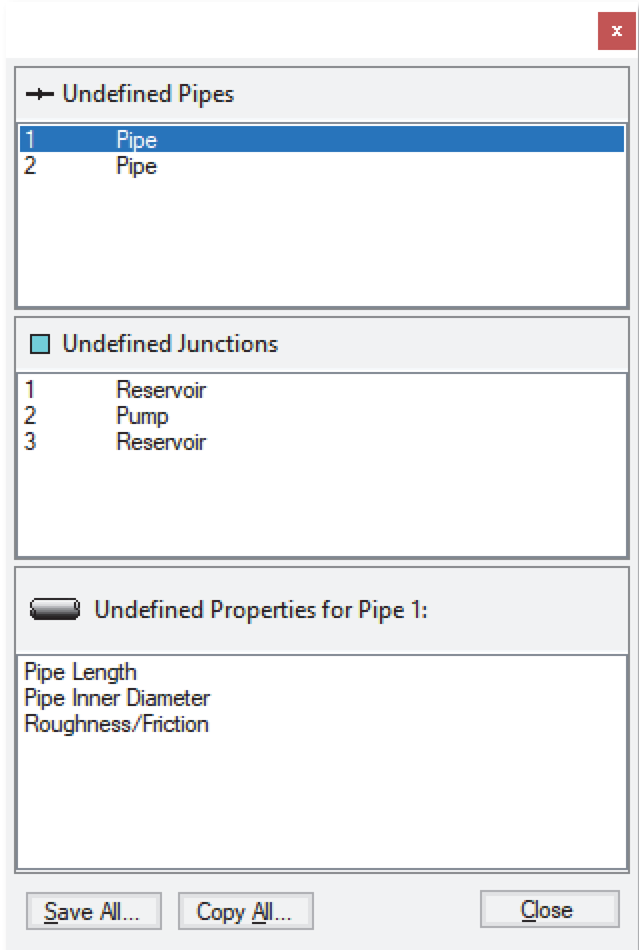


Figure 2.8 The Undefined Objects list lets you see the undefined properties for each undefined object

A. Define Reservoir J1

➢ **To define the first reservoir,** open the J1 Reservoir Properties window (Figure 2.9) by double-clicking on the J1 icon. The pipe table should show you that Pipe 1 is connected to this reservoir (click the Pipe Depth & Loss Coefficients tab to display the pipe table).

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/Junction Window icon on the Workspace Toolbar.

- **Enter a Liquid Surface Pressure of 0 psig, Pipe Depth of 10 feet, and a Liquid Surface Elevation of 10 feet.** This reservoir will represent the supply reservoir upstream of the pump. You can assign any unit of pressure or elevation found in the adjacent dropdown list of units.

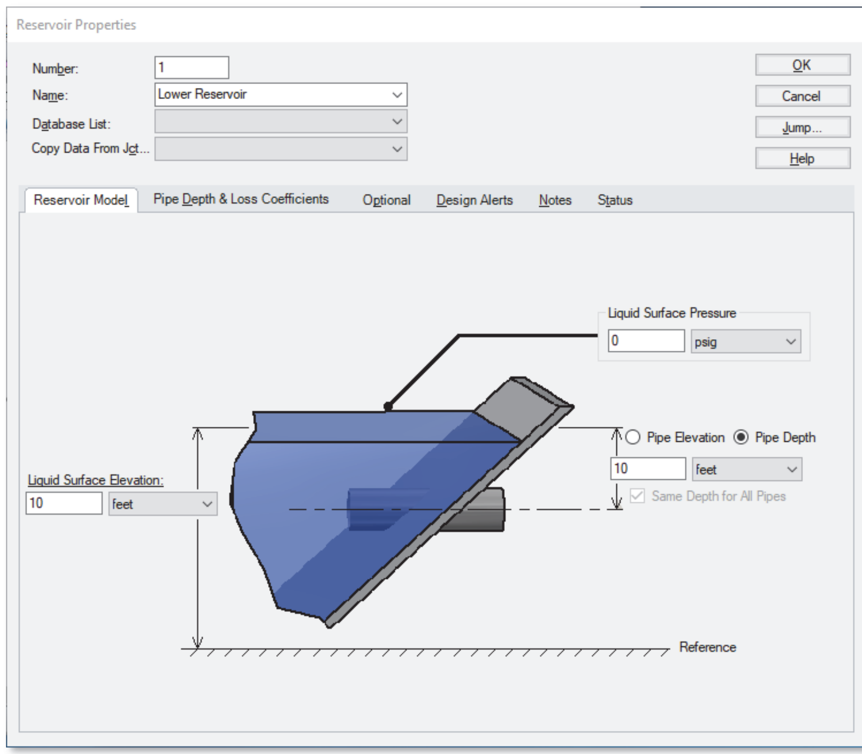


Figure 2.9 Properties window for Reservoir J1. You may or may not see additional tabs called Transient and Cost. These tabs are only visible if the XTS module is active or system costs are being calculated, respectively

Note: You can choose default units for many parameters (such as feet for length) in the User Options window under Preferred Units.

You can give the component a name, if desired, by entering it in the Name field at the top of the window. In Figure 2.9, the name of this reservoir is Lower Reservoir. By default, the junction's name indicates the junction type. The name can be displayed on the Workspace, Visual Report and in the Output.

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

The "Copy Data From Jct…" list will show all the junctions of the same type in the model. This will copy the user-selected parameters from a specified existing junction in the model to the current junction.

Using the folder tabs in the Properties windows

The information in the Properties windows is grouped into several categories (or folders) and placed on separate tabs. Click the folder tab to bring its information forward. Figure 2.9 is an example of a reservoir's Properties window.

If there is only one pipe connected to the reservoir, the depth can be entered directly on the diagram on the Reservoir Model tab for convenience. The depth can also be entered no matter the number of pipes on the Pipe Depth & Loss Coefficients tab. The pipe table allows you to specify entrance and exit loss factors for each pipe connected to the tank (in this case there is only one). The default selection is for the loss factors to be specified as zero. To change the loss factors later, click within the pipe table and enter the loss.

- **Enter a pipe depth of 10 feet in the table.**

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.

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

The highlight feature displays all the required information in the Properties window in light blue. The highlight is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window above the folder tabs or by pressing the F2 key. The highlight feature can also be turned on or off by selecting "Highlight in Pipe and Jct Windows" from the View menu. Note that the highlight feature is turned off in the Examples models.

➢ **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.

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 the J1 reservoir icon and hold down the right mouse button. The inspection window appears, as shown in Figure 2.10.

Inspecting is a faster way of examining the input in an object than opening the Properties window.

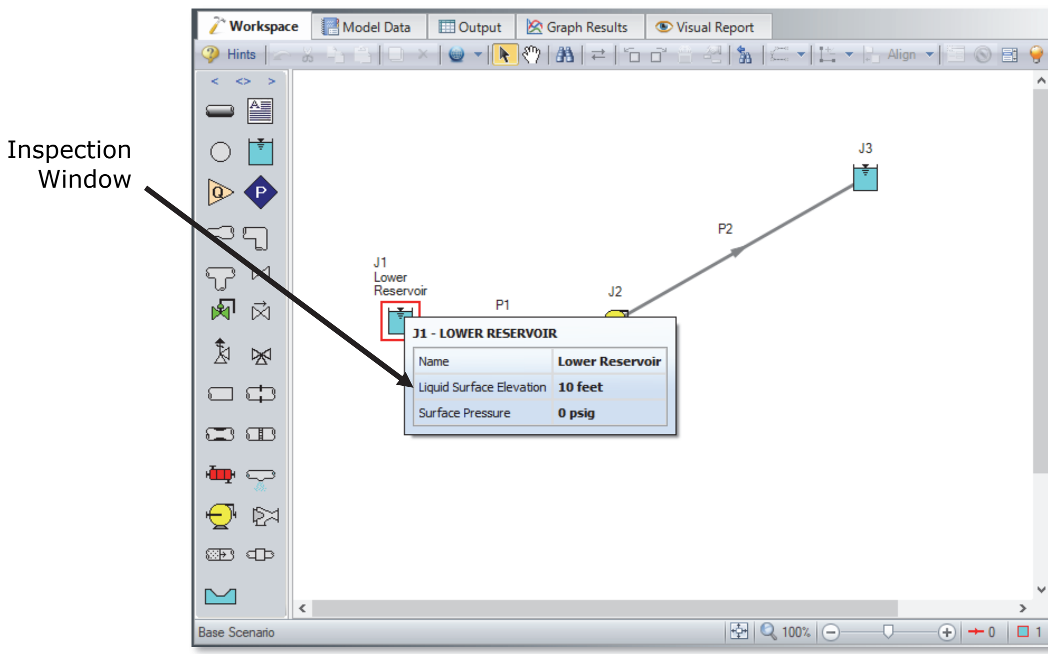


Figure 2.10 Inspecting from the Workspace with the right mouse button

B. Define other junctions

➢ **Next, open the Properties window for Pump junction J2.** Remember that the pump in this example is a centrifugal pump that is being sized. Select Centrifugal (Rotodynamic) and then Sizing for the Analysis Type. The Pump Sizing Parameter should be set to Volumetric Flow Rate. Enter the Fixed Flowrate as 500 gal/min, and the elevation as 0 feet (see Figure 2.11).

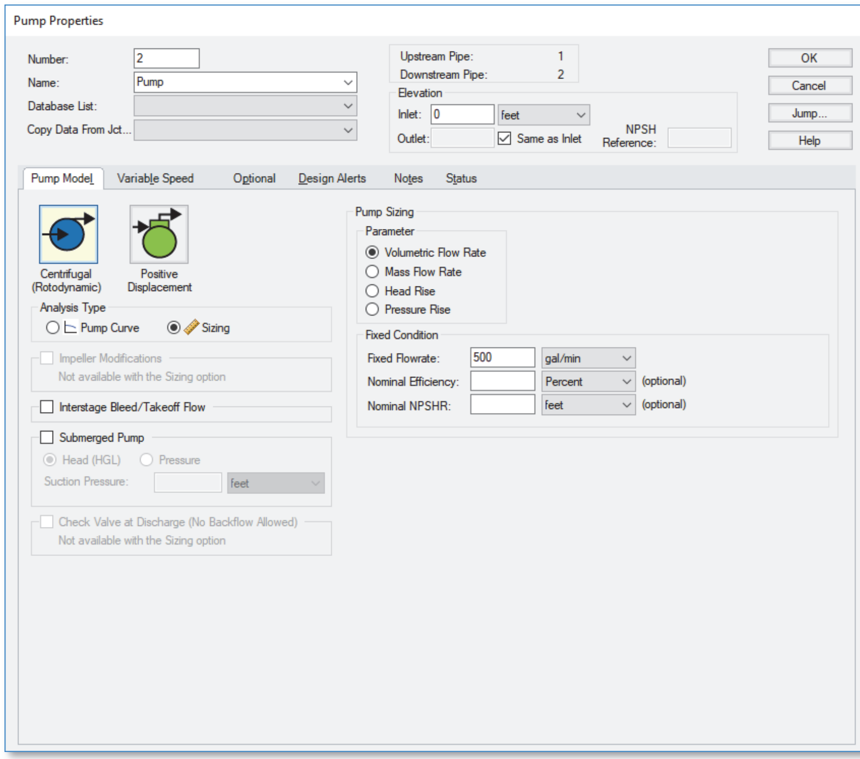


Figure 2.11 Properties window for Pump J2

➢ **Open the J3 Reservoir junction** and change the name to Upper Reservoir. Then enter a Liquid Surface Pressure of 0 psig, a Liquid Surface Elevation of 200 feet, and a pipe depth of 10 feet.

➢ **Save the model** again before proceeding.

C. 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 2.12). For Pipe P1, enter Steel - ANSI, 4 inch, STD (schedule 40), Standard friction model, and a length of 10 feet.

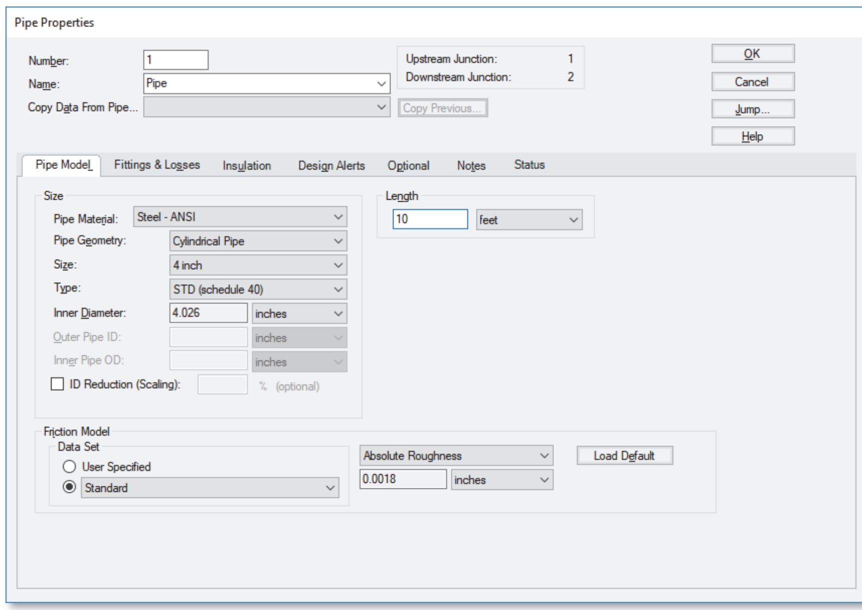


Figure 2.12 Properties window for Pipe P1. You may or may not see an additional tab called Cost. This tab is only visible if the system costs are being calculated.

The Pipe Properties window

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

The Inspect feature can be accessed not only from pipes and junctions located on the Workspace, but also from *within* the Properties window of each pipe (and certain junctions). This is helpful when you want to quickly check the properties of objects that connect to a pipe or junction whose Properties window you already have open.

To Inspect a junction connected to a given pipe, position the mouse pointer on the connected junction's ID number in that pipe's Properties window (located at the top right of the Pipe Properties window) and hold down the right mouse button. This process can be repeated for any junctions that state the upstream and downstream pipe in the junction's Properties window by holding the right mouse button on the pipe's ID number.

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

D. Define Pipe P2

➢ **Open the Properties window for Pipe P2** and specify Steel - ANSI, 4 inch, STD (schedule 40), Standard friction model, and 990 feet in length. In addition, there are some elbows and fittings in this pipe. For simplicity, we are going to represent these as having a total K factor of 25. This is entered on the Fittings & Losses tab in the Total K factor field (Figure 2.13).

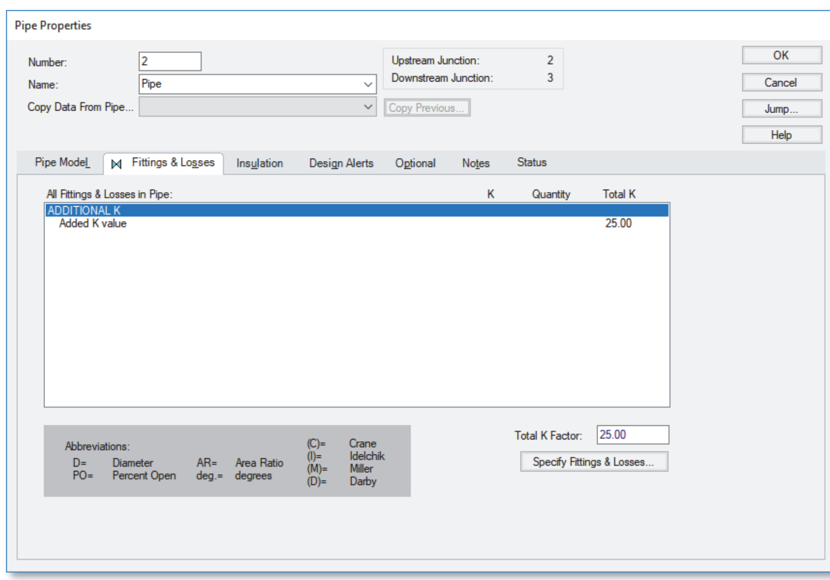


Figure 2.13 Values for losses on pipes due to fittings or other types of losses can be entered on the Fittings & Losses tab in the Pipe Properties Window

After completing Pipe P2, the Checklist should now be complete.

E. Check pipes and junctions

- **Check if all the pipes and junctions are defined.** If all data is entered, the "Define Pipes and Junctions" Checklist item on the Quick Access Panel will have a check mark. If not, turn on the "Show Object Status" from the View menu, and open each undefined pipe and junction. The Status tab on each Properties window will indicate what information is missing.

F. Review Model Data

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

Reviewing input in the Model Data window

The Model Data window is shown in Figure 2.14. To change to this window, you can click on the Model Data tab, select it from the Window menu, or press 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, exported into Excel, saved to a formatted file, printed to an Adobe™ PDF, or printed out for review.

Data is displayed in three general sections. The top is called the General Data section, the middle is the Pipe Data section and the bottom is the Junction Data section. Each section is collapsible using the buttons at the top left of the section. Further, each section can be resized.

The Model Data window allows access to all Properties windows by double-clicking on any input parameter column in the row of the pipe or junction you want to access. You may want to try this right now.

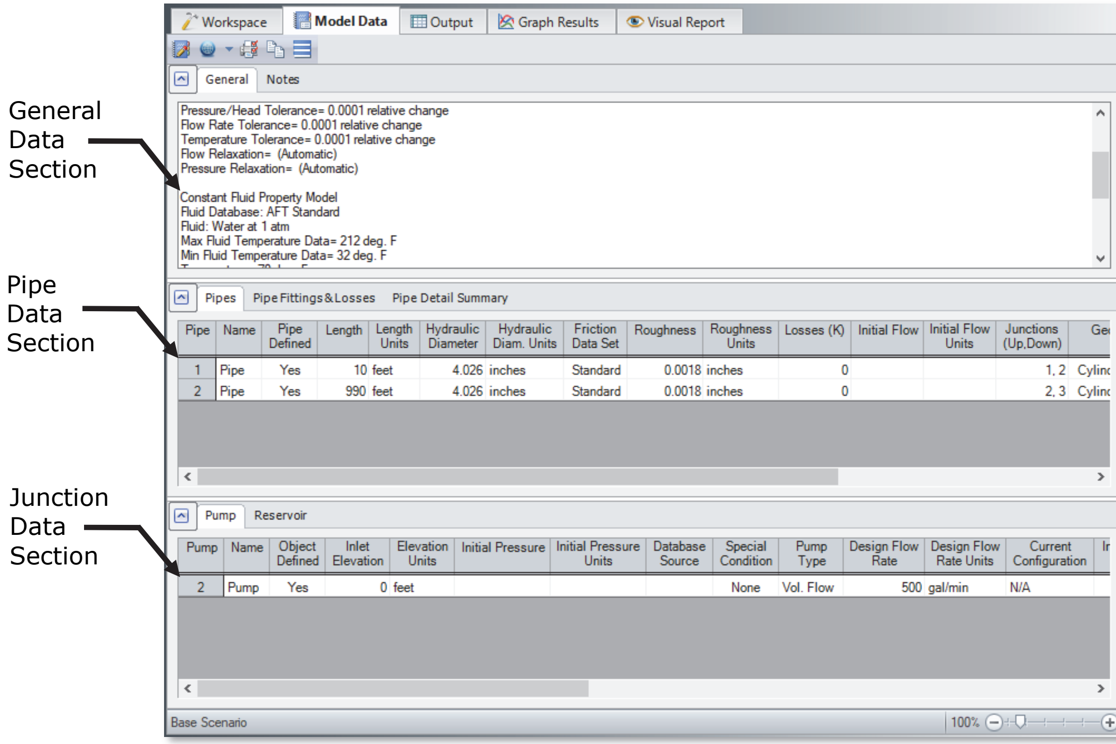


Figure 2.14 The Model Data window shows all input in text form

Step 5. Run the solver

➢ **Choose "Run Model" from the Analysis menu** or click the arrow icon on the Main Toolbar. During execution, the Solution Progress window displays (see Figure 2.15). You can use this window to pause or cancel the Solver's activity. When the solution is complete, click the Output button and the text-based Output window will appear (see Figure 2.17). 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, or printed out on the printer.

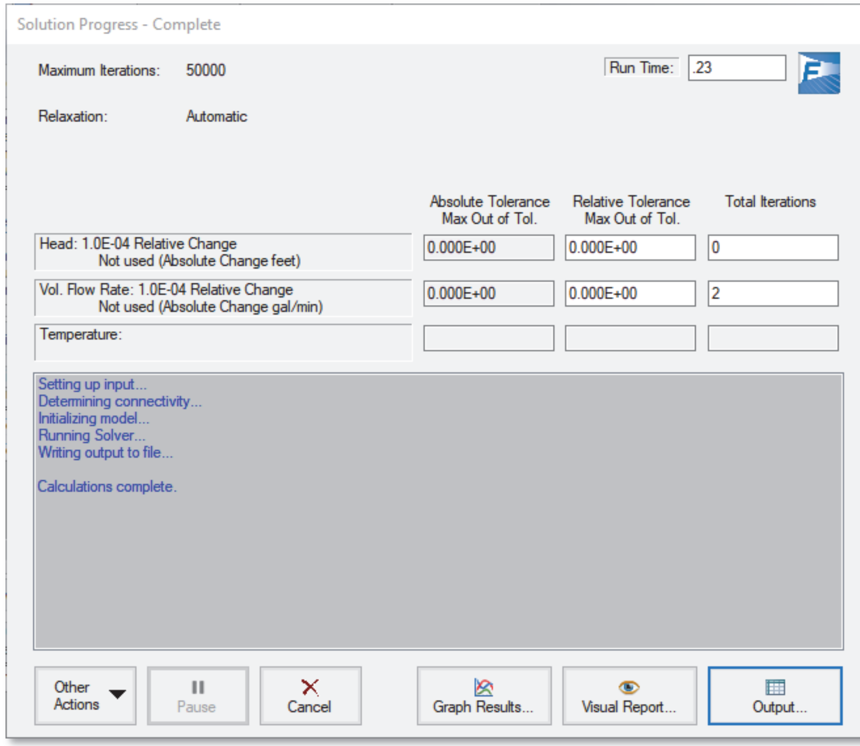


Figure 2.15 The Solution Progress window displays the convergence progress

Step 6. Customize the output

The Output Control window (Figure 2.16) allows you to select the specific output parameters you want in your output. You also can choose the units for the output.

➢ **Select Output Control** from the Tools menu or the Main Toolbar to open the Output Control window. On the Display Parameters tab, select the Pipes button. The list of selected pipe output parameters will be displayed on the right-hand side.

Click the General tab, enter a new title (if you like, you can title this “Water Transfer System”), then click OK to accept the title and other default data.

If you do not change any of the Output Control settings, default Output Control parameters and a default title are assigned.

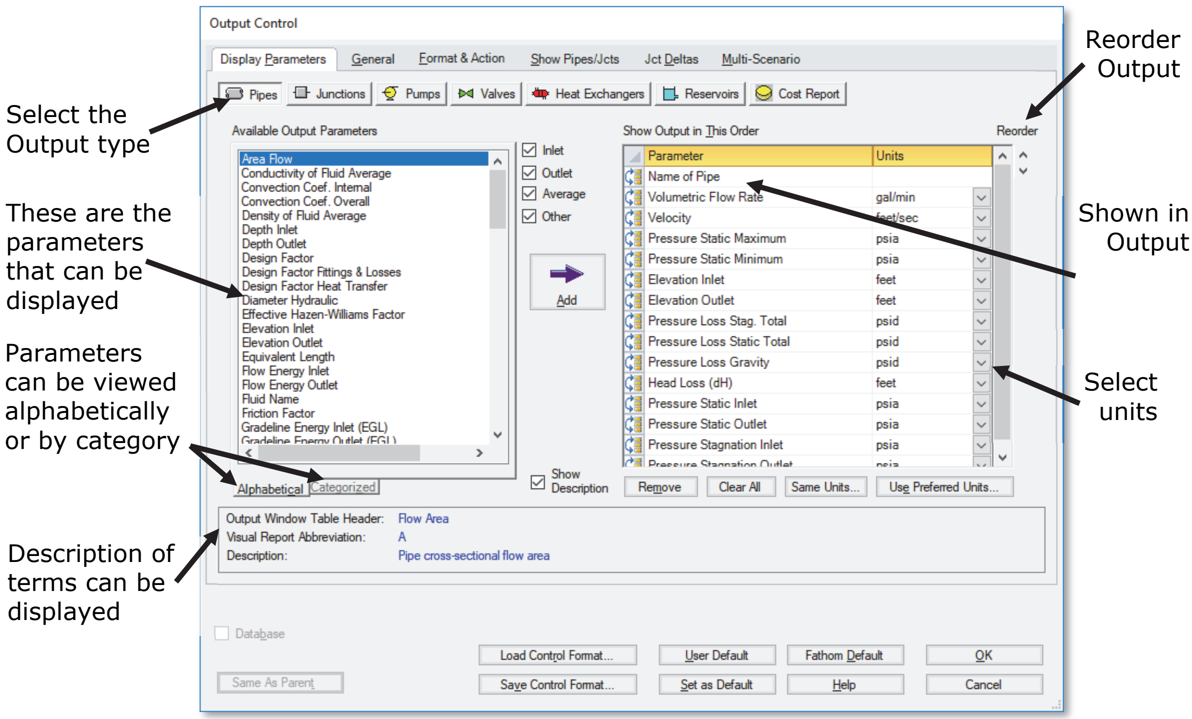


Figure 2.16 The Output Control window lets you customize the output

Step 7. Review the output

The Output window (Figure 2.17) is similar in structure to the Model Data window. Three areas are shown, and you can expand or collapse each area by dragging the boundary between the areas up or down or by clicking the arrow beside the area label. The items displayed in the tables are those items you chose in the Output Control window.

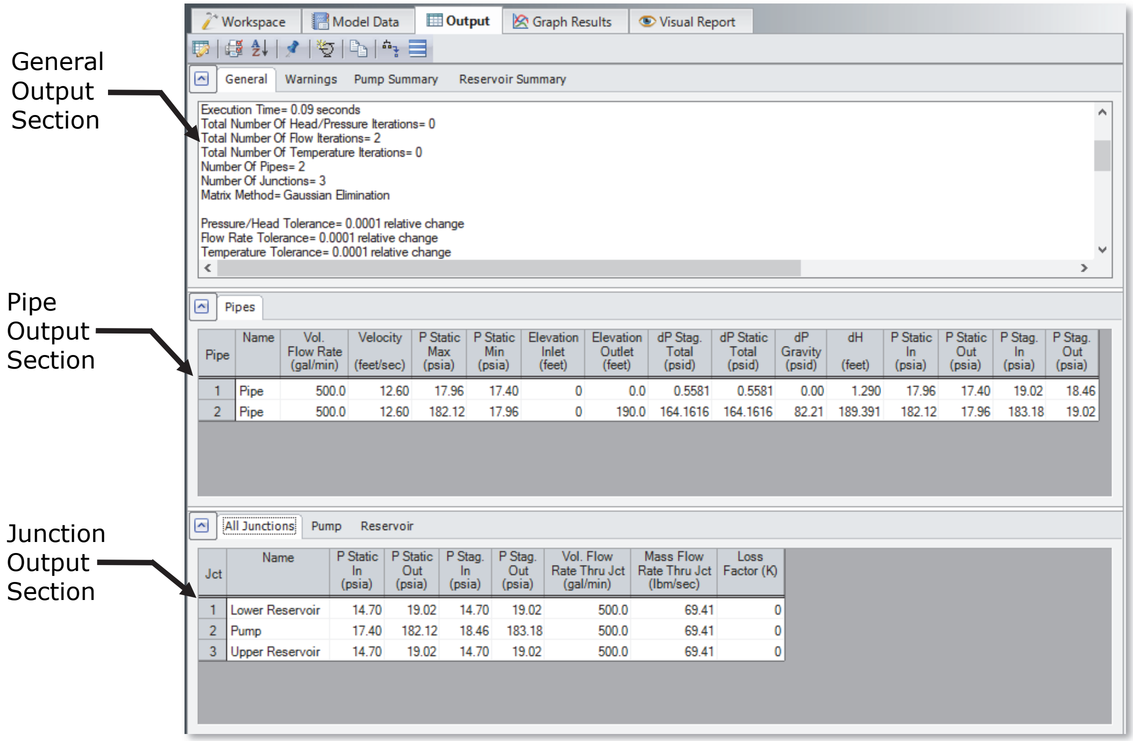
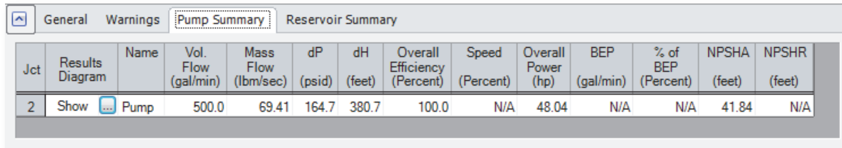


Figure 2.17 The Output window displays output in text form

When pumps are included in a model, AFT Fathom gathers important pump data together in one convenient location in the Output window. This information can be found on the Pump Summary tab located in the General Output section of the output window, as shown in Figure 2.18.

The Pump Summary shows that, for this system, the head requirement of the pump is 380.7 feet. The difference in elevation of the reservoirs is 190 feet, hence this is the minimum required head rise regardless of the flow rate or pipe size. The friction loss is also approximately 190 feet. This can be seen in the dH column of the Pipe output window.



General | Warnings | Pump Summary | Reservoir Summary

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)
2	Show	Pump	500.0	69.41	164.7	380.7	100.0	N/A	48.04	N/A	N/A	41.84	N/A

Figure 2.18 The Output window displays the Pump Summary tab whenever pumps are included in the system

This Pump Summary can also be displayed in a Pump Results Diagram, as shown in Figure 2.19, by selecting the button next to “Show” in the Results Diagram column, double-clicking anywhere in the row, or right-clicking anywhere in the row and selecting Pump Results Diagram.

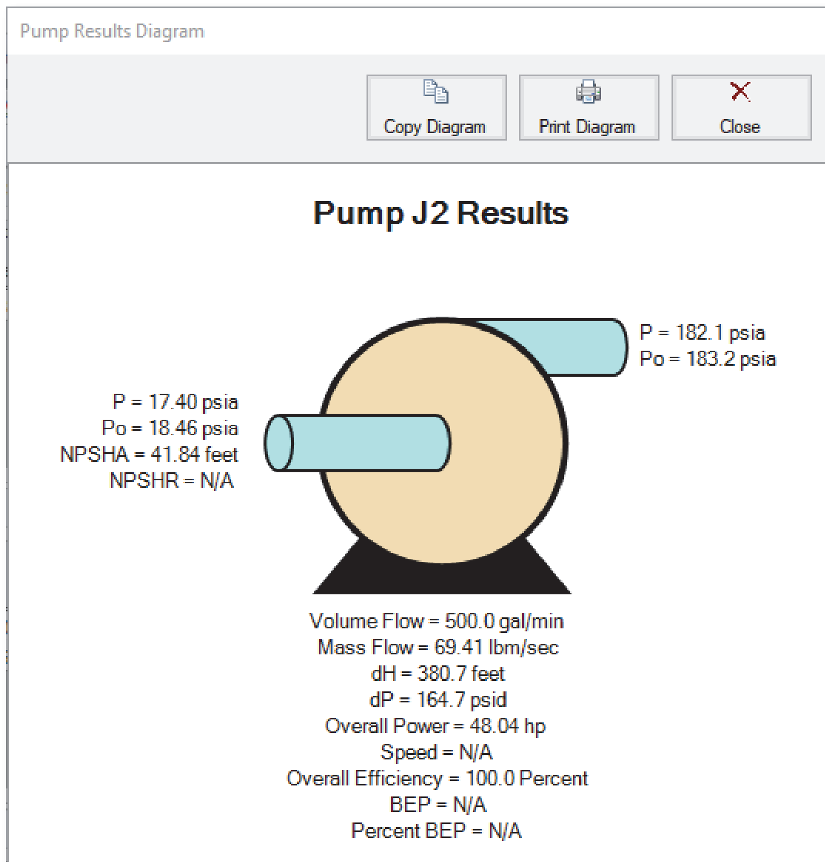


Figure 2.19 Pump Results Diagram as Opened from the Pump Summary (see Fig. 2.18)

A. 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 with units of gal/min.

- **Select Output Control from the Tools menu** or Main Toolbar to open the Output Control Window (see Figure 2.16). On the Display Parameters tab, select the Pipes button. The list of selected pipe output parameters will be displayed on the right-hand side. Change the units for Volumetric Flow Rate by selecting gal/hr (gallons per hour) from the unit list shown beside the parameter.

- **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.** Select the Display Parameters tab and select the Pipe button. The Reorder scroll bar on the far right allows you to reorder parameters in the list. You can also click on the reorder symbol on the left side of each row to manually drag-and-drop the rows to change the parameter order (see Figure 2.16).

- **Select the Velocity parameter** and use the Reorder scroll bar or drag-and-drop 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, the second column now contains the Pipe Name, and the third column 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.

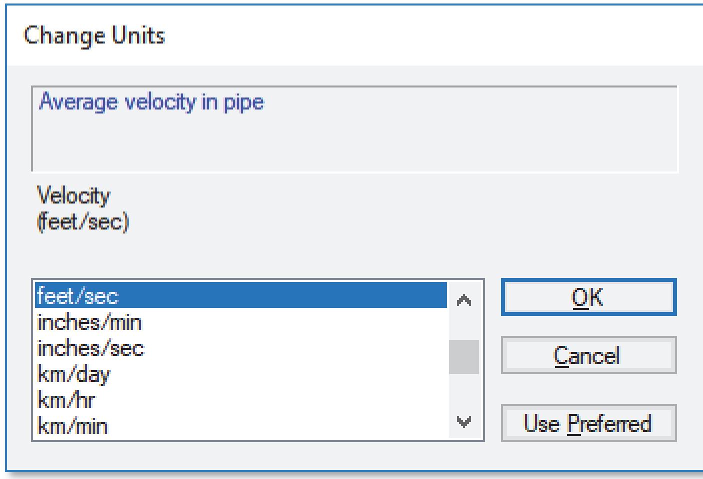


Figure 2.20 The Change Units window is opened from the Output window tables by double-clicking the column header

- **Select Output Control from the Tools menu once again.** Select the Display Parameters tab and select the Pipes button. From the list on the left select Volumetric Flow Rate. Click the Add button to add it to the list on the right and change its units to gal/min. Then use the reorder button or row drag-and-drop to move it below the previous Volumetric Flow Rate item.

- **Click OK to display the changes to the current results.** You will see two columns of Volumetric Flow Rate, each with different units. This is helpful when you want to see an output parameter in different sets of units simultaneously.

- **Lastly, double-click the column header for Velocity on the Output window Pipe Results Table.** This will open a window in which you can change the units again if you prefer (see Figure 2.20). These changes are extended to the Output Control parameter data you have previously set.

B. View the Visual Report

The Visual Report allows you to show model input and output next to the model itself. Pipes can be colored based on ranges of output values (or simply one specific value) to aid in your understanding of the modeled system's hydraulics.

- **Change to the Visual Report window** by clicking on the Visual Report tab, choosing it from the Window menu, or pressing CTRL-I. This window allows you to integrate your text results with the graphic layout of your pipe network.

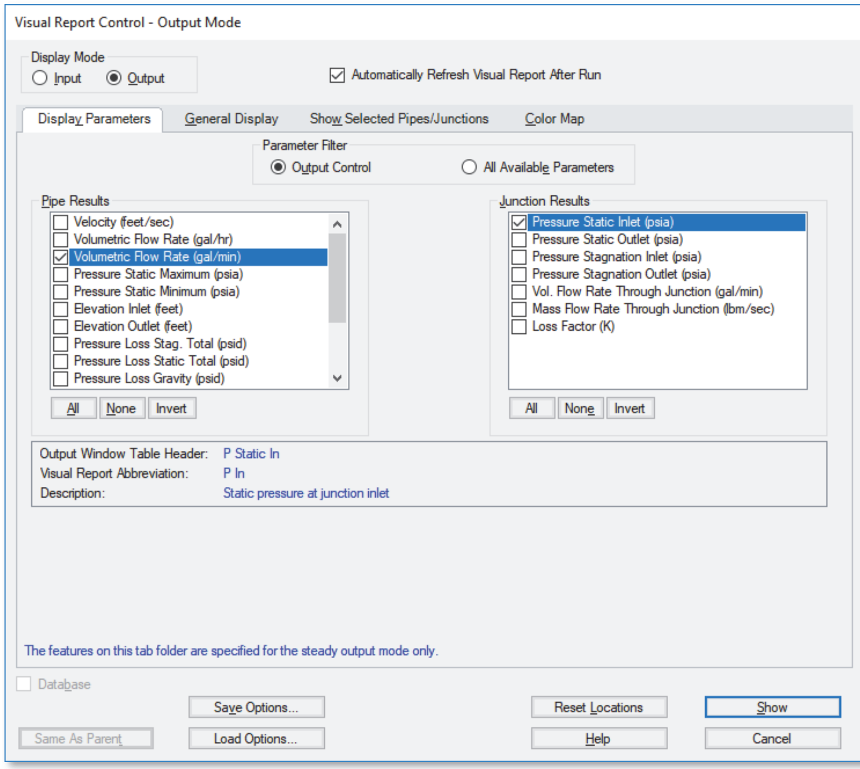


Figure 2.21 The Visual Report Control window selects content for the Visual Report window

- **Click the Visual Report Control button on the Visual Report Toolbar** (or View menu) and open the Visual Report Control window, shown in Figure 2.21. Select Volumetric Flow Rate in the Pipe Results area and Pressure Static Inlet in the Junction Results area. Click the Show button. The Visual Report window graphic is generated (see Figure 2.22).

  It is common for the text in the Visual Report window to overlap when first generated. You can change this by selecting smaller fonts or by dragging the text to a new area to increase clarity. You can also use the Visual Report Control window to display units in a legend to increase the clarity of the display. These adjustments have already been done in Figure 2.22. This window can be printed or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.

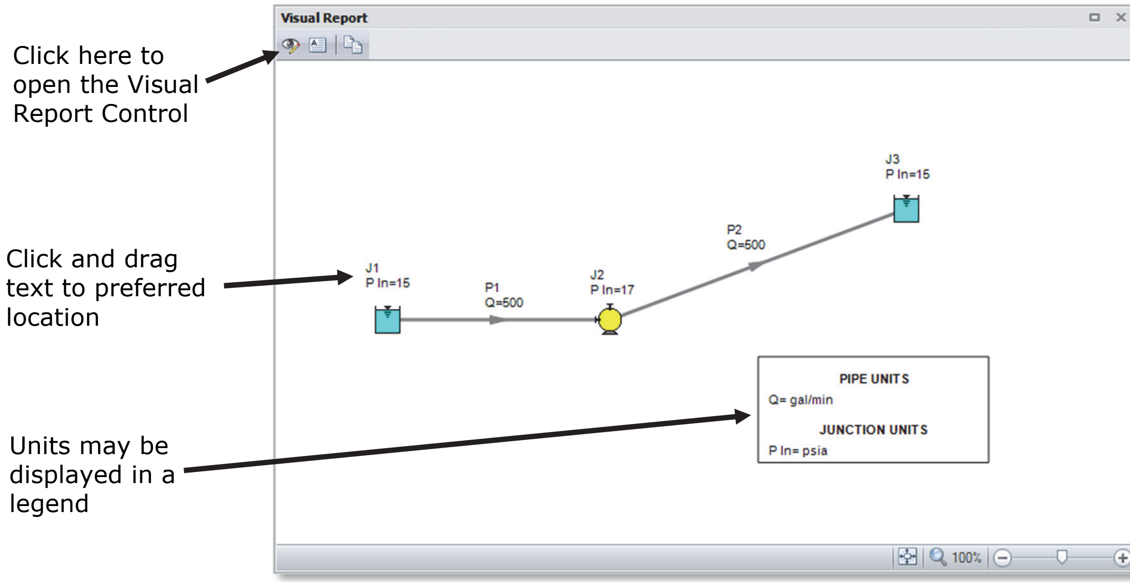


Figure 2.22 The Visual Report integrates results with model layout

C. Graph the results

➢ **Change to the Graph Results window** by clicking on the Graph Results tab or choosing it from the menu. The Graph Results window offers full-featured Windows plot preparation.

AFT Fathom's Graph Guide, accessed by clicking on the "What Would You Like to Do?" button located at the top right of the Graph area, provides assistance by guiding you through the creation of a "Quick and Simple" graph, or an "Advanced" graph (Figure 2.23). You can create a graph by following the prompts on the Graph Guide, by manually specifying the graphing parameters on the Graph Control tab on the Quick Access Panel, or from the "Select Graph Parameters" icon located in the top left corner of the Graph Results toolbar. For the remainder of this Quick Start Guide, the Graph Guide will be hidden, but keep in mind that you can use it whenever you would like assistance in creating a graph.

Figure 2.24 shows the Graph Results window with the Graph Control tab enabled on the Quick Access Panel.

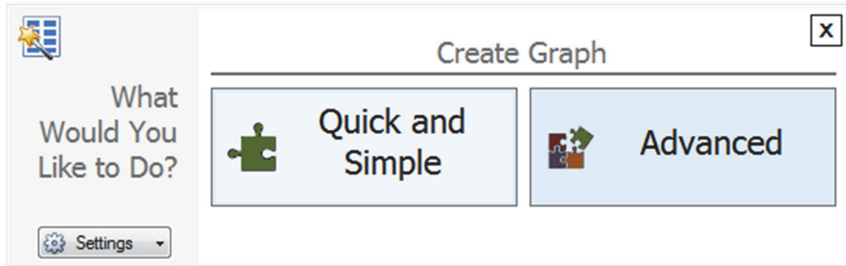


Figure 2.23 The Graph Guide can be toggled on and off by clicking the "What Would You Like to Do?" button on the Graph Results window

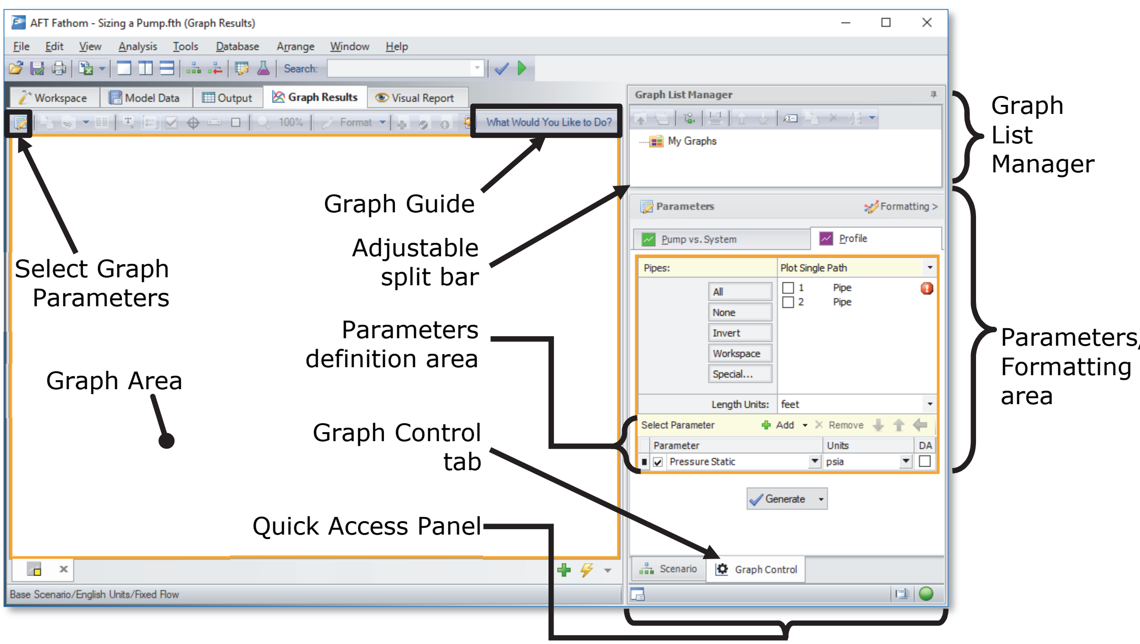


Figure 2.24 The Graph Results window and Quick Access Panel Graph Control tab is where various system parameters can be graphed

AFT Fathom gives you the ability to create “stacked graphs”. These are graphs that are displayed on top of each other with the same X-axis but with different parameters on the Y-axis. This feature is very helpful when you want to look the behavior of multiple hydraulic parameters in the same location of your model simultaneously without having to create completely separate graphs.

In this example, you will create stacked graphs of the pressure and flow along the pipeline.

- **Click the Profile tab on the Quick Access Panel**. In the Pipes selection list, click the All button to select all the pipes.
- **For the Length Units,** select feet.
- **From the Parameters definition area, select “Pressure Static”** and specify units of psig.
- **In order to add a stacked graph with the static pressure graph showing the volumetric flowrate along the pipeline,** click the “Add” button, which is the green “+” icon next to “Select Parameter”. A new row under the Parameters definition area will appear.
- **Choose “Volumetric Flow Rate” in this new parameter row,** and select units of gal/min.
- **Click the Generate button** to create the stacked pressure and volumetric flow rate graphs.
- **To format the legend font size, right-click** on each legend and use the scroll bar to decrease the font size to 9 for both graphs. Drag the static pressure graph legend to the upper right corner of the graph to improve visibility of the graphed data.
- **To format each axis font size, right-click** on each axis title and use the scroll bar to decrease the size until the font size on each axis appears as you like.

Figure 2.25 shows the input in the Parameters/Formatting area on the Quick Access Panel.

Figure 2.26 shows the stacked graphs detailing the static pressure and volumetric flowrate along the entire pipeline modeled in this simple system. Note that the pump adds approximately 165 psig of pressure in order to maintain a flow of 500 gpm throughout the pipeline.

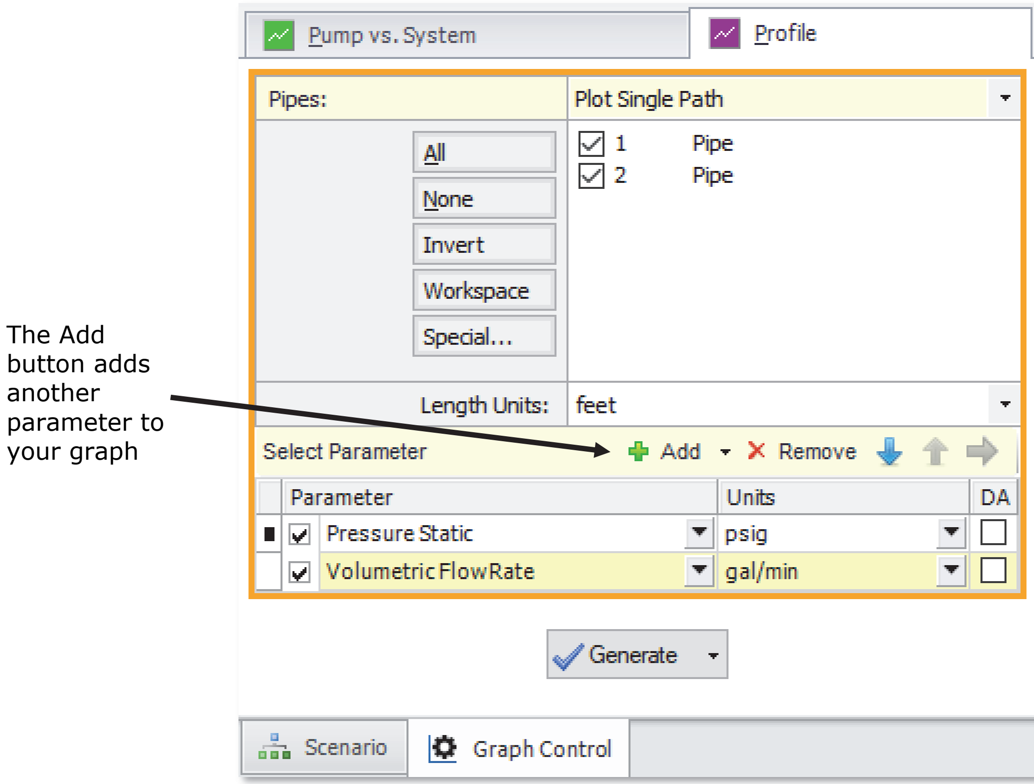


Figure 2.25 The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area

The graph colors, fonts and other elements can be modified using the Formatting area on the Graph Control tab on the Quick Access Panel. 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.

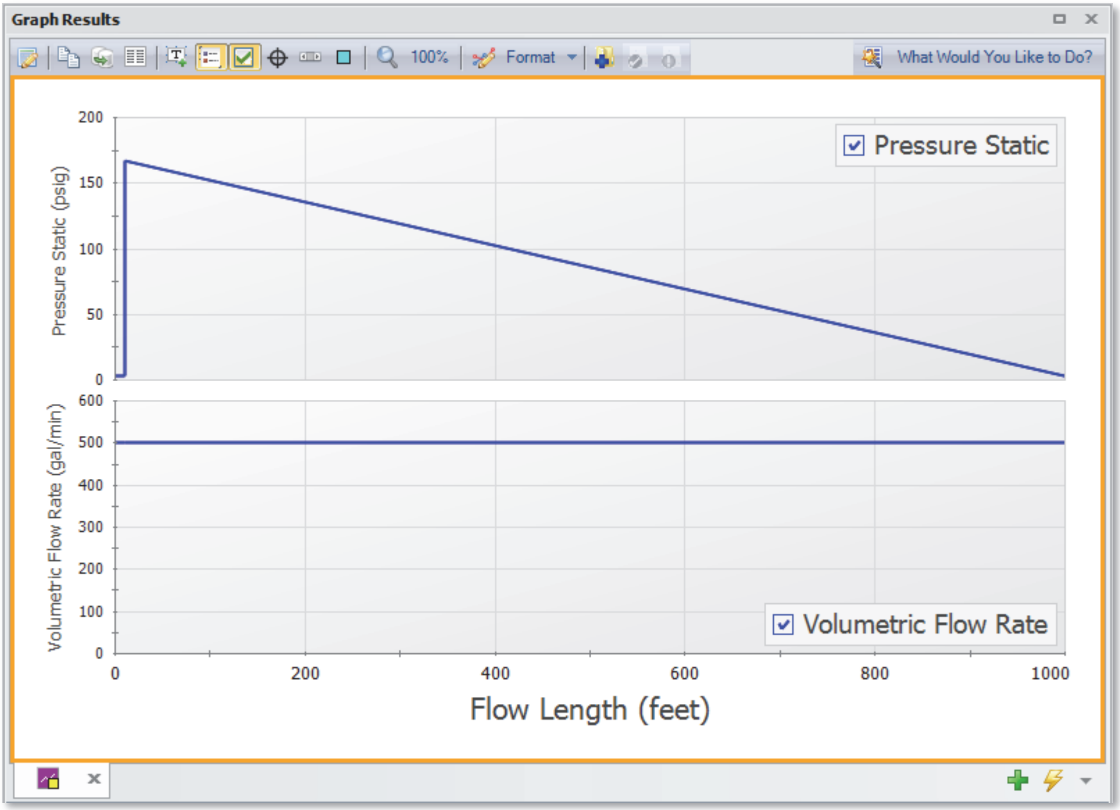


Figure 2.26 Graph Results window offers full-featured graphing. Here the static pressure and volumetric flow along the entire length of the pipeline is shown

Step 8. 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 component by adding the pump curve data to the pump junction.

Open the Pump Properties window. Change the Analysis type from Sizing to Pump Curve. Then select the Enter Curve Data button. This will open the Pump Configuration window.

Data for the pump curve is entered in the Raw Data table. After the data has been entered, a curve fit for the data must be created by selecting the Generate Curve Fit Now button. Enter the following pump curve data, as shown in Figure 2.27.

1. Flow Parameter: Volumetric (gal/min)
2. Pressure/Head Parameter: Head (ft)
3. Pump Curve Data:
    a. 400 ft @ 0 gal/min
    b. 381 ft @ 500 gal/min
    c. 250 ft @ 1000 gal/min
4. Curve Fit Order: 2

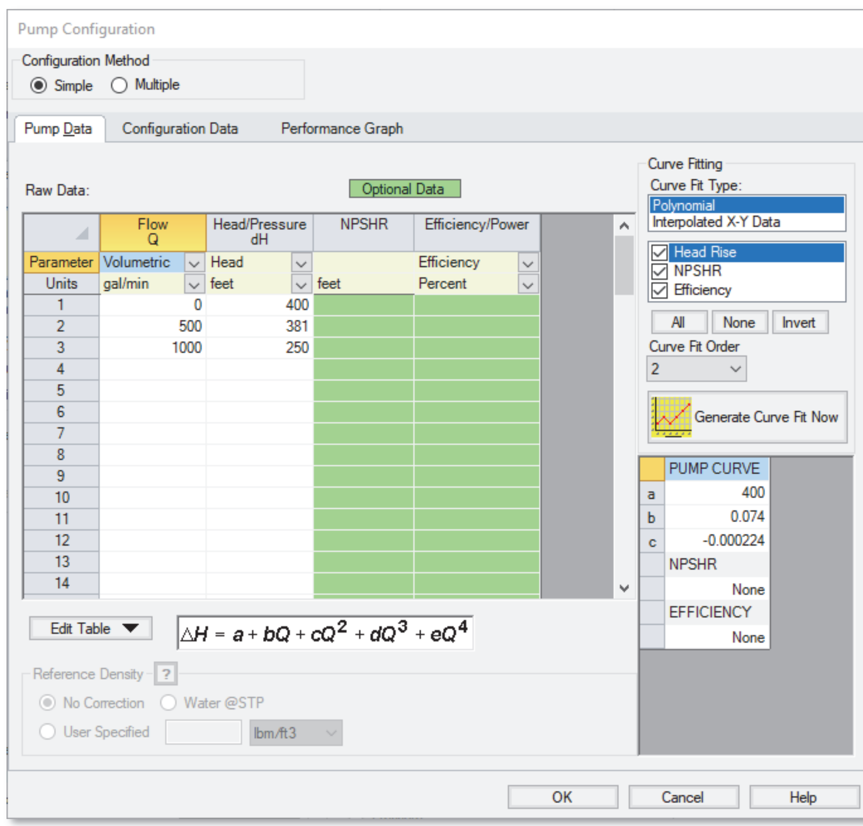


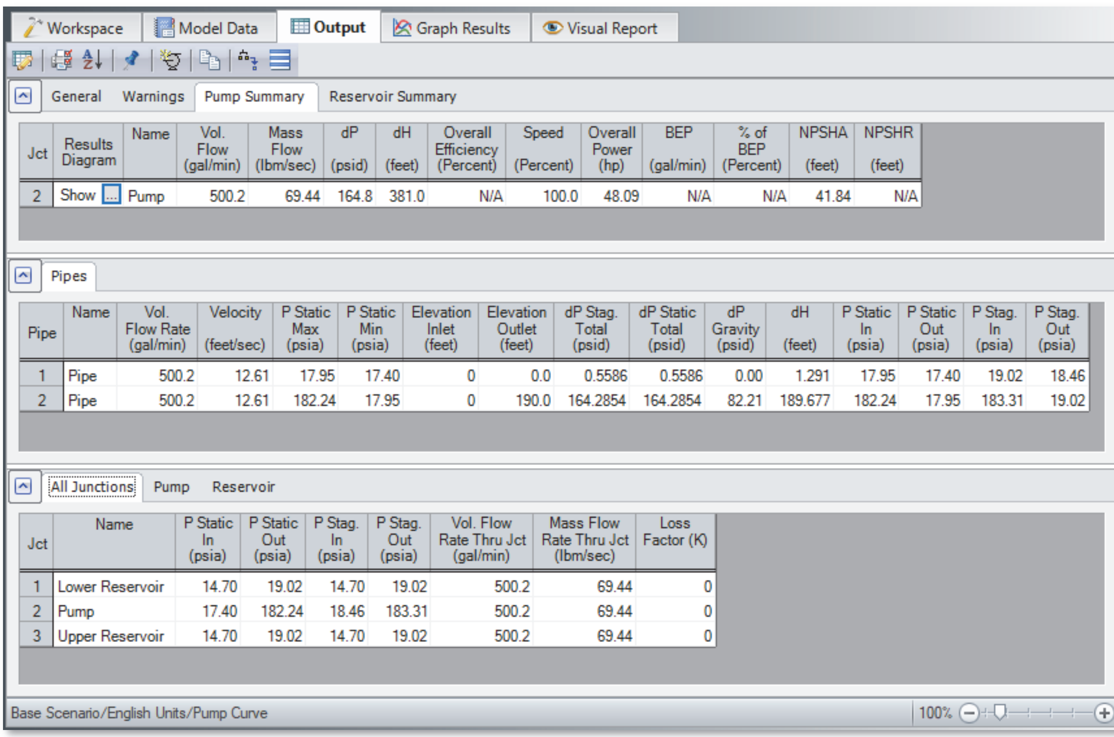
Figure 2.27 The Pump Configuration window is used to enter pump curve data

After the curve data is entered, click the Generate Curve Fit Now button and then the OK button.

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. See Chapter 4 for an example that illustrates how to use the Scenario Manager.

Re-run this model. Examine the Pump Summary in the General Output Section. The output (Figure 2.28) shows the pump operating at a flow rate of 500.2 gal/min, and a head rise of 381 ft, which is acceptably close to the previous sizing calculation.

Change to the Graph Results window, and choose Pump vs. System on the Graph Control tab on the Quick Access Panel (Figure 2.29). Here you can specify that a pump vs. system curve be created as shown in Figure 2.30.



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)
2	Show	Pump	500.2	69.44	164.8	381.0	N/A	100.0	48.09	N/A	N/A	41.84	N/A

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	500.2	12.61	17.95	17.40	0	0.0	0.5586	0.5586	0.00	1.291	17.95	17.40	19.02	18.46
2	Pipe	500.2	12.61	182.24	17.95	0	190.0	164.2854	164.2854	82.21	189.677	182.24	17.95	183.31	19.02

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	19.02	14.70	19.02	500.2	69.44	0
2	Pump	17.40	182.24	18.46	183.31	500.2	69.44	0
3	Upper Reservoir	14.70	19.02	14.70	19.02	500.2	69.44	0

Figure 2.28 Output when running with a pump curve

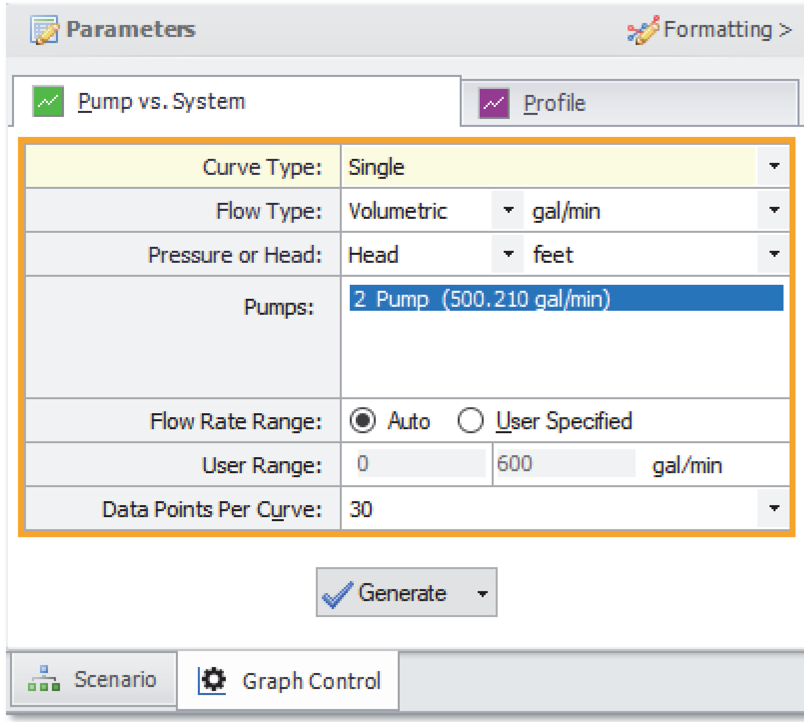


Figure 2.29 The Graph Control tab on the Quick Access Panel can be used to generate a Pump vs. System Curve

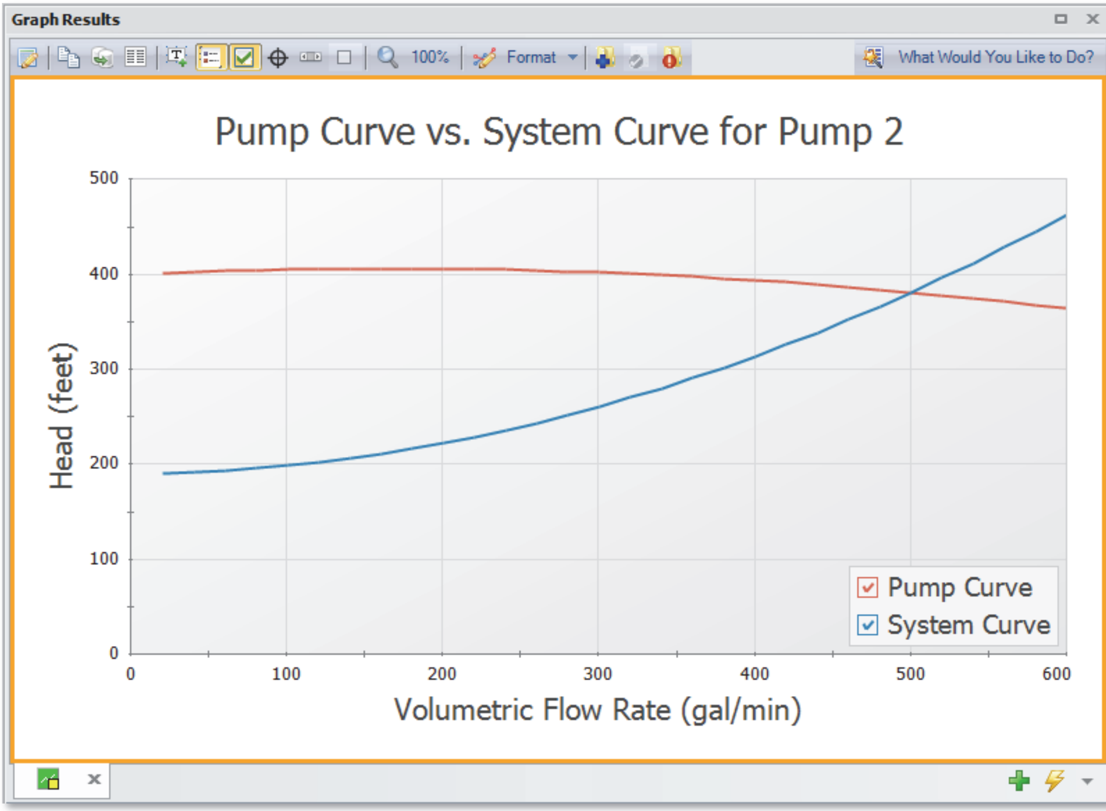


Figure 2.30 Pump vs. system curve using selected pump

Conclusion

You have now used AFT Fathom's five Primary Windows on a simple model to size and select a pump.

CHAPTER 3

Spray Discharge System Example

The objective of this example is to find the minimum supply pressure needed to provide a system of eight spray discharge heads with at least 100 gal/min to each spray head.

Topics covered

This example will cover the following topics:

- Use of spray discharge junctions
- Global Pipe Editing and Copy Junction Data From tool
- Using manual iterative techniques to arrive at a desired result
- Use of Isometric Pipe Drawing Mode

Required knowledge

This example assumes that the user has some familiarity with AFT Fathom, such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the Sizing a Pump Example in Chapter 2 for more information on these topics.

Model file

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

- *Spray Discharge.fth* - AFT Fathom model file

Problem statement

A spray discharge system consisting of eight spray discharge nozzles is supplied by a constant pressure supply. The system has one main supply line that branches into two supply lines that supply four sprays on each side. Determine the minimum supply pressure necessary to ensure a minimum volumetric flow rate of 100 gal/min through each of the spray junctions.

Step 1. Start AFT Fathom

From the Start Menu, choose AFT Products and AFT Fathom.

Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu.
2. On the Fluid Data tab, select the AFT Standard database and then select “Water at 1 atm” in the Fluids Available in Database list.
3. Click “Add to Model” to select water for use in this model.
4. Now type in 70 deg. F in the fluid temperature box and click “Calculate Properties”. This calculates the fluid properties to use in the model.
5. Click OK.

Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to “Define Pipes and Junctions”. In the Workspace window, assemble the model as shown in Figure 3.1.

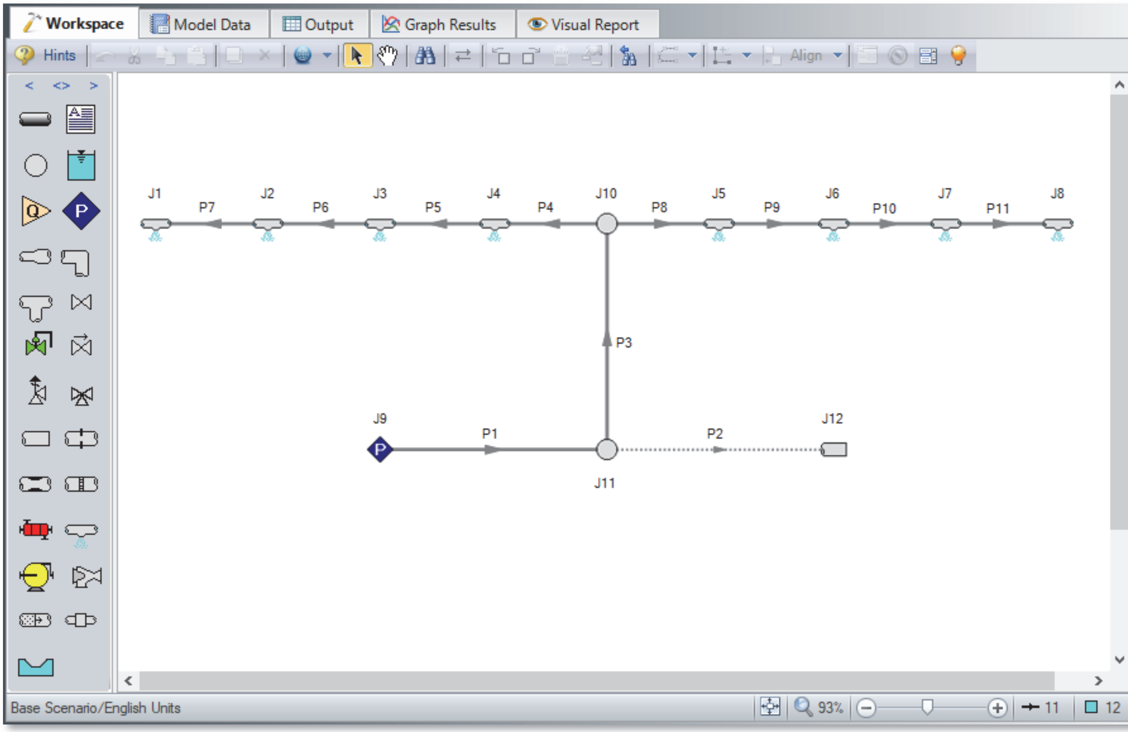


Figure 3.1 Layout of pipe system for Spray Discharge System Example

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click pipe objects P1-P3 and enter the following data in the Properties window (pipes P4-P11 are addressed below using Global Editing).

Pipes P1-P11 are Steel - ANSI with standard roughness and the following data:

Pipe	Length (feet)	Size	Type
P1	500	8 inch	Schedule 40
P2	500	8 inch	Schedule 40
P3	9	4 inch	Schedule 40
P4	10	1-1/2 inch	Schedule 40
P5	10	1-1/2 inch	Schedule 40
P6	10	1-1/2 inch	Schedule 40
P7	10	1-1/2 inch	Schedule 40
P8	10	1-1/2 inch	Schedule 40
P9	10	1-1/2 inch	Schedule 40
P10	10	1-1/2 inch	Schedule 40
P11	10	1-1/2 inch	Schedule 40

Because pipes P4-P11 are the same, you can use the Global Pipe Editing tool to speed up the process of entering the data. The Global Pipe Editing tool allows you to apply changes to multiple pipes at the same time. Select Global Pipe Edit from the Edit menu. This will open the Global Pipe Edit window. Select pipes P4-P11 from the list of pipes, as shown in Figure 3.2.

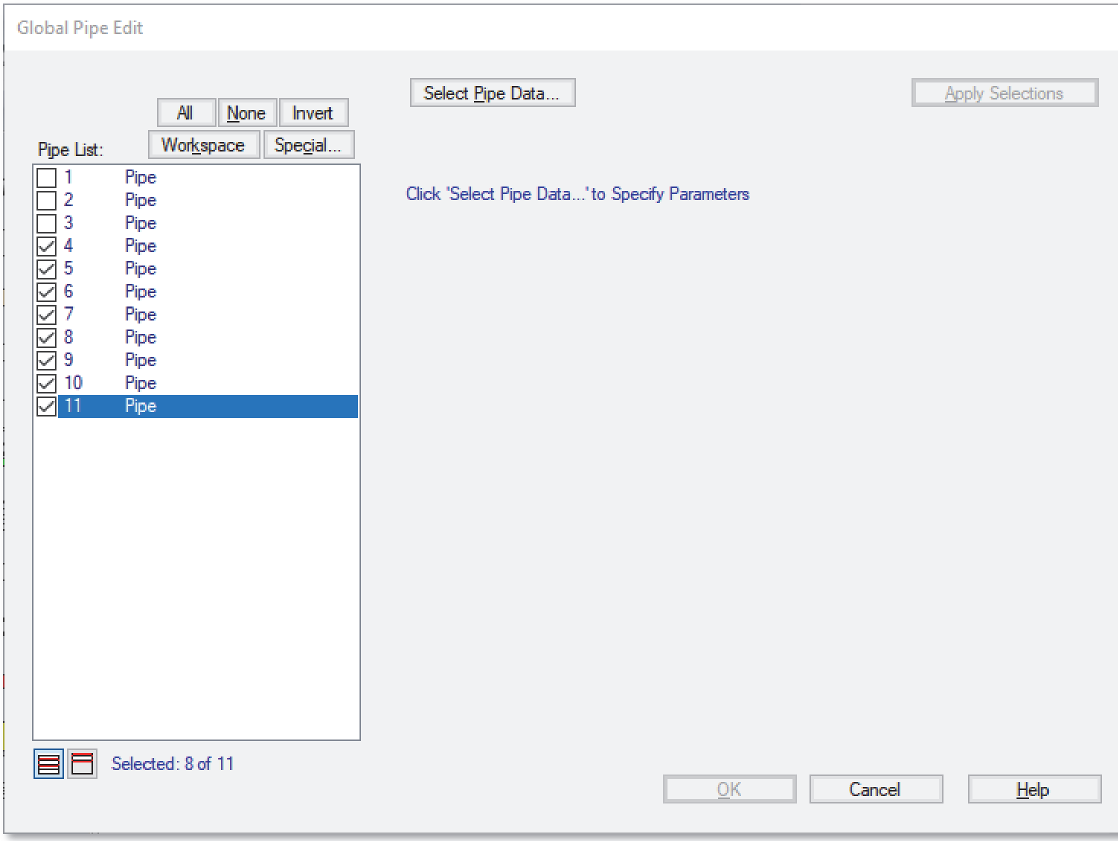


Figure 3.2 Global Pipe Edit window for Spray Discharge System Example

After selecting the pipes, click the Select Pipe Data button. This will open a Pipe Properties window for entering the data that is to be applied to all of the selected pipes. Fill out the data for pipes P4-P11. When you have entered all of the data for the pipes, close the Pipe Properties window by selecting OK. The Global Pipe Edit window will now display a list of all of the parameters that may be applied to the selected pipes. The parameters are categorized as they are displayed on the tabs on the Pipe Properties window. For this example, you want all of the parameters for all of the selected pipes to be updated to the specified values. This can be accomplished by selecting the check box beside each parameter, or by selecting the All button above the list of parameters. The Global Pipe Edit window should now appear as shown in Figure 3.3.

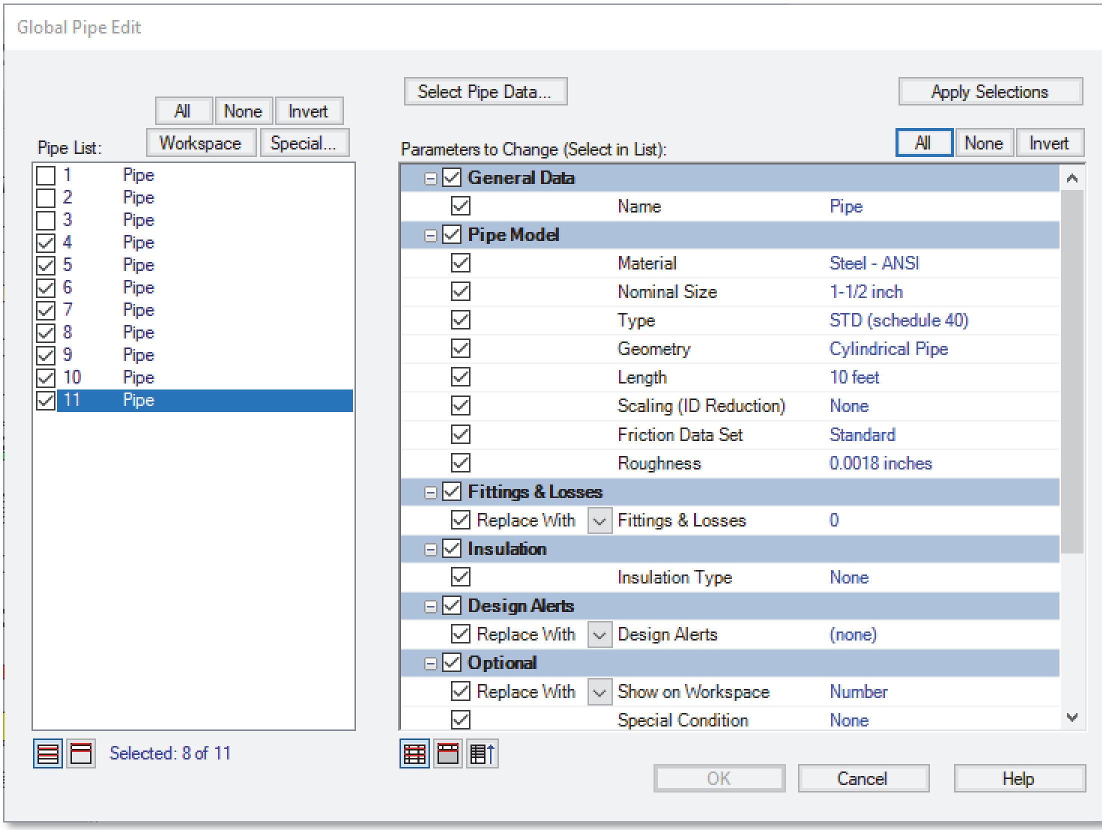


Figure 3.3 Global Pipe Edit window for Spray Discharge System Example with all pipe parameters selected for updating

Apply the changes to the selected pipes by clicking the Apply Selections button. AFT Fathom will notify you that the changes have been completed. Select OK. At this point, all of the changes have been applied. If you wish to cancel all of the changes you just implemented, you may do so by clicking Cancel on the Global Pipe Edit window. Accept the changes by clicking OK.

C. Enter the spray data (J1-J8)

Because all of the spray discharge junctions have the same input data, the Global Junction Edit tool could be used to input the data for all of the junctions at the same time. The Global Junction Edit tool works in much the same way as the Global Pipe Edit tool.

Another tool that can be used to speed up data entry for items with the same input data is the Copy Data From Junction tool. Enter the data for spray discharge junction J1, as listed below, by double-clicking the junction icon on the Workspace to open the Spray Discharge Properties window.

1. Elevation = 10 feet
2. Loss Model = Cd Spray (Discharge Coefficient)
3. Geometry = Spray Nozzle
4. Exit Pressure = 0 psig
5. Cd (Discharge Coefficient) = 0.6
6. Discharge Flow Area = 0.5 in2

After the data has been entered for J1, and the Properties window has been closed, open the Properties window for spray discharge J2. Near the top of the Properties window is a drop-down box labeled Copy Data From Jct. This box lists all of the spray discharge junctions on the Workspace. Select junction J1 from the list. This will open the “Copy Data From Junction” window, which, similar to the Global Edit window, shows a list of all of the parameters for junction J1, and their values. By checking the box beside a parameter, the current junction will copy the parameter value from J1. In this case, you want all of the parameters for junction J2 to be the same as J1, so select the All button at the top of the list to select all of the parameters, as shown in Figure 3.4.

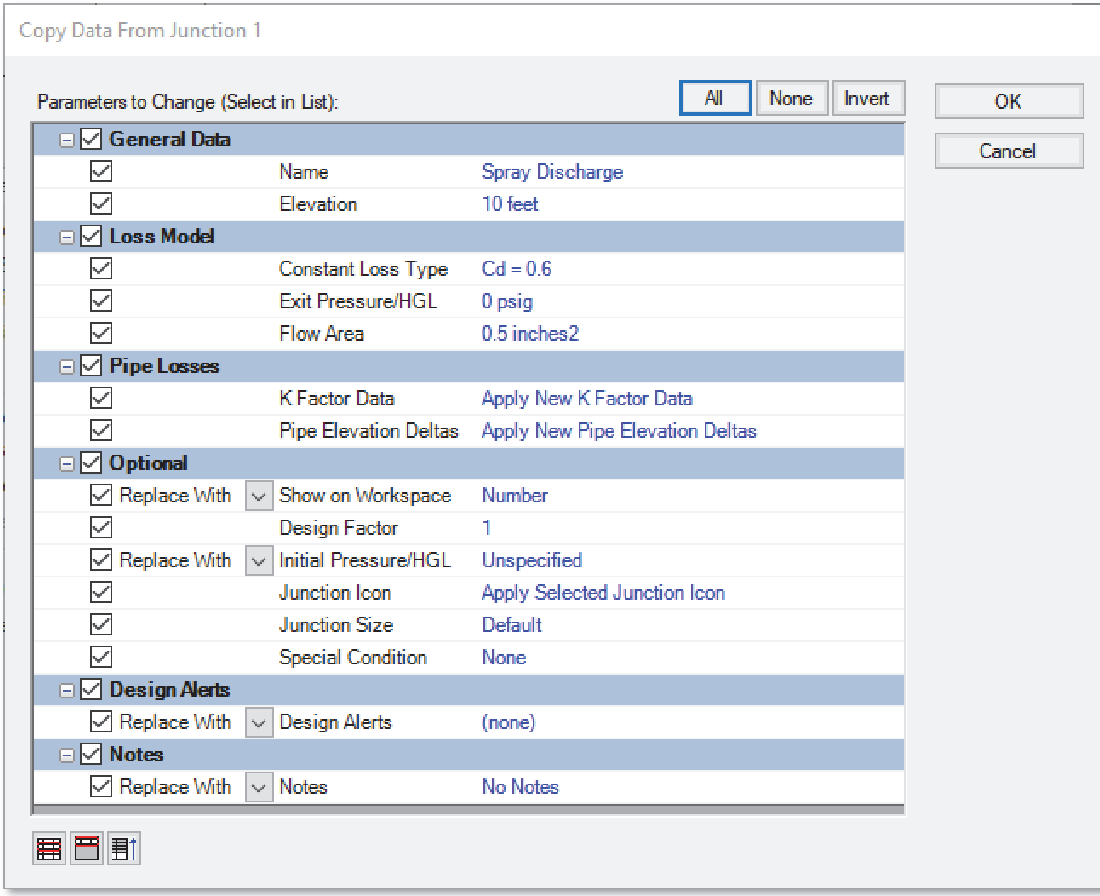


Figure 3.4 Copy Data from Junction window used for Copy Data from Junction tool

After clicking the OK button in the Copy Data From Junction window, all of the junction J2 parameters are updated to the selected J1 parameter values. Repeat this process for all of the spray discharge junctions.

There is a similar function available from the Pipe Properties window tool.

D. Enter other junction data

J9 - Assigned Pressure

1. Elevation = 1 feet
2. Pressure = 200 psig, Stagnation

J10 - Branch

Elevation = 10 feet

J11 - Branch

Elevation = 1 feet

J12 - Dead End

Elevation = 1 feet

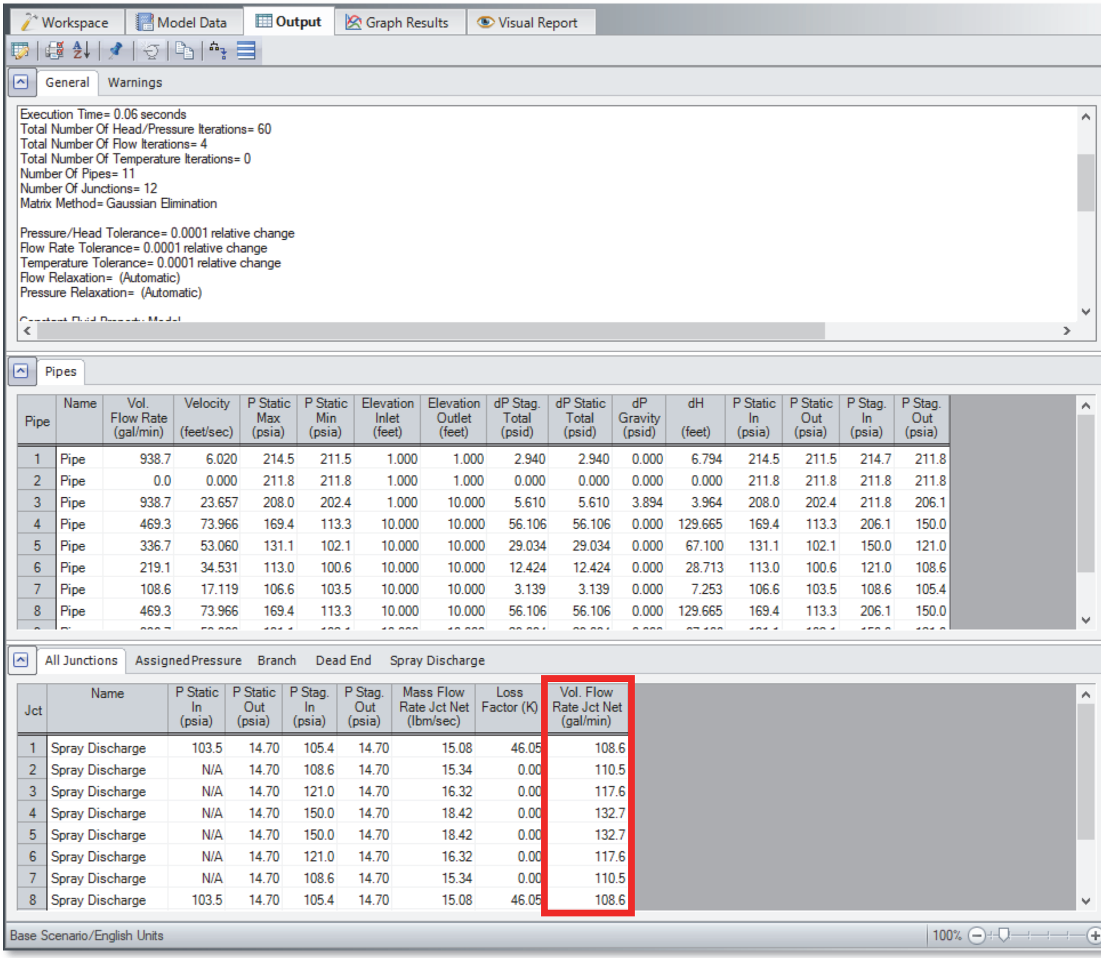
E. Check if the pipe and junction data is complete

Turn on “Show Object Status” from the View menu to verify that all the necessary data is entered. If all data is entered, the Define All Pipes and Junctions Checklist item on the Quick Access Panel 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. You can also open the List Undefined Objects window from the View menu to see what data is missing.

Step 4. Run the model and iterate

Select Run Model in the Analysis menu. This will launch the Solution Progress window. This window allows you to watch as the AFT Fathom Solver converges on the answer. When the solution has converged, view the results by pressing View Output at the bottom of the Solution Progress window.

Open the Output Control window from the Tools menu. Add “Vol. Flow Rate Net at Junction” to the junction Output.



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	938.7	6.020	214.5	211.5	1.000	1.000	2.940	2.940	0.000	6.794	214.5	211.5	214.7	211.8
2	Pipe	0.0	0.000	211.8	211.8	1.000	1.000	0.000	0.000	0.000	0.000	211.8	211.8	211.8	211.8
3	Pipe	938.7	23.657	208.0	202.4	1.000	10.000	5.610	5.610	3.894	3.964	208.0	202.4	211.8	206.1
4	Pipe	469.3	73.966	169.4	113.3	10.000	10.000	56.106	56.106	0.000	129.665	169.4	113.3	206.1	150.0
5	Pipe	336.7	53.060	131.1	102.1	10.000	10.000	29.034	29.034	0.000	67.100	131.1	102.1	150.0	121.0
6	Pipe	219.1	34.531	113.0	100.6	10.000	10.000	12.424	12.424	0.000	28.713	113.0	100.6	121.0	108.6
7	Pipe	108.6	17.119	106.6	103.5	10.000	10.000	3.139	3.139	0.000	7.253	106.6	103.5	108.6	105.4
8	Pipe	469.3	73.966	169.4	113.3	10.000	10.000	56.106	56.106	0.000	129.665	169.4	113.3	206.1	150.0

Jct	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Mass Flow Rate Jct Net (lbm/sec)	Loss Factor (K)	Vol. Flow Rate Jct Net (gal/min)
1	Spray Discharge	103.5	14.70	105.4	14.70	15.08	46.05	108.6
2	Spray Discharge	N/A	14.70	108.6	14.70	15.34	0.00	110.5
3	Spray Discharge	N/A	14.70	121.0	14.70	16.32	0.00	117.6
4	Spray Discharge	N/A	14.70	150.0	14.70	18.42	0.00	132.7
5	Spray Discharge	N/A	14.70	150.0	14.70	18.42	0.00	132.7
6	Spray Discharge	N/A	14.70	121.0	14.70	16.32	0.00	117.6
7	Spray Discharge	N/A	14.70	108.6	14.70	15.34	0.00	110.5
8	Spray Discharge	103.5	14.70	105.4	14.70	15.08	46.05	108.6

Figure 3.5 Output Window for Spray Discharge System Example

The Output window contains all the data that was specified in the Output Control window. The output for the Spray Discharge example is shown in Figure 3.5, for a supply pressure of 200 psig, stagnation. The last column in the junction output table shows the volumetric flow rates for all of the spray discharge junctions are well above 100 gal/min required in the problem statement.

Reduce the supply pressure and run the model again. Repeat this process until you have determined the minimum supply pressure required to supply 100 gal/min at every spray junction.

Step 5. Graph the results

AFT Fathom allows you to create dual Y-axis graphs, which plot two parameters on the same graph.

In this example, you will graph the static pressure and volumetric flowrate from the constant pressure supply J9 through spray nozzle J1 on a dual Y-axis graph to help you graphically determine whether the minimum flow through each spray nozzle is at least 100 gal/min.

Note: You can repeat this process for the flow path from the constant pressure junction to spray nozzle J8, or you can graph both flow paths on this same graph using the Plot Multiple Paths Using Groups feature.

1. From the Graph Control tab, choose the Profile tab in the Parameters/Formatting area, specify “Plot Single Path” in the Pipes definition area, then select pipes 1, 3, 4, 5, 6, and 7. These pipes make up the path from the constant pressure supply to the J1 spray nozzle.
2. Ensure that “feet” is selected for the Length Units.

You will need to decide what parameter you wish to plot on the primary Y-axis and which parameter you want to plot on the secondary Y-axis. In this example, we will plot static pressure on the primary Y-axis and volumetric flowrate on the secondary Y-axis.

3. To do this, select “Pressure Static” in the Parameter definition area and choose “psig” as the static pressure unit.
4. Add a parameter by clicking on the “Add” button next to “Select Parameter” in the Parameters definition area.
5. Change this added parameter to “Volumetric Flowrate” and choose gal/min as the unit.
6. To specify that the volumetric flowrate will be plotted on the secondary Y-axis, click the “Make Secondary Y-Axis” button, which is the blue, right arrow icon located at the top right over the selected parameters. Alternatively, you can double-click on the column in front of “Volumetric Flowrate” in the Parameter definition area. Note that “Volumetric Flowrate” becomes indented under “Pressure Static”, and that the black arrow next to “Volumetric Flowrate” turns to point towards the graph. See Figure 3.6 to see how these graphing parameters are defined.

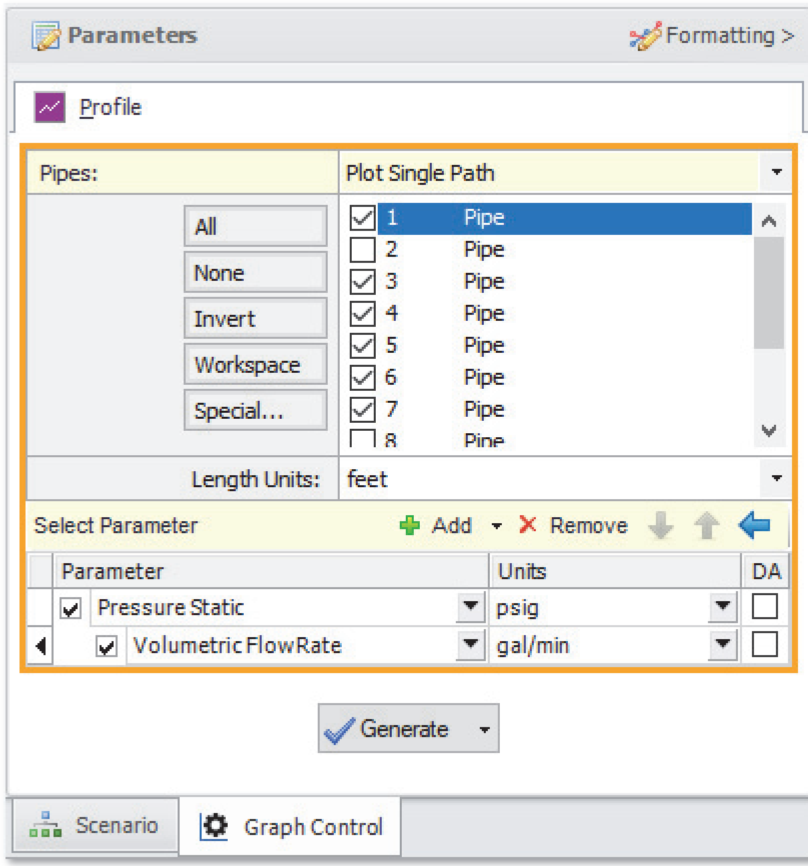


Figure 3.6 Parameter definition for dual Y-axes graph of static pressure (Primary Y-axis) and volumetric flowrate (Secondary Y-axis) on the Graph Control tab on the Quick Access Panel

Click the Generate button. You will see that the static pressure and volumetric flowrate drop, as expected, when some flow leaves the system through each spray nozzle. See the graph in Figure 3.7.

Note that you can also plot the same parameter with different units on the secondary Y-axis.

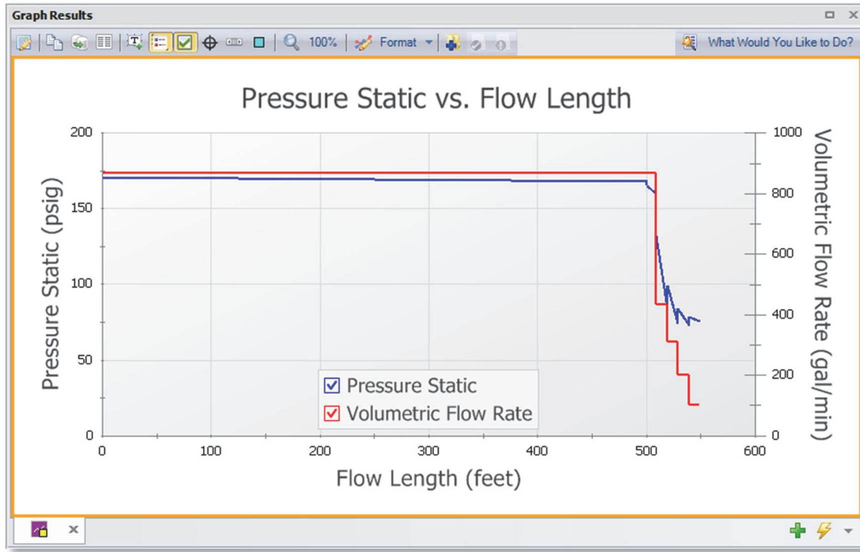


Figure 3.7 Pressure and volumetric flowrate from the constant pressure supply (J9) to the spray nozzle at one end of the model (J1)

Analysis summary

The minimum required supply pressure necessary to supply a volumetric flow rate of 100 gal/min to all of the spray junctions is about 171 psig, stagnation.

By iterating on values for the supply pressure, you were able to use AFT Fathom to determine the minimum required supply pressure for this system. This information can now be used to design the supply reservoir for this system.

The goal seeking capabilities in the optional AFT Fathom GSC Module eliminate the manual iteration required for this problem. The supply pressure would be set as a variable, and the minimum flow rate of 100 gal/min discharge for the spray discharges would be set as the goal (this is called a group max/min goal). GSC would then modify the supply pressure until the goal was met, thus eliminating the need to run AFT Fathom multiple times to arrive at the final solution.

More discussion on the GSC Module is given in Chapter 5.

Isometric Pipe Drawing Mode

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. Figure 3.8 shows the spray discharge system with the 2D grid displayed (select Show Grid from the Arrange menu).

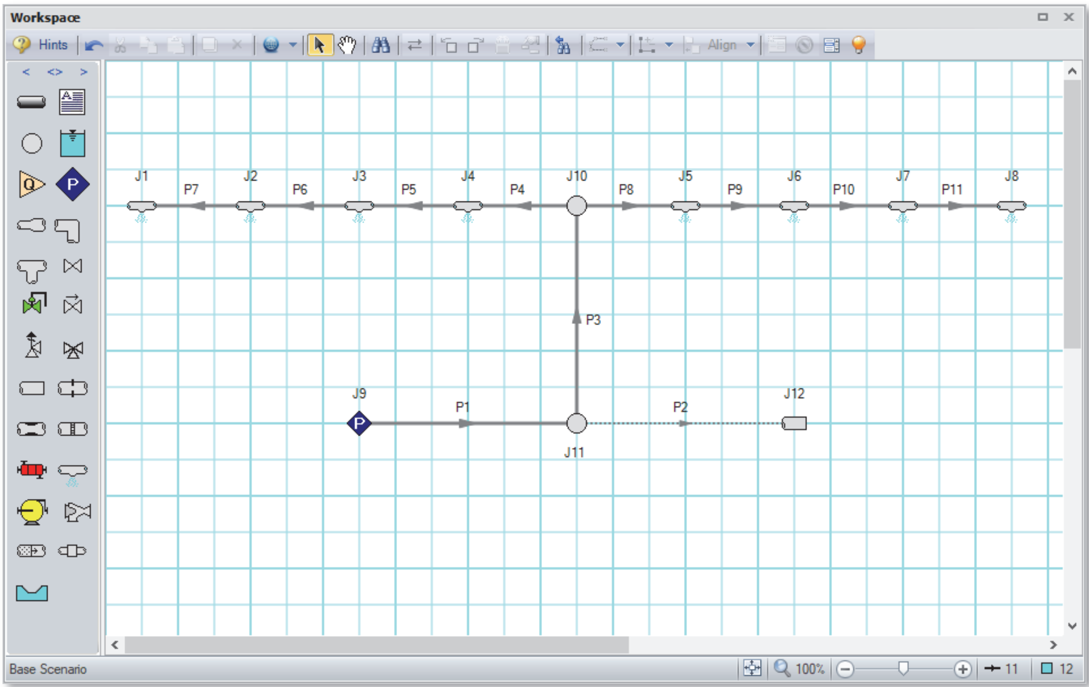


Figure 3.8 Spray discharge system with 2D grid shown

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 Pipe Drawing Mode for these cases. The isometric grid has three gridlines that are offset by 60°, representing the x, y, and z axes. Figure 3.9 shows the spray discharge system built on an isometric grid.

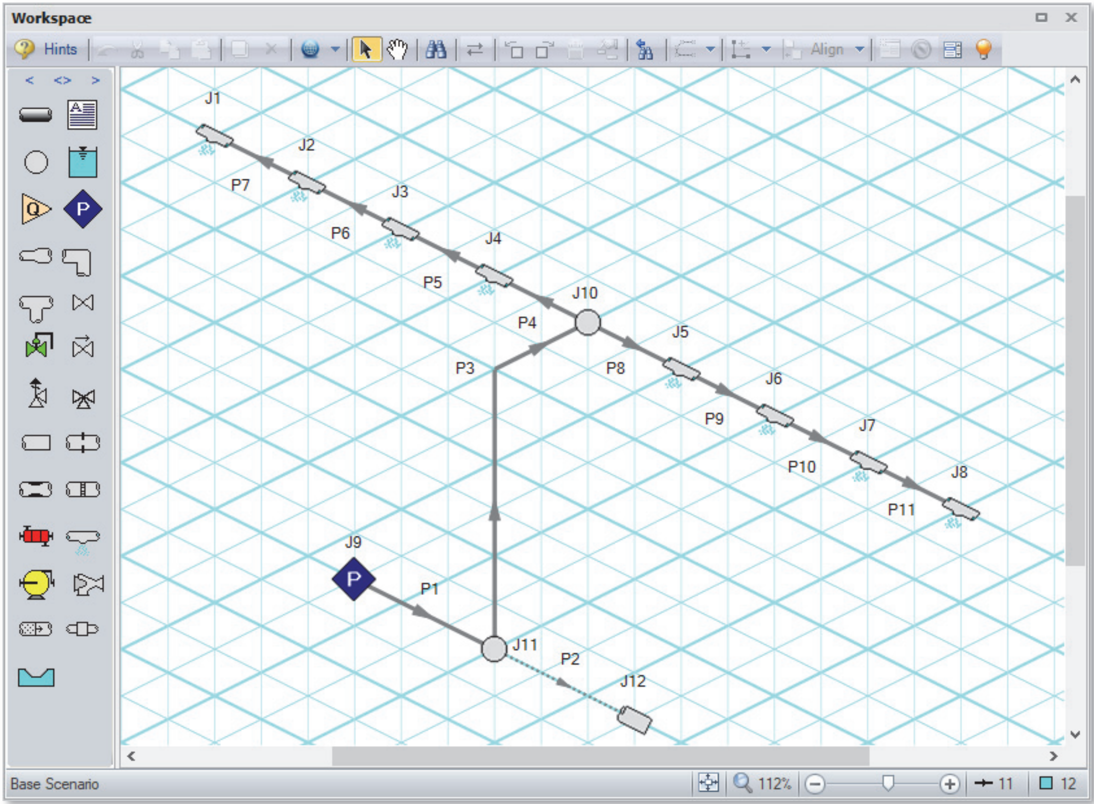


Figure 3.9 Spray discharge system with the isometric grid shown

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. From the Arrange menu, Show Grid and choose Isometric under Pipe Drawing Mode.
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 3.10.

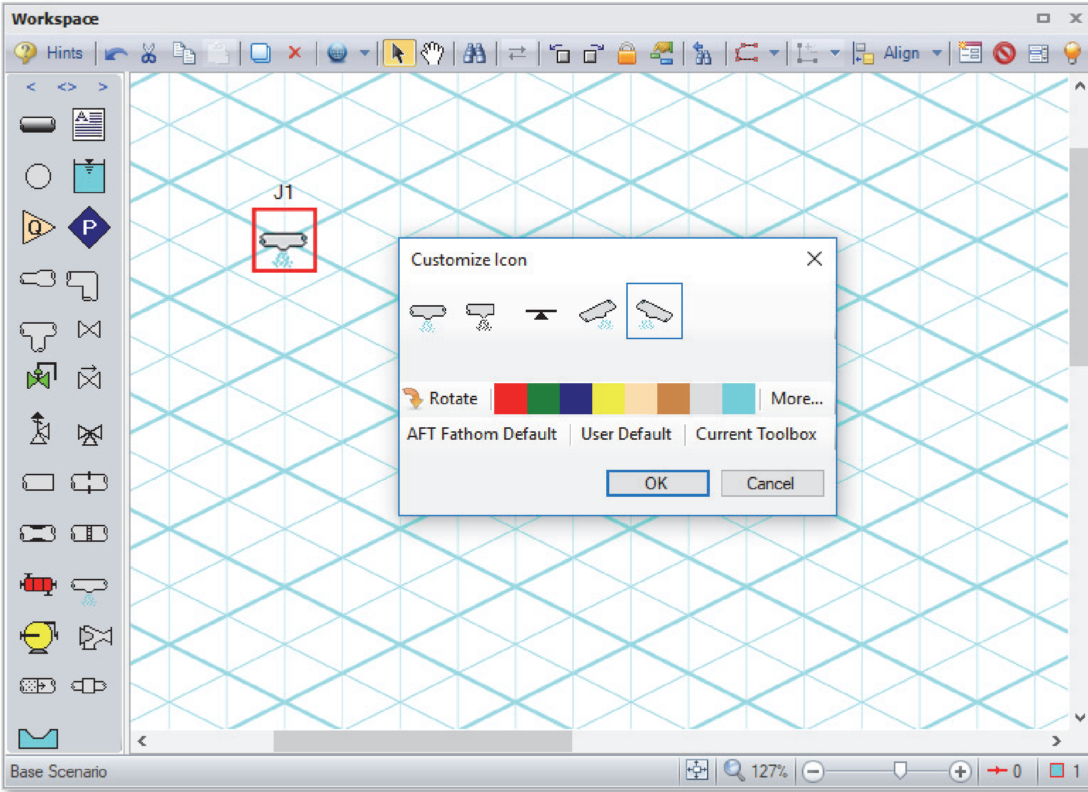


Figure 3.10 Right click on the junction to open the Customize Icon window and select the preferred icon and rotation

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 3.11.

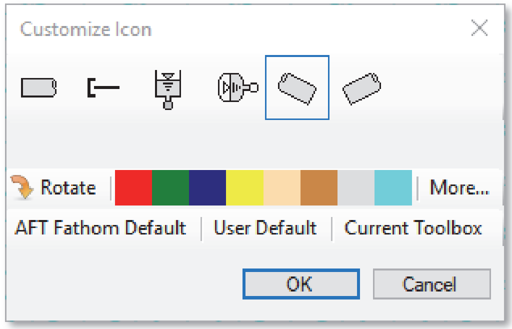


Figure 3.11 Use the Rotate button to obtain the preferred icon and rotation

10. Draw Pipes 1 and 2, as shown in Figure 3.9.
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 3.12 shows Pipe 3 being drawn with the preview line.

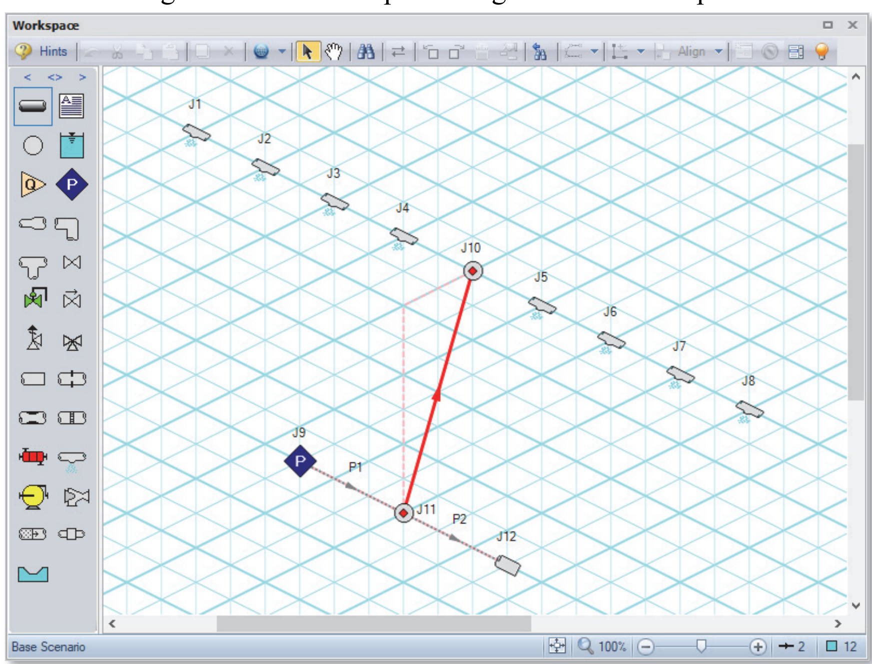


Figure 3.12 A preview line shows when drawing or adjusting pipes

Note: You can hold the "Alt" key while adjusting a pipe by the endpoint to add an additional segment. 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 3.9.
13. The grid can be shown or turned off in the Arrange menu.

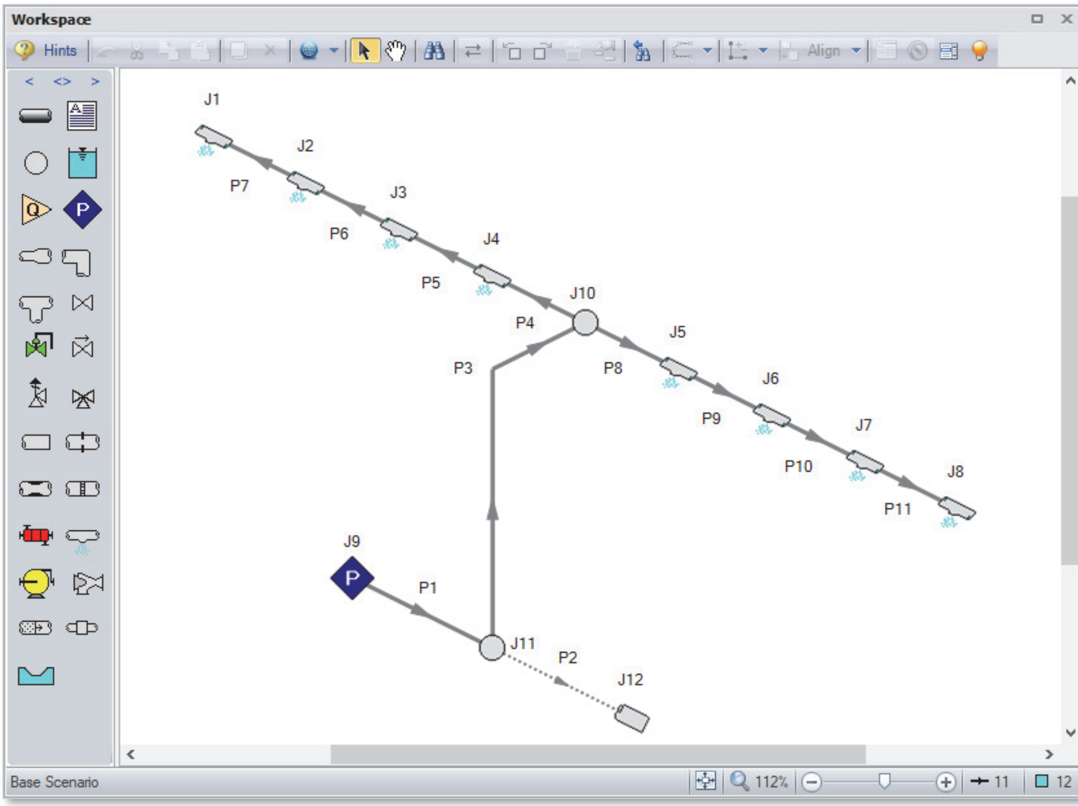


Figure 3.13 Isometric spray discharge system with the grid not shown

CHAPTER 4

Freon Delivery System with Heat Transfer Example

This example demonstrates how to use heat transfer modeling on pipes and heat exchangers. The example involves a review of the use of different refrigerant fluids to achieve certain design goals.

Topics covered

This example will cover the following topics:

- Using the NIST REFPROP database to define fluids
- Setting up a heat transfer model in System Properties
- Specifying convective heat transfer data for pipes
- Specifying thermal characteristics of heat exchangers
- Entering pump curves
- Entering resistance curves
- Using Scenario Manager

Required knowledge

This example assumes that the user has some familiarity with AFT Fathom such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Sizing a Pump Example in Chapter 2 for more information on these topics.

Model file

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

- *Freon System with HT.fth* - AFT Fathom model file

Problem statement

This example will model a Freon delivery system that is 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. You will also calculate the temperature of the hot water supply needed for the heat exchangers and select the refrigerant that requires the lowest water temperature.

Step 1. Start AFT Fathom

From the Start Menu choose AFT Products and AFT Fathom.

Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu (see Figure 4.1).
2. On the Fluid Data tab, select the NIST REFPROP database and then click Search and type “R11.” Select R11 in the Search Results.
3. Click “Add to Model” to select R11 for use in this model.
4. Now type in 150 in the fluid pressure box and -50 in the fluid temperature box. Select “psig” and "deg. F" for the units and click “Calculate Properties”. This calculates the default fluid properties to use in the model, although when modeling heat transfer AFT Fathom adjusts these properties during the simulation in accordance with calculated temperatures.
5. Select the “Heat Transfer With Energy Balance (Single Fluid)” option. This enables AFT Fathom's heat transfer capabilities.
6. Click OK.

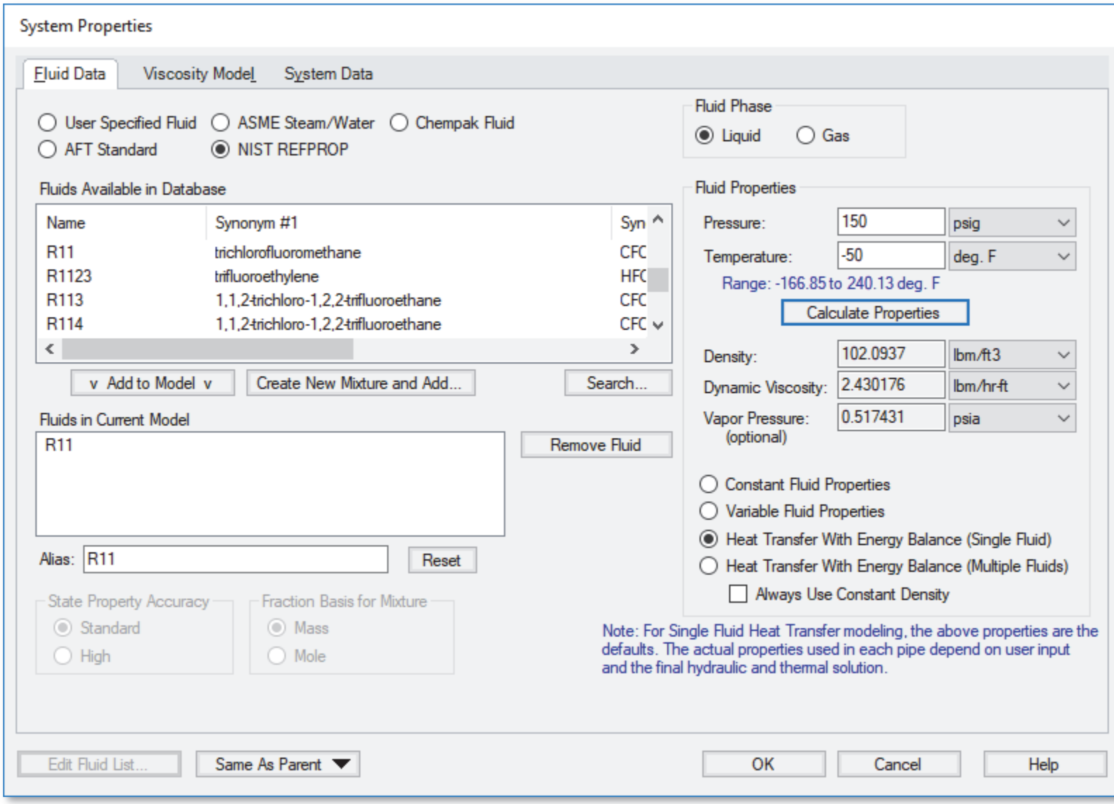


Figure 4.1 System Properties window with R11 selected and the "Heat Transfer with Energy Balance (Single Fluid)" selected.

Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to "Define Pipes and Junctions". In the Workspace window, assemble the model as shown in Figure 4.2.

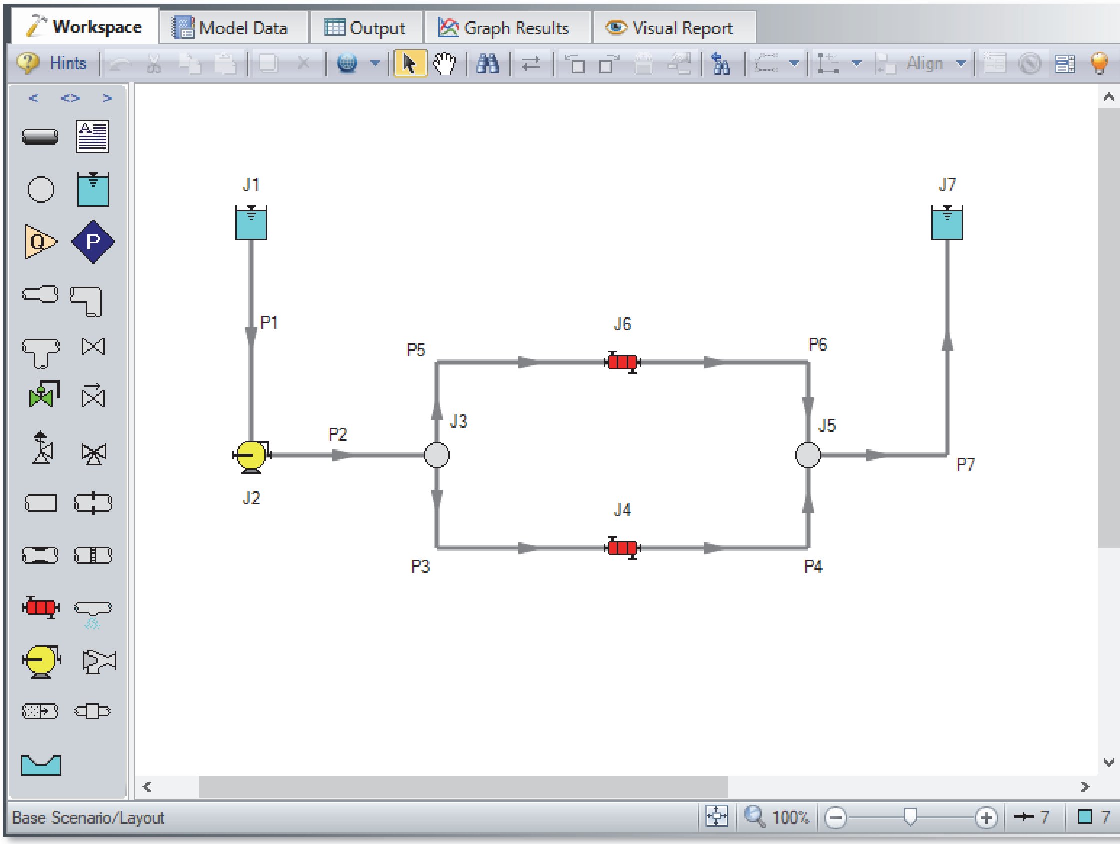


Figure 4.2 Layout of pipe system for Freon Delivery System with Heat Transfer Example. Pipes 3-7 have graphical bends that can be added using the Pipe Segment tool on the Arrange menu.

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Properties window (or use the Global Pipe Edit window, as discussed in Chapter 3).

Pipes P1-P7 are Steel - ANSI with standard roughness and the following data:

Pipe	Length (feet)	Size	Type
P1	10	3 inch	Schedule 40
P2	10	3 inch	Schedule 40
P3	5	3 inch	Schedule 40
P4	5	3 inch	Schedule 40
P5	5	3 inch	Schedule 40
P6	5	3 inch	Schedule 40
P7	30	3 inch	Schedule 40

Heat transfer data for piping is entered on the Heat Transfer tab on the Pipe Properties window. From the Heat Transfer tab, you first select the appropriate heat transfer model, then enter the required data for each type. For this example, you will be using the Convective Heat Transfer model.

Enter the following data on the Heat Transfer tab for each pipe, as shown for P1 in Figure 4.3:

1. Heat Transfer Model = Convective Heat Transfer
2. Number of Insulation Layers = 1 External
3. Ambient Temperature = 60 deg. F
4. Use default values for Fluid Internal and Pipe Wall
5. Insulation #1 (external) Data:
    a. Conductivity/Convection Data Source = User Specified
    b. Conductivity = 0.4 Btu/hr-ft-R
    c. Thickness = 1 inch
6. External Data:
    a. Conductivity/Convection Data Source = User Specified
    b. Convection Coefficient = 10 Btu/hr-ft^2-R

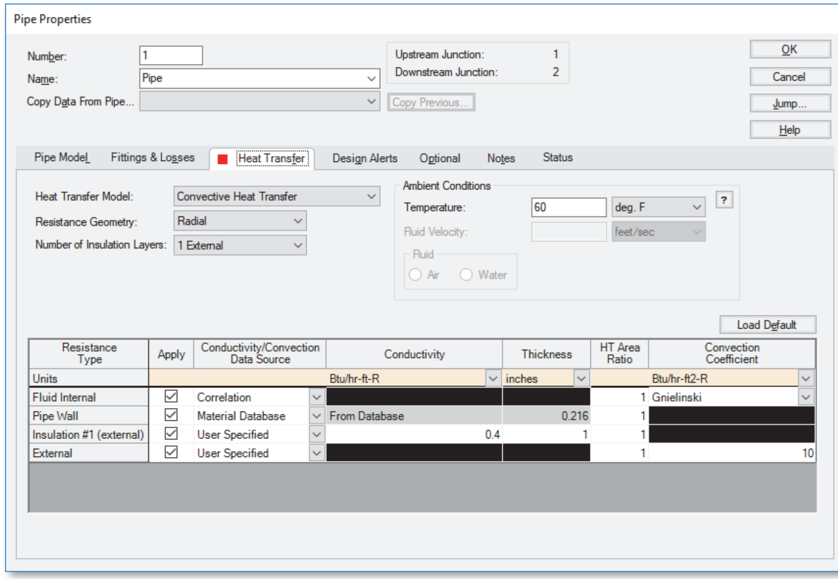


Figure 4.3 Pipe Heat Transfer tab for Freon Delivery System Example

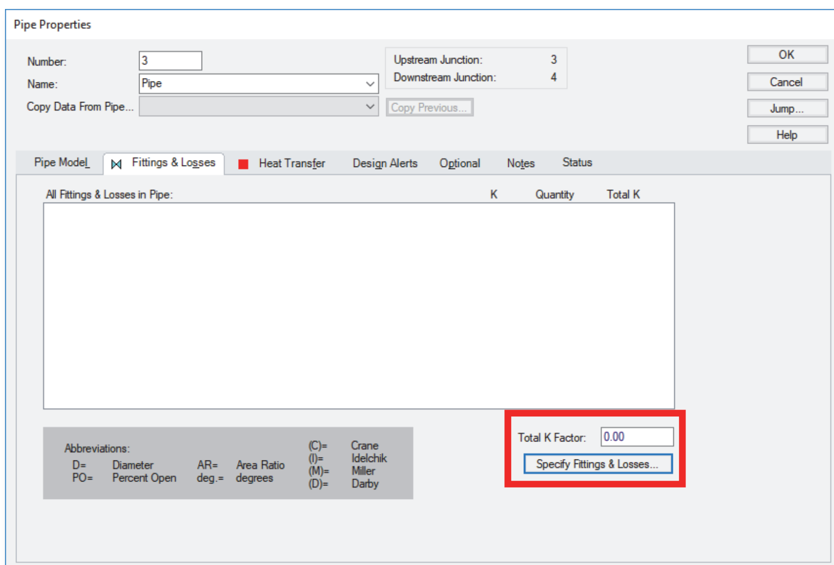


Figure 4.4 The Pipe Fittings & Losses Window is opened by clicking the Specify Fittings & Losses button on the Pipe Properties Window.

Note: This example uses a User Specified External Convection Coefficient. Alternatively, there are built-in Correlations available that will calculate free or forced convection coefficients for you.

Pipes P3, P4, P5, and P6 have an additional loss due to an elbow fitting. The losses due to these fittings are added by clicking the Specify Fittings & Losses button on the Fittings & Losses tab on the Pipe Property window (Figure 4.4). This opens the Pipe Fittings & Losses Window. Note these could have been added as junctions and the fitting losses entered there. The two approaches yield the same results and are offered for user flexibility.

For pipes P3-P6, open the Pipe Fittings & Losses Window, select the Bends tab, and enter a Quantity of 1 for a Standard Threaded, General, 90 deg bend, and then click the OK button to add this loss to each pipe (Figure 4.5). Global pipe editing could also be used to make this change to all four pipes simultaneously.

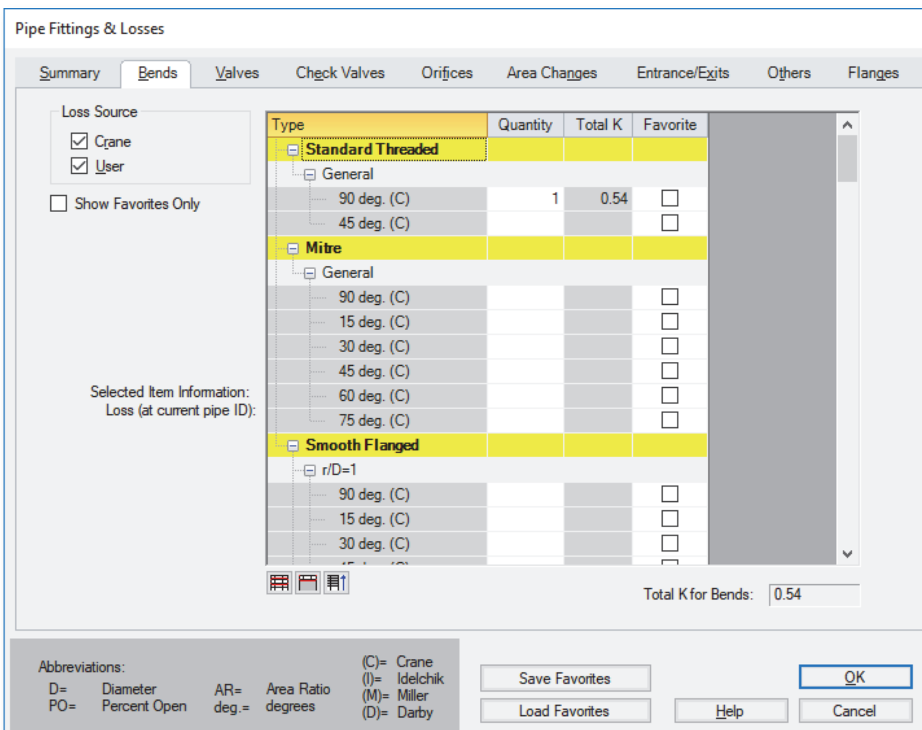


Figure 4.5 Fitting & Loss values are added to pipes by specifying fitting quantities on the Pipe Fittings & Losses Window.

C. Enter junction data

J1 - Supply Reservoir

1. Elevation = 10 ft
2. Surface pressure = 150 psig
3. Temperature = -50 F
4. Pipe depth = 0 feet

J7 - Receiving Reservoir

1. Elevation = 10 ft
2. Surface pressure = 150 psig
3. Temperature = 75 F
4. Pipe depth = 0 feet

J3, J5 - Branch Junctions

1. Elevation = 2 feet

J2 - Pump Junction

1. Elevation = 2 feet
2. Pump Model = Pump Curve
3. Enter the pump curve data (see Figure 4.6):
    a. Click the "Enter Curve Data" button
    b. On the Pump Configuration window, select the "Flow Parameter" as Volumetric, and the "Pressure Parameter" as Head.
    c. Enter the following pump curve data in the Raw Data table:

Volumetric Flow Rate (gal/min)	Head (feet)
0	25
200	24
500	15

d. Select "Generate Curve Fit Now" to create the curve fit.

e. Close the Pump Configuration window by clicking OK.

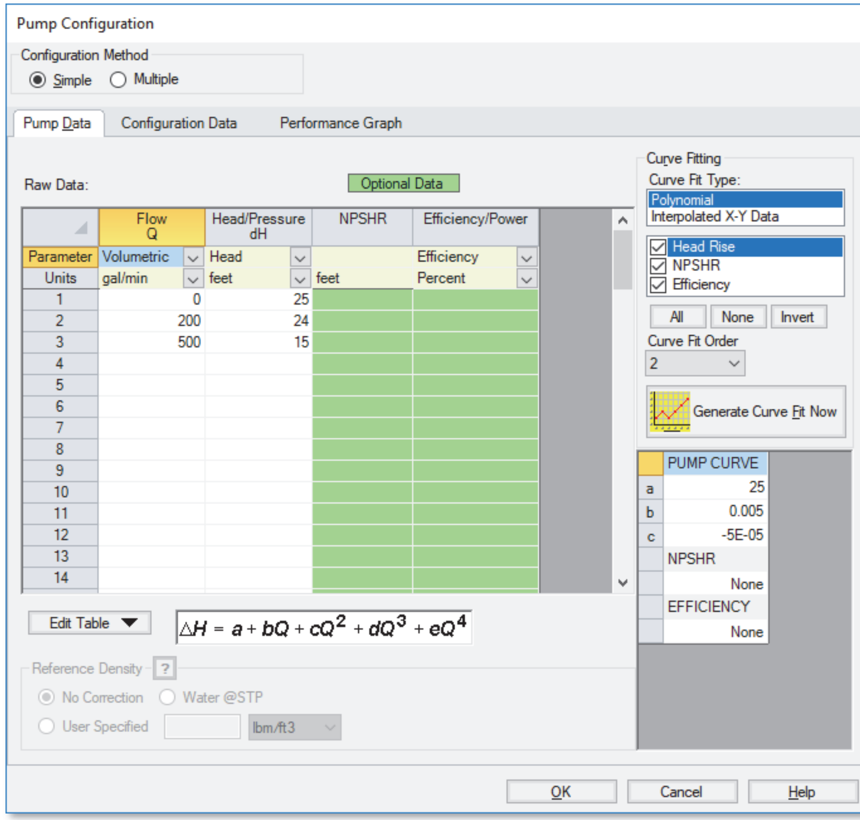


Figure 4.6 Pump Configuration Window for the Freon Delivery System Example

J4, J6 - Heat Exchangers

1. Elevation = 2 feet
2. Loss Model = Resistance Curve
3. Enter resistance curve data (see Figure 4.7):
   a. Click the "Enter Curve Data" button
   b. Select the "Flow Parameter" as Volumetric
   c. Enter 250 gpm and 2 psid in the Raw Data table

d. Click the “Fill as Quadratic” button to create data for a “square law” loss curve, which is typical of heat exchangers and other components.

e. Select “Generate Curve Fit Now” to create a quadratic curve fit

f. Close the Heat Exchanger Loss Curve Fit window by clicking OK.

4. Enter the following data on the Thermal Data tab:

a. Thermal Model = Counter-Flow

b. Heat Transfer Area = 80 feet^2

c. Overall Heat Transfer Coefficient = 500 Btu/hr-ft^2-R

d. Secondary Fluid Flow Rate = 30 lbm/sec

e. Secondary Fluid Specific Heat = 1 Btu/lbm-R

f. Secondary Fluid Inlet Temperature = 200 deg. F

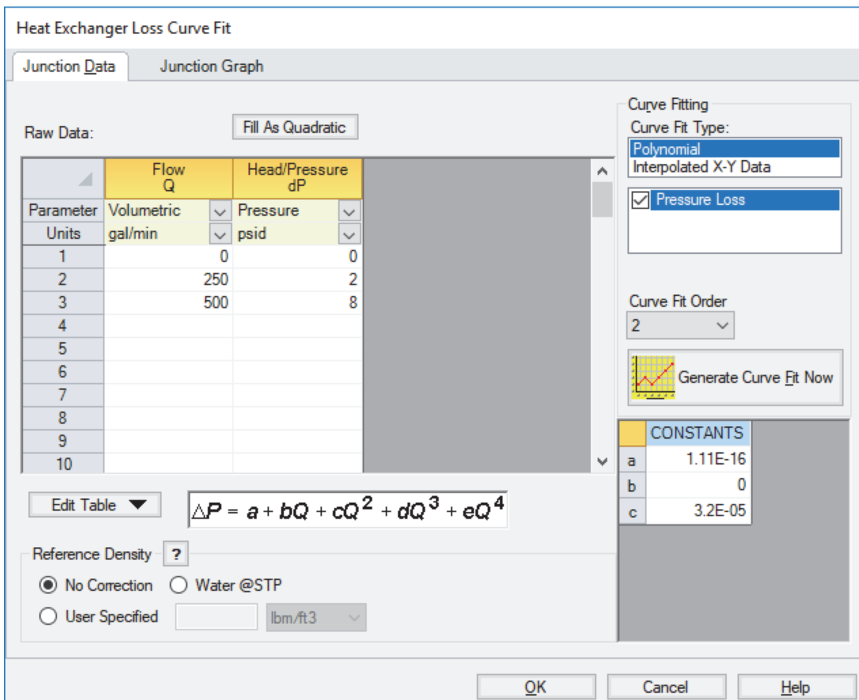


Figure 4.7 Heat Exchanger Loss Curve Fit Window for the Freon Delivery System Example

To solve this problem, you will have to guess the inlet temperature of the secondary fluid (hot water) for both heat exchangers. Start with 200 deg. F. You will then have to run the model, examine the results, and adjust the secondary fluid inlet temperature until you achieve the desired Freon outlet temperature of 75 deg. F.

D. Check if the pipe and junction data is complete

Turn on “Show Object Status” from the View menu to verify that all the necessary data is entered. If so, the “Define Pipes and Junctions” Checklist item on the Quick Access Panel 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. You can also open the List Undefined Objects window from the View menu to see what data is missing.

Step 4. Create a scenario for each refrigerant

For this problem, you are required to determine which refrigerant is the best one for this application by minimizing the temperature of the hot water used to heat the Freon to the delivery temperature. To do this, you will use the Scenario Manager to examine the different results for each refrigerant in the system.

The Scenario Manager is a powerful tool for managing variations of a model, referred to as scenarios. The Scenario Manager allows you to:

- Create, name and organize scenarios
- Select the scenario to appear in the Workspace (the ‘current’ scenario)
- Delete, copy and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario’s properties
- Pass changes from a scenario to its variants

You will create two scenarios to model these cases. Scenarios are created using the Scenario Manager. The Scenario Manager can be accessed from the Tools menu, or from the Quick Access Panel which is located on the right side of the Workspace area.

Click the "Create Child" button in the Scenario Manager on the Quick Access Panel. Name the child “R11”. A new scenario will appear below the Base Scenario in the list shown in the Scenario Panel. Select the Base Scenario, create another child, and call it “R123”. See Figure 4.8.

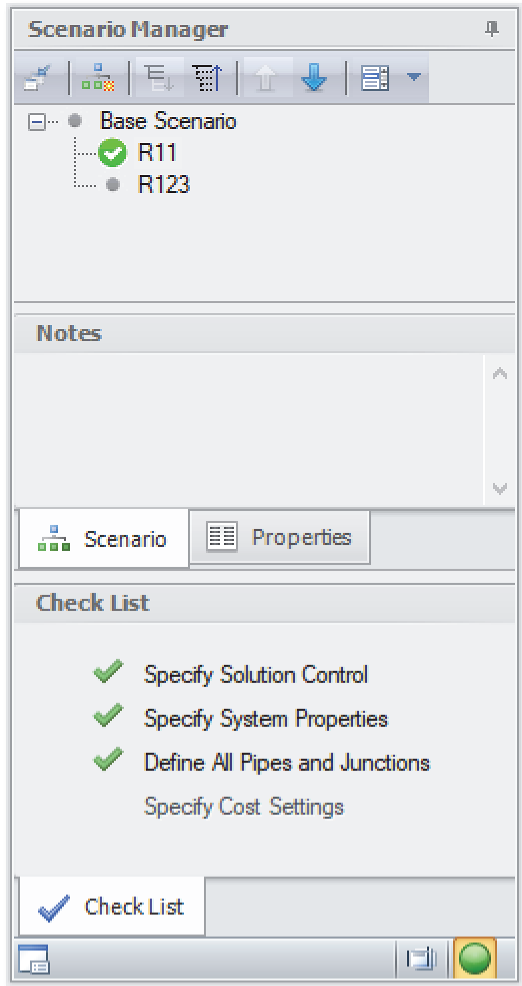


Figure 4.8 The Scenario Manager on the Quick Access Panel allows you to create model variants and keep them organized within the same model file

Step 5. Set up the R11 case

The base scenario contains all of the basic information for the model, such as the pipe and junction layout, pipe lengths, system properties etc. Now you must modify the system conditions for each scenario to use the refrigerant that will be analyzed for that case.

Child scenarios "inherit" data from their ancestors. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent. In this example, you entered R11 as the fluid in the System Properties window in the Base Scenario, so the R11 scenario has inherited the same fluid properties as the Base Scenario. Therefore, no changes need to be made to the R11 scenario.

Double-click the "R11" scenario in the list in the Scenario Manager on the Quick Access Panel. This will load the R11 scenario into the Workspace as the current scenario. The currently loaded scenario is displayed in the list with a green check mark (Figure 4.8).

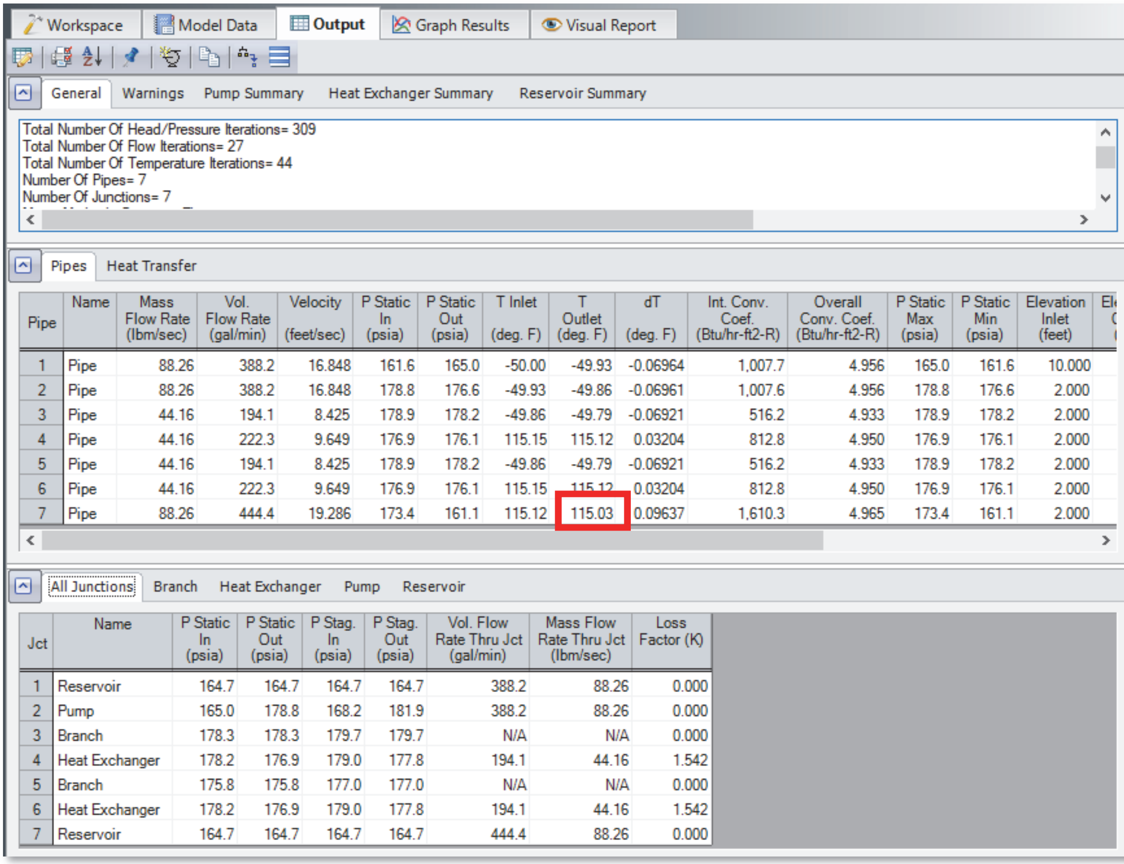
Step 6. Run the R11 case

Starting with the initial guess of 200 deg. F for the Secondary Fluid Temperature, select Run Model in the Analysis menu. This will launch the Solution Progress window. This window allows you to watch as the AFT Fathom Solver converges on the answer. Once the model has converged, you can view the results by pressing View Output at the bottom of the Solution Progress window.

Step 7. Examine the R11 results

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

Examination of the R11 delivery temperature at the outlet of pipe P7 shows a temperature of 115 deg. F (see Figure 4.9, also visible with additional detail in Figure 4.10 and 4.11). This is well above the required minimum delivery temperature of 75 deg. F. The inlet temperature of the Secondary Fluid must be decreased, and the model must be run again to determine the outlet temperature at the new value. This process must be repeated until the Secondary Fluid Inlet Temperature that delivers the Freon at 75 deg. F is determined.



Workspace | Model Data | Output | Graph Results | Visual Report

General | Warnings | Pump Summary | Heat Exchanger Summary | Reservoir Summary

Total Number Of Head/Pressure Iterations= 309
Total Number Of Flow Iterations= 27
Total Number Of Temperature Iterations= 44
Number Of Pipes= 7
Number Of Junctions= 7

Pipes | Heat Transfer

Pipe	Name	Mass Flow Rate (lbm/sec)	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static In (psia)	P Static Out (psia)	T Inlet (deg. F)	T Outlet (deg. F)	dT (deg. F)	Int. Conv. Coef. (Btu/hr-ft2-R)	Overall Conv. Coef. (Btu/hr-ft2-R)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)
1	Pipe	88.26	388.2	16.848	161.6	165.0	-50.00	-49.93	-0.06964	1,007.7	4.956	165.0	161.6	10.000
2	Pipe	88.26	388.2	16.848	178.8	176.6	-49.93	-49.86	-0.06961	1,007.6	4.956	178.8	176.6	2.000
3	Pipe	44.16	194.1	8.425	178.9	178.2	-49.86	-49.79	-0.06921	516.2	4.933	178.9	178.2	2.000
4	Pipe	44.16	222.3	9.649	176.9	176.1	115.15	115.12	0.03204	812.8	4.950	176.9	176.1	2.000
5	Pipe	44.16	194.1	8.425	178.9	178.2	-49.86	-49.79	-0.06921	516.2	4.933	178.9	178.2	2.000
6	Pipe	44.16	222.3	9.649	176.9	176.1	115.15	115.12	0.03204	812.8	4.950	176.9	176.1	2.000
7	Pipe	88.26	444.4	19.286	173.4	161.1	115.12	115.03	0.09637	1,610.3	4.965	173.4	161.1	2.000

All Junctions | Branch | Heat Exchanger | Pump | Reservoir

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	Reservoir	164.7	164.7	164.7	164.7	388.2	88.26	0.000
2	Pump	165.0	178.8	168.2	181.9	388.2	88.26	0.000
3	Branch	178.3	178.3	179.7	179.7	N/A	N/A	0.000
4	Heat Exchanger	178.2	176.9	179.0	177.8	194.1	44.16	1.542
5	Branch	175.8	175.8	177.0	177.0	N/A	N/A	0.000
6	Heat Exchanger	178.2	176.9	179.0	177.8	194.1	44.16	1.542
7	Reservoir	164.7	164.7	164.7	164.7	444.4	88.26	0.000

Figure 4.9 Output for Freon Delivery System for R11 with a Secondary Fluid Inlet Temperature of 200 deg. F

Pipes | Heat Transfer

Pipe	Results Diagram	Name	T Fluid Inlet (deg. F)	T Fluid Outlet (deg. F)	T Inlet Wall Inside (deg. F)	T Outlet Wall Inside (deg. F)	T Inlet Wall Outside (deg. F)	T Outlet Wall Outside (deg. F)	T Ambient (deg. F)
1	Show ...	Pipe	-50.00	-49.93	-49.46	-49.39	-49.16	-49.09	60.00
2	Show ...	Pipe	-49.93	-49.86	-49.39	-49.32	-49.09	-49.02	60.00
3	Show ...	Pipe	-49.86	-49.79	-48.81	-48.74	-48.52	-48.45	60.00
4	Show ...	Pipe	115.15	115.12	114.82	114.79	114.66	114.63	60.00
5	Show ...	Pipe	-49.86	-49.79	-48.81	-48.74	-48.52	-48.45	60.00
6	Show ...	Pipe	115.15	115.12	114.82	114.79	114.66	114.63	60.00
7	Show ...	Pipe	115.12	115.03	114.95	114.86	114.80	114.70	60.00

Figure 4.10 Heat Transfer tab in Output window shows detailed heat transfer results for pipes

The pipe heat transfer results from the Heat Transfer tab (see Figure 4.10) can also be displayed in a Results Diagram, as shown in Figure 4.11, by selecting the button next to Show in the Results Diagram column, double-clicking anywhere in the row, or right-clicking anywhere in the row and selecting Results Diagram.

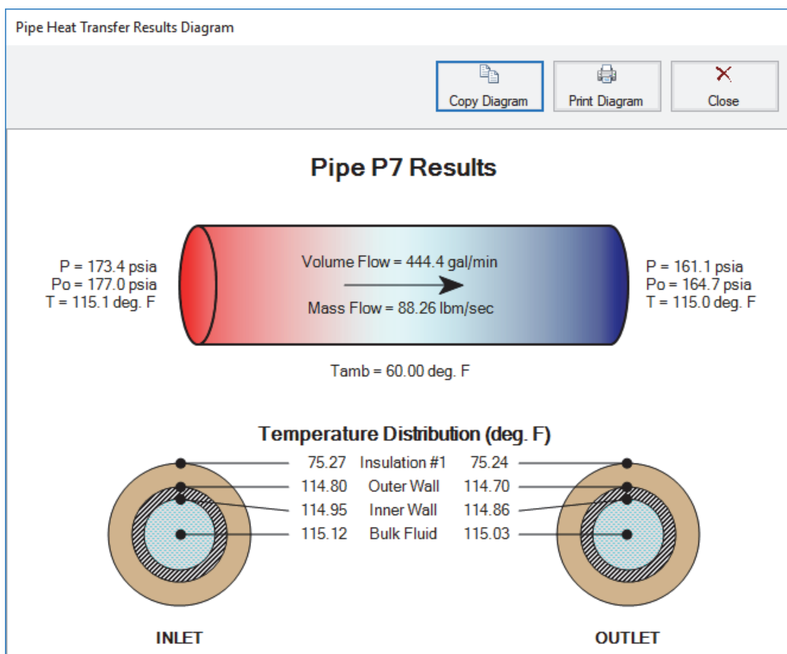


Figure 4.11 Pipe Heat Transfer Results Diagram Opened from the Heat Transfer Table (see Fig. 4.10)

Step 8. Set up the R123 case

Double-click the R123 scenario in the list in the Scenario Manager on the Quick Access Panel to load it as the current scenario.

Open the System Properties window from the Analysis menu and select R123 similar to Step 2.

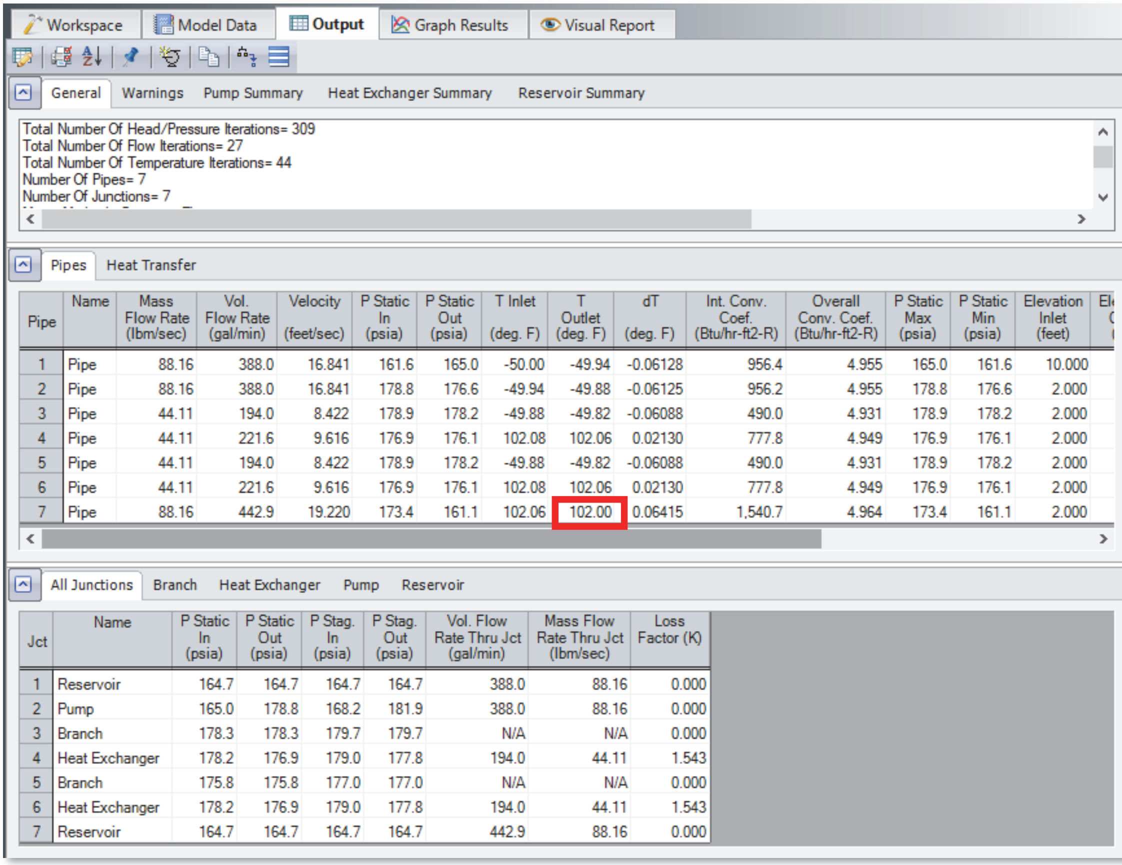
Step 9. Run the R123 case

Starting with the initial guess of 200 deg. F for the Secondary Fluid Temperature, select Run Model in 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. Once the model has converged, you can view the results by pressing View Output at the bottom of the Solution Progress window.

Step 10. Examine the R123 results

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

Examination of the R123 delivery temperature at the outlet of pipe P7 shows a temperature of 102 deg. F (see Figure 4.11). This is well above the required minimum delivery temperature of 75 deg. F. The inlet temperature of the Secondary Fluid must be decreased, and the model must be run again to determine the outlet temperature at the new value. This process must be repeated until the Secondary Fluid Inlet Temperature that delivers the Freon at 75 deg. F is determined.



Pipe	Name	Mass Flow Rate (lbm/sec)	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static In (psia)	P Static Out (psia)	T Inlet (deg. F)	T Outlet (deg. F)	dT (deg. F)	Int. Conv. Coef. (Btu/hr-ft2-R)	Overall Conv. Coef. (Btu/hr-ft2-R)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)
1	Pipe	88.16	388.0	16.841	161.6	165.0	-50.00	-49.94	-0.06128	956.4	4.955	165.0	161.6	10.000
2	Pipe	88.16	388.0	16.841	178.8	176.6	-49.94	-49.88	-0.06125	956.2	4.955	178.8	176.6	2.000
3	Pipe	44.11	194.0	8.422	178.9	178.2	-49.88	-49.82	-0.06088	490.0	4.931	178.9	178.2	2.000
4	Pipe	44.11	221.6	9.616	176.9	176.1	102.08	102.06	0.02130	777.8	4.949	176.9	176.1	2.000
5	Pipe	44.11	194.0	8.422	178.9	178.2	-49.88	-49.82	-0.06088	490.0	4.931	178.9	178.2	2.000
6	Pipe	44.11	221.6	9.616	176.9	176.1	102.08	102.06	0.02130	777.8	4.949	176.9	176.1	2.000
7	Pipe	88.16	442.9	19.220	173.4	161.1	102.06	102.00	0.06415	1,540.7	4.964	173.4	161.1	2.000

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	Reservoir	164.7	164.7	164.7	164.7	388.0	88.16	0.000
2	Pump	165.0	178.8	168.2	181.9	388.0	88.16	0.000
3	Branch	178.3	178.3	179.7	179.7	N/A	N/A	0.000
4	Heat Exchanger	178.2	176.9	179.0	177.8	194.0	44.11	1.543
5	Branch	175.8	175.8	177.0	177.0	N/A	N/A	0.000
6	Heat Exchanger	178.2	176.9	179.0	177.8	194.0	44.11	1.543
7	Reservoir	164.7	164.7	164.7	164.7	442.9	88.16	0.000

Figure 4.11 Output for Freon Delivery System for R123 with a Secondary Fluid Inlet Temperature of 200 deg. F

Analysis summary

The results of the iterative solutions show the following:

Refrigerant	Secondary Fluid Inlet Temp (deg F)
R11	138.5
R123	154.6

In terms of heat exchanger hot water temperature, R11 is the better choice for a refrigerant delivery temperature of 75 deg. F. However, there may be other factors, such as environmental regulations, that must be considered in the selection of a refrigerant.

Similar to the spray system in Chapter 3, this example requires manual iteration to obtain the results. The goal seeking capabilities in the optional AFT Fathom GSC Module would allow this problem to be solved directly without manual iteration. The variables would be specified as the heat exchanger secondary inlet temperature (these variables would be linked), and the goal would be the pipe exit temperature.

More discussion on the GSC Module is given in Chapter 5.

CHAPTER 5

AFT Fathom Add-on Modules Examples

This chapter covers two examples. This first example demonstrates how to use the Goal Seek and Control (GSC) and Extended Time Simulation (XTS) optional add-on modules to AFT Fathom. Later in the chapter, a second example using the Settling Slurry (SSL) module will be covered.

The Automated Pipeline Sizing (APS) and Automated Network Sizing (ANS) modules are covered in a separate Quick Start Guide.

The user can only perform these examples if access to the relevant module is available.

Topics covered

These examples will cover the following topics:

- Goal Seek and Control Manager
- Defining GSC Variables and Goals
- XTS Transient Control
- Defining system transients
- Defining settling slurry properties

Required knowledge

This example assumes that the user has some familiarity with AFT Fathom such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Sizing a Pump Example in Chapter 2 for more information on these topics. To model the heat transfer in the

GSC example, the user may need to refer to Chapter 4 for more detailed explanations on modeling heat transfer.

Model files

These examples use the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *Controlled HX Temperature.fth* - AFT Fathom model file (GSC and XTS Examples)
- *Pump Sizing for Sand Transfer System - SSL.fth* - AFT Fathom model file (SSL Example)

Problem statements

GSC Example

A heat exchanger system has a bypass line that is used to divert flow around the heat exchangers. This is sometimes necessary to maintain a constant downstream temperature of 120 deg. F. Use the GSC module to determine the valve open percentage required to maintain the proper delivery temperature for a supply temperature of 200 deg. F.

XTS Example

The discharge reservoir is a finite sized tank, 20 feet high, with a cross sectional area of 80 ft^2. 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?

SSL Example

Use the settling slurry module to size a slurry pump. 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.

Step 1. Start AFT Fathom

From the Start Menu choose AFT Products and AFT Fathom.

Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu
2. On the Fluid Data tab, select the AFT Standard database and then select “Water at 1 atm” in the Fluids Available in Database list
3. Click “Add to Model” to select water for use in this model
4. Type in 70 in the fluid temperature box and specify the units as “degrees F”
5. Click “Calculate Properties”
6. Select “Heat Transfer With Energy Balance (Single Fluid)”
7. Select OK

Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to “Define Pipes and Junctions”. In the Workspace window, assemble the model as shown in Figure 5.1.

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Properties window (or use the Global Pipe Editing window, as discussed in Chapter 3).

Pipes P1-P10 are Steel - ANSI with standard roughness and the following data:

Pipe	Length (feet)	Size	Type
P1	20	8 inch	Schedule 40
P2	5	8 inch	Schedule 40
P3	20	8 inch	Schedule 40
P4	20	8 inch	Schedule 40
P5	20	8 inch	Schedule 40
P6	20	8 inch	Schedule 40
P7	20	8 inch	Schedule 40
P8	20	8 inch	Schedule 40
P9	200	8 inch	Schedule 40
P10	100	8 inch	Schedule 40

All of the pipes are very well insulated, so select the Adiabatic option on the Heat Transfer tab.

C. Enter the junction data

J1, J9 - Reservoirs

1. Tank Model = Infinite Reservoir (only visible if XTS module is enabled)
2. Liquid Surface Pressure = 40 psig
3. Liquid Surface Elevation = 10 feet
4. Liquid Temperature = 200 F
5. Update Density On Temperature Changes = Checked (the Liquid Density will be calculated automatically to 60.10897 lbm/ft3)
6. Pipe Depth = 0 feet

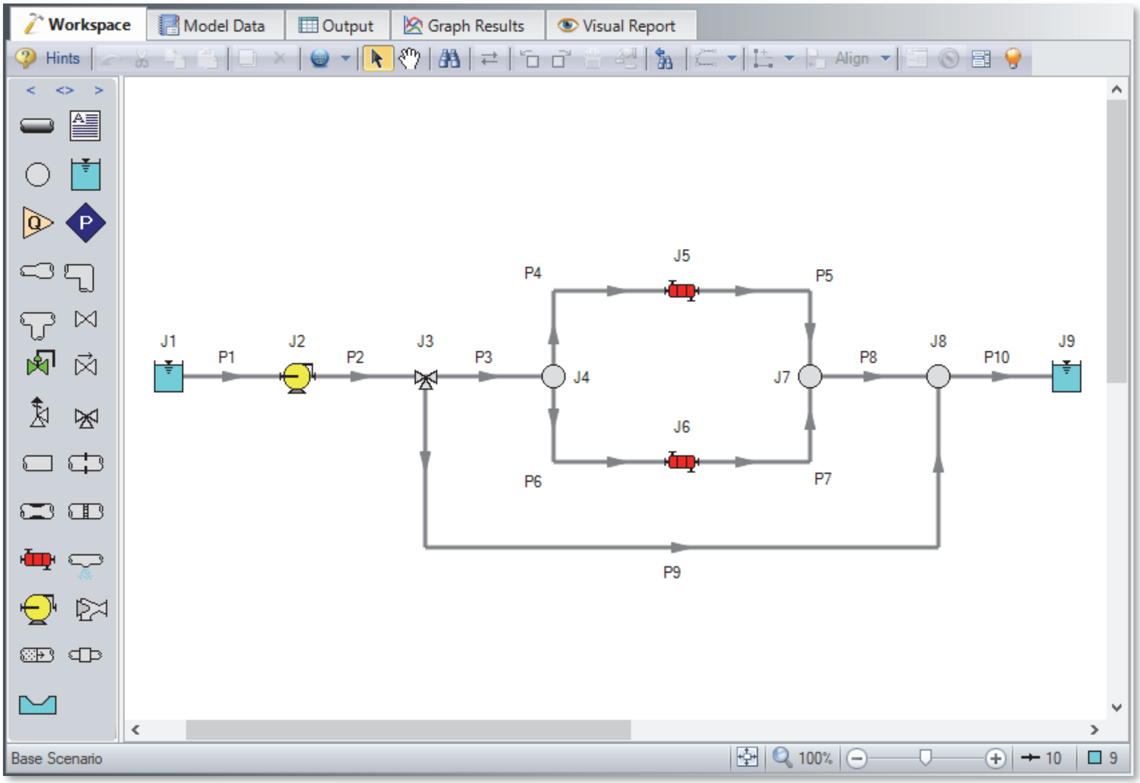


Figure 5.1 Layout of pipe system for Controlled Heat Exchanger Temperature Example

J2 - Pump

1. Elevation = 0 feet
2. Type = Pump Curve
3. Pump Curve Data:

Q (gal/min)	dH (feet)
0	70
500	60
1000	40

J3 - Three-Way Valve

1. Elevation = 0 feet
2. Combined Flow Pipe = P2
3. Split Flow Pipe #1 = P3
4. Split Flow Pipe #2 = P9
5. Valve Cv Data (ignore area data):

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

6. Actual Percent Open = 75%

J4, J7, J8 - Branch Junctions

Elevation = 0 feet

J5, J6 - Heat Exchangers

1. Elevation = 0 feet
2. Loss Model = K Factor (Constant)
3. K = 1400
4. Thermal Model = Counter-Flow
5. Heat Transfer Area = 1600 ft^2
6. Overall Heat Transfer Coefficient = 50 Btu/hr-ft^2-R
7. Secondary Fluid Flow Rate = 200 lbm/sec
8. Secondary Fluid Specific Heat = 1 Btu/lbm-R
9. Secondary Fluid Inlet Temperature = 40 F

D. Check if the pipe and junction data is complete

Turn on “Show Object Status” from the View menu to verify that all the necessary data is entered. If so, the “Define Pipes and Junctions” Checklist item on the Quick Access Panel 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. You can also open the List Undefined Objects window from the View menu to see what data is missing.

GSC problem statement

Use the Goal Seek and Control module to determine the valve open percentage required to maintain a 120 deg. F temperature at branch junction J9 for a reservoir supply temperature of 200 deg. F.

GSC Step 1. Create a scenario for GSC case

Create a new scenario from the Base Scenario for the GSC Case. Refer to Chapter 4 for directions on how to add a scenario.

GSC Step 2. Activate the GSC Module

Activate the GSC Module by choosing Activate Modules from the Tools menu and selecting the GSC Module in the list.

GSC Step 3. Open the Goal Seek and Control Manager

When the GSC Module is activated, Specify Goal Seek and Control appears as an additional Checklist item.

The Goal Seek and Control is specified by entering data for variables and goals in the Goal Seek and Control Manager window. Open the Goal Seek and Control Manager from the Analysis Menu. After opening the Goal Seek and Control Manager, the user specifies all of the system variables, as well as the desired goals. The Goal Seek and Control Manager is shown in Figure 5.2 below.

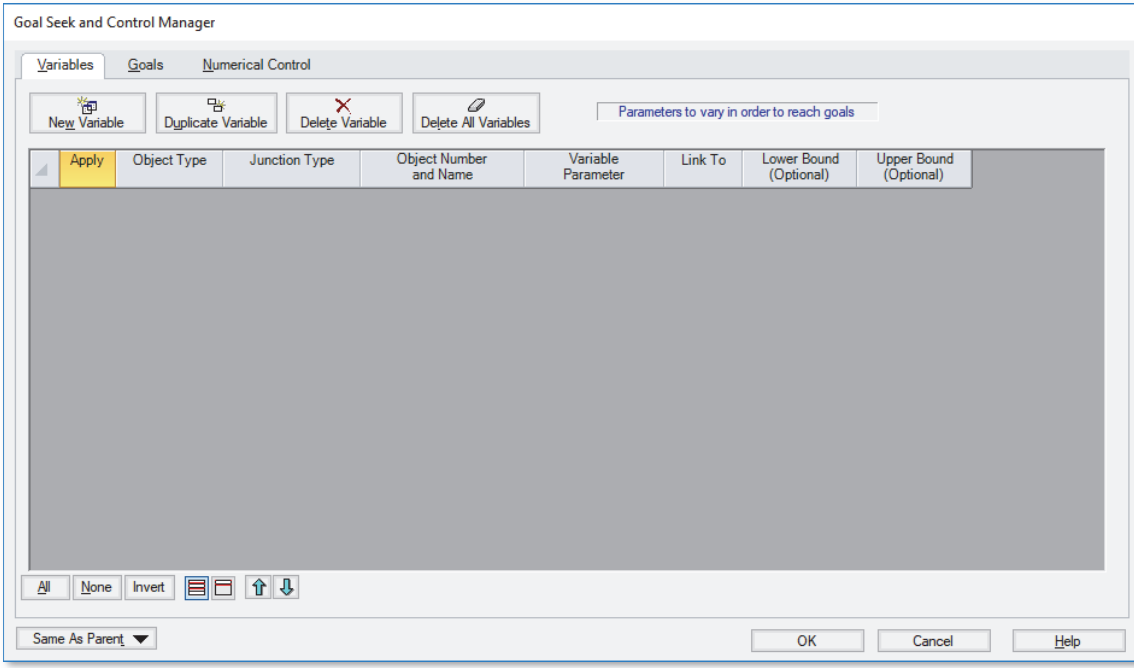


Figure 5.2 The Goal Seek and Control Manager is used to define GSC Variables and Goals

GSC Step 4. Add a variable

In the GSC module, variables are the parameters that AFT Fathom will modify in order to achieve the specified goals. To obtain a unique answer there should be one variable applied for each goal. You may define as many variables and goals as you wish.

Select the Variables tab on the Goal Seek and Control Manager window. The Variables tab allows users to create and modify the system variables. On the Variables tab, you are able to select the object and junction types, the name and number of the object to which the variable applies, and the object parameter that is to be varied. Additionally, you can elect to apply certain variables while omitting others.

For this example, you will be adding a variable for the Three-Way Valve Percent Open. Select the “New Variable” button, and input the following variable 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 and Upper Bounds: Use defaults provided

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 the lower bound as 0 %, and the upper bound as 100 %.

After entering the data, the Variable tab should appear as shown in Figure 5.3.

GSC Step 5. Add a goal

Goals are the parameters you want to achieve. The goals are achieved as AFT Fathom modifies the variables. To obtain a unique answer there should be one goal applied for each variable. You may define as many variables and goals as you wish.

Select the Goals tab on the Goal Seek and Control Manager window. The Goals tab allows users 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. The user then selects the object to which the goal applies, and, if applicable, the location on the object at which the goal applies (e.g., the inlet or outlet of a pipe object).

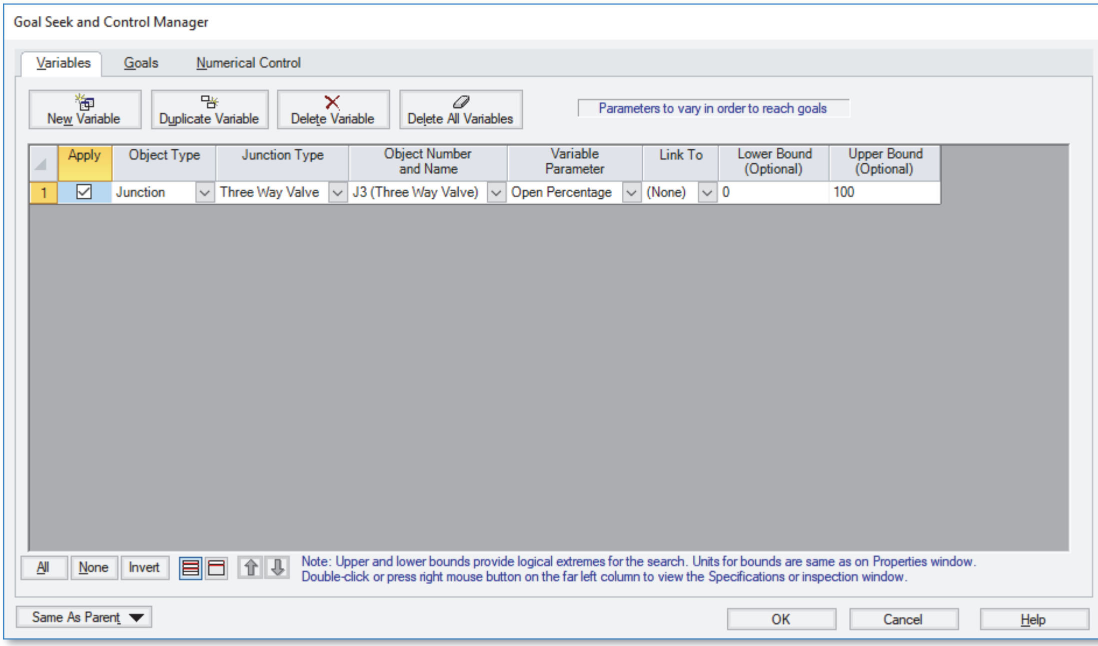


Figure 5.3 GSC Variables are parameters that are changed by AFT Fathom to achieve the defined goals

Create a new goal for the downstream temperature as defined 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 tab should appear as shown in Figure 5.4.

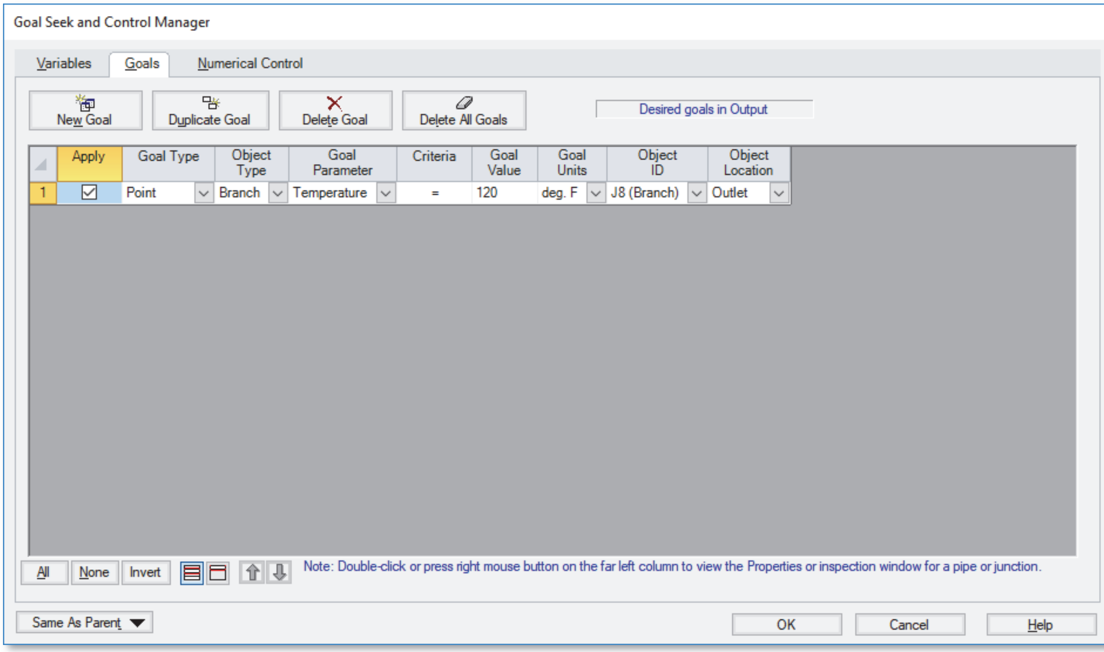


Figure 5.4 GSC Goals are the parameter values set by the user that AFT Fathom adjusts the variables to achieve

As variables and goals are added to a model, AFT 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. This is illustrated in Figure 5.5.

Note: The goal symbol is not displayed next to objects that are part of a group goal.

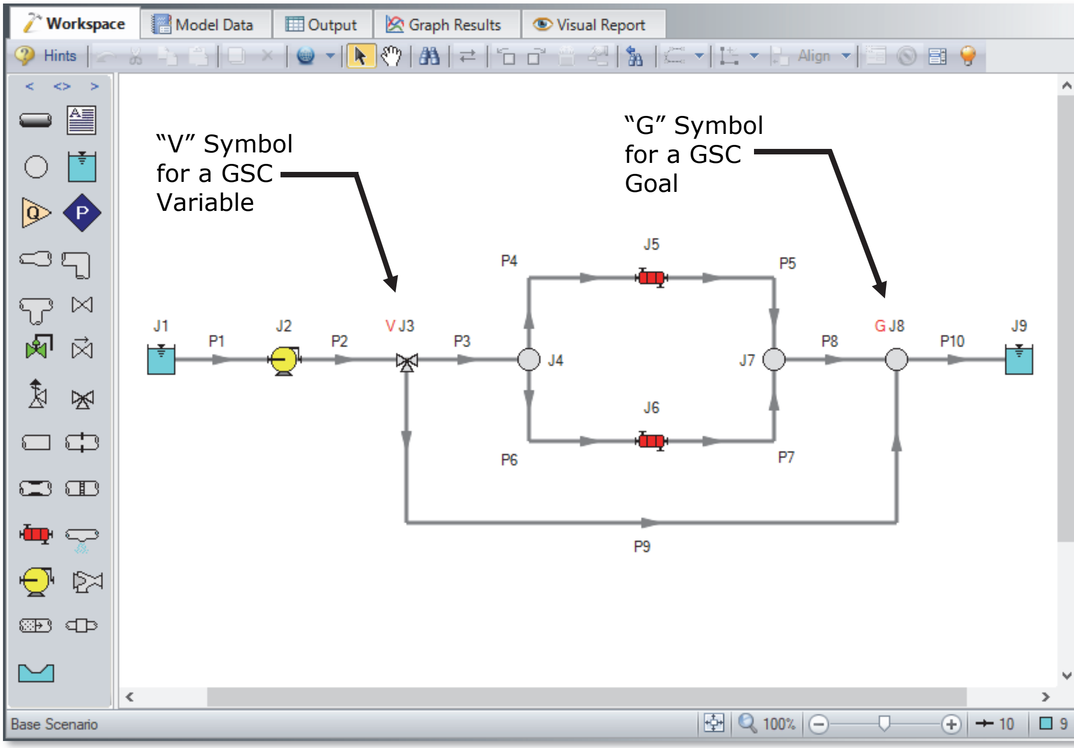


Figure 5.5 AFT Fathom displays symbols next to objects on the Workspace that have goals or variables defined

GSC Step 6. Activate GSC analysis

After the GSC goals and variables have been defined, GSC must be enabled using the Analysis menu, as shown in Figure 5.6.

GSC Step 7. Run the model

Select Run Model in 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.

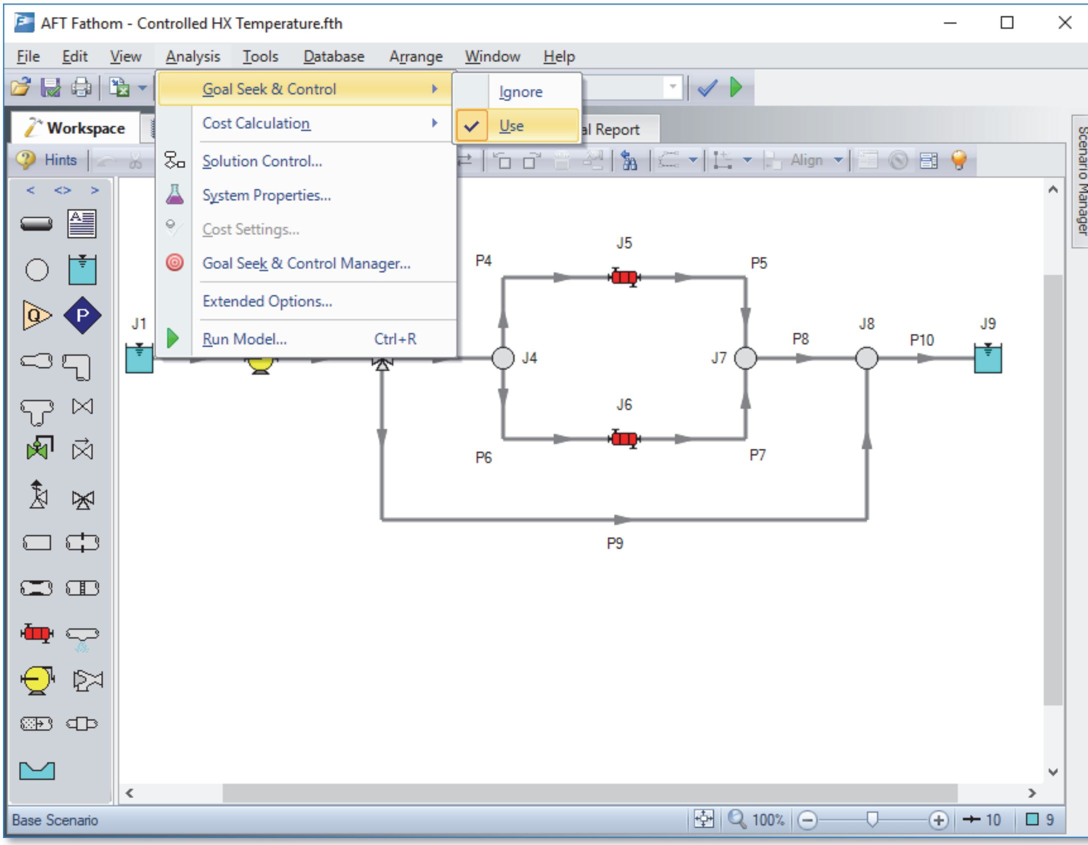


Figure 5.6 Select "Use" from the Goal Seek & Control menu item on the Analysis menu to instruct AFT Fathom to use goal seeking when it runs

Note: When using the GSC module, there is a new area displayed in Solution Progress that shows the specific progress of the GSC module. As it makes progress, the Best (Lowest) value will decrease towards zero. The field in the far right displays how many complete hydraulic solutions have been run.

After completion, click the View Output button at the bottom of the Solution Progress window.

GSC Step 8. Examine the results

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.

The GSC Variables tab shows the final values for the variable parameters, as shown in Figure 5.7. The GSC Goals tab shows the values achieved for the goals, as shown in Figure 5.8.

Heat Exchanger Summary | Reservoir Summary | GSC Variables | GSC Goals

Variable	Object Type	Object	Parameter	Value	Units
1	Three Way Valve	J3	Open Percentage	85.87	Percent

Figure 5.7 The final GSC Variable values are shown on the GSC Variables tab in the Output window General section

Heat Exchanger Summary | 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	Branch	J8 (Branch)	Outlet	Temperature	120	120.0	deg. F

Figure 5.8 The final GSC Goal values are shown on the GSC Goals tab in the Output window General section. The Actual and User values should be close if GSC was succesful. If not, a warning will appear

GSC analysis summary

For this example, the goal of 120 F was achieved for the temperature at branch junction J8 as shown in Figure 5.8. The open percentage required for the Three-Way Valve to achieve a temperature of 120 F at Branch J9 is 85.87%, as shown in Figure 5.7.

XTS problem statement

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 sized tank, 20 feet high, with a cross sectional area of 80 ft^2. 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?

XTS Step 1. Create a scenario for XTS case

Create a new scenario from the Base Scenario for the XTS Case. Refer to Chapter 4 for directions on how to add a scenario.

XTS Step 2. Activate the XTS Module

Activate the XTS Module by choosing Activate Modules from the Tools menu and selecting the XTS Module in the list.

When the XTS Module is activated, Specify Transient Control appears as an additional Checklist item.

Important: The XTS and GSC modules can be used in the same run to simulate control functions over time, but we are not going to do this here. Before proceeding, disable the GSC parameters by selecting Goal Seek & Control from the Analysis menu and then *Ignore*.

XTS Step 3. Modify the existing model

A. Enter the new junction and system data

When using XTS, reservoirs may be defined as finite sized tanks. Modify reservoir J9 to be a finite tank with the following data:

J9 - Reservoir

1. Name = Discharge Reservoir
2. Tank Model = Finite Open Tank
3. Known Parameters Initially = Liquid Surface Level, Surface Pressure

4. Cross-Sectional Area = Constant, 80 ft^2
5. Liquid Height = Height from Bottom, 0 feet
6. Liquid Surface Pressure = 40 psig
7. Tank Height = 20 feet
8. Pipe Elevation = 10 feet
9. Tank Bottom Elevation = 10 feet

J3 - Three-Way Valve

1. Actual Percent Open = 86 %

Heat transfer cannot be calculated when using XTS. Open the System Properties window and select “Constant Fluid Properties” to turn off the heat transfer option. If heat transfer is active when you try to run an XTS model, AFT Fathom will notify you that heat transfer calculations cannot be made. You will then be given the option of continuing the run without heat transfer or stopping the run altogether.

B. Check if the pipe and junction data is complete

Turn on “Show Object Status” from the View menu to verify that all the necessary data is entered. If so, the “Define Pipes and Junctions” Checklist item on the Quick Access Panel 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. You can also open the List Undefined Objects window from the View menu to see what data is missing.

XTS Step 4. Specify transient output time format

The format and units used to display the transient data in the Output window are controlled from the Format & Action tab on the Output Control window. Open the Output Control window from the Tools Menu and select the Format & Action tab. Set the Time Simulation Formatting units to minutes, and then select OK to close the window.

XTS Step 5. Activate transient analysis

The XTS module may be turned on and off so users can run steady-state models using AFT Fathom. It is often useful to check the model steady-state solutions before attempting to run a transient case to ensure the model is realistic.

In order to run a transient analysis using XTS, the analysis must first be set to Transient. This is done by selecting Time Simulation from the Analysis Menu, as shown in Figure 5.9.

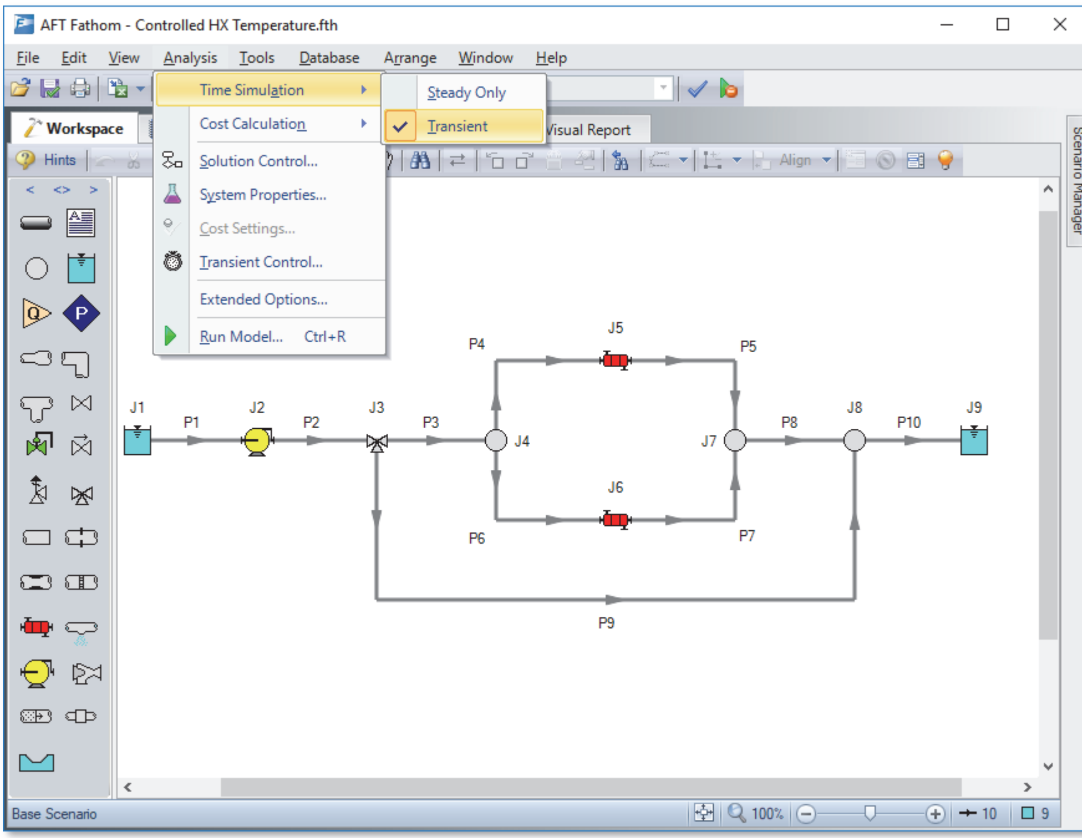


Figure 5.9 Transient Analyses are activated from the Analysis menu.

XTS Step 6. Open transient control

XTS module transient simulations are controlled through the Transient Control window. Activate the Transient Control window by selecting Transient Control from the Analysis Menu, or by clicking the Model Status light and selecting Transient Control from the list. Transient Control will not appear in the list if AFT Fathom is set to perform steady-state analyses. When AFT Fathom is set to perform a transient analysis, the Transient Control becomes a new Checklist item required to run the model.

The Transient Control window 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 Transient Control window, 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.

For this example, the simulation will run for 45 minutes, with a time step of 1 minute. The output will be saved to the output file for every time step. The Transient Control window should appear as shown in Figure 5.10.

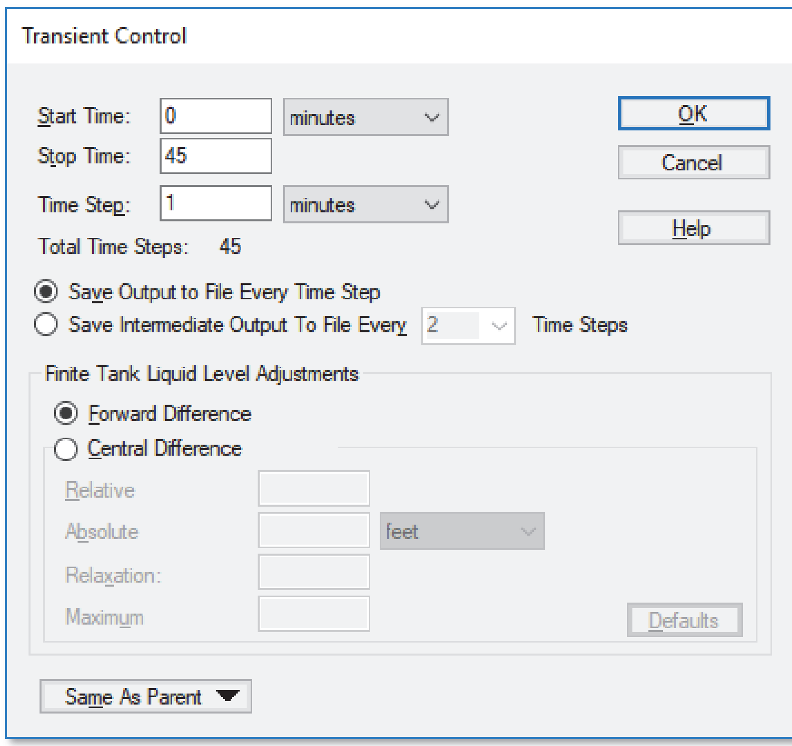


Figure 5.10 The Transient Control window is used to specify transient run times and time step sizes

XTS Step 7. Set up the pump transient

In the XTS module, transients can be defined for junctions. These transients include valves opening and closing, and pumps starting and tripping. 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 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 transient events which occur one after the other, without repeating.

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.

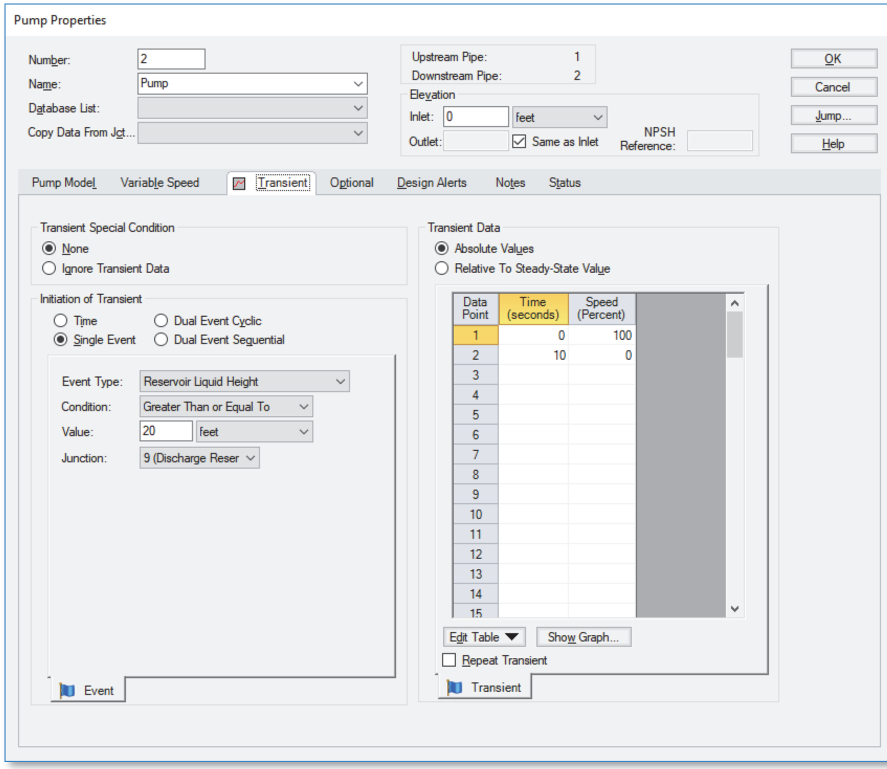


Figure 5.11 Junction transients are specified on the Transient tabs on the junction Properties windows

Open the Pump Properties window, and select the Transient Data tab. Then select Single Event for the Initiation of Transient. Enter the Transient data as shown in Figure 5.11.

XTS Step 8. Run the model

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. Select Run Model in 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.

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 5.12.

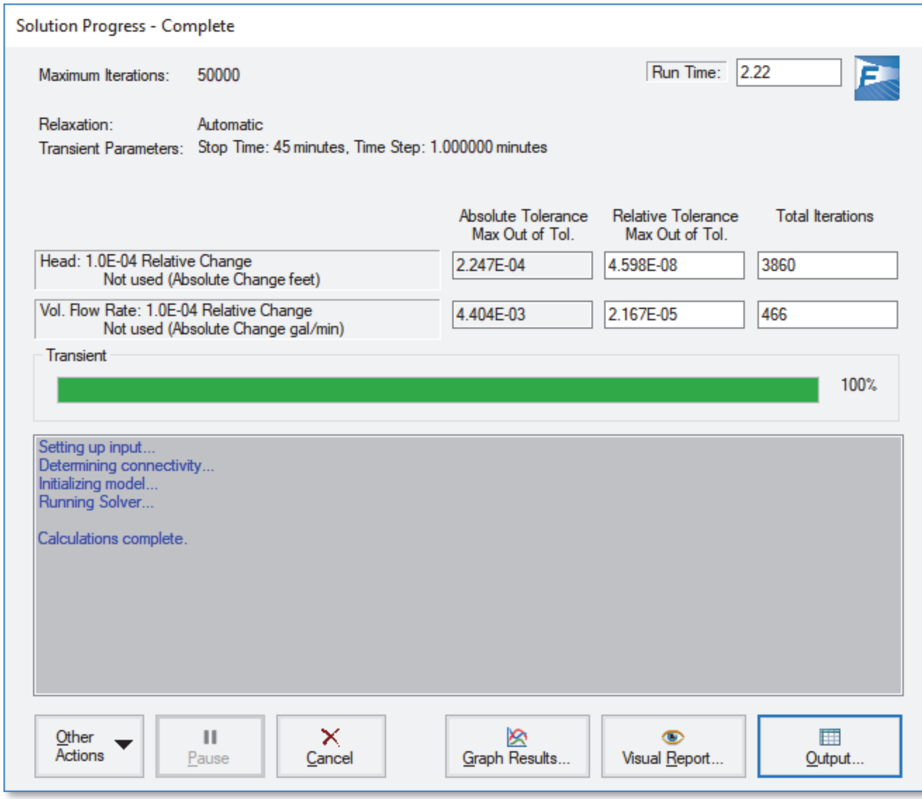


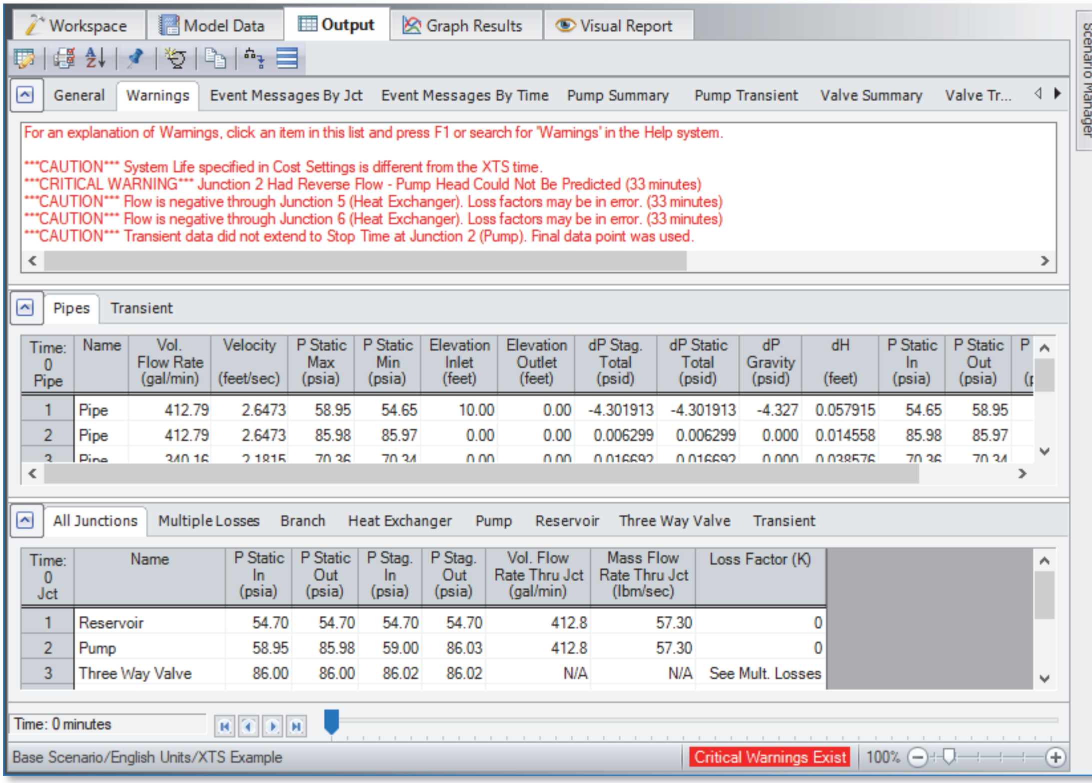
Figure 5.12 The Solution Progress window shows the progress of the transient analysis

After the run has completed, the results can be reviewed by clicking the View Output button.

XTS Step 9. Examine the transient results

The first thing that you may notice after clicking the View Output button is that “Critical Warnings Exist,” as shown in Figure 5.13. For this scenario, there is a critical warning because “Junction 2 Had Reverse Flow – Pump Head Could Not Be Predicted (33 minutes).”

A critical warning is very important and should be reviewed. By examining the transient results of this scenario, you can understand why this warning occurs and how it affects the model.



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)
1	Pipe	412.79	2.6473	58.95	54.65	10.00	0.00	-4.301913	-4.301913	-4.327	0.057915	54.65	58.95
2	Pipe	412.79	2.6473	85.98	85.97	0.00	0.00	0.006299	0.006299	0.000	0.014558	85.98	85.97

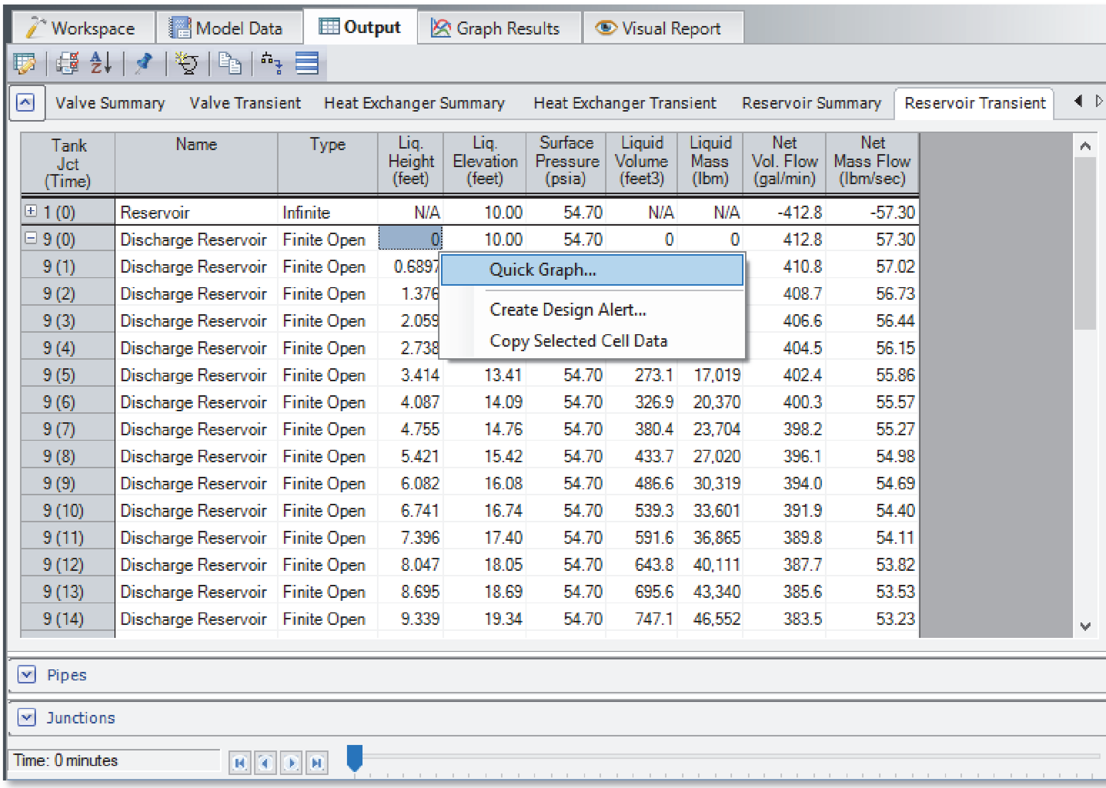
Time: 0 Jct	Name	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
1	Reservoir	54.70	54.70	54.70	54.70	412.8	57.30	0
2	Pump	58.95	85.98	59.00	86.03	412.8	57.30	0
3	Three Way Valve	86.00	86.00	86.02	86.02	N/A	N/A	See Mult. Losses

Figure 5.13 A Critical Warning Exists because there is reverse flow through the pump. The warnings and transient results need review.

To begin examining the transient results, it is important to understand that the AFT Fathom XTS module displays transient output in a number of places 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 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 5.14 shows the Reservoir Transient summary tab, with the data for Reservoir J1 collapsed, and Reservoir J9 expanded.

The Quick Graph feature can be used to conveniently plot transient data for quick examination. To use the Quick Graph feature, place the mouse cursor over the column of transient data you wish to examine. Then, right click with the mouse, and select Quick Graph from the list of options. Figure 5.14 illustrates how to do this for J9 Reservoir Liquid Height vs. Time. Figure 5.15 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 transient output.



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)	Reservoir	Infinite	N/A	10.00	54.70	N/A	N/A	-412.8	-57.30
9 (0)	Discharge Reservoir	Finite Open	0	10.00	54.70	0	0	412.8	57.30
9 (1)	Discharge Reservoir	Finite Open	0.6897					410.8	57.02
9 (2)	Discharge Reservoir	Finite Open	1.376					408.7	56.73
9 (3)	Discharge Reservoir	Finite Open	2.059					406.6	56.44
9 (4)	Discharge Reservoir	Finite Open	2.738					404.5	56.15
9 (5)	Discharge Reservoir	Finite Open	3.414	13.41	54.70	273.1	17,019	402.4	55.86
9 (6)	Discharge Reservoir	Finite Open	4.087	14.09	54.70	326.9	20,370	400.3	55.57
9 (7)	Discharge Reservoir	Finite Open	4.755	14.76	54.70	380.4	23,704	398.2	55.27
9 (8)	Discharge Reservoir	Finite Open	5.421	15.42	54.70	433.7	27,020	396.1	54.98
9 (9)	Discharge Reservoir	Finite Open	6.082	16.08	54.70	486.6	30,319	394.0	54.69
9 (10)	Discharge Reservoir	Finite Open	6.741	16.74	54.70	539.3	33,601	391.9	54.40
9 (11)	Discharge Reservoir	Finite Open	7.396	17.40	54.70	591.6	36,865	389.8	54.11
9 (12)	Discharge Reservoir	Finite Open	8.047	18.05	54.70	643.8	40,111	387.7	53.82
9 (13)	Discharge Reservoir	Finite Open	8.695	18.69	54.70	695.6	43,340	385.6	53.53
9 (14)	Discharge Reservoir	Finite Open	9.339	19.34	54.70	747.1	46,552	383.5	53.23

Figure 5.14 The Quick Graph feature can be accessed from any transient data column using the right mouse button.

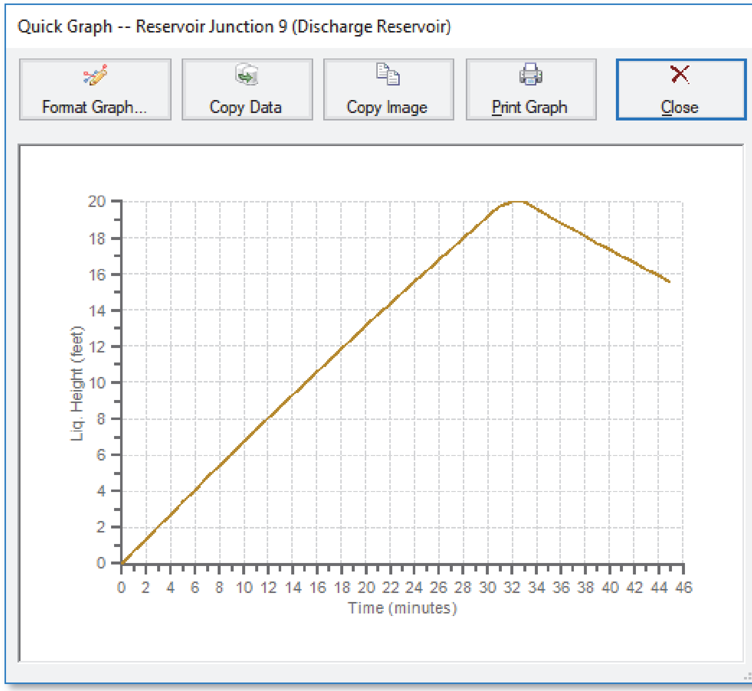
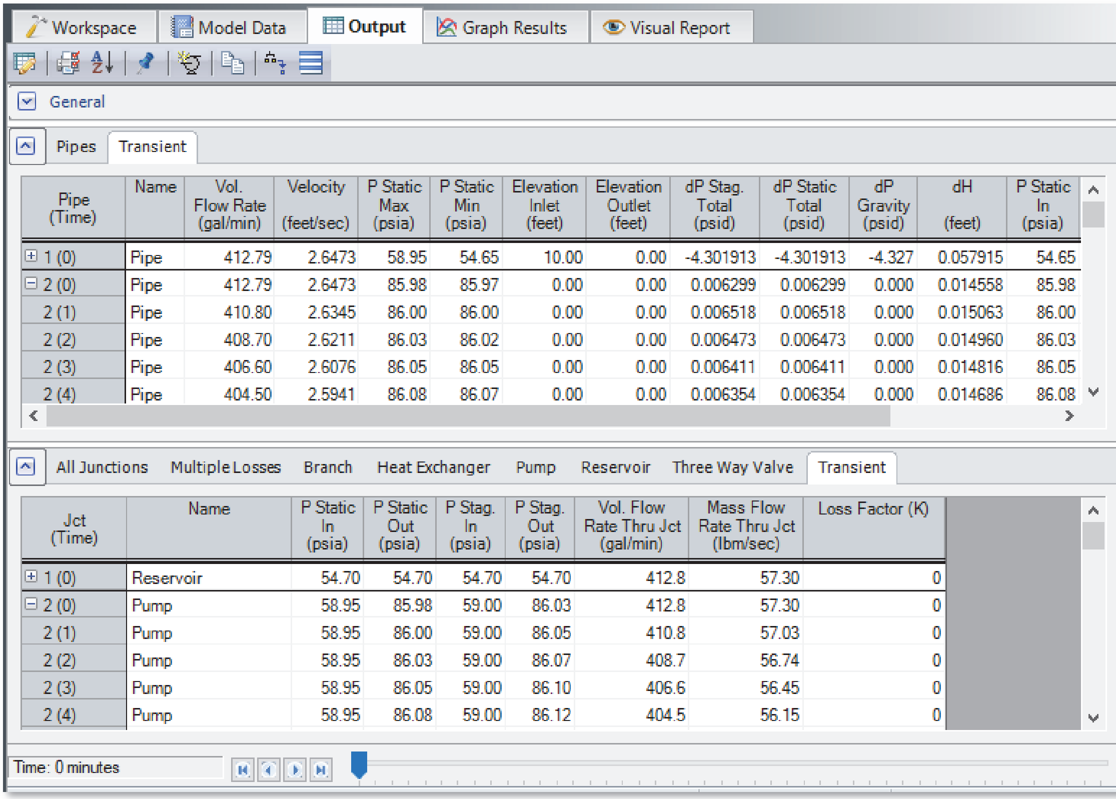


Figure 5.15 The Quick Graph feature can be used to quickly plot transient data from the Output for review.

Consider Figure 5.15 in terms of the Critical Warning, which states that pump head could not be predicted at 33 minutes due to reverse flow through the pump. The tank liquid height increases until the time when the pump turns off. The graph then shows that there is back flow into the system, which causes the Critical Warning. This is discussed further in the XTS Analysis Summary after further examination of the transient results.

Similar to the transient summary 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 5.16. This transient data can be expanded or collapsed in the same manner as in the General Output transient summaries.

Figure 5.17 shows the transient output data for the pipes and junctions, found on the Pipes and All Junctions tabs at Time = 5 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.



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)
1 (0)	Pipe	412.79	2.6473	58.95	54.65	10.00	0.00	-4.301913	-4.301913	-4.327	0.057915	54.65
2 (0)	Pipe	412.79	2.6473	85.98	85.97	0.00	0.00	0.006299	0.006299	0.000	0.014558	85.98
2 (1)	Pipe	410.80	2.6345	86.00	86.00	0.00	0.00	0.006518	0.006518	0.000	0.015063	86.00
2 (2)	Pipe	408.70	2.6211	86.03	86.02	0.00	0.00	0.006473	0.006473	0.000	0.014960	86.03
2 (3)	Pipe	406.60	2.6076	86.05	86.05	0.00	0.00	0.006411	0.006411	0.000	0.014816	86.05
2 (4)	Pipe	404.50	2.5941	86.08	86.07	0.00	0.00	0.006354	0.006354	0.000	0.014686	86.08

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)	Reservoir	54.70	54.70	54.70	54.70	412.8	57.30	0
2 (0)	Pump	58.95	85.98	59.00	86.03	412.8	57.30	0
2 (1)	Pump	58.95	86.00	59.00	86.05	410.8	57.03	0
2 (2)	Pump	58.95	86.03	59.00	86.07	408.7	56.74	0
2 (3)	Pump	58.95	86.05	59.00	86.10	406.6	56.45	0
2 (4)	Pump	58.95	86.08	59.00	86.12	404.5	56.15	0

Figure 5.16 The Transient tabs in the Pipe and Junction sections show results for all time steps. The first number in the left column is the pipe/junction number and the second (in parentheses) is the time (in minutes)

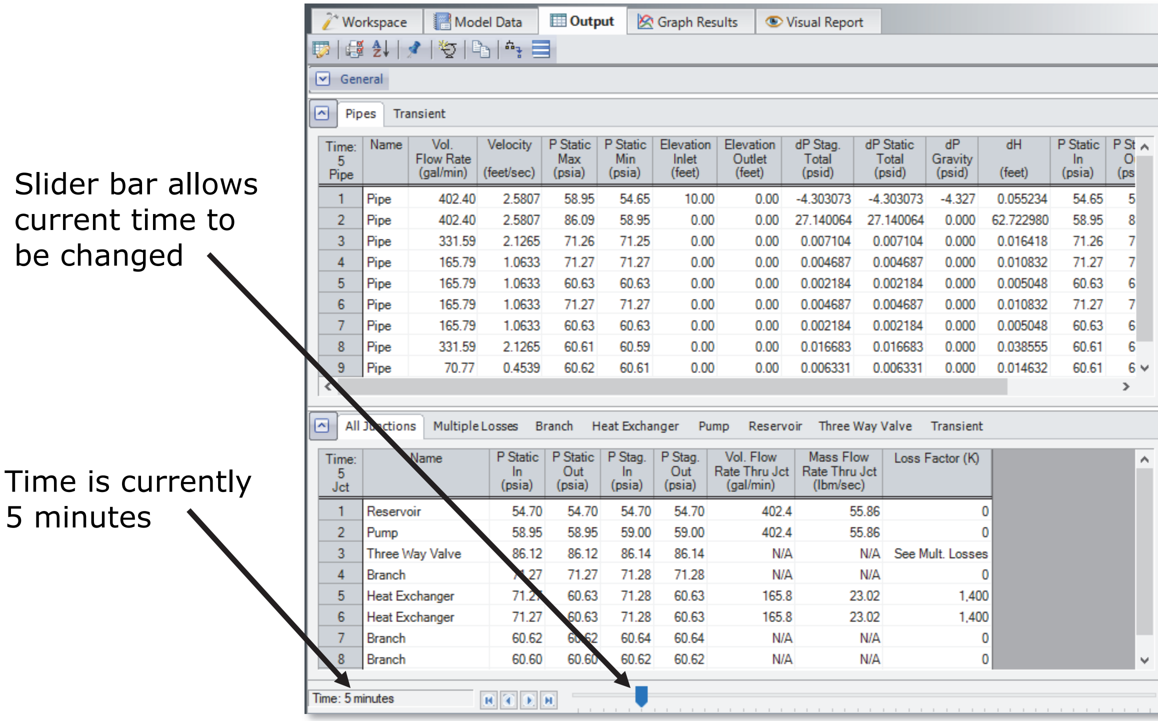


Figure 5.17 The Pipe and Junction output can be displayed at any time step using the slider bar at the bottom of the output window

When junction event transients occur, these events are recorded and displayed in the general output section of the Output Window. 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 5.18.

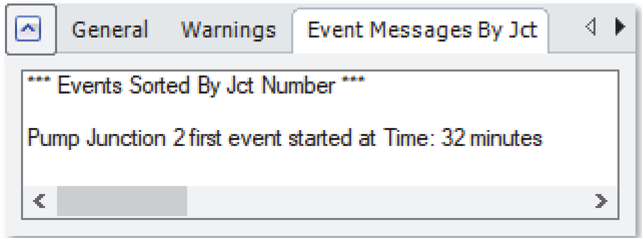


Figure 5.18 The occurrence of event transients is recorded in the General Output section

XTS analysis summary

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.

The Quick Graph plot (Figure 5.15) shows the 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. When there is reverse flow through the pump, the pump head cannot be predicted, and a critical warning is displayed in the Output. A check valve would need to be added to the system to prevent this situation in the case of pump shutdown.

SSL problem statement

The piping for a slurry system which 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.

SSL Step 1. Start AFT Fathom

From the Start menu, choose AFT Products and AFT Fathom. When prompted to activate modules, choose to activate the SSL module. If not prompted, then choose Activate Modules from the Tools menu.

SSL Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu
2. Select Basic Water Slurry Input at the upper right (this should be the default selection). See Figure 5.19.
3. Select the Detailed option for Slurry Calculation Method
4. Enter a temperature for water of 70 deg. F.
5. In the Solids Specifications area, select the “User Specified Solids Added” button
6. Select the calculated *M* and V_t/V_{ts} factor of 1
7. Enter a solids density of 2.9 S.G. water, a d50 of 0.015 inches, a d85 of 0.03 inches
8. Set the Amount Solids Added to 25% by Volume
9. Select OK

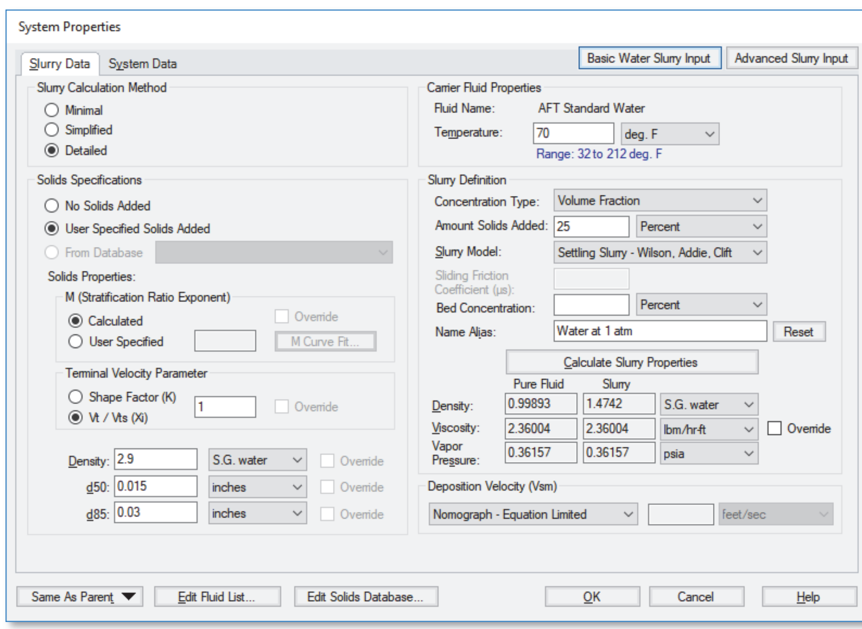


Figure 5.19 Data entry for Sand slurry properties

SSL Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to Define Pipes and Junctions. In the Workspace window, assemble the model as shown in Figure 5.20.

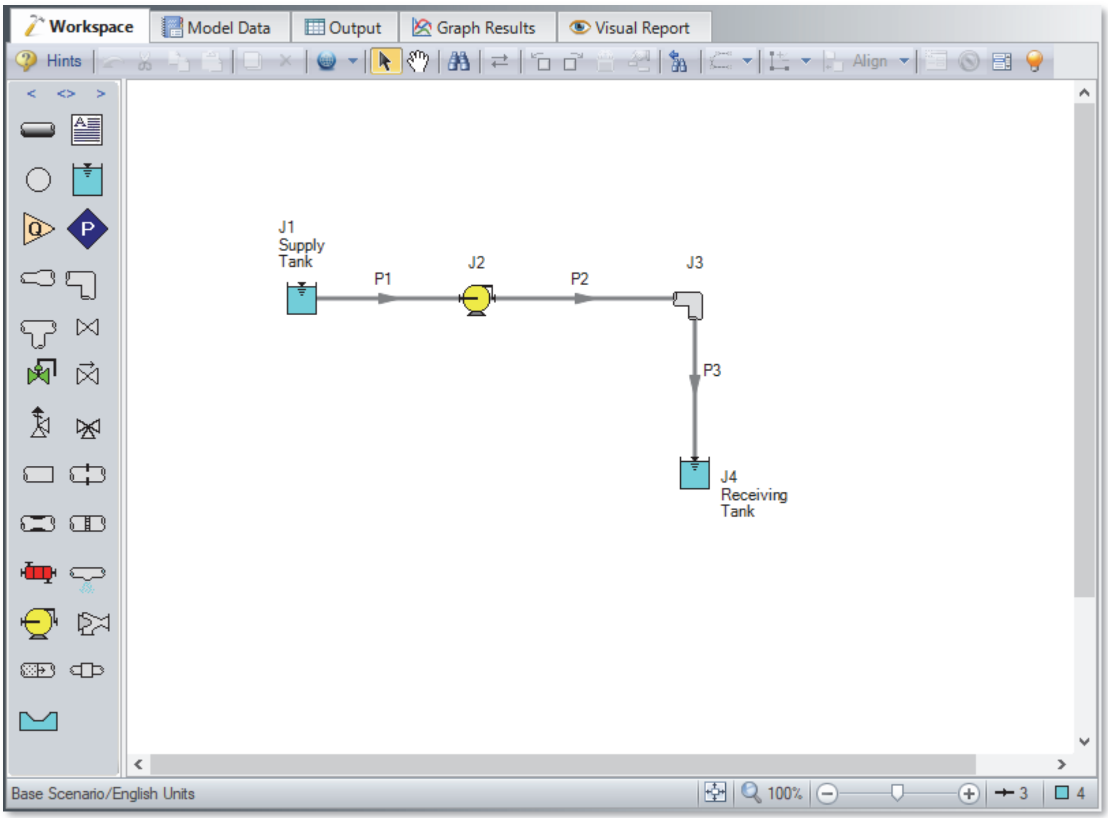


Figure 5.20 Layout of pipe system for Pump Sizing for Sand Transfer System Example

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Properties window (or use the Global Pipe Editing window).

All pipes are Steel - ANSI, with standard roughness and this data:

Pipe	Length (feet)	Size	Type
P1	20	12 inch	STD
P2	500	12 inch	STD
P3	50	12 inch	STD

C. Enter the junction data

J1 - Reservoir

1. Name = Supply Tank
2. Liquid Surface Elevation = 5 ft
3. Liquid Surface Pressure = 10 psig
4. Pipe Depth = 5 ft

J4 - Reservoir

1. Name = Receiving Tank
2. Liquid Surface Elevation = 10 ft
3. Liquid Surface Pressure = 30 psig
4. Pipe Depth = 10 ft

J2 - Pump

1. Elevation = 0 feet
2. Pump Model = Mass Flow Rate Fixed
3. Fixed Flow Rate = 1,400 tons/hour of solids
4. Slurry De-rating use ANSI/HI Standard 12.1-12.6-2005 method
5. Impeller size is 8 inches

J3 - Elbow

1. Elevation = 0 feet
2. Type = standard

D. Check if the pipe and junction data is complete

Turn on "Show Object Status" from the View menu to verify that all the necessary data is entered. If so, the "Define Pipes and Junctions" Checklist item on the Quick Access Panel 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. You can also open the List Undefined Objects window from the View menu to see what data is missing.

SSL Step 4. Run the model

Select Run Model in 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.

After completion, click the View Output button at the bottom of the Solution Progress window.

SSL Step 5. Examine the results

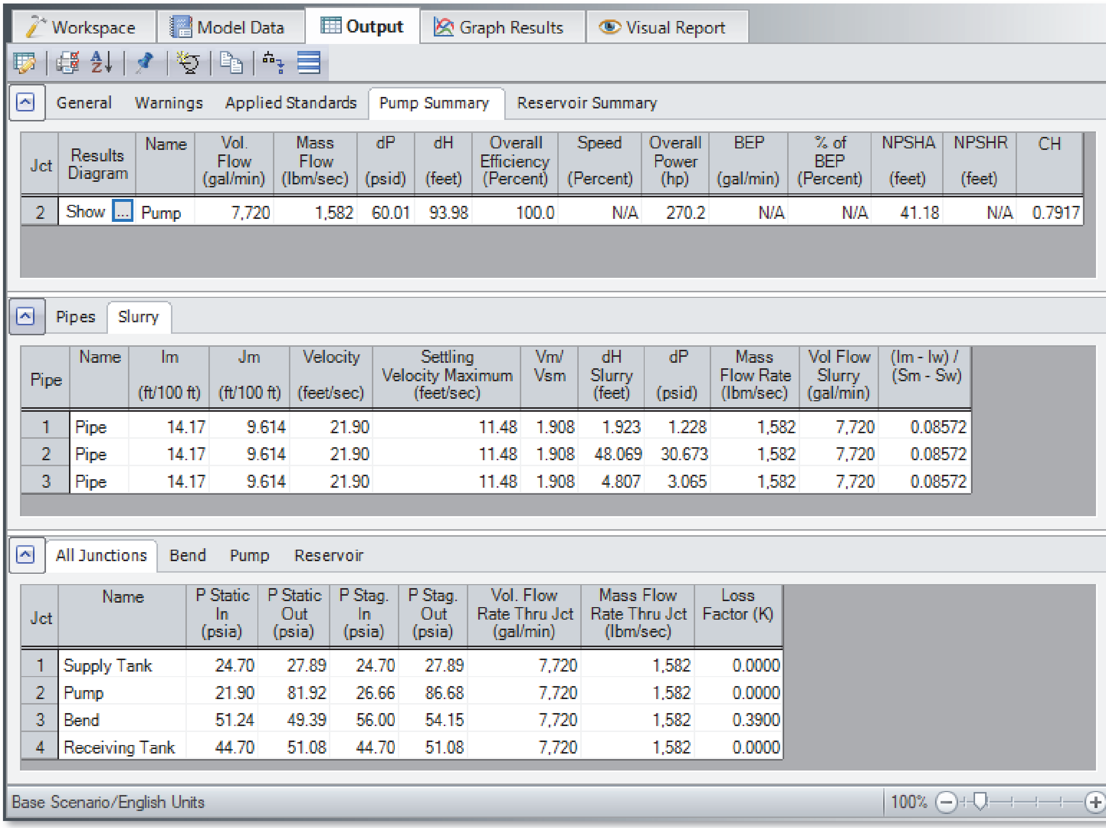
The Output window contains all the data that was specified in the Output Control window.

The Slurry tab shows the slurry results, as shown in Figure 5.21. Here, one can see that the velocity is 21.9 feet/sec, which is safely above the 11.5 feet/sec settling velocity. Other pressure drop and relevant slurry data can be viewed here.

Also shown in Figure 5.21 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 0.79. Hence, a pump should be selected that can generate 93.98/0.79 = 119 feet of head based on water.

Also of interest is the system curve. Select the Graph primary tab, and then the Graph Parameters tab on the Quick Access Panel. Then select the Slurry System Curve tab on the right. See Figure 5.22.

By default, the parameter to graph is j_m vs. velocity. Select the units of j_m as ft/ft which will make it completely dimensionless as is typically done. Check the box for cross-plotting the water curve and then click the Show button. The curves are shown in Figure 5.23.



Workspace | Model Data | Output | Graph Results | Visual Report

General | Warnings | Applied Standards | Pump Summary | Reservoir Summary

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)	CH
2	Show ...	Pump	7,720	1,582	60.01	93.98	100.0	N/A	270.2	N/A	N/A	41.18	N/A	0.7917

Pipes | Slurry

Pipe	Name	Im (ft/100 ft)	Jm (ft/100 ft)	Velocity (feet/sec)	Settling Velocity Maximum (feet/sec)	Vm/ Vsm	dH Slurry (feet)	dP (psid)	Mass Flow Rate (lbm/sec)	Vol Flow Slurry (gal/min)	(Im - Iw) / (Sm - Sw)
1	Pipe	14.17	9.614	21.90	11.48	1.908	1.923	1.228	1,582	7,720	0.08572
2	Pipe	14.17	9.614	21.90	11.48	1.908	48.069	30.673	1,582	7,720	0.08572
3	Pipe	14.17	9.614	21.90	11.48	1.908	4.807	3.065	1,582	7,720	0.08572

All Junctions | Bend | Pump | Reservoir

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	Supply Tank	24.70	27.89	24.70	27.89	7,720	1,582	0.0000
2	Pump	21.90	81.92	26.66	86.68	7,720	1,582	0.0000
3	Bend	51.24	49.39	56.00	54.15	7,720	1,582	0.3900
4	Receiving Tank	44.70	51.08	44.70	51.08	7,720	1,582	0.0000

Base Scenario/English Units 100%

Figure 5.21 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

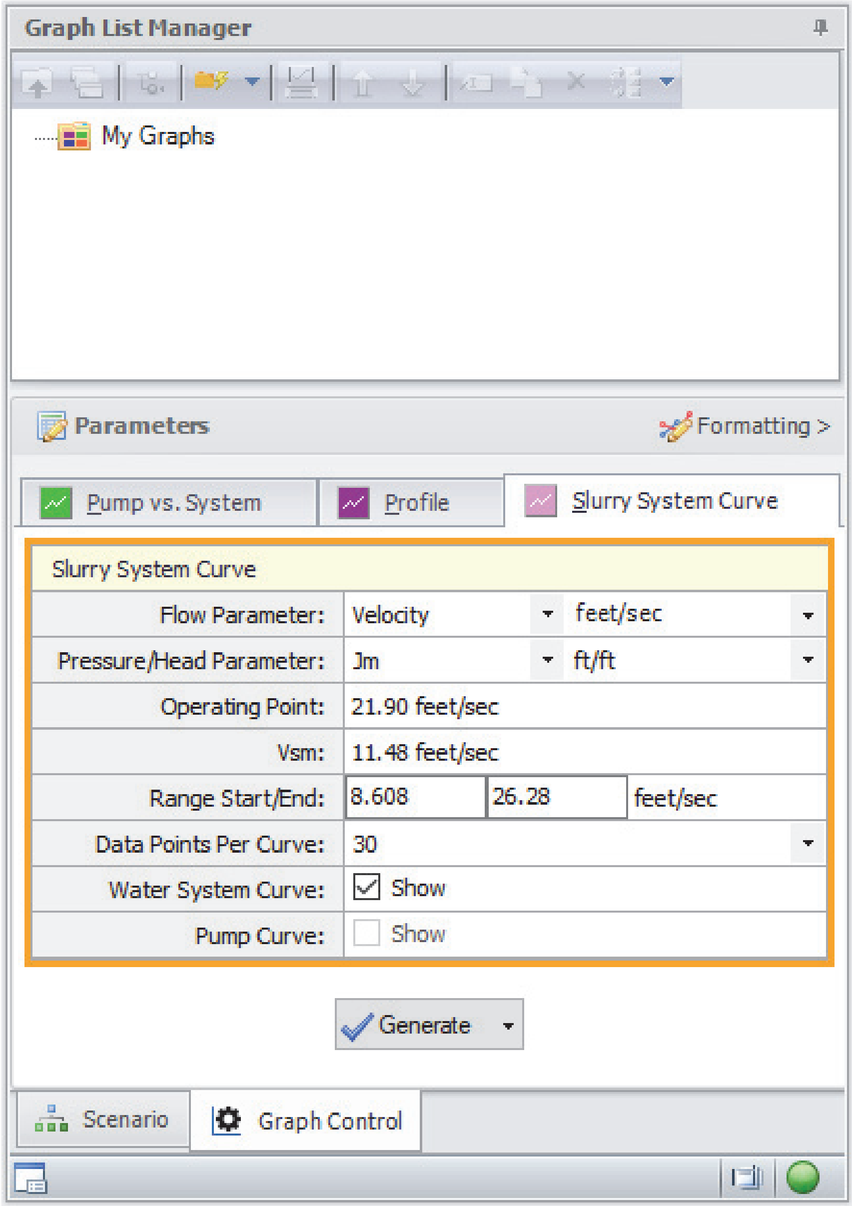


Figure 5.22 Data selection for graphing slurry system curve

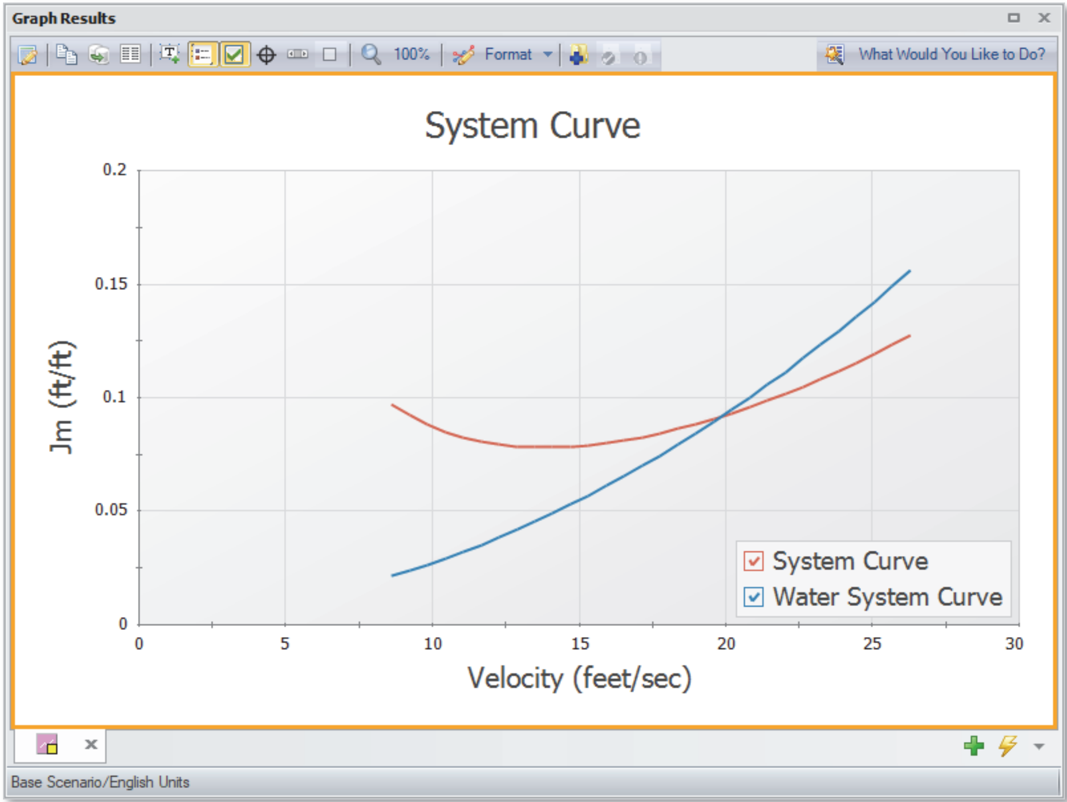


Figure 5.23 Slurry system curve and water curve in Graph Results window

SSL analysis summary

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.

CHAPTER 6

Other AFT Fathom Capabilities

This Quick Start Guide necessarily omitted or had brief coverage of a number of important AFT Fathom capabilities. This chapter briefly describes some of the important capabilities not covered.

Microsoft Excel™ data integration

AFT Fathom now includes enhanced Excel importing and robust Excel exporting. An Excel spreadsheet can be used to vary selected input parameters in multiple scenarios. Selected output can also be exported to specified Excel sheets and cells.

Integration with other software and data standards

AFT Fathom includes a number of importing and exporting capabilities. Piping layouts and dimensional data can be imported from GIS Shapefiles (SHP) to build a model. Additionally, CAESAR II Neutral Files (CII) can be imported into AFT Fathom and EPANET Files (INP) can be both imported and exported.

Piping Component Files (PCF) from AutoCAD Plant 3D, SmartPlant, PDS, CADWorx, and other software can be imported into AFT Fathom. These options are all accessed from the File Menu.

Control valve modeling

AFT Fathom can model four types of control valves. They are Pressure Reducing Valves (PRV’s), Pressure Sustaining Valves (PSV’s), Flow Control Valves (FCV’s), and Pressure Drop Control Valves (PDCV's).

Fitting library

AFT Fathom offers a library of about 400 fitting losses that can be added to pipes.

Variable speed pumps

AFT Fathom allows you to model controlled pumps where the control parameter is flowrate or pressure. The pump speed is varied to deliver the flow rate or pressure you set.

Pump impeller trim effects

AFT Fathom can automatically modify pump curve data for user specified impeller trims. The impeller variations can be based on a percentage from the specified curve or, if using multiple pump configurations, actual impeller dimensions can be entered.

AFT Fathom uses standard affinity laws to modify the pump curves. When multiple curves are entered, AFT Fathom modifies the pump curve by interpolating between the curves based on impeller size.

Pump viscosity effects

When fluids are viscous, pump performance degrades. AFT Fathom uses methods published by the Hydraulics Institute for rotodynamic pumps to automatically adjust manufacturer pump curves.

Model heat transfer between loops

Heat transfer between loops which are hydraulically isolated can be modeled. The loops can have different fluids. This is called thermal linking.

Modeling of gas flow in HVAC and ventilation duct systems

AFT Fathom can model incompressible gas flow. AFT Fathom offers support for both cylindrical as well as rectangular ducts.

Varied pipe geometries including rectangular duct

Five different pipe geometries can be modeled in AFT Fathom. The default geometry is the cylindrical pipe. The other four geometries are the rectangular duct, the cylindrical annulus, the noncylindrical pipe, and the helical tube.

Insulation on inside of pipe/duct

Insulation can be placed on the inside of a pipe or duct, and the effect of reduced flow area on the pressure drop will be automatically determined. If modeling heat transfer, the effect of the insulation on the inside will be accounted for.

View input and output data from different scenarios concurrently

Input data from different scenarios can be viewed concurrently in the Model Data window. Similarly, output data from different runs can be viewed concurrently in the Output window.

Online pump manufacturer equipment selection

AFT Fathom includes a two-way link to Intelliquip™ enabled pump manufacturer Internet websites to select pumps. Operating data determined by AFT Fathom can be passed to the website, a pump model selected by the user, and the pump performance data transferred back to AFT Fathom for immediate use in the model.

Network databases

Junction components and pipe materials can be saved to databases for later reuse. Databases can be located on local PC's or deployed across local or wide area networks. The Database Manager allows users to connect to relevant databases for their specific pipe system design.

Darby 3-K method for laminar flow

The 3-K method developed by Darby can be optionally used in valves and elbows. This has particular application in the laminar and transition regions.

Equivalent Length

Standard equivalent lengths can be specified for pipe fittings, valves and elbows. This is often used in fire protection systems.

Index

Q

R

S

T

U

V

W

X

AFT Fathom Add-on Modules

GSC

Goal Seek & Control

Identifies input parameters that yield desired output values and simulates control functions

XTS

Extended Time Simulation

Models dynamic system behavior and how critical system parameters vary over time

SSL

Settling Slurries

Models the effects of pumping fluids containing settling solids using the Wilson/GIW method

---

For Product Support:

Call: (719) 686-1000
Email: support@aft.com

Applied Flow Technology

2955 Professional Place, Suite 301
Colorado Springs, CO 80904 USA
(719) 686 1000
info@aft.com
www.aft.com