AFT Fathom™ 13
Quick Start Guide
Metric Units

Dynamic solutions for a fluid world ™

AFT
Applied Flow Technology

AFT Fathom™
Quick Start Guide

Metric Units

AFT Fathom Version 13
Incompressible Pipe Flow Modeling

Published: September 14, 2023

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.

Contents

C H A P T E R 1

Introducing AFT Fathom

Welcome to AFT Fathom, 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, pumps, and the resultant effect on fluid properties such as viscosity.

AFT Fathom has four available Add-on Modules for purchase which extend the modeling capabilities into new areas.

- The Goal Seek & Control (GSC) module allows the engineer to perform multivariable goal seeking and simulate control system functions.

- The Extended Time Simulation (XTS) module allows the engineer to model time-varying system behavior via a series of steady-state snapshots.

- 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 design requirements, such as maximum pressure or minimum flow, to reduce cost or weight.

Modeling capabilities

Standard version capabilities

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Pressure driven 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/or 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)

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

AFT Fathom provides hundreds of standard loss values from handbook sources for pipe system components, and also allows the engineer to

enter their 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.

Engineering assumptions in AFT Fathom

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

- Incompressible flow

- Steady-state conditions (XTS Module consists of steady snapshots)

- One-dimensional flow

- System is fluid-full

- Fluid properties are single phase

- No chemical reactions

Thermophysical property data

AFT Fathom offers several sources from which thermophysical properties can be derived:

- ASME Steam/Water - Properties are calculated with the equations of state from the IAPWS Industrial Formulation (1997), also used to generate the ASME steam/water tables.

- Chempak™ - Optional add-on for purchase which contains data for approximately 700 fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.

- NIST REFPROP - The Reference Fluid Thermodynamic and Transport Properties Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via equations of state.

REFPROP supports user-specified fluid mixtures.

- AFT Standard - Common fluids from various industry standard sources have been curve fit to optimize calculation speed.

- User Specified Fluid - Custom fluid properties can be entered if the desired working fluid is not available via the other sources. Fathom requires density and viscosity at a minimum. Vapor pressure can be optionally entered to check for cavitation.

- User Library Fluid - A custom fluid can be saved to the User Library for repeated use in the future. Heat transfer can be modeled if additional parameters are specified.

AFT Fathom Primary Windows

AFT Fathom has several Primary 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*.

Workspace

The Workspace Primary Window is where the user creates the graphical representation of the system. Engineering data is input on pipe and junction property windows which are opened by double-clicking the objects.

The Workspace Layers are also located on this Primary Window and allow the user to display both input and output data superimposed on the pipe system schematic. Workspace Layers replaces the deprecated Visual Report Primary Window from previous versions.

Model Data

The Model Data Primary Window summarizes the engineering input data from the Workspace into categorized tables. This allows for easy review of the system inputs, and double clicking cells will open the pipe or junction properties for quick editing.

Output

The Output Primary Window offers a powerful and diverse range of features for reviewing analysis results which are organized into categorized spreadsheets and lists. The Output allows the user to easily compare multiple design scenarios with color coded rows, details errors or design requirement violations, and prepare the results for documentation.

Graph Results

The Graph Results Primary Window allows the user to create a wide array of graphs with various parameters which are integral to gaining a deeper understanding of the pipe system's behavior. The user can adjust various formatting settings such as axis ranges, labels, colors, and create dual y-axis or stacked graphs. The settings can be saved into templates called Graph List Items so that the graphs can be easily reproduced.

Input vs. Output

Table 1.1 summarizes how each Primary Window relates to input and output data. The Workspace can display output data by using Workspace Layers after the simulation has been run.

Table 1.1   Primary Window Input vs. Output Data

Primary Window	Can Display Input	Can Display Output
Workspace	✓	✓
Model Data	✓	
Output		✓
Graph Results		✓

Note: AFT Fathom supports multiple monitors. You can click and drag any of the Primary Window tabs off of the main AFT Fathom window, where it can then be moved anywhere on the screen, including onto a second monitor. To add the Primary Window back to the main AFT Fathom program window bar, simply close the Primary Window.

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

Sizing a Pump Example

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 Fluid properties
- Specifying Output Control
- Using Workspace Layers
- Graphing results
- Sizing a pump and entering pump curve data

Required knowledge

No prior knowledge is required for this example.

Model file

This example is intended to be built from scratch to help familiarize yourself with the steps to define a complete model in AFT Fathom. For reference purposes only, the completed example model file can be found in the Examples folder as part of the AFT Fathom installation:

C:\AFT Products\AFT Fathom 13\Examples

- *Metric - Sizing a Pump.fth*

Problem statement

A pump transfers 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 3 meters long, and the second pipe leading from the pump to the discharge reservoir is 300 meters long. Both pipes are Steel - ANSI, 4 inch diameter (10.2 cm ID), and STD (schedule 40). Assume constant fluid properties.

The supply reservoir surface pressure is 0 barG (0 kPa (g)), and the liquid surface is at an elevation of 3 meters. The discharge reservoir surface pressure is 0 barG (0 kPa (g)), and the liquid surface is at an elevation of 60 meters. The pipes for both reservoirs connect at a depth of 3 meters below the liquid surface.

Determine the head requirement for a pump to supply flow in this system at 115 m3/hr.

Step 1. Start AFT Fathom

➢ **To start AFT Fathom,** open the all applications list in the Windows Start Menu, expand the AFT Fathom 13 folder, and choose AFT Fathom 13. This refers to the standard menu items created by a standard installation; you may have chosen to specify a different installation location.

As AFT Fathom starts, the AFT Fathom Startup Window appears, as shown in Figure 2.1. This window provides you with several options to choose from before you start building a model.
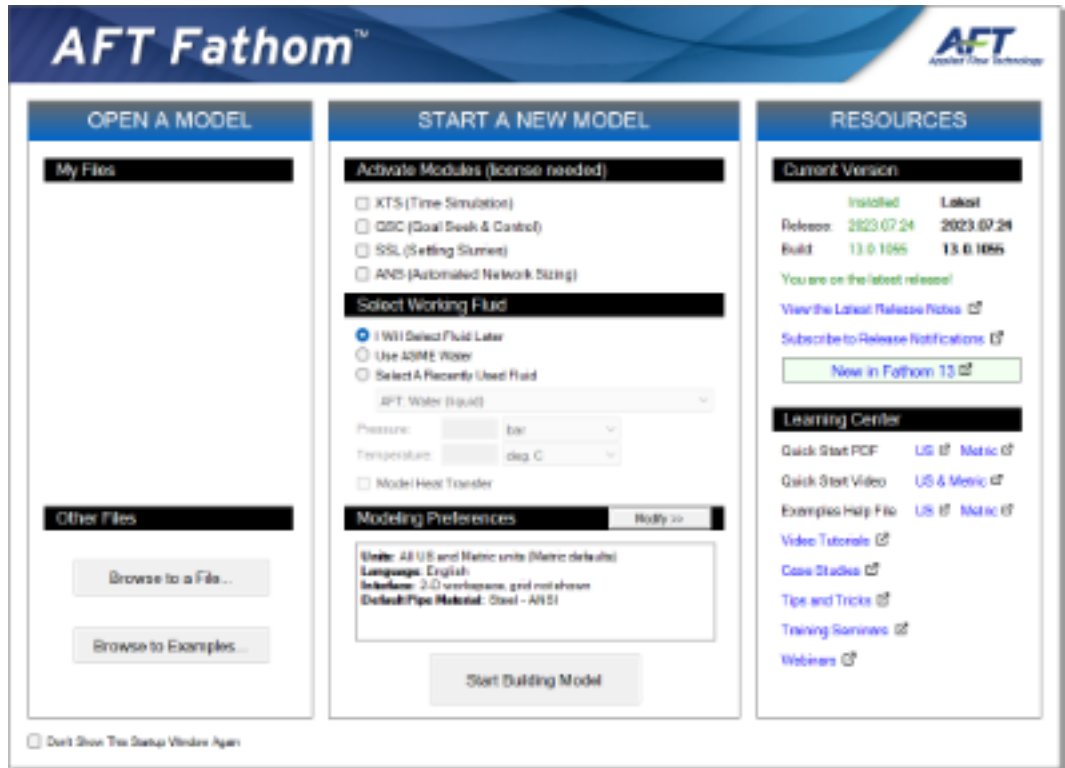
Figure 2.1   Startup Window with Modeling Preferences collapsed

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

- Review or modify Modeling Preferences: select a Unit System, filter units, choose a Grid Style, select a Default Pipe Material, and more

- Access other Resources such as Help Files and Video Tutorials

The first time AFT Fathom launches, Modeling Preferences will be expanded in the middle section of the Startup Window, as shown in Figure 2.2. Otherwise, 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 the Start a New Model section. 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 Metric Defaults under Unit System.** The Filter section can be used to limit the number of units that are shown in the interface.

The Common Plus Selected Industries option adds units from the industries you select. You can use the Show Sample Units option to see the model units based on your selections. After modifying your Modeling Preferences, click Remember My Preferences and Hide.

➢ **Click Start Building Model.**

The Workspace window is shown initially, as seen in Figure 2.3. The tabs in the AFT Fathom window represent the Primary Windows. Each Primary Window contains its own toolbar displayed directly beneath the Primary Window tabs.

The Workspace window

The Workspace window is the Primary Window for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox consists of the object icons on the far left. The Quick Access Panel is on the far right and gives easy access to the Scenario Manager, object Properties, and Workspace Layers.
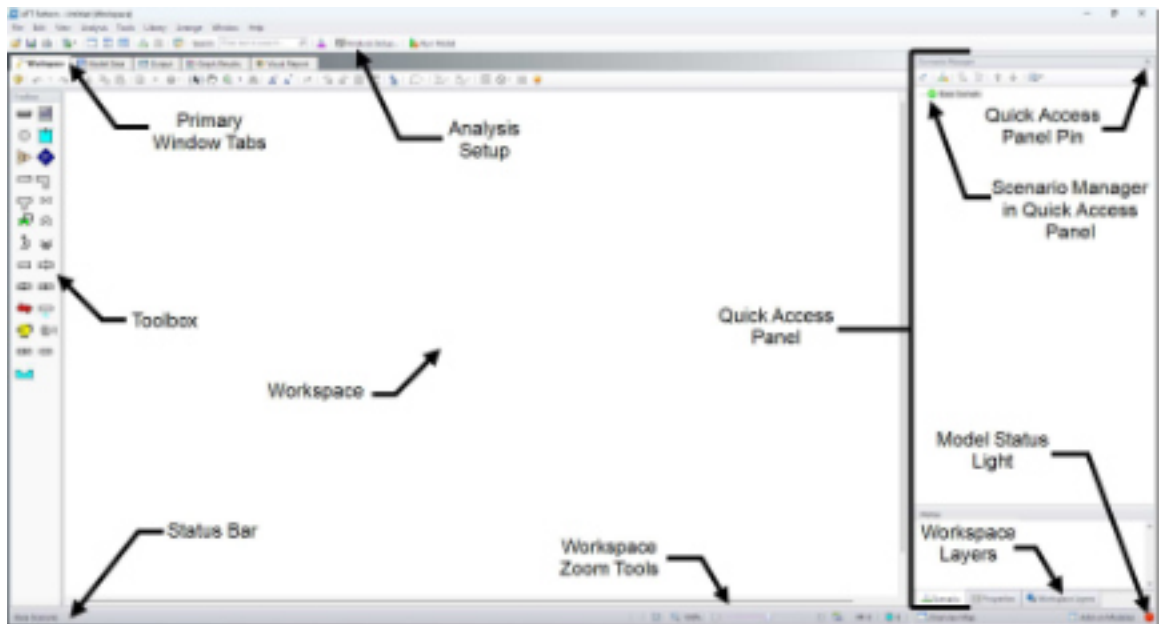
Figure 2.3   The Workspace Primary Window

You will build your model on the Workspace using the Toolbox. The Pipe Drawing tool on the upper-left is used to draw new pipes on the Workspace. The Annotation tool beside it allows you to create annotations and auxiliary graphics.

Below the two drawing tools are the different types of junctions available in AFT Fathom. Junctions connect pipes and also influence the pressure or flow behavior of the pipe system. Drag and drop any of the junction icons from the Toolbox to add them to the Workspace.

Holding your mouse pointer over a Toolbox icon identifies the tool's function.

Hiding the Quick Access Panel

By default, the Quick Access Panel is pinned to the Workspace window so it is always displayed. The Quick Access Panel can be unpinned so it is only displayed when the mouse is moved over a tab displayed on the right edge of the Workspace window. Unpin the Quick Access Panel by clicking on the pin icon displayed in the top-right corner of the Quick Access Panel (see Figure 2.3). The Quick Access Panel will be pinned for the remainder of the examples in this guide.

Step 2. Build the model

To lay out this example model, you will place several 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 2.4 shows the Workspace with one reservoir.

Objects and ID numbers

Items placed on the Workspace are called *objects*. AFT Fathom uses three types of objects: *pipes*, *junctions*, and *annotations*.

All pipe and junction objects on the Workspace have an associated ID number. Junctions are prefixed with the letter J, and pipes are prefixed with the letter P. You can assign a name in the object's Properties window, and the visibility of both the name and ID number can be controlled through the use of Workspace Layers. You can also drag the ID number/name text to a different location to improve visibility.

The reservoir you created on the Workspace will take on the default ID number of 1. You can change this to any desired integer between 1 and 99,999 that is not already assigned.

Editing on the Workspace

You can move junctions on the Workspace to new locations. You can also cut, copy, paste, and duplicate junctions from the Edit menu or using keyboard shortcuts. Undo is available for most arrangement actions on the Workspace.
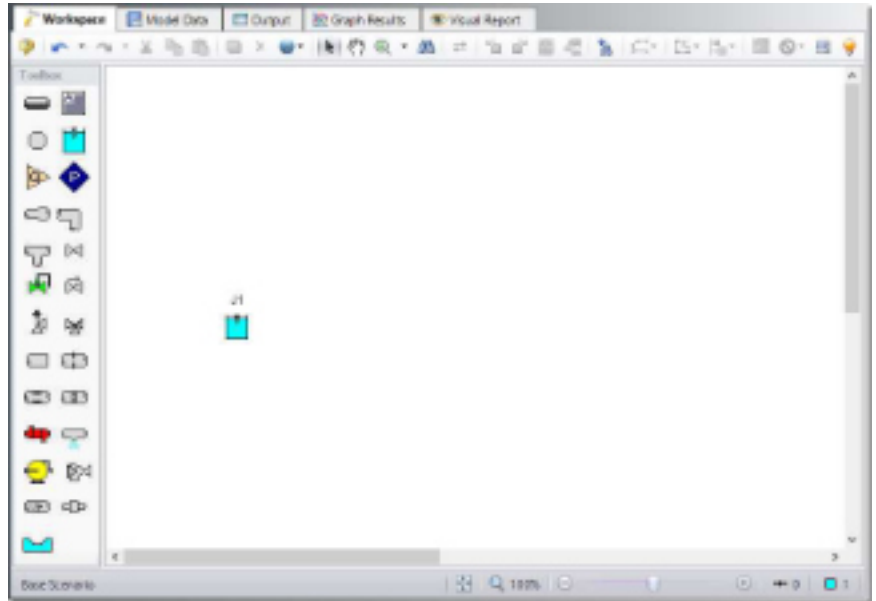
Figure 2.4   Pump Sizing model with the first junction placed

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 2.5).

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 or by using CTRL+D. 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 2.5).

> **Note:** The relative location of objects is not important. Pipe lengths and junction elevations are defined through property windows. The relative object locations on the Workspace establish the connectivity of the objects but have *no bearing on the actual length or elevation relationships*.

Even if the icons do not line up exactly, the results will not change. However, your model may have a better appearance if the icons do line up. You can align the icons from the Arrange menu using the Align feature.

Your model should now appear similar to that shown in Figure 2.5.
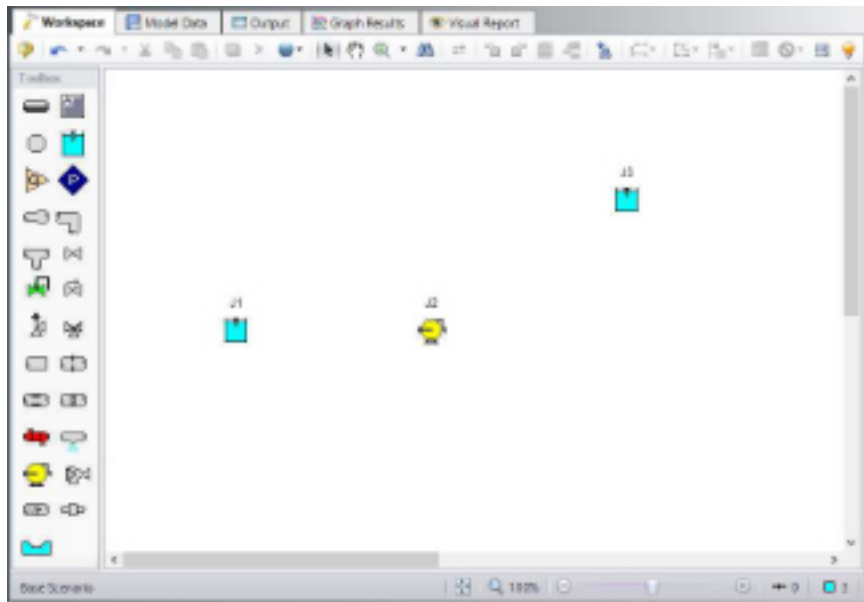
Figure 2.5   Pump Sizing model with all junctions placed

➢ **Before continuing, save your work.** Press CTRL+S, use the Save button on the Common Toolbar, or go to the File menu and select Save. AFT Fathom models use the *.fth file extension.

C. Draw a pipe between J1 and J2

Now that you have all of the junctions placed, you need to connect them with pipes.

Connecting Junctions with Pipes

➤ **To create a pipe,** click the Pipe Drawing Tool icon on the Toolbox. The mouse pointer will change to a crosshair when you move it over the Workspace. Left-click and drag to draw a pipe below the junctions, similar to that shown in Figure 2.6.

The pipe object on the Workspace has an ID number (P1) shown near the center of the pipe.
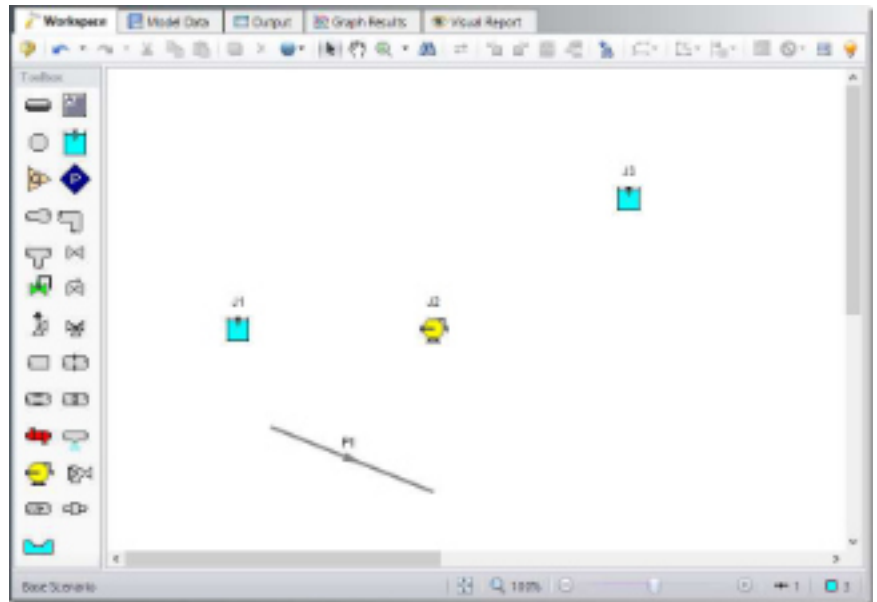
Figure 2.6   Pump Sizing model with the first pipe drawn

➤ **To place the pipe between J1 and J2,** click and drag the pipe so that the left endpoint falls within the J1 Reservoir junction (see Figure 2.7). Next, grab the right endpoint of the pipe and drag it until the endpoint falls within the J2 Pump icon (see Figure 2.8).
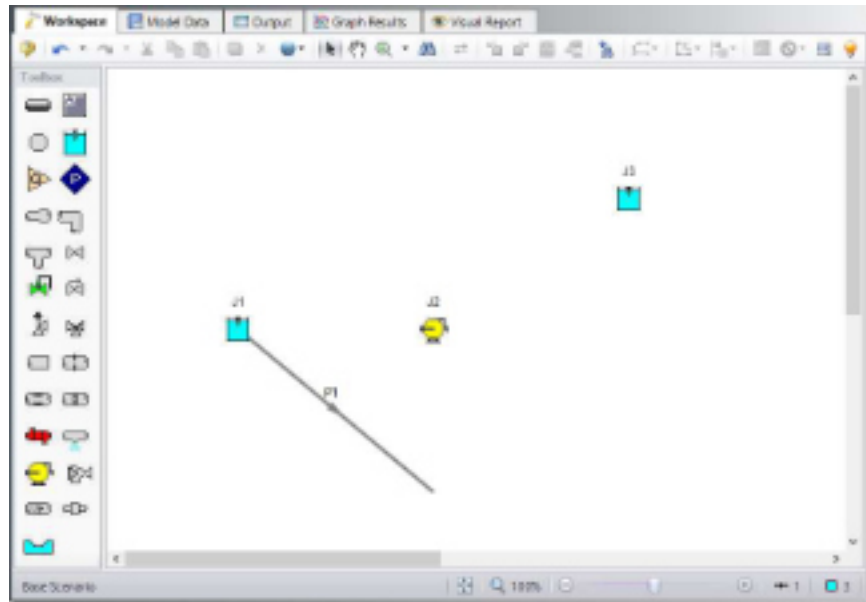
**Figure 2.7   Pump Sizing model with the first pipe partially
connected**
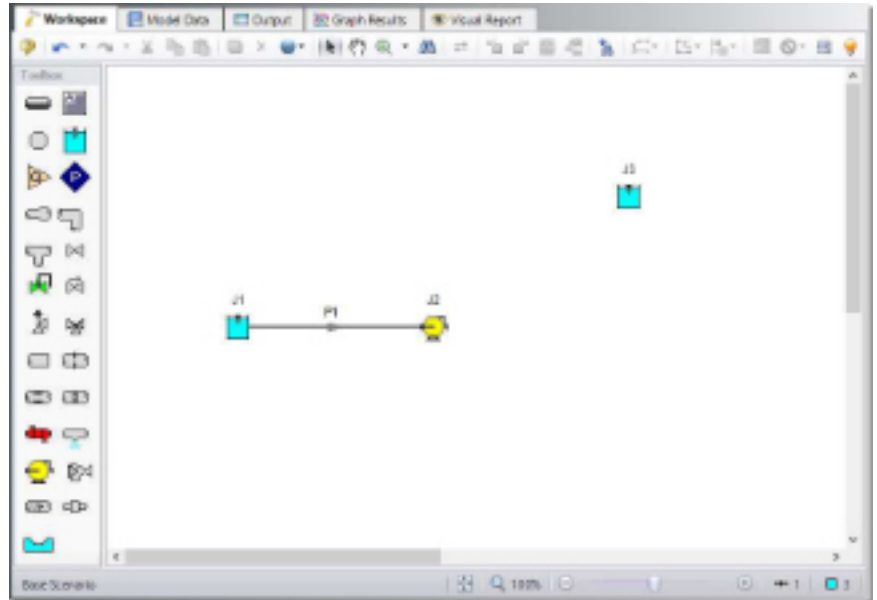
Figure 2.8   Pump Sizing model with the first pipe fully connected

Reference positive flow direction

The arrow on each pipe indicates the reference positive flow direction for the pipe. The direction does not influence the direction that the flow may go, it influences sign convention only. If the direction of flow calculated by the Solver is the opposite of this reference flow direction, then the output will show the flow rate as a *negative number*.

You can reverse the reference positive flow direction by choosing Reverse Direction from the Arrange menu, selecting the Reverse Pipe Direction button on the Toolbar, or pressing F3.

Although the reference positive flow direction does not need to point in the direction of the actual flow, when used with pumps, the reference direction determines the suction side and discharge side of the pump. Therefore, for the remainder of this example make sure the positive flow direction of Pipe P1 points from the J1 Reservoir to the J2 Pump.

Note: Examples of other junctions that depend on pipe directions include Control Valves, Check Valves, and Area Changes.

D. Add the remaining pipe

For faster placement, draw the pipe directly between the two 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 2.9.
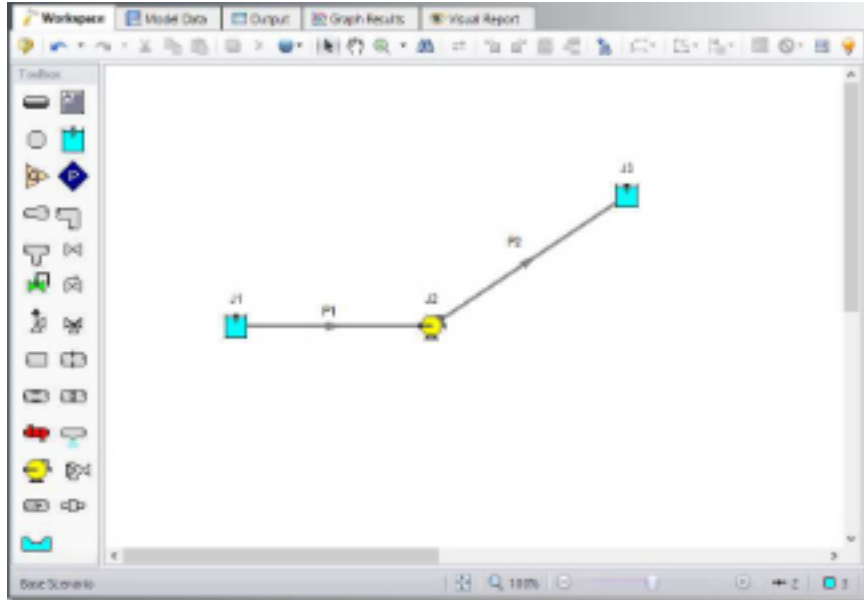
Figure 2.9   Pump Sizing model with pipes and junctions placed

At this point all the objects in the model are graphically connected. Save the model again.

Note: Some users find it desirable to lock objects to the Workspace once placed. This prevents accidental movement and disruption of the connections. You can lock all the objects by first selecting all objects by using CTRL+A or the Edit menu by clicking Select All. Then use CTRL+L or go to the Arrange menu and selecting Lock Object. The lock button on the Workspace Toolbar will appear depressed, indicating it is in an enabled state. Alternatively, you can use the grid feature and snap to grid. The grid options can be modified through the Arrange menu or User Options window.

Step 3. Specify Analysis Setup

The Analysis Setup window allows the user to define fluid properties, view undefined pipe/junction properties, adjust solution control, enable Add-on Modules if purchased and available on the current license, and specify other simulation settings from a single central location.

➤ **Next, click Analysis Setup on the Common Toolbar.** This opens the Analysis Setup window (see Figure 2.10). The Analysis Setup window contains various *groups* depending on which Add-on Modules you may be using. Each group has one or more *items*. A fully defined group has every item within it defined with all required inputs. Complete every group to run the Solver. A group or item that needs more information will have a red exclamation point to the left of the group or item name. After completing a group, the group icon shows a green circle with a checkmark. After defining all groups, thus completing the Analysis Setup, the Model Status light in the lower-right corner turns from red to green.

A. Specify the Modules group

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 2.10. Here you can enable and disable Add-on Modules. These Add-on Modules include Goal Seek & Control (GSC), Settling Slurry (SSL), Extended Time Simulation (XTS), and Automated Network Sizing (ANS). By default, the model will have all modules disabled. Therefore, make no changes to this panel for this example.
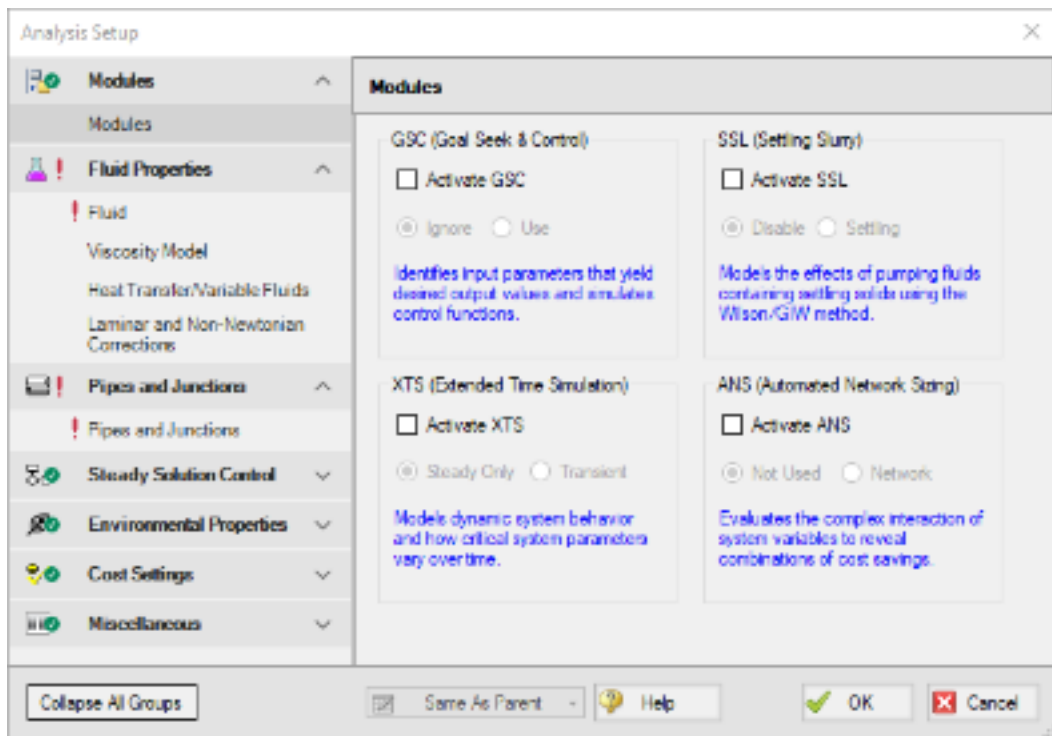
Figure 2.10   On the Modules panel, you can enable or disable any module

B. Specify the Fluid Properties group

The Fluid Properties group consists of four items: Fluid, Viscosity Model, Heat Transfer/Variable Fluids, and Laminar and Non-Newtonian Corrections. On the Fluid panel, you specify the fluid used in the model. On the Viscosity Model panel, you can select which viscosity model to use – Newtonian by default. On the Heat Transfer/Variable Fluids panel, you can specify constant fluid properties (default selection), variable fluid properties, or single or multiple fluid heat transfer with energy balance. Finally, on the Laminar and Non-Newtonian Corrections panel, you can specify the type of corrections to apply with laminar flow or Non-Newtonian fluids.

By default, only the Fluid panel requires input. Click on the Fluid item to open the Fluid panel, shown in Figure 2.11. You can directly enter

density and viscosity data (using User Specified Fluid), select a fluid from the standard AFT Fathom fluid library (AFT Standard), or select Water Data from ASME Steam/Water Tables. Additionally, you can select from fluids and/or create mixtures using the NIST REFPROP library (included in AFT Fathom) or the Chempak library (an optional add-on to AFT Fathom). You can also add your own custom fluids to the User Library.
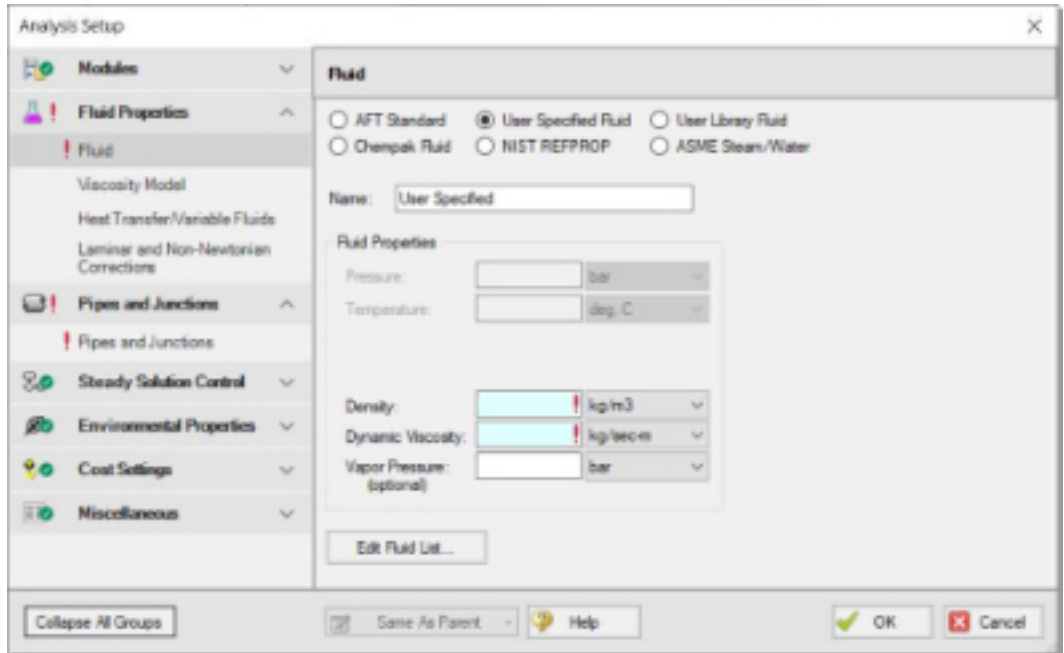
Figure 2.11   Specify the fluid in the model on the Fluid panel

➢ **Select AFT Standard to see the available fluids** (see Figure 2.12). Select Water (liquid) in the library list, then click the Add to Model button below the list. On the right, enter a temperature of 21 deg. C. This satisfies the required inputs for the Fluid panel and Fluid Properties group.
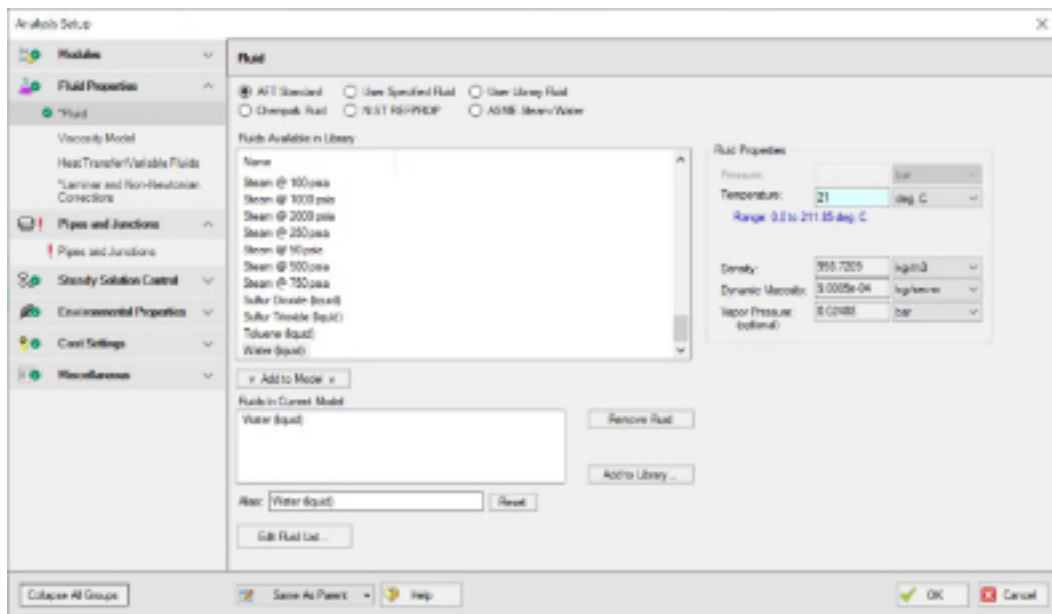
Figure 2.12   The fully defined Fluid panel

C. Specify the Pipes and Junctions group

The Pipes and Junctions group is less straightforward to define compared to the previous two. This group requires full definition for all pipe and junction properties. Click on the Pipes and Junctions item to open the panel shown in Figure 2.13. Here, you will see all the undefined pipes and junctions. By selecting a pipe or junction in the top or middle section, the remaining properties that need definition appear in the bottom section. Select Pipe 1 for example. Pipe Length, Pipe Inner Diameter, and Roughness/Friction are undefined. To define these parameters, you can double-click on Pipe 1 from the top section, or go to the Workspace and open the properties window from there (double-clicking the pipe object). However, before closing the Analysis Setup window, take a look at the final four groups.
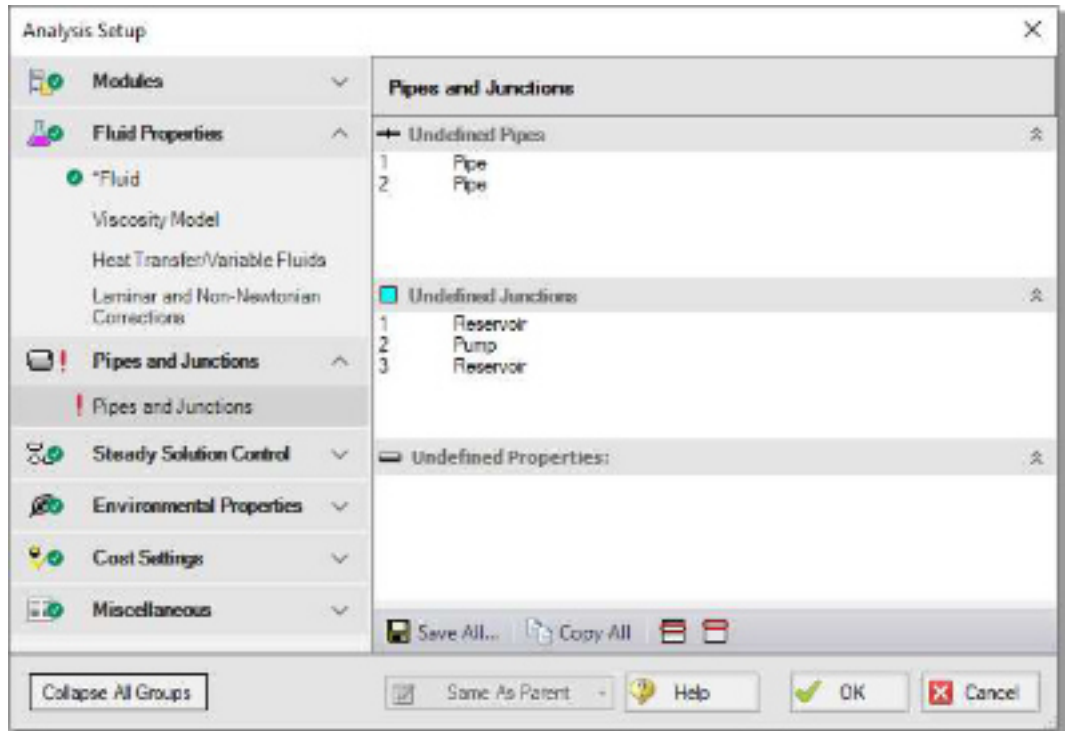
Figure 2.13   The Pipes and Junctions panel lists the remaining undefined objects before the model can run

D. Specify the final four groups

The remaining groups are Steady Solution Control, Environmental Properties, Cost Settings, and Miscellaneous. In the Steady Solution Control group, you can adjust the tolerance and iteration settings for the Solver; however, the default settings will suffice for the vast majority of use cases, and caution should be taken if editing these settings. On the Environmental Properties panel, you can adjust the ambient pressure, gravitational acceleration, and transition Reynolds Numbers. The Cost Settings panel allows you to select which costs to include as well as define energy costs. The Miscellaneous group has items to turn on NFPA, specify heat transfer sectioning, as well as change other advanced parameters. The default selections for all four of these groups will suffice to fully define these panels.

> ➢ **Click OK to save the changes in Analysis Setup** and to return to the Workspace. The remaining undefined Pipes and Junctions group requires input before the model can run. Complete this item by opening the properties window of the objects directly from the Workspace.

Step 4. Define the pipes and junctions

Check an object's status to see if a pipe or junction has all the required input.

Object status

Every pipe and junction has an object status. The object status tells you whether the object has all the necessary inputs. 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 toggles on or off.

With Show Object Status on, the ID numbers for all undefined pipes and junctions display in red on the Workspace. Completely defined objects 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 example problem, all the object ID numbers will change to red when you turn on Show Object Status.

Showing undefined objects

Additionally, you can see which remaining objects need input in the Undefined Objects list, similar to what was shown in the Pipes and Junctions panel of the Analysis Setup.

Open the Undefined Objects list (Figure 2.14) using the List Undefined Objects button on the Workspace Toolbar or from the View menu by clicking on List Undefined Objects. The icon on the Workspace Toolbar is located at the far right end next to the Show Object Status lightbulb icon. This list shows all objects with incomplete information. Clicking on an undefined pipe or junction will display the missing property data. Click the Close button (black X icon) on the bottom-right to stop showing this window.
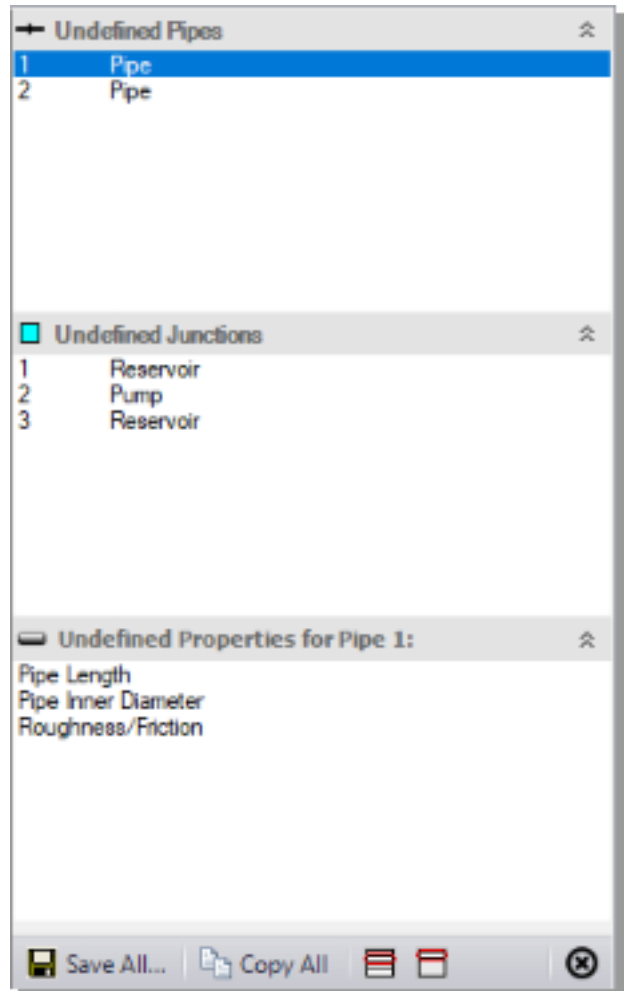
Figure 2.14   The List Undefined Objects window lets you see the undefined properties for each undefined object

A. Define the J1 Reservoir

➢ **To define the first reservoir,** open the J1 Reservoir Properties window (Figure 2.15) by double-clicking on the J1 icon. The pipe table should show you that Pipe 1 connects 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.

➢ **On the Reservoir Model tab, enter a Liquid Surface Pressure of 0 barG (0 kPa (g)), Pipe Depth of 3 meters, and a Liquid Surface Elevation of 3 meters.** 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. For this example, keep the units as specified above.
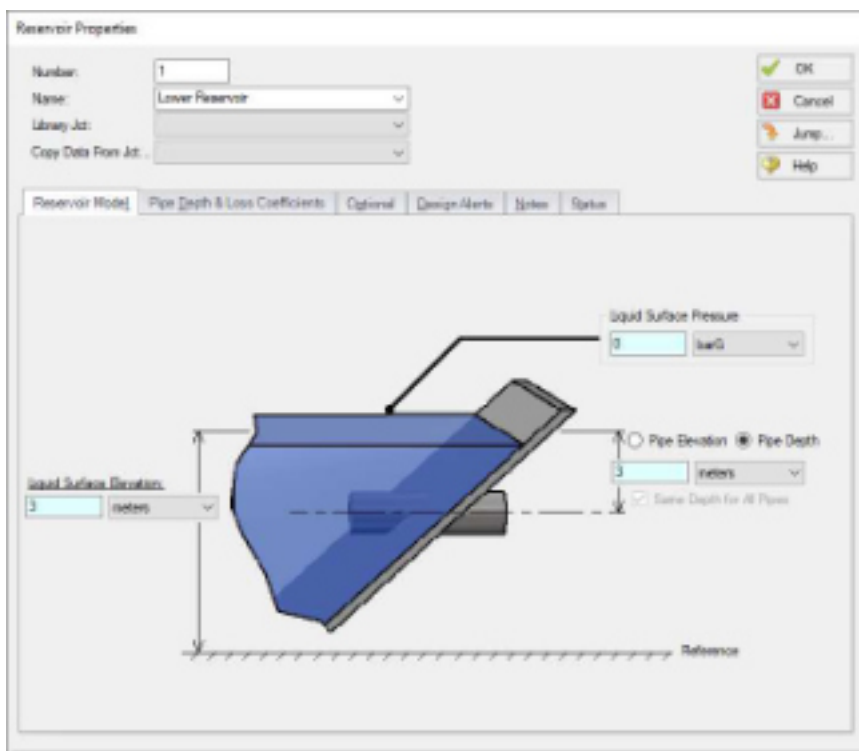
Figure 2.15   Properties window for the J1 Reservoir

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

You can give the junction a name, if desired, by entering it in the Name field at the top of the window. By default, the junction's name indicates the junction type, and will be displayed in the Model Data and Output. It can optionally be displayed on the Workspace via the use of Workspace Layers which will be discussed later in this chapter. Name this reservoir Lower Reservoir, as shown in Figure 2.15.

You can add most junction types into a custom library. This saves the junction inputs for repetitive use within the current model, a completely separate model, or with another user. To select a junction from the custom library, choose the desired junction from the Library Jct list in the junction's property window. The current junction will get the properties from the library.

The Copy Data From Jct list at the top of the properties window shows all the junctions of the same type in the model. This copies the user-selected parameters from an existing junction in the model to the current junction.

Using the tabs in the properties windows

Separate tabs in the Properties windows contain the junction information and input fields. Click any tab to bring its information forward. Figure 2.15 shows a Reservoir junction Properties window.

With only one pipe connected to the reservoir, enter the depth directly on the diagram on the Reservoir Model tab. Alternatively, enter the depth, regardless of the number of pipes, on the Pipe Depth & Loss Coefficients tab (also called the pipe table). The pipe table allows you to specify entrance and exit loss factors for each pipe connected to the reservoir (only one in this case). AFT Fathom defaults to lossless entrance and exits.

➢ **Confirm the pipe depth of 3 meters in the pipe table.**

The Optional tab allows you to enter different types of optional data. Depending on the junction type, you can enter initial guesses for the Solver, specify Design Factors, set Special Conditions (like closing a valve or turning off a pump), and change the junction icon.

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

The highlight feature (on by default) displays all the required information in the Properties window in light blue. You can toggle the highlight on and off by pressing F2 with the properties window open or by double-clicking an empty area above the tabs of the properties window. The highlight feature can also be toggled on and off using the View menu, Highlight in Pipe and Jct Windows option.

➢ **Click OK** to close the Reservoir Properties window. With Show Object Status turned on, you should see the J1 ID number and name turn black again, informing you that the junction is fully 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 2.16.

Inspecting is a faster way of examining the input (and output if results are available) than opening the Properties window.
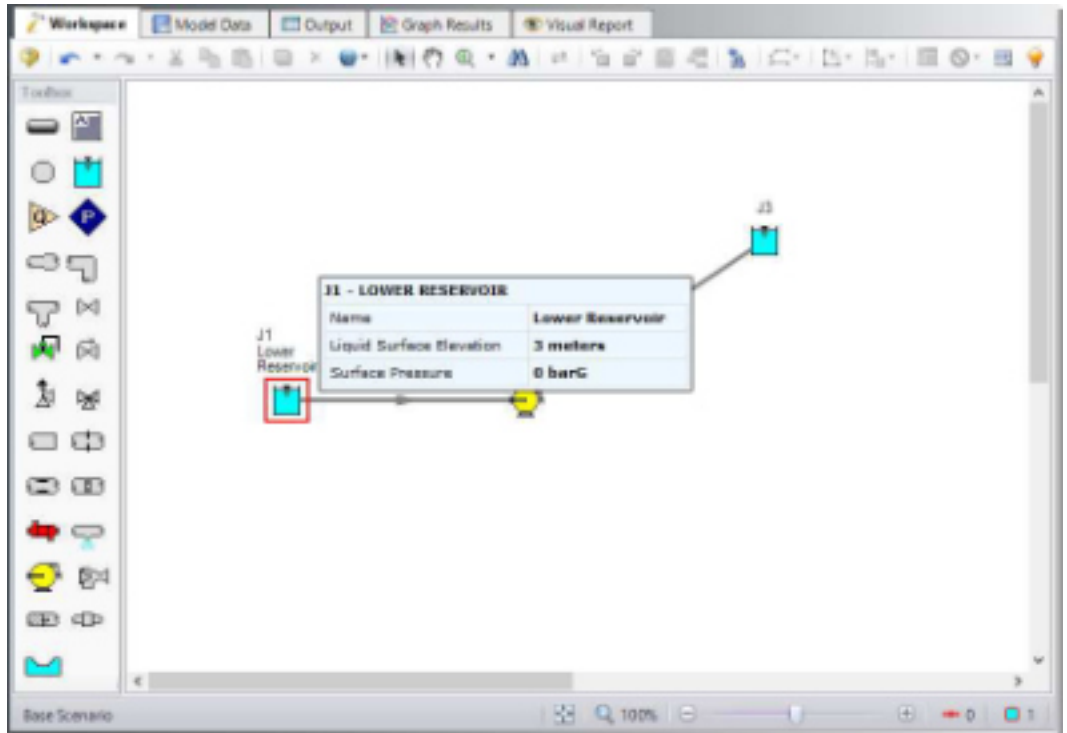
Figure 2.16   Inspect junction properties from the Workspace by clicking and holding the right mouse button

B. Define other junctions

➢ **Next, open the Properties window for the J2 Pump junction.**
Remember this example sizes a centrifugal pump. By default, the junction assumes a Centrifugal (Rotodynamic) pump, so select Sizing for the Analysis Type. For the Pump Sizing Parameter, use Volumetric Flow Rate, and enter the Fixed Flow Rate as 115 m3/hr. Set the junction Inlet Elevation as 0 meters (see Figure 2.17). Click OK to close the window.
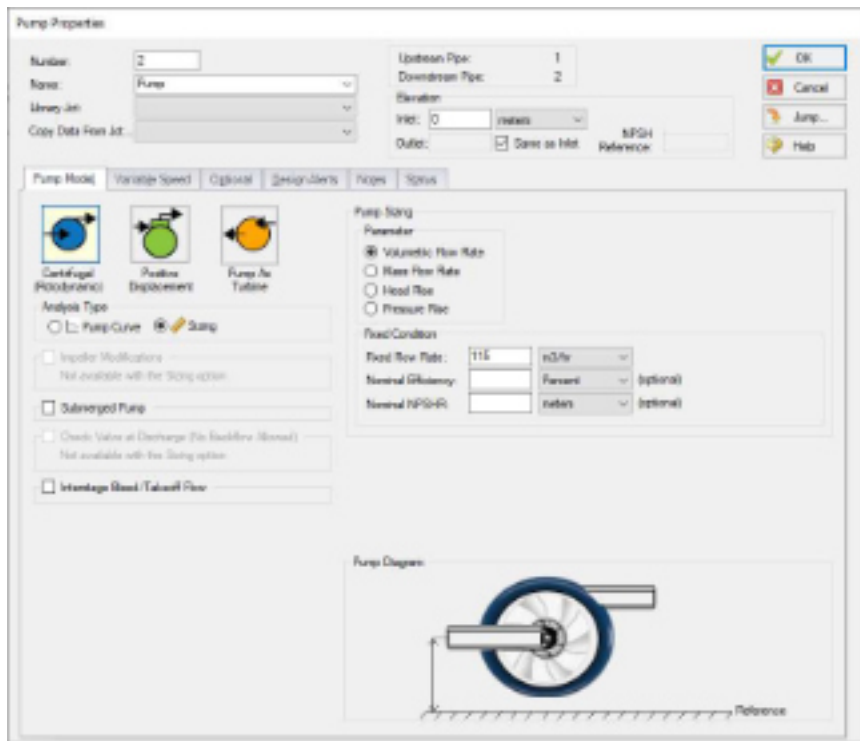
Figure 2.17   Properties window for the J2 Pump

➢ **Open the J3 Reservoir junction** and change the name to Upper Reservoir. Then enter a Liquid Surface Pressure of 0 barG (0 kPa (g)), a Liquid Surface Elevation of 60 meters, and a Pipe Depth of 3 meters. Click OK.

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

C. Define Pipe P1

You can enter data for pipes and junctions in any order. In this example, we defined the junctions first. Next, define all the pipes. To open the Pipe Properties window, double-click the pipe on the Workspace.

➢ **First, open the Pipe Properties window for Pipe P1.** The Pipe Properties window offers control over all important hydraulic parameters that are related to pipes. For Pipe P1, click the dropdown

arrow for Pipe Material to select Steel - ANSI. Next, for Size, select 4 inch (10.2 cm ID), and for Type select STD (schedule 40). For the Friction Model Data Set, select Standard. Finally, enter a length of 3 meters (see Figure 2.18) and click OK.
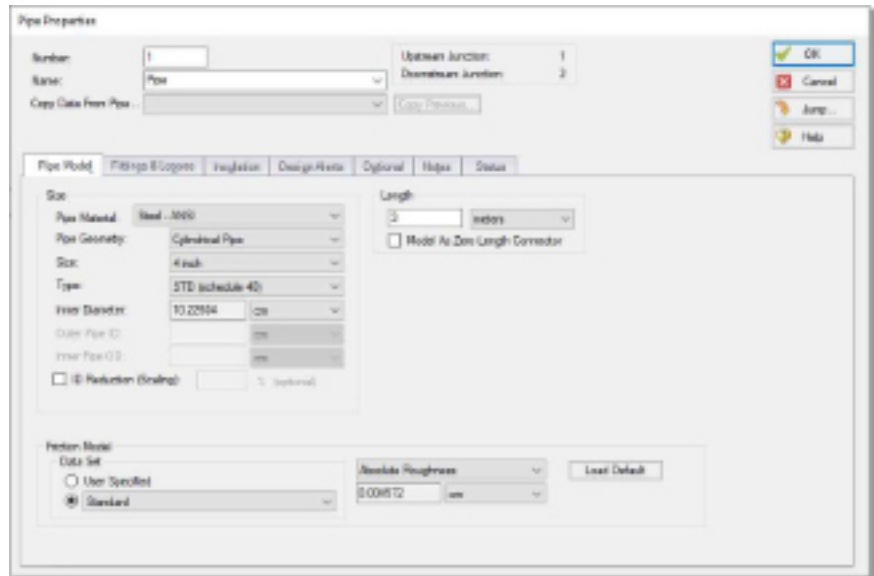
Figure 2.18   Properties window for Pipe P1

The Inspect feature capability expands to *within* the Properties window of each pipe (and certain junctions). Use this 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 junction that states the upstream and downstream pipe in the junction's Properties window. You can jump to that junction's Properties window by double-clicking the connected junction's ID number, or use the Jump button to jump to any other part of the model.

D. Define Pipe P2

➢ **Open the Properties window for Pipe P2** and select Steel - ANSI, 4 inch, STD (schedule 40), Standard friction model, and enter 300 meters in length. Note that some of these fields already have values populated based on the input you entered from Pipe P1.

This pipe has elbows and fittings to specify. For simplicity, represent these as a total K Factor of 25. Enter this on the Fittings & Losses tab in the Total K Factor field (Figure 2.19). Click OK to save your changes.

Note: A red ampersand symbol (&) will now appear next to the pipe number for Pipe P2. This indicates the pipe has Fittings & Losses entered. If desired, the symbol can be hidden via Workspace Layers. Furthermore, the symbols can be edited or disabled from the User Options.
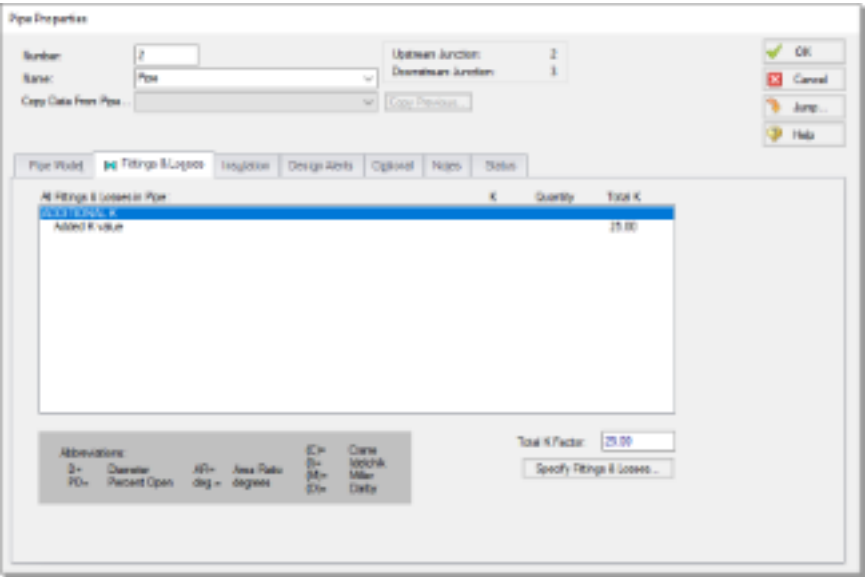

Figure 2.19   Losses in pipes due to fittings are entered on the Fittings & Losses tab in the Pipe Properties Window

E. Check pipes and junctions

➤ **Check for any remaining undefined pipes or junctions.** If all the panels in the Analysis Setup have their required inputs, the Status Light on the Quick Access Panel will be green. If not, view the Undefined Objects list from the Analysis Setup, or turn on Show Object Status from the View menu, and open each undefined pipe and junction. The Status tab on each Properties window will indicate the missing information.

F. Review Model Data

➤ **Before running the model, save it one more time.** Review the input using the Model Data window.

Navigate to the Model Data window shown in Figure 2.20 by clicking on the Model Data tab or pressing CTRL+M. The Model Data window gives you a table-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to a PDF, or printed out for review.

Data displays in three sections: the General Data section at the top, the Pipe Data section in the middle, and the Junction Data section at the bottom. Each section collapses 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 cell in the row of the pipe or junction you want to access.
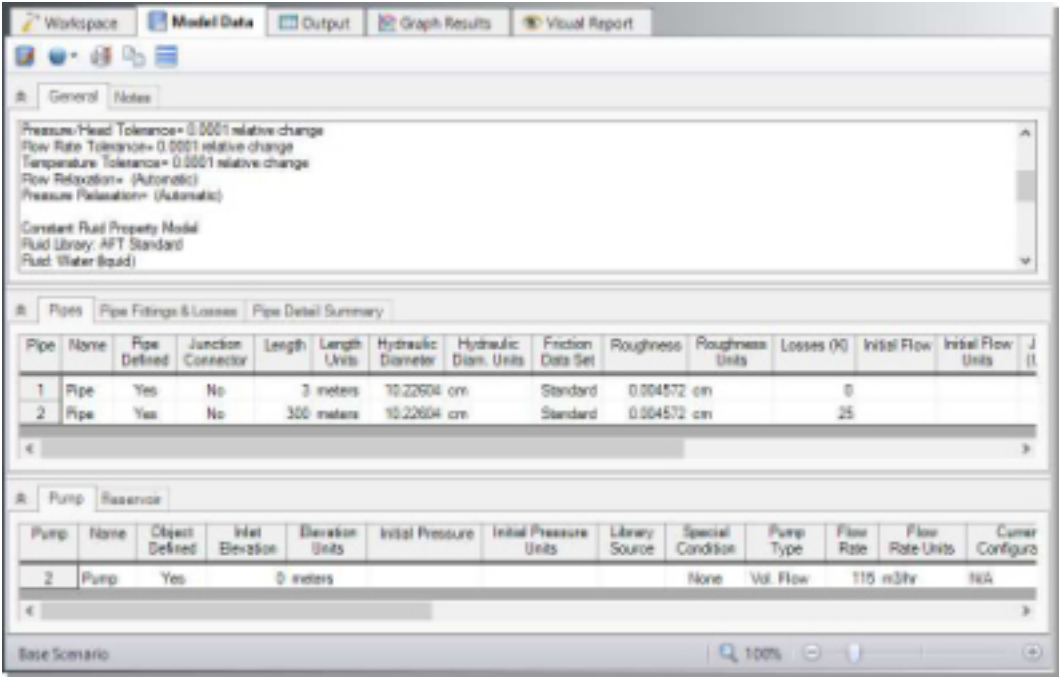
Figure 2.20   The Model Data window shows all input in table form

Step 5. Run the model

> ➤ **Run the Model: click the green arrow icon on the Common Toolbar,** press CTRL+R, or open the Analysis menu and click Run Model. During execution, the Solution Progress window is displayed and can be used to pause or cancel the Solver's activity (see Figure 2.21). When the solver finishes, click the Output button and the table-based Output window will appear (see Figure 2.23). Besides just viewing the results in the Output window, you can save to file, export to Excel or a spreadsheet-ready format, copy to the clipboard, print to PDF, or print out on the printer.
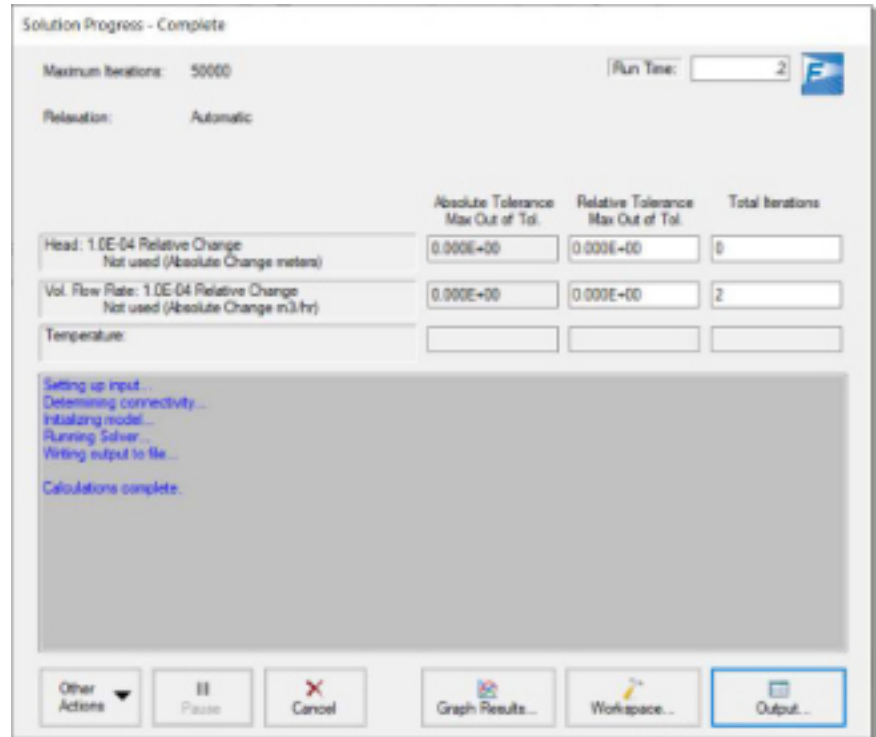
Figure 2.21   The Solution Progress window displays the Solver's convergence progress

The output file

When the Solver runs, the output data is written to a file. This file is given the same name as the model itself with a number appended to the name, and with an *.out extension appended to the end. For all data processing, graphing, etc., the data is extracted from this file. The number is appended because users are able to build different scenarios all within this model. Each scenario will have its own output file; thus, the files need to be distinguishable from each other.

The output file will only remain on disk if you save the model after it has been run. Once saved, it will remain until the user erases it or the model input data is modified.

➤ **Save the model file again** such that the output file is saved to the disk. This means that if you were to close your model right now and then

reopen it, you could proceed directly to the Output window for data review without rerunning your model.

Step 6. Customize the output

Select the specific output parameters and units that you want in your output report using the Output Control window (Figure 2.22).

➢ **Select Output Control** from the Common Toolbar or Tools menu to open the Output Control window. On the Display Parameters tab, Pipes output type, the right-hand side lists the pipe output parameters to display in the output.

Click the General tab, enter the title Water Transfer System, then click OK to save the title.

If you do not change any of the Output Control settings, the output uses the default Output Control parameters and title.
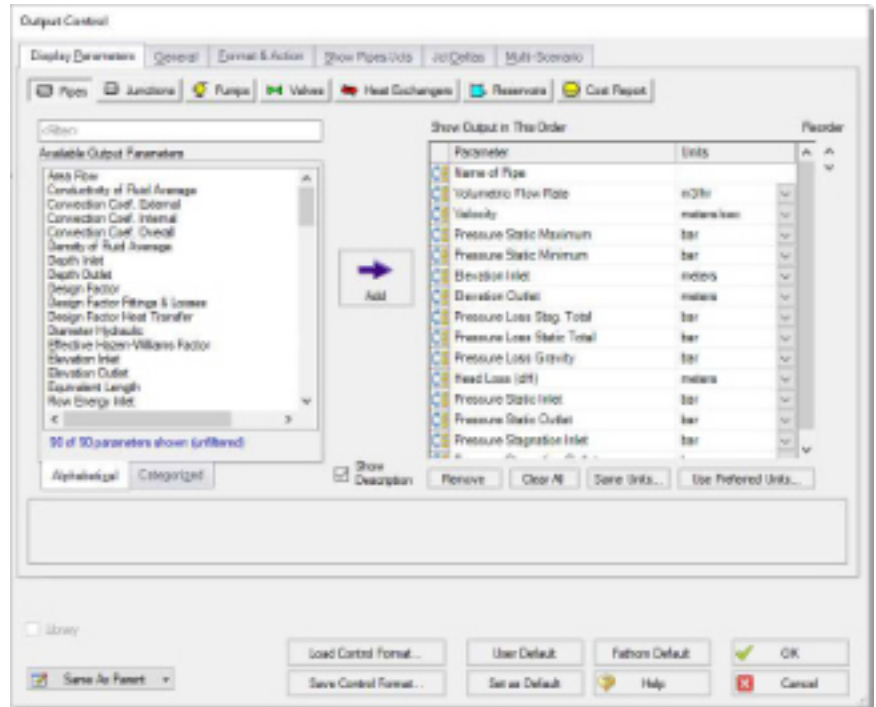
Figure 2.22   Customize the output using the Output Control window

Step 7. Review the output

Similar to the Model Data window, the Output window (Figure 2.23) has a similar structure with three sections. You can expand or collapse each section by dragging the boundary between the sections up or down or by clicking the arrow beside the section label. The parameters and units chosen in the Output Control display in the tables.
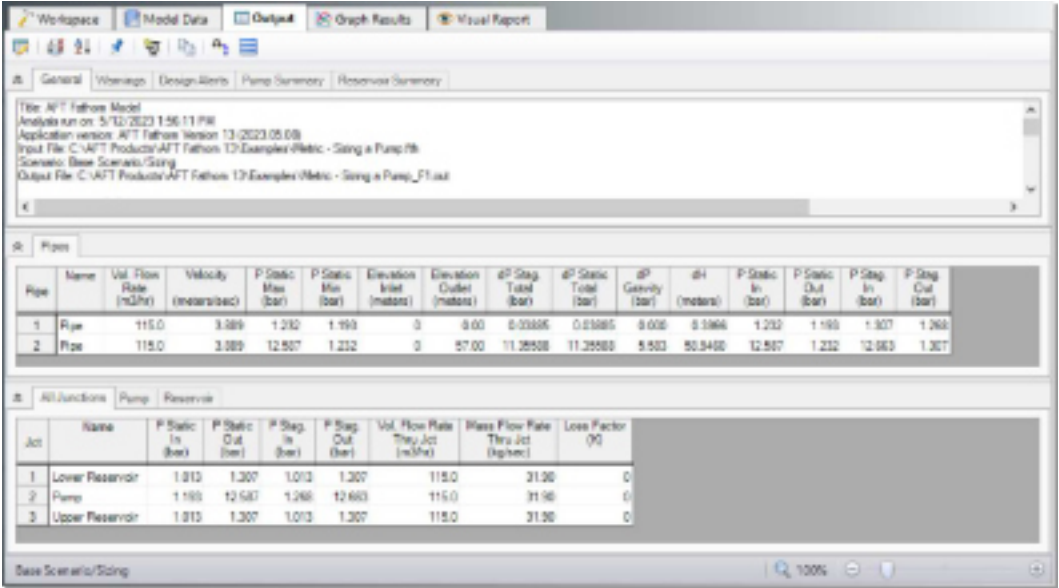
Figure 2.23   The Output window displays output in table form

With pumps in the model, AFT Fathom gathers important pump output together in one convenient location in the Output window on the Pump Summary tab. Go to this tab located in the General Output section of the Output window, as shown in Figure 2.24.

The Pump Summary shows that the pump needs to generate 116.3 meters of head. The *minimum* required head rise equals the 57-meter elevation difference of the reservoirs regardless of the flow rate or pipe size. This leaves approximately 59 meters of friction loss through the pipes. The dH column on the Pump Summary tab of the Pipe output window shows this output.

The Pump Results Diagram shows the Pump Summary output in diagram form, as shown in Figure 2.25. Select the "…" button next to Show in the Results Diagram column, double-click anywhere in the row, or right-click anywhere in the row and select Pump Results Diagram to view the diagram.

Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show ☐	Pump	115.0	31.90	11.39	116.3	100.0	N/A	36.38	N/A	N/A	12.69	N/A

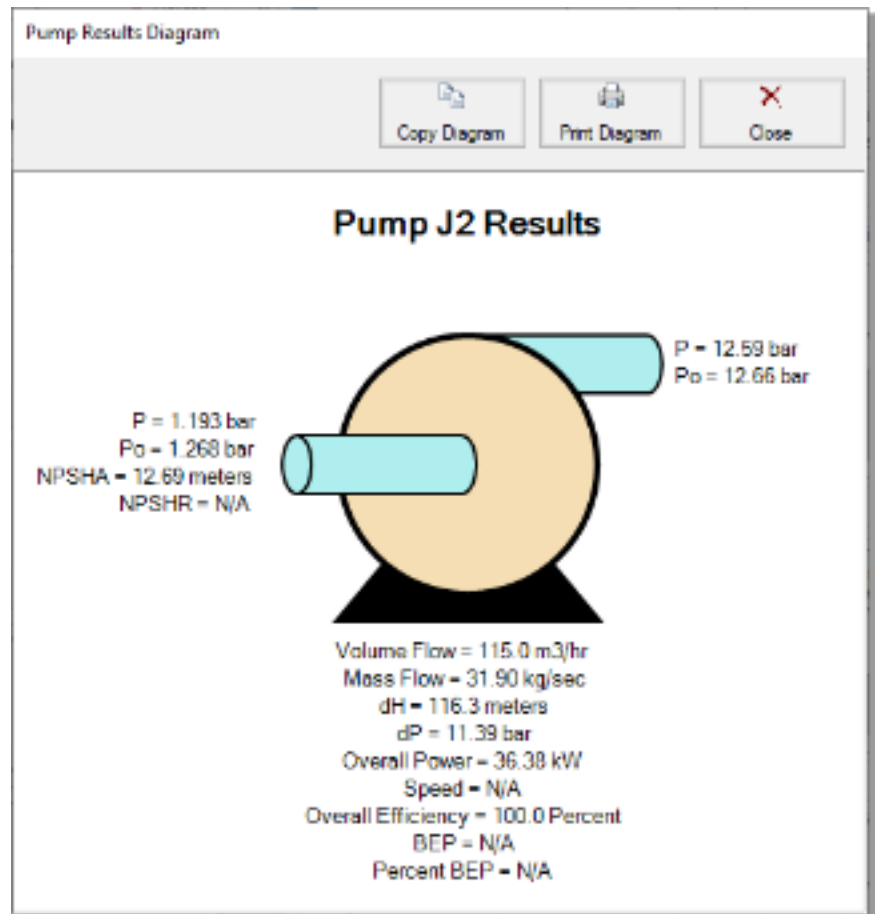Figure 2.24   The Output window displays the Pump Summary tab when a model has pumps in it

Figure 2.25   Pump Results Diagram as opened from the Pump Summary (see Figure 2.24

A. Modify the output format

The Output Control window allows you to obtain the parameters, units, and display 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.

By default, the Pipe Results table in the output shows the volumetric flow rate in the second column with units of m3/hr. Next, we will modify the units and order of parameters shown in the output.

➢ **Select Output Control from the Common Toolbar** or from the Tools menu to open the Output Control Window (see Figure 2.22). On the Display Parameters tab, Pipes output type, the right-hand side lists the pipe output parameters to display in the output. Change the units for Volumetric Flow Rate by selecting the dropdown arrow next to m3/hr, then selecting m3/sec (cubic meters per second) from the unit list.

➢ **Click OK to display changes to the current results.** You should see the volumetric flow rate pipe results section, still in the second column, in units of m3/sec. Notice the Velocity results in the third column.

➢ **Select Output Control from the Common Toolbar again.** On the Display Parameters tab, Pipes output type, 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 2.22).

➢ **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.** The first column of the Pipe Results table now displays Velocity, the second displays Pipe Name, and the third displays Volumetric Flow Rate.

➢ **Select Output Control from the Common Toolbar** one more time. On the Display Parameters tab, Pipes output type, select Volumetric Flow Rate from the list on the left. Click the Add button to add it to the list on the right and make sure its units are m3/hr. Then use the reorder button or row drag-and-drop to move it below the previous Volumetric Flow Rate parameter.

➢ **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 opens a window to change the units of the column without opening the Output Control window (see Figure 2.26). These changes are automatically updated in the Output Control window. Keep the selected units as is, and click OK to close the window.
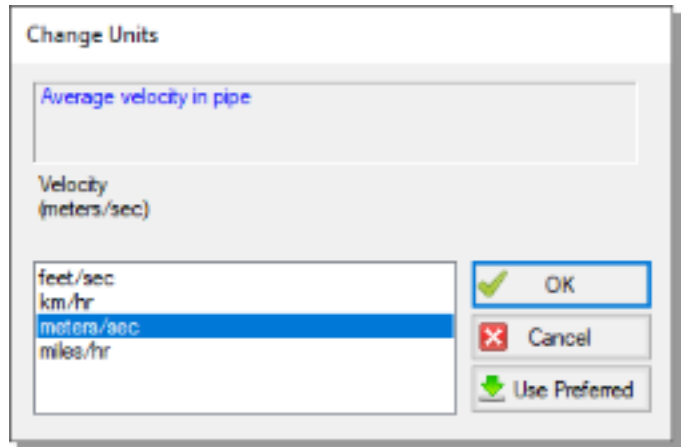
Figure 2.26   Open the Change Units window from the Output window tables by double-clicking the column header

B. View the Workspace Layers

Workspace Layers can display input and output values superimposed over the model, and color pipes automatically based on output values.

➢ **Navigate to the Workspace window** by clicking on the Workspace tab or pressing CTRL+W. Select the Workspace Layers tab in the bottom-right of the Quick Access Panel. The Workspace Layers panel contains two sections named Layer Presets and Layers.

Within the Layers section, there are two expandable or collapsible groups which contain the two *layer types* named Color Maps and Standard layers. We will integrate the model results with the graphical layout of the pipe network by using the All Objects Layer located under the Standard group. The All Objects Layer is the bottommost layer for

all models and shows all pipes, junctions, and annotations when the layer is visible. Layer visibility can be toggled on/off using the eye icon next to the layer. The combination of layers and toggling visibility allows for a wide range of visual customization, and the All Objects Layer will always provide a safety net to keep track of all objects in the model.

➢ **Open the Layer Settings window for the All Objects Layer** by double-clicking the All Objects Layer, clicking the gear icon to the left of the layer name, or selecting the layer and clicking the single gear icon on the Layers Toolbar (Figure 2.27).

The Layer Settings window provides flexible options for customizing the appearance of the model. The Pipes & Junctions group allows you to show or hide junctions, and change display properties like color, size, and thickness. Note that you cannot hide individual objects on the All Objects Layer, but you can hide objects on new layers, and then hide the entire All Objects Layer below. The Label Control group allows you to display input and output parameters next to objects on the Workspace. The Annotations group allows you to customize annotations.
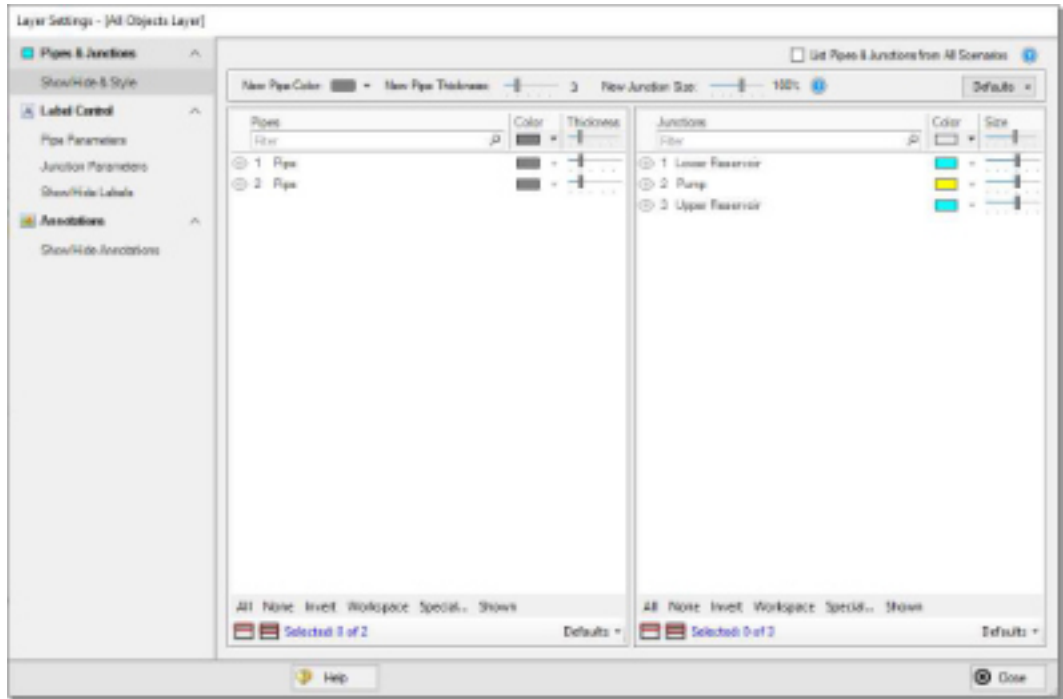
Figure 2.27   The Layer Settings window specifies content to be superimposed on the Workspace

➢ **Select the Pipe Parameters item under the Label Control group.** The left pane lists input and output parameters available for display, and the right pane lists the parameters currently being displayed by the layer on the Workspace.

➢ **In the left pane, expand the Pipe Parameters list.** Then, scroll to the bottom of the list and select the Volumetric Flow Rate parameter. Add it to the right pane by either double-clicking the parameter, or selecting the parameter and clicking the arrow between the panes. Set the units to m3/hr.

➢ **Select the Junction Parameters item under the Label Control Group.** Expand the Junction Parameters list and add the Pressure Static Inlet parameter with units of barG. Expand the Pump Summary Parameters list and add the Head Rise parameter with units of meters. Expand the Commonly Used Junction Parameters list and add the

Junction Name parameter. Select the Junction Name parameter and reorder it above the pressure and head parameters using the up arrow in the top-right. Close the Layer Settings using the button in the bottom-right.

➢ **Add additional clarity by repositioning the labels and adding a legend.** It is common for the text on the Workspace Layers to overlap with objects when first generated. Labels can be dragged to a new area or a smaller font can be selected using the Font Decrease icon on the Workspace Toolbar. Parameter units can be consolidated in a legend which can be added via the Global Layer Settings. In the Layers Toolbar, click the double gear icon to open the Global Layer Settings. Then, enable the checkbox for Show Units in Legend. The result of these adjustments is shown in Figure 2.28.
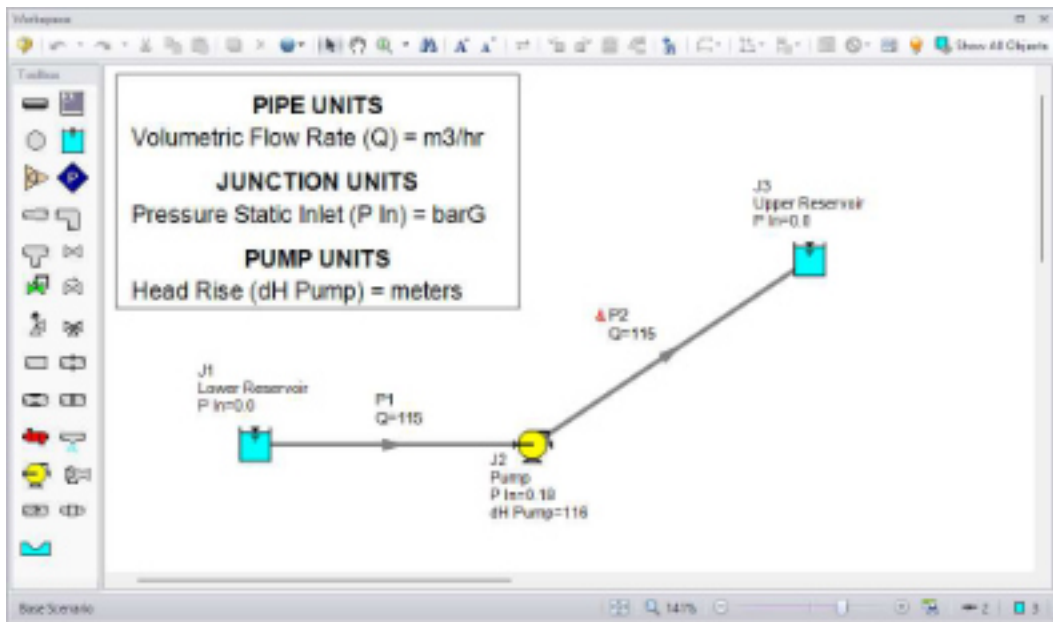

Figure 2.28   Results are superimposed on the model layout via Workspace Layers
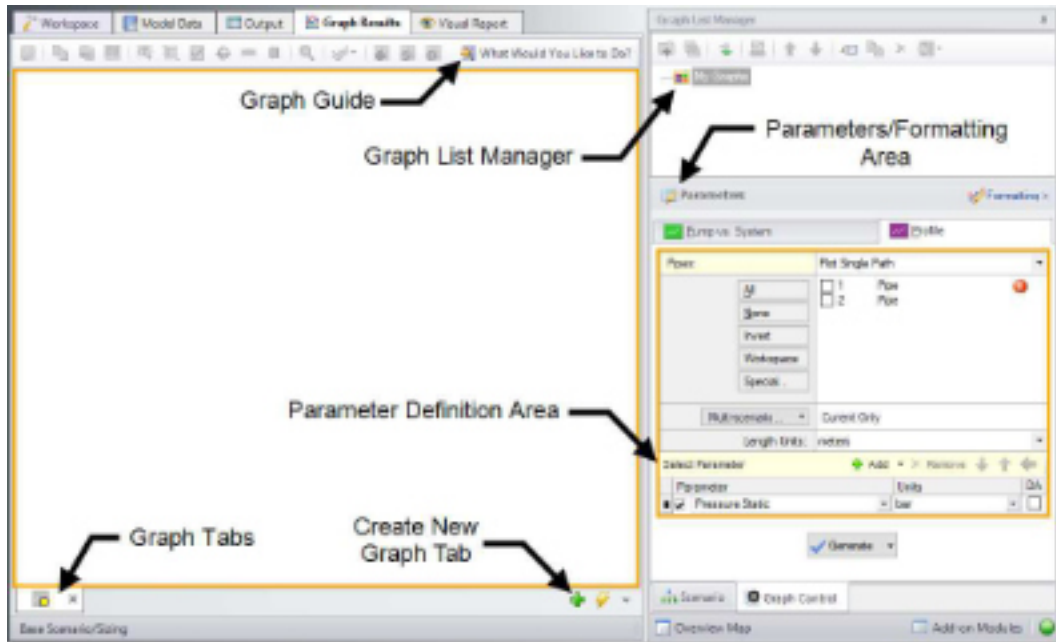
C. Graph the results

➤ **Navigate to the Graph Results window** by clicking on the Graph
Results tab or pressing CTRL+G. The Graph Results window offers
versatile graph generation and 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 2.29). 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 2.30 shows the Graph Results window with the Graph Control
tab enabled on the Quick Access Panel.
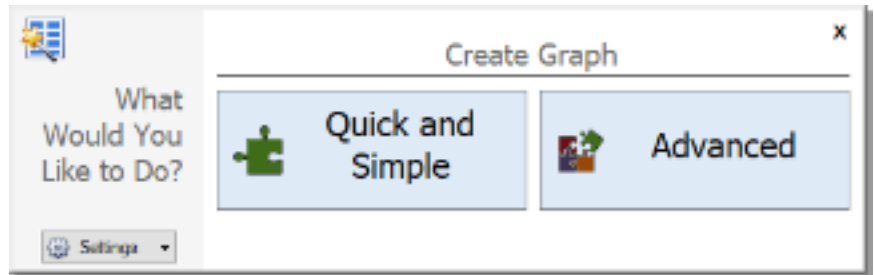
**Figure 2.29   The Graph Guide can be toggled on and off by
clicking the What Would You Like to Do? button on
the Graph Results window**

Figure 2.30   The Graph Results window 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 helps show the output of multiple hydraulic parameters in the same location of your model simultaneously without creating completely separate graphs.

Create stacked graphs of the pressure and flow along the pipeline.

➢ **Click the Profile tab on the Parameters pane.** In the Pipes selection list, click the All button to select all the pipes.

➢ **For the Length Units,** keep the default units of meters.

➢ **From the Parameters definition area, keep Pressure Static selected** and select units of bar.

➢ **To add a stacked graph,** click the Add button with the green "+" icon next to Select Parameter, shown in Figure 2.31. A new row under the Parameters definition area will appear.

➢ **Choose Volumetric Flow Rate in this new parameter row,** and keep the default units of m3/hr.

➢ **Click Generate** to create the stacked static pressure and volumetric flow rate graphs, shown in Figure 2.32. Note that the pump adds approximately 12 bar of pressure in order to maintain a flow of 115 m3/hr throughout the pipeline.

➢ **To format the legend font size,** right-click on a legend and use the slider bar to decrease the font size to 12 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 slider bar to decrease the size until the font size on each axis appears as you like.
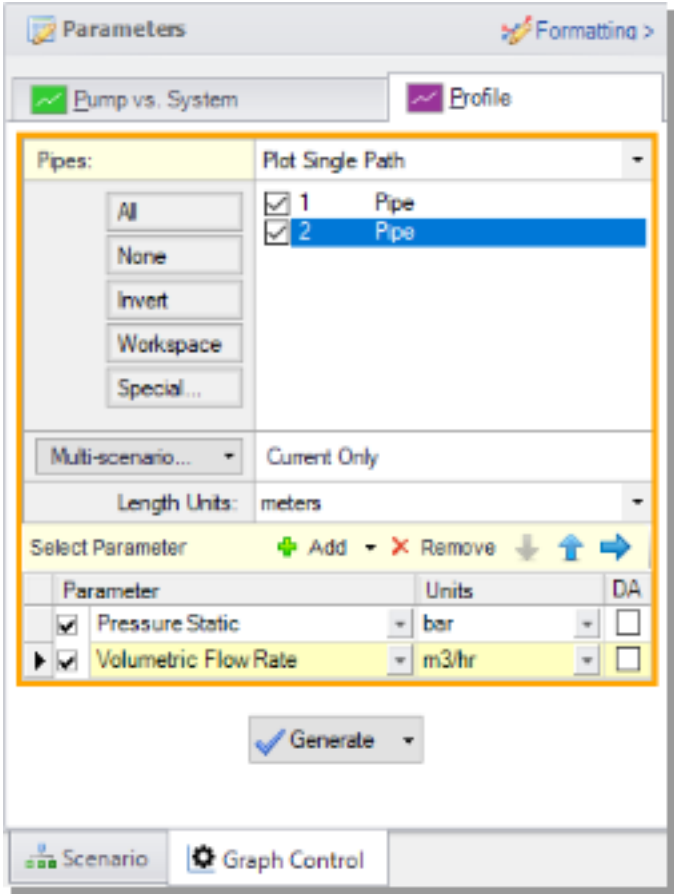
Figure 2.31   The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area

The Formatting area on the Graph Control tab can be used to modify the graph colors, fonts, and other elements. Options to copy the graph image or x-y data are available on the Graph Results Toolbar or by right-clicking the graph. The graph image or x-y data can be saved from the File menu.
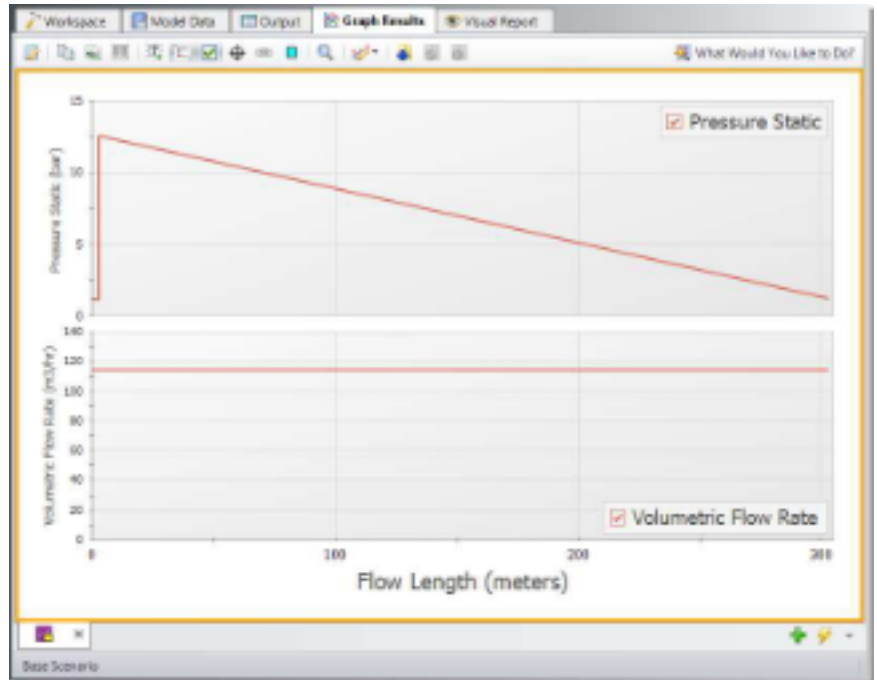
Figure 2.32   The Graph Results window showing a graph of the static pressure and volumetric flow rate along the entire length of the pipeline

Step 8. Add a pump curve

With the known pump head requirement of 116.3 meters at the design flow rate of 115 m3/hr, you can purchase a pump of the correct size to meet this requirement. With the actual pump curve, add the pump characteristics to the junction.

Open the J2 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.

Enter the following pump curve data, as shown in Figure 2.33, in the Raw Data table. After the data has been entered, click Generate Curve Fit Now to fit a curve to the data. Then, click the OK button to close the Pump Configuration window, then the OK button again to close the

Pump Properties window. If prompted, click OK to acknowledge that any previous output data will be erased.

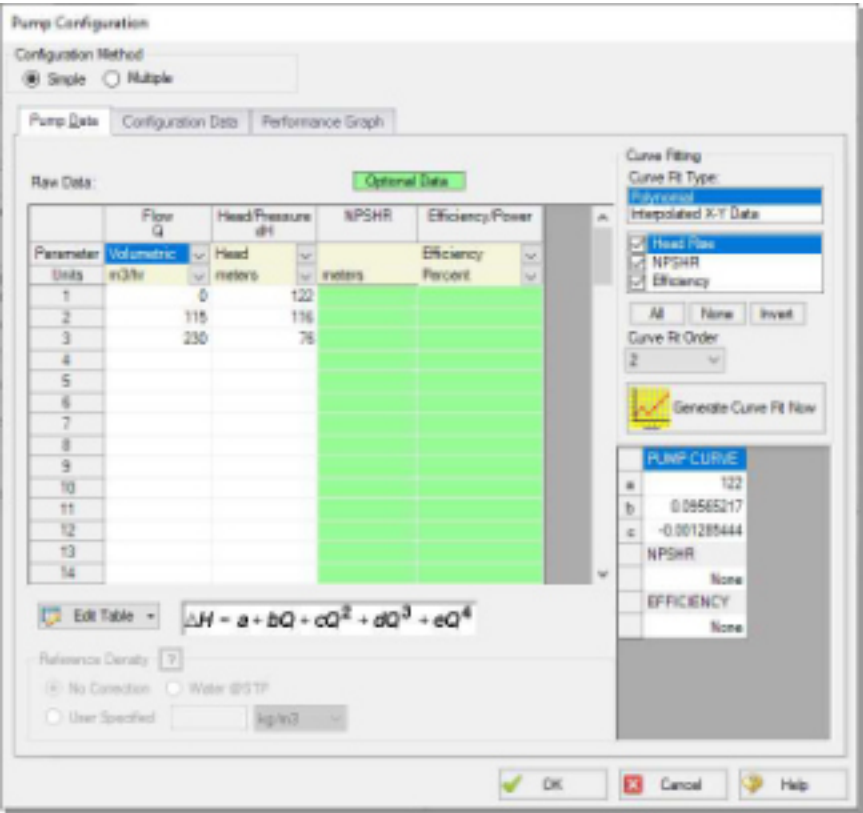| Q (m3/hr)	dH (meters)
0	122
115	116
230	76

Figure 2.33   Enter the pump curve data in the Pump Configuration window

Note: In this situation 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 the model. Examine the Pump Summary in the General Output Section. The output (Figure 2.34) shows the pump operating at a flow rate of 114.7 m3/hr and a head rise of 116.1 meters which is acceptably close to the previous sizing calculation.

Navigate to the Graph Results window and to the Pump vs. System tab on the Parameters pane of the Quick Access Panel (Figure 2.35). Generate a pump vs. system curve using the default selection as shown in Figure 2.36.
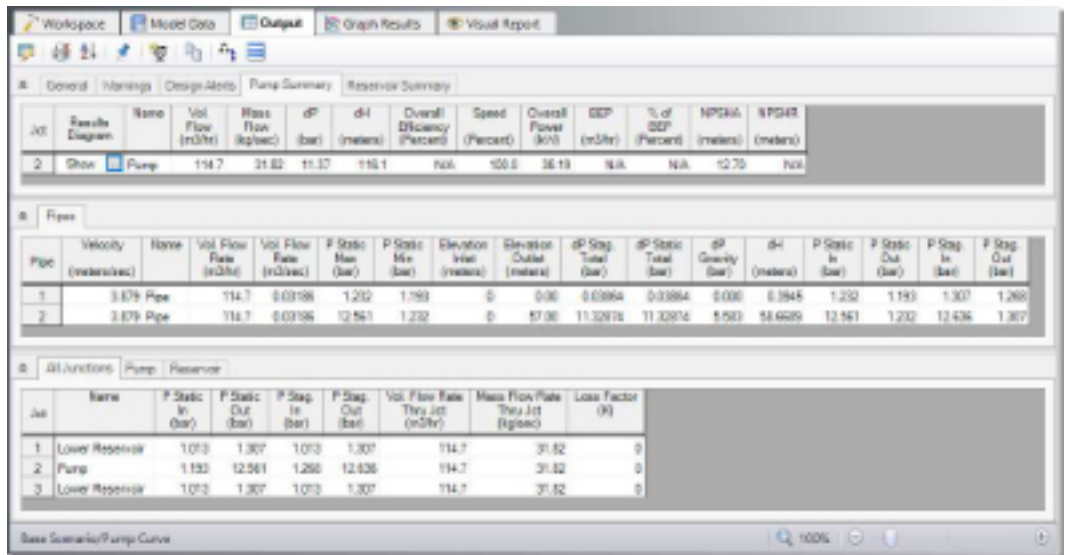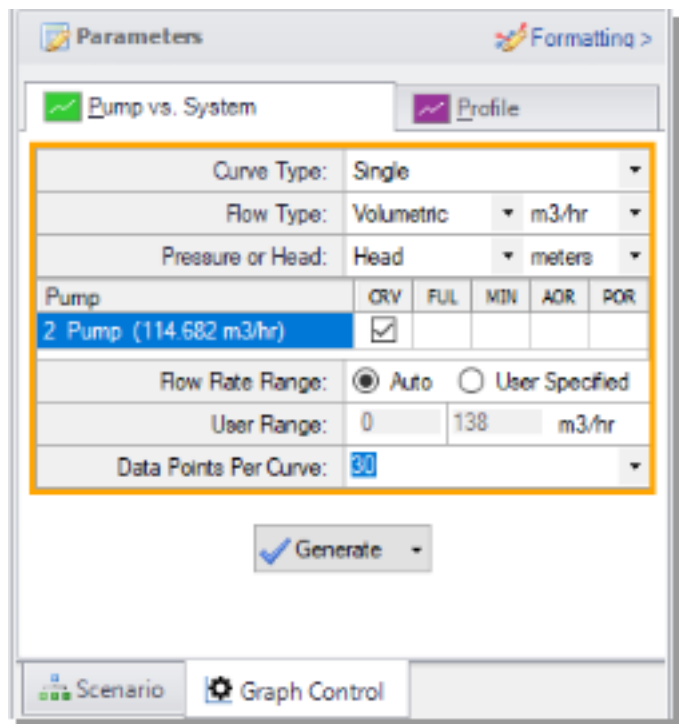

Figure 2.34   Output using a pump with a pump curve

Figure 2.35   The Graph Control tab on the Quick Access Panel can generate a Pump vs. System Curve
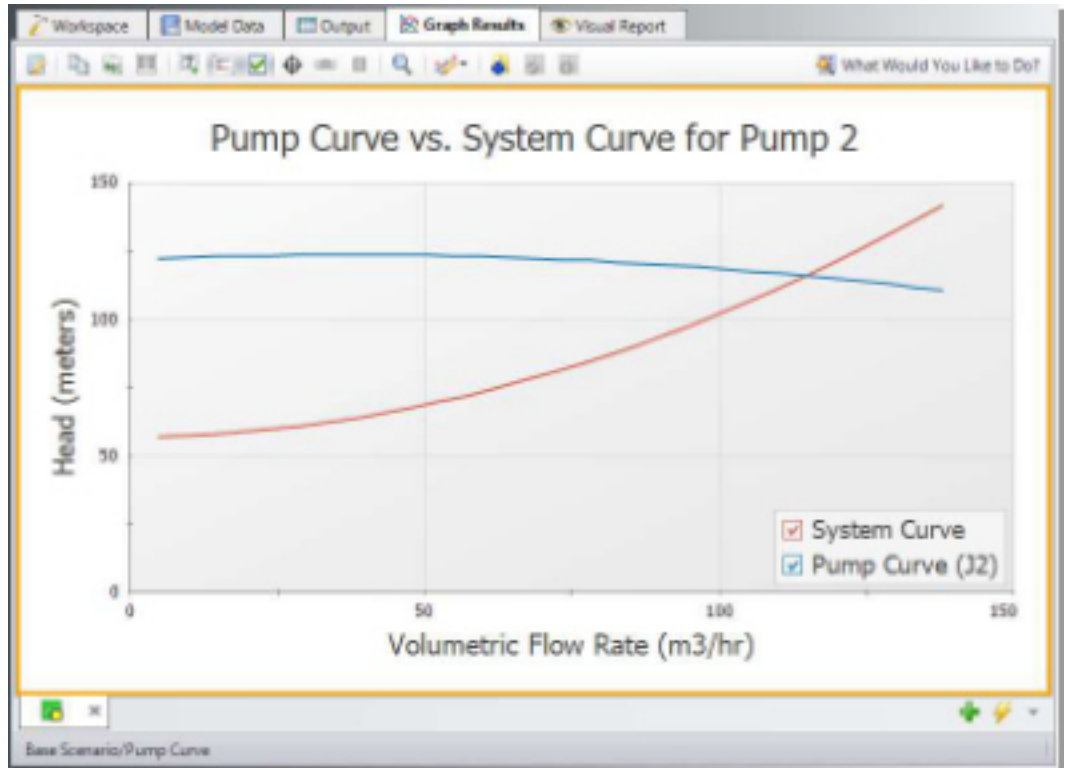
Figure 2.36   Pump Curve vs. System Curve for Pump 2

Conclusion

You have now used AFT Fathom's Primary Windows to build and analyze a simple model, and to size a pump.

Spray Discharge System Example

This example demonstrates how to find the minimum supply pressure needed to provide a system of eight spray discharge heads with at least 25 m3/hr to each spray head.

Topics covered

- Using spray discharge junctions

- Using the Global Pipe Edit and Copy Junction Data From tools

- Using manual iterative techniques to arrive at a desired result

- Using the 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 example in Chapter 2 for more information on these topics.

Model file

A completed version of the model file can be referenced from the Examples folder as part of the AFT Fathom installation:

C:\AFT Products\AFT Fathom 13\Examples

- *Metric - Spray Discharge System.fth*

Problem statement

A constant pressure source supplies a spray discharge system consisting of eight spray discharge nozzles. 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 25 m3/hr through each of the spray junctions.

Step 1. Start AFT Fathom

➢ **To start AFT Fathom,** open the all applications list in the Windows Start Menu, expand the AFT Fathom 13 folder, and choose AFT Fathom 13. This refers to the standard menu items created by a standard installation; you may have chosen to specify a different installation location.

Step 2. Specify fluid properties

1. Open Analysis Setup from the Common Toolbar or from the Analysis menu.

2. Under the Fluid Properties group, click the Fluid item to open the Fluid panel. Select the AFT Standard fluid library and then select Water (liquid) in the Fluids Available in Library list.

3. Click Add to Model to select water for use in this model.

4. Enter 21 deg. C for the temperature. The fluid properties will be automatically calculated.
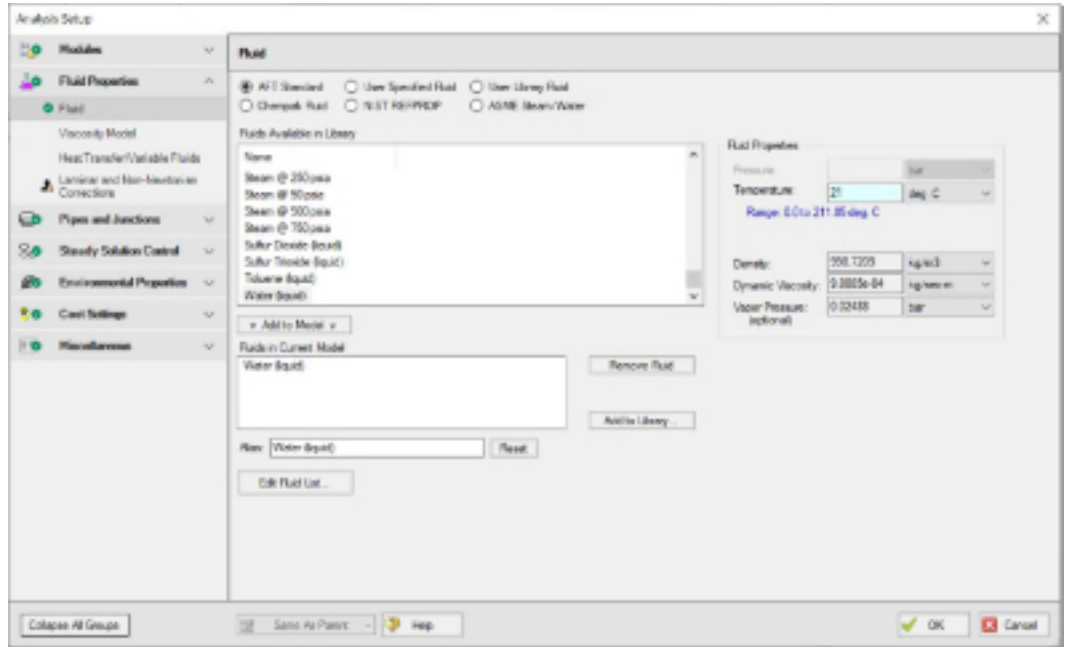
5. The Fluid panel should resemble Figure 3.1. Click OK.

Figure 3.1   Fully specified fluid panel in the Analysis Setup

Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 3.2.

Note: Multiple of the same junction type (or pipes) can be quickly placed by double-clicking that junction's icon in the Toolbox, then clicking consecutively on the Workspace to place the junctions. When finished, press the ESC key to exit the multiple object drawing mode. This is useful for placing multiple Spray Discharge junctions for this model, and for drawing many pipes rapidly.
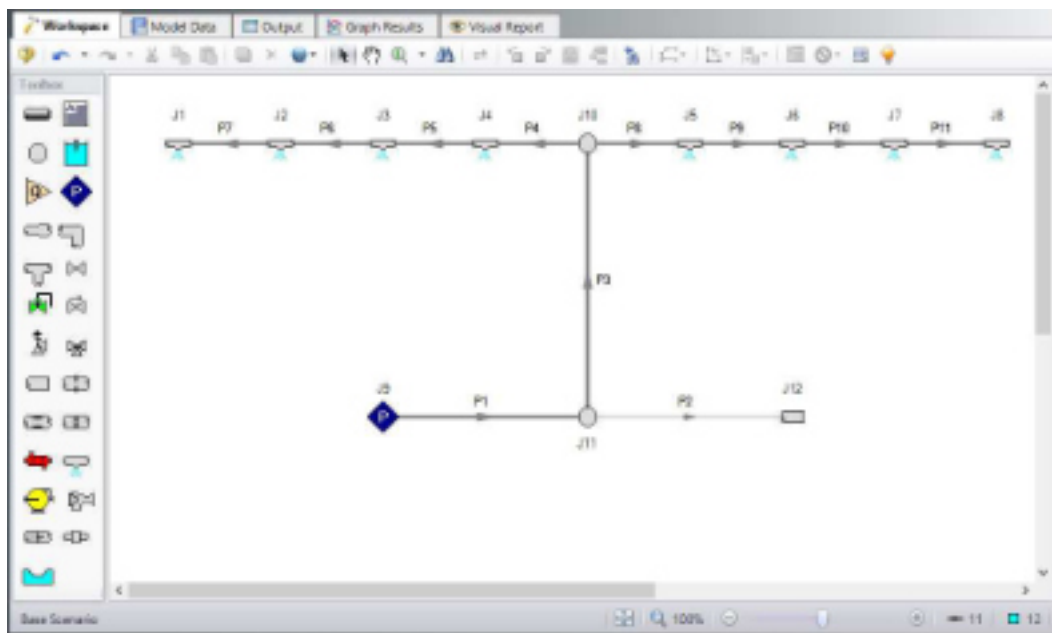
Figure 3.2   Layout of the pipe system for the Spray Discharge System Example

B. Define the pipes

With the objects in place, define the input data for the pipes and junctions. Double-click Pipes P1-P3 and enter the following inputs in the Properties window (use Global Edit, addressed below, for Pipes P4-P11).

For all pipes, use Pipe Material = Steel - ANSI, Type = STD (schedule 40), Friction Model = Standard, and the following parameters:

Pipe	Size (inch)	Length (meters)
P1	8	150
P2	8	150
P3	4	3
P4	1-1/2	3

Pipe	Size (inch)	Length (meters)
P5	1-1/2	3
P6	1-1/2	3
P7	1-1/2	3
P8	1-1/2	3
P9	1-1/2	3
P10	1-1/2	3
P11	1-1/2	3

Use Global Pipe Edit to quickly define identical pipes such as P4-P11. Global Pipe Edit applies changes to multiple pipes at the same time. Select Global Pipe Edit from the Workspace Toolbar or Edit menu. The Global Pipe Edit window opens. Click the All button above the list, then deselect Pipes P1-P3, such that only Pipes P4-P11 have the checkboxes enabled, as shown in Figure 3.3.
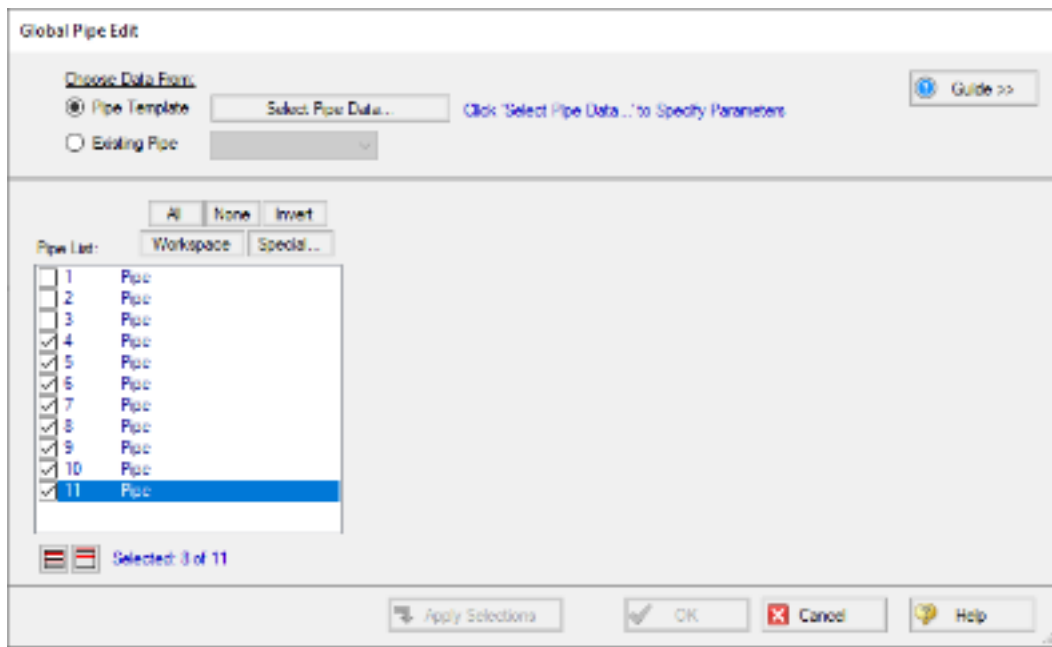
**Figure 3.3   Global Pipe Edit for the Spray Discharge System
Example**

After selecting the pipes, click the Select Pipe Data button. A template
Pipe Properties window opens to input the pipe information that will
apply to all of the selected pipes. Fill out the properties window
according to the previous table for Pipes P4-P11. After entering all of
the parameters, close the Pipe Properties window by selecting OK. The
Global Pipe Edit window now displays a list of all possible parameters
to apply to the selected pipes. The parameter categories in the list match
the tabs on the Pipe Properties window. For this example, you want all
of the parameters to apply to the selected pipes. To do this, click the All
button above the list of parameters or the check box beside each
parameter. The Global Pipe Edit window now appears as shown in
Figure 3.4.
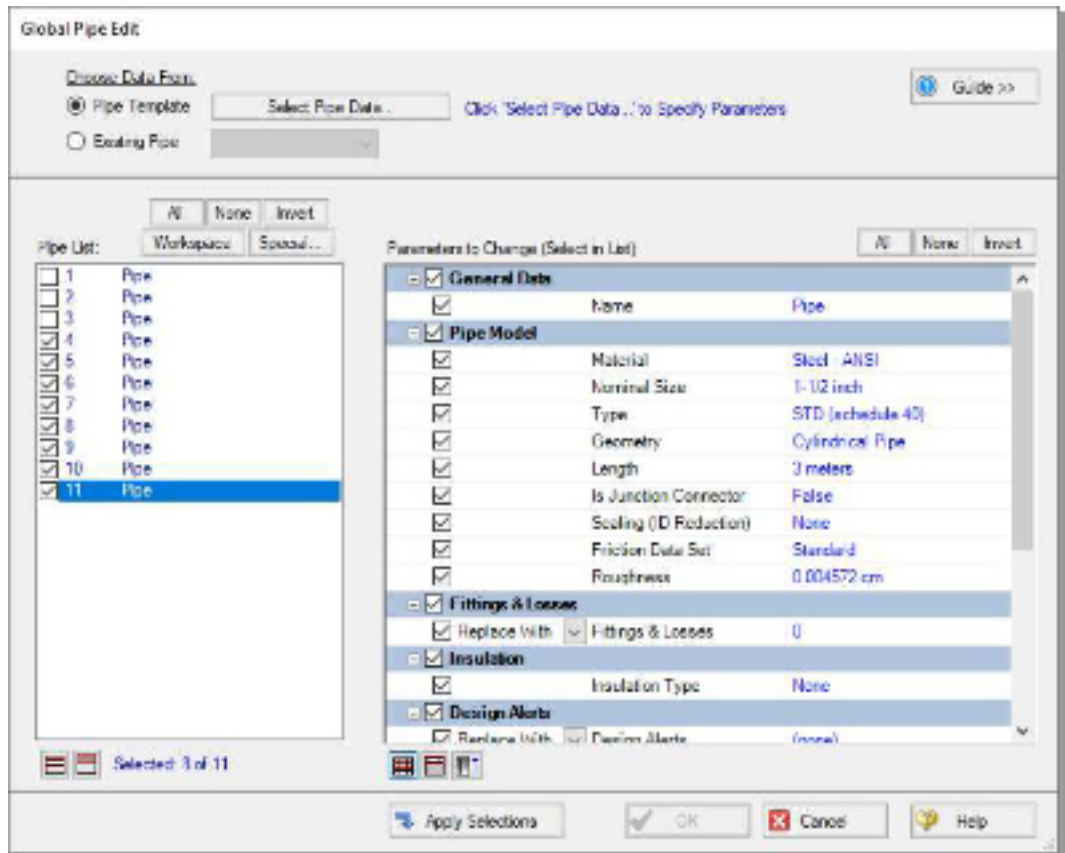
Figure 3.4   Global Pipe Edit for the Spray Discharge System Example with all pipe parameters selected for updates

Apply the changes to the selected pipes by clicking the Apply Selections button. AFT Fathom will notify you that the changes were successfully applied with a message in the bottom-left of the Global Pipe Edit window. If you wish to cancel all of the changes you applied, click Cancel on the Global Pipe Edit window. Otherwise, accept the changes by clicking OK.

Note: The Guide button in the Global Edit windows may be used as a reminder of the required steps for the global edit process if needed.

C. Define the spray discharge junctions

All of the spray discharge junctions have the same input. Use the Global Junction Edit tool to define all of the spray discharge junctions at the same time. The Global Junction Edit tool works in much the same way as the Global Pipe Edit tool.

Another tool to speed up the time it takes to define objects is the Copy Data From Junction tool. Define the J1 Spray Discharge junction, as listed below.

1. Elevation = 3 meters

2. Loss Model = Cd Spray (Discharge Coefficient)

3. Geometry = Spray Nozzle

4. Exit Properties = Pressure

5. Exit Pressure = 0 barG

6. Cd (Discharge Coefficient) = 0.6

7. Discharge Flow Area = 3.5 cm2

After defining J1, click OK to close the properties window, and open the Properties window for spray discharge J2. Locate the drop-down box near the top of the properties window labeled Copy Data From Jct. This box lists all of the spray discharge junctions on the Workspace. Select Junction 1 from the list. This will open the Copy Data From Junction 1 window, which, similar to the Global Edit window, lists all parameters for J1. Check the box beside a parameter to apply the parameter value to J2. In this case, you want all of the parameters for junction J1 to apply to J2, so select the All button at the top of the list to select all of the parameters, as shown in Figure 3.5.

Figure 3.5   Copy Data from Junction window used for the Copy Data from Junction tool

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

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

D. Enter other junction data

J9 - Assigned Pressure

1. Elevation = 0.3 meters

2. Pressure Specification = Stagnation

3. Pressure = 14 barG (1400 kPa (g))

J10 - Branch

Elevation = 3 meters

J11 - Branch

Elevation = 0.3 meters

J12 - Dead End

Elevation = 0.3 meters

E. Check if the pipe and junction data is complete

Check the Model Status light in the bottom-right of the Quick Access Panel. A green light at this point indicates a fully defined model. A red light indicates that there are undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

Step 4. Run the model

Click Run Model from the Common Toolbar or Analysis menu. The Solution Progress window opens and begins iterating. This window displays the AFT Fathom Solver converging on the answer. After the solver converges, view the results by clicking Output at the bottom of the Solution Progress window.

Open the Output Control from the Common Toolbar or Tools menu and select the Junctions output type. Add the Vol. Flow Rate Net at Junction parameter by double-clicking the parameter name in the list on the left, then clicking OK. In this example, the net volumetric flow rate represents the flow rate exiting the nozzle to the ambient surroundings.

Figure 3.6 Output Window for the Spray Discharge System Example

The Output window contains all the results specified in the Output Control window. The output for this example, shown in Figure 3.6, displays a supply pressure of 14 barG, stagnation, for the inlet of Pipe P1. The last column on the Spray Discharge tab of the junction output table shows a volumetric flow rate above 25 m3/hr for all of the spray discharge junctions.

Reduce the supply pressure and run the model again. Repeat this process until you have determined the minimum supply pressure required to supply 25 m3/hr at every spray junction.

Step 5. Graph the results

AFT Fathom can create *dual y-axis graphs*, which plot two parameters on the same graph with the same x-axis. 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 flow rate on the secondary y-axis.

Graph the static pressure and volumetric flow rate from the constant pressure supply J9 through spray nozzle J1 on a dual y-axis graph to graphically determine whether the minimum flow through each spray nozzle is at least 25 m3/hr.

Note: You can repeat this process for the flow path from the constant pressure junction J9 to spray nozzle J8, or you can graph both flow paths on this same graph using the Plot Multiple Paths Using Groups feature.

1. Navigate to the Graph Results tab. On the Quick Access Panel, Graph Control tab, use the default selection of Plot Single Path. In the Pipes definition area, select pipes 1, 3, 4, 5, 6, and 7. These pipes make up the path from the constant pressure supply J9 to the J1 spray nozzle. Select these pipes quickly by first selecting them in the Workspace (using Ctrl + left-click), then clicking the Workspace button on the Graph Control tab.

2. Use meters as the Length Units.

3. Use Pressure Static for the parameter with barG as the unit.

4. Add a parameter by clicking the Add button next to Select Parameter in the Parameters definition area.

5. Change this added parameter to Volumetric Flow Rate and choose m3/hr as the unit.

6. To set the volumetric flow rate on the secondary y-axis, click the Make Secondary Y-Axis button, the blue arrow icon pointing right located above the selected parameters. Alternatively, double-click on the thin column in front of Volumetric Flow Rate in the Parameter definition area. Note that Volumetric Flow Rate

becomes indented under Pressure Static, and that the black arrow next to Volumetric Flow Rate turns to point to the left. See Figure 3.7 to see how these graphing parameters are defined.
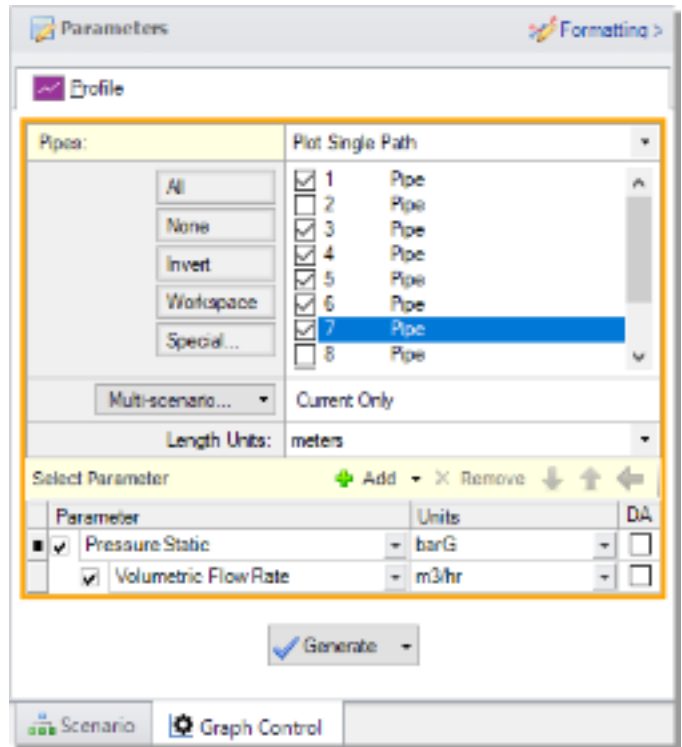
Figure 3.7　**Parameter definition for a dual y-axis graph of static pressure (primary y-axis) and volumetric flow rate (secondary y-axis) on the Graph Control tab on the Quick Access Panel**

Click the Generate button. After the graph generates, note that the static pressure and volumetric flow rate drop, as expected, when some flow leaves the system through each spray nozzle. See the graph in Figure 3.8.
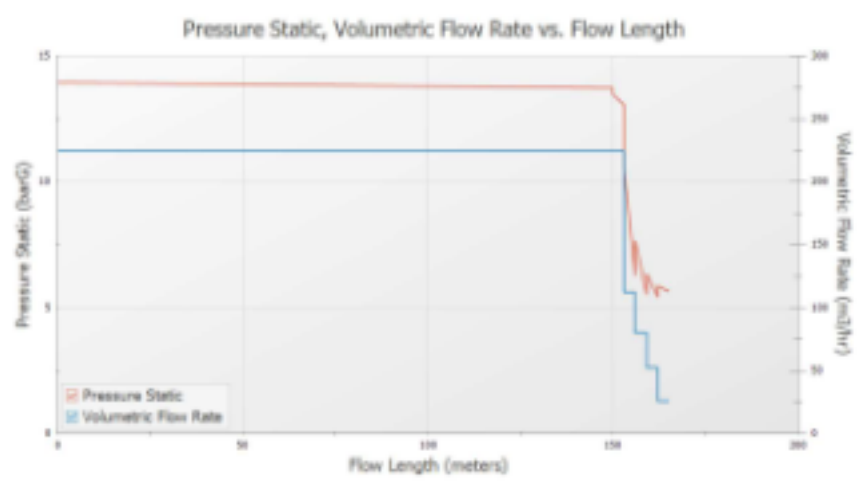
**Figure 3.8   Pressure and volumetric flow rate from the constant
pressure supply (J9) to the spray nozzle at one end of
the model (J1)**

Analysis summary

The minimum required supply pressure necessary to supply a volumetric
flow rate of 25 m3/hr to all of the spray junctions is about 13.2 barG,
stagnation.

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

The goal seeking capabilities in the optional AFT Fathom GSC Add-on
Module eliminate the manual iteration required for this problem. With
the supply pressure set as a variable and the minimum flow rate of 25
m3/hr for the spray discharges set as the goal (this is called a group
max/min goal), the AFT Fathom GSC Module then iterates on the
supply pressure until the goal is achieved. This eliminates the need to
run AFT Fathom multiple times to arrive at the final solution.

Chapter 5 provides more information on the AFT Fathom GSC Module.

Isometric Pipe Drawing Mode

AFT Fathom uses a 2D grid by default for objects on the Workspace, as was the case with this example. Figure 3.9 shows the spray discharge system with the 2D grid displayed (select Show Grid from the Arrange menu).
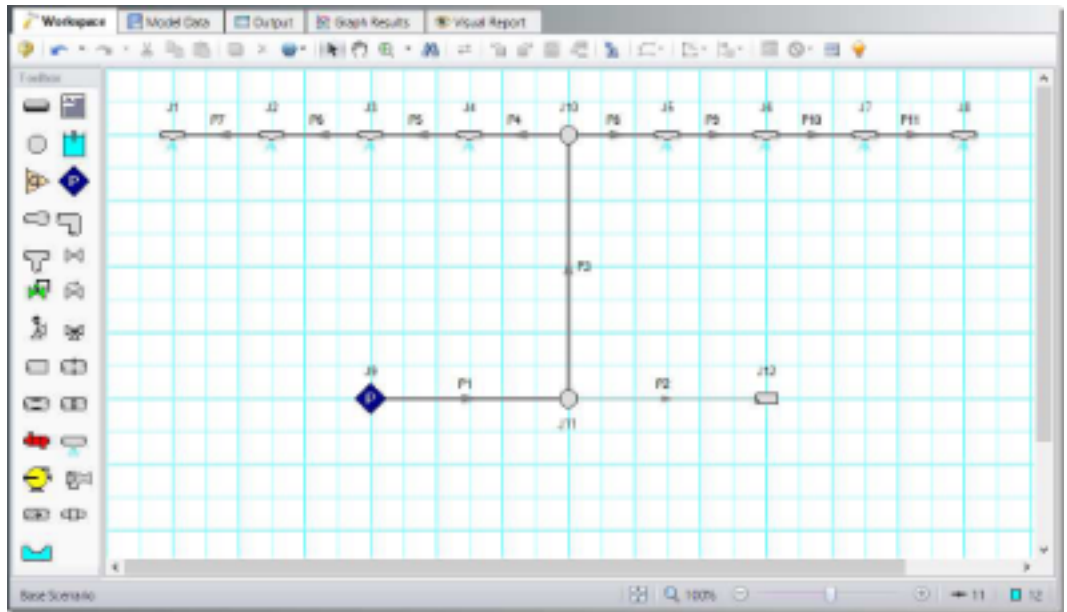
Figure 3.9   Spray discharge system with the 2D grid shown

At times it may be convenient to demonstrate the three-dimensional nature of a system. For example, you may be 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 3.10 shows the spray discharge system built on an isometric grid.
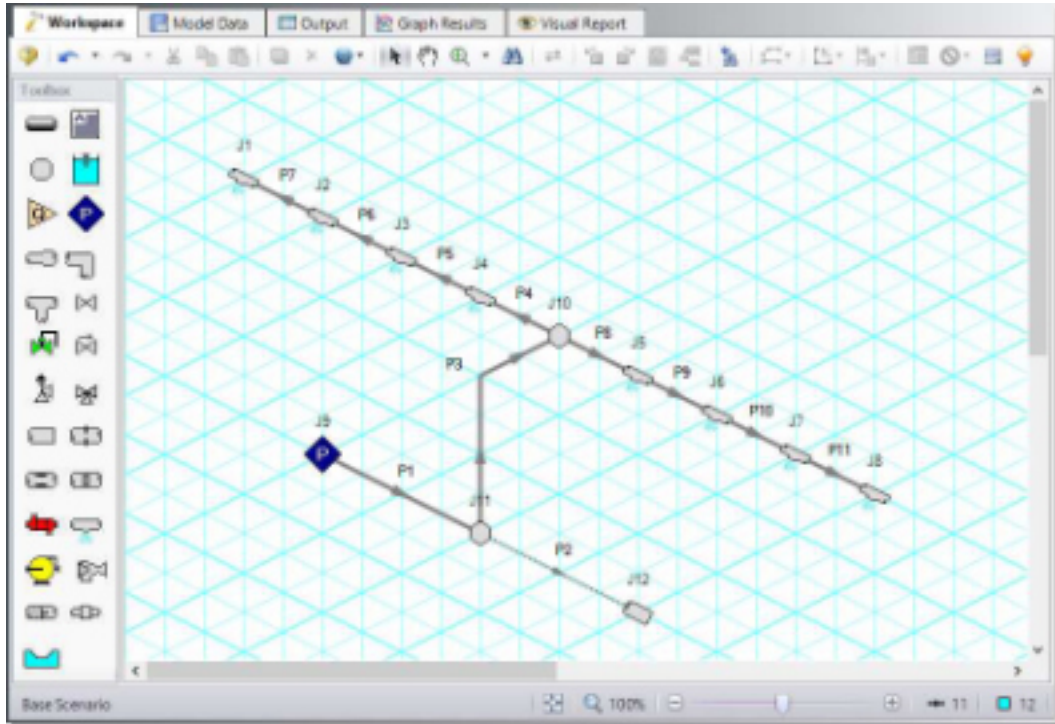
Figure 3.10   Spray discharge system with the isometric grid shown

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

Create the spray discharge system on an isometric grid to see how to use this feature.

1. Go to the File menu and select New.

2. From the Arrange menu, choose Show Grid.

3. From the Arrange menu again, go to Pipe Drawing Mode and select Isometric.

4. Place a Spray Discharge junction, J1, on the Workspace. 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. The preferred icon and rotation must be selected based on the pipe connections. We will address this later.

5.   Double-click the Spray Discharge icon in the Toolbox, then place junctions J2-J8.

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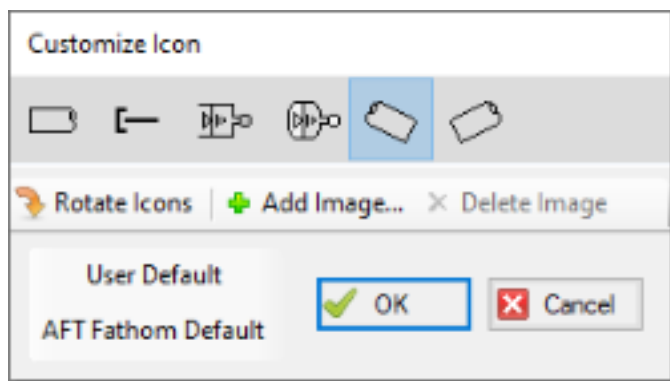
Figure 3.11   Use the Rotate button to obtain the preferred icon and rotation

10.   Draw Pipes P1 and P2 as shown in Figure 3.10.

11.   Draw Pipe P3 from J11 to J10. A red-dashed preview line will show how the pipe will display on the isometric grid. 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 P3 with the preview line.

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 P4-P11 as shown in Figure 3.10.

13. Right-click on junctions J1 and J8 and select Customize Icon. Select the appropriate orientation, as shown in Figure 3.10.

14. Select all of the remaining Spray Discharge Junctions. Right-click one of the junctions and select Auto-rotate Icon. The junctions will automatically align with the connected pipes.

15. Turn the grid off in the Arrange menu as shown in Figure 3.13.
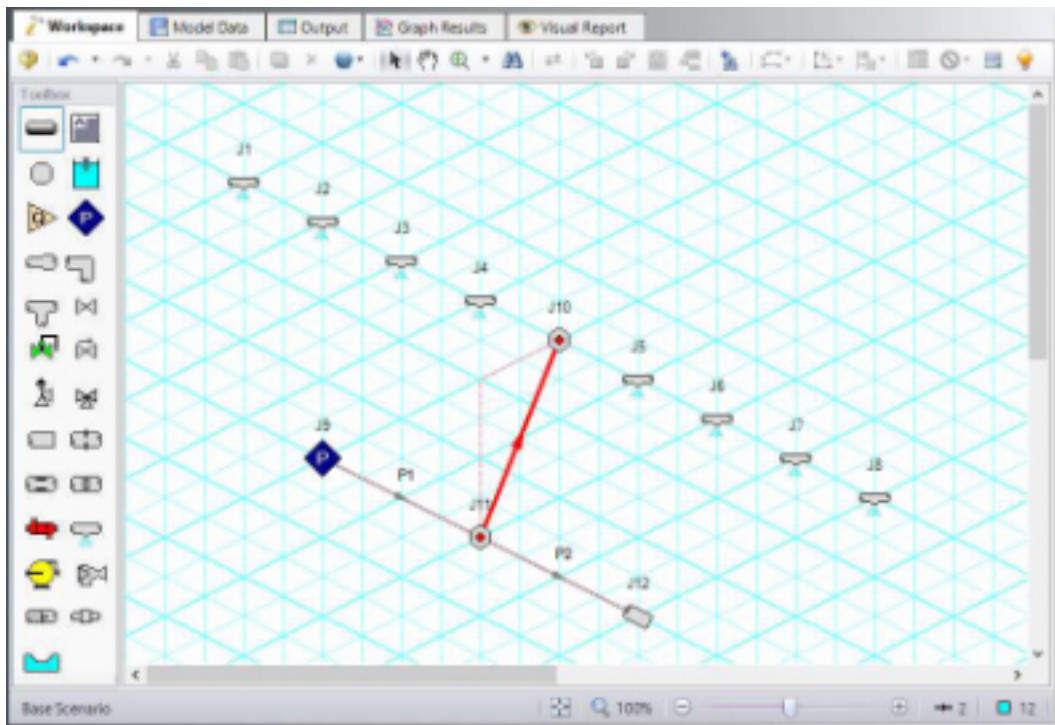
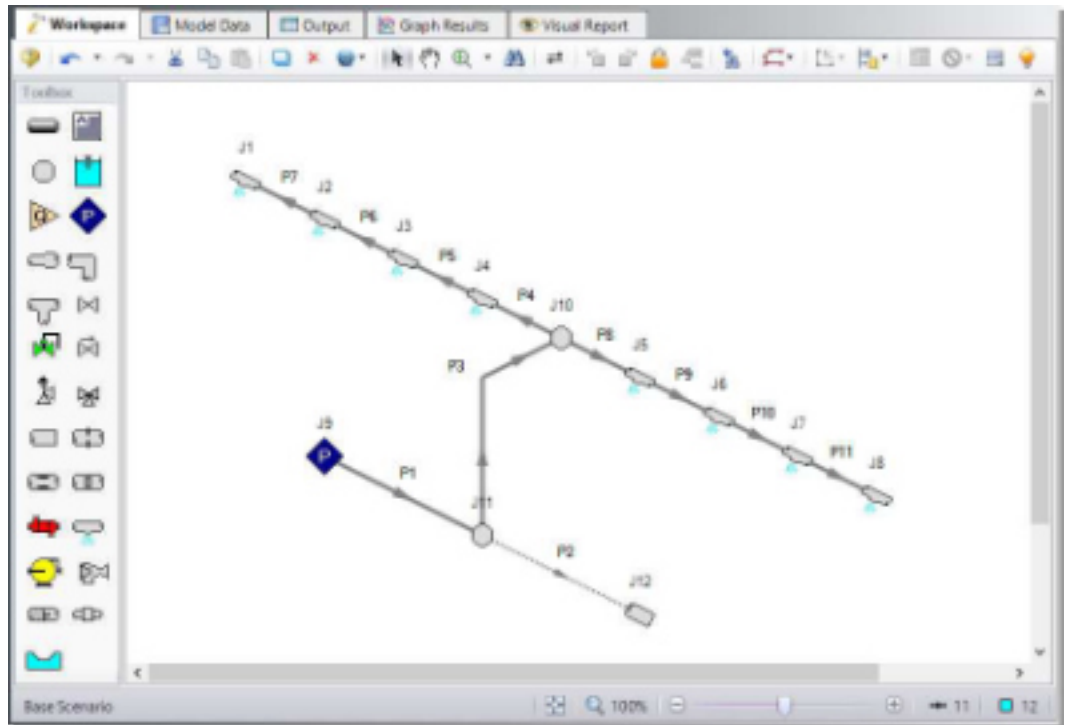**Figure 3.12   A preview line is shown when drawing or adjusting pipes in the Isometric Drawing Mode**

Figure 3.13   Isometric spray discharge system with the grid not shown

Freon Delivery System with Heat Transfer Example

This example demonstrates how to use heat transfer modeling on pipes and heat exchangers and using different refrigerant fluids to achieve certain design goals.

Topics covered

- Using the NIST REFPROP library to define fluids
- Setting up a heat transfer model in Analysis Setup
- Specifying convective heat transfer data for pipes
- Specifying thermal characteristics of heat exchangers
- Entering pump curves
- Entering resistance curves
- Using the 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 the example in Chapter 2 for more information on these topics.

Model file

A completed version of the model file can be referenced from the Examples folder as part of the AFT Fathom installation:

C:\AFT Products\AFT Fathom 13\Examples

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

Problem statement

A delivery system supplies Freon at 25 deg. C. Evaluate two candidate refrigerants (Refrigerant 11 and 123) to determine the best choice. Additionally, 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

➢ **To start AFT Fathom,** open the all applications list in the Windows Start Menu, expand the AFT Fathom 13 folder, and choose AFT Fathom 13. This refers to the standard menu items created by a standard installation; you may have chosen to specify a different installation location.

Step 2. Specify fluid properties

1. Open Analysis Setup by selecting Analysis Setup from the Common Toolbar or from the Analysis menu.

2. Under the Fluid Properties group, click the Fluid item to open the fluid panel. Select the NIST REFPROP fluid library, click Search, type R11 in the search field, click OK, click R11 in the Search Results, then click the Select button.

3. Click Add to Model to select R11 for use in this model.

4. In the Fluid Properties section on the right, enter 10.5 barG for pressure and -45 deg. C for temperature (see Figure 4.1). Default fluid properties are automatically calculated. The Solver adjusts the

fluid properties during the simulation when modeling heat transfer.

5. Navigate to the Heat Transfer/Variable Fluids panel by clicking on the Heat Transfer/Variable Fluids item on the left under the Fluid Properties group.

6. Select the option for Heat Transfer With Energy Balance (Single Fluid). This enables AFT Fathom's heat transfer capabilities.

7. Click OK to close Analysis Setup.
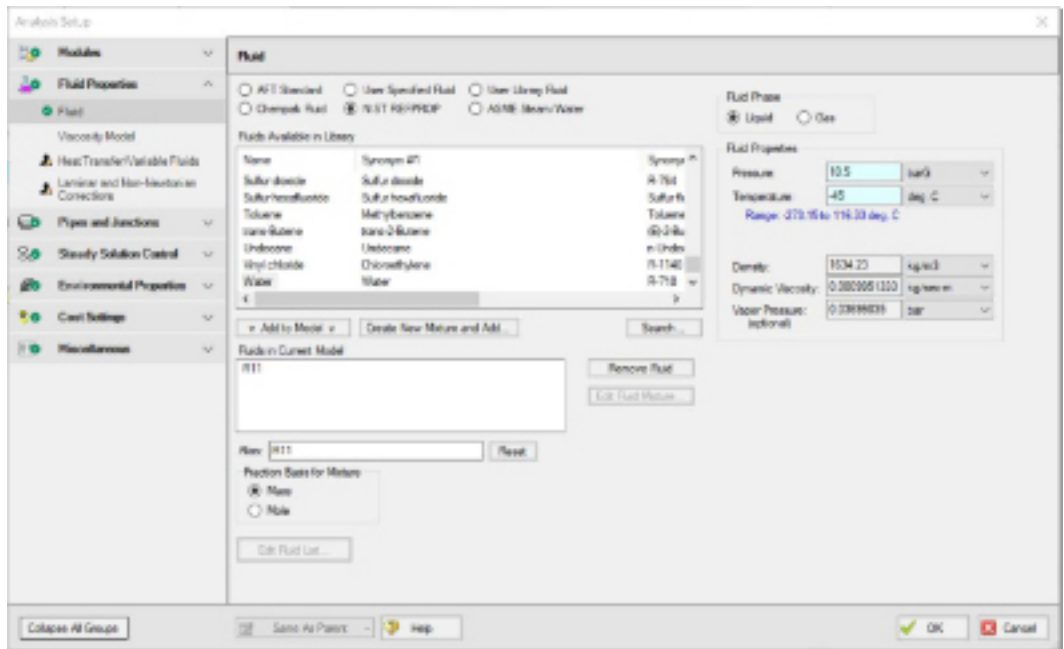
Figure 4.1   Fluid panel with R11 selected and fluid pressure and temperature specified

Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 4.2. To add graphical bends for Pipes 3-7, select a pipe, open the Arrange menu, go to Add/Remove Segments, then click Add Segment. Move the segment point to the desired location on the Workspace. Alternatively, enable the 2D Orthogonal Pipe Drawing Mode to automatically draw pipes along an orthogonal grid, or, after selecting a pipe, press CTRL+SHIFT+Y to add a pipe segment.
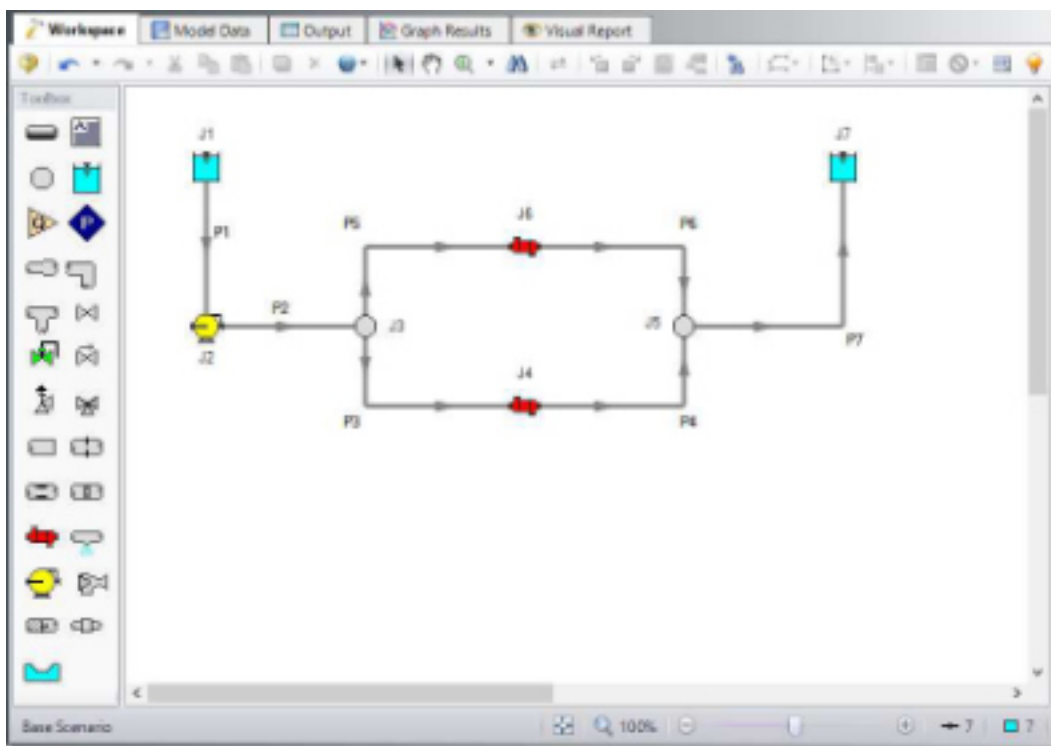
Figure 4.2   Layout of the pipe system for the Freon Delivery System with Heat Transfer Example

B. Define the pipes

With the objects in place, define 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).

For Pipes P1-P7, use

- Pipe Material = Steel - ANSI

- Size = 3 inch (7.8 cm ID)

- Type = STD (schedule 40)

- Friction Model = Standard

- Length = use the table below

Pipe	Length (meters)
P1	3
P2	3
P3	1.5
P4	1.5
P5	1.5
P6	1.5
P7	9

Specify the piping heat transfer parameters on the Heat Transfer tab in the Pipe Properties window, as shown below for Pipe P1 in Figure 4.3. After entering the parameters below for Pipe P1, click OK.

1. Heat Transfer Model = Convective Heat Transfer

2. Resistance Geometry = Radial

3. Number of Insulation Layers = 1 External

4. Ambient Conditions:

      a.   Temperature = 16 deg. C

      b.   Fluid Velocity = 8 km/hr

      c.   Fluid = Air

5.   Use default values for Fluid Internal, Pipe Wall, and External

6.   Insulation #1 (external):

      a.   Apply = Checked

      b.   Conductivity/Convection Data Source = User Specified

      c.   Conductivity (W/m-K) = 0.7

   d.   Thickness (cm) = 3

      e.   HT Area Ratio = 1
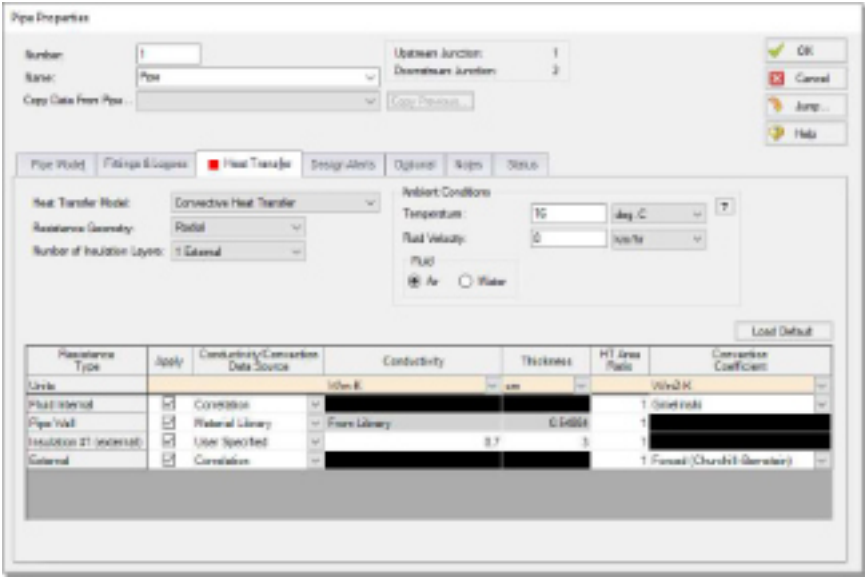
Figure 4.3   Pipe Heat Transfer tab for the Freon Delivery System Example

After specifying the piping heat transfer for Pipe P1, open the Global Pipe Edit window, select the Existing Pipe option under the Choose Data From section at the top, in the dropdown choose data from Pipe P1, check the box next to Heat Transfer in the Parameters to Change

list, select all pipes in the Pipe List, and click Apply Selections before clicking OK.

Add an additional loss due to an elbow fitting to Pipes P3-P6. To add this loss, open the Pipe Properties window, go to the Fittings & Losses tab, and click the Specify Fittings & Losses button (Figure 4.4). This opens the Pipe Fittings & Losses window.
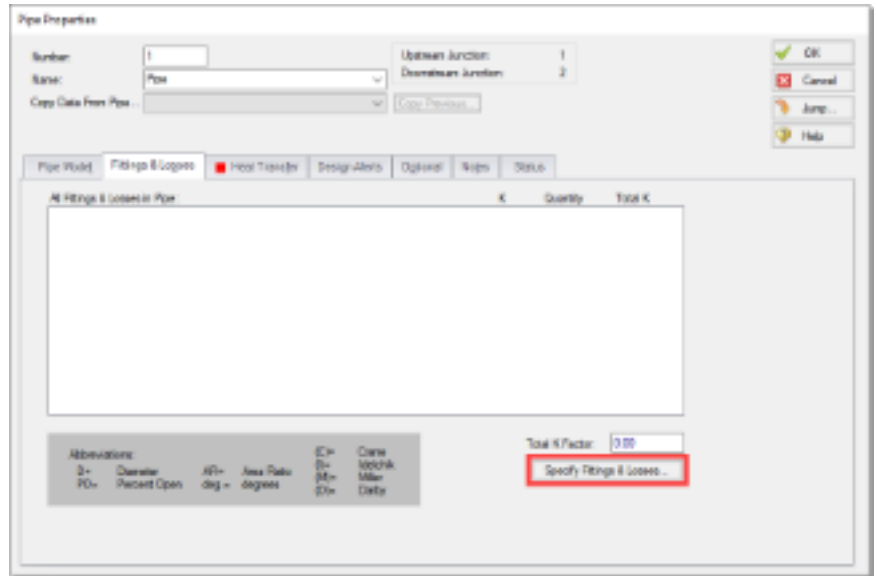
Figure 4.4   The Pipe Fittings & Losses window is opened by clicking the Specify Fitting & Losses button on the Pipe Properties window

Go to the Bends tab and under Standard Threaded, General, 90 deg. (C), enter a quantity of 1 (Figure 4.5). Click the OK button to add this loss to the pipe, then repeat for Pipes P4-P6. Global pipe editing could also be used to make this change to all four pipes simultaneously.
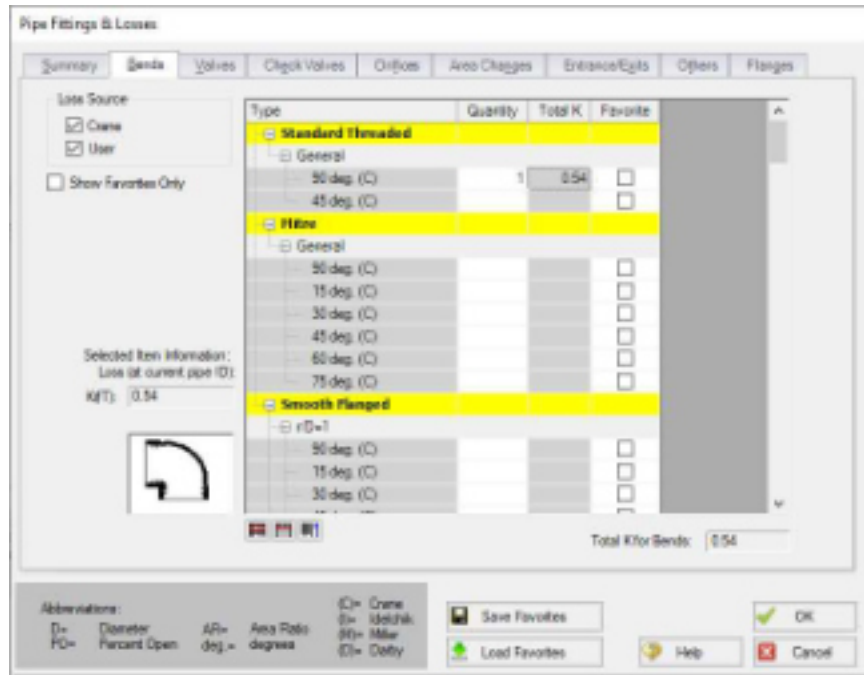
**Figure 4.5   Add Fittings & Losses to pipes by specifying fitting
quantities on the Pipe Fittings & Losses window**

C. Define the junctions

J1 - Reservoir Junction

1.   Name = Supply Reservoir

2.   Liquid Surface Pressure = 10.5 barG (1050 kPa (g))

3.   Pipe Depth = 0 meters

4.   Liquid Surface Elevation = 3 meters

5.   Liquid Temperature = -45 deg. C

6.   Liquid Density = keep default value

J7 - Reservoir Junction

1. Name = Receiving Reservoir

2. Liquid Surface Pressure = 10.5 barG (1050 kPa (g))

3. Pipe Depth = 0 meters

4. Liquid Surface Elevation = 3 meters

5. Liquid Temperature = 24 deg. C

6. Liquid Density = keep default value

J3, J5 - Branch Junctions

1. Elevation = 0.5 meters

J2 - Pump Junction

1. Inlet Elevation = 0.5 meters

2. Pump Model = Centrifugal (Rotodynamic)

3. Analysis Type = Pump Curve

4. Enter the pump curve data (see Figure 4.6):

    a. Click Enter Curve Data.

    b. Enter the following pump curve data in the Raw Data table:

Volumetric	Head
m3/hr	meters
0	7.5
45	7
115	4.5

    c. Click Generate Curve Fit Now to create the curve fit.

    d. Close the Pump Configuration window by clicking OK.
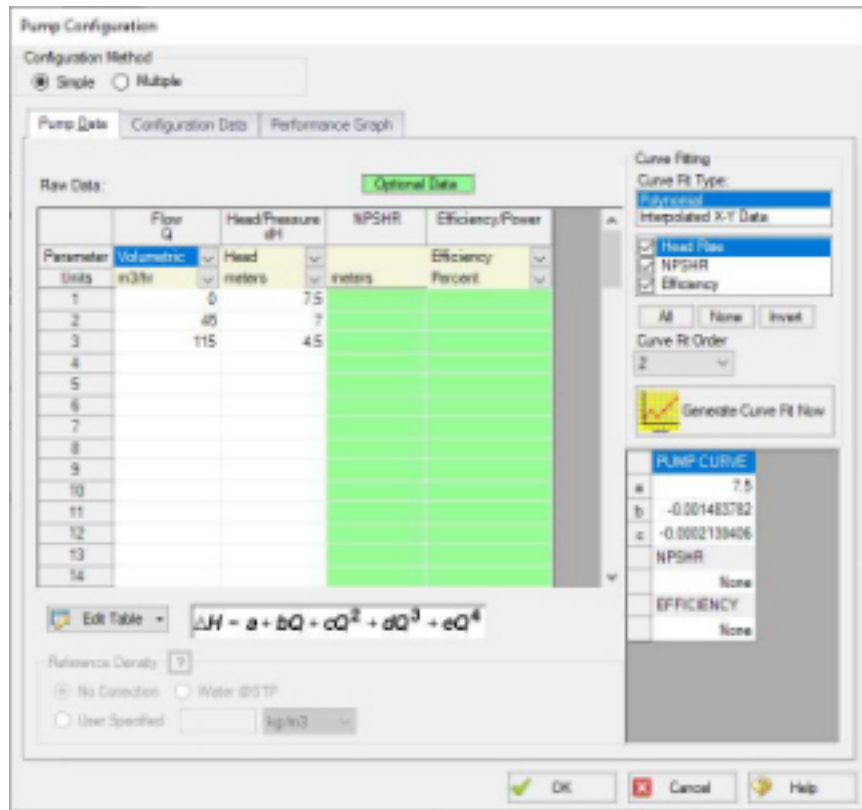
Figure 4.6   Pump Configuration window for the Freon Delivery System Example

J4, J6 - Heat Exchangers

1.   Inlet Elevation = 0.5 meters

2.   Loss Model = Resistance Curve

3.   Enter resistance curve data (see Figure 4.7):

     a.   Click Enter Curve Data.

     b.   Change the Flow Parameter to Volumetric.

     c.   Enter 55 m3/hr and 0.15 bar (25 kPa) in the Raw Data table.

    d.   Click Fill As Quadratic above the data table to create data for a square law loss curve, which is typical of heat exchangers.

    e.   Click Generate Curve Fit Now to create a quadratic curve fit.

    f.   Click OK to save and close.

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

    a.   Thermal Model = Counter-Flow

    b.   Heat Transfer Area = 7.5 meters2

    c.   Overall Heat Transfer Coefficient = 2800 W/m2-K

    d.   Secondary Fluid Flow Rate = 15 kg/sec

    e.   Secondary Fluid Specific Heat = 4.2 kJ/kg-K

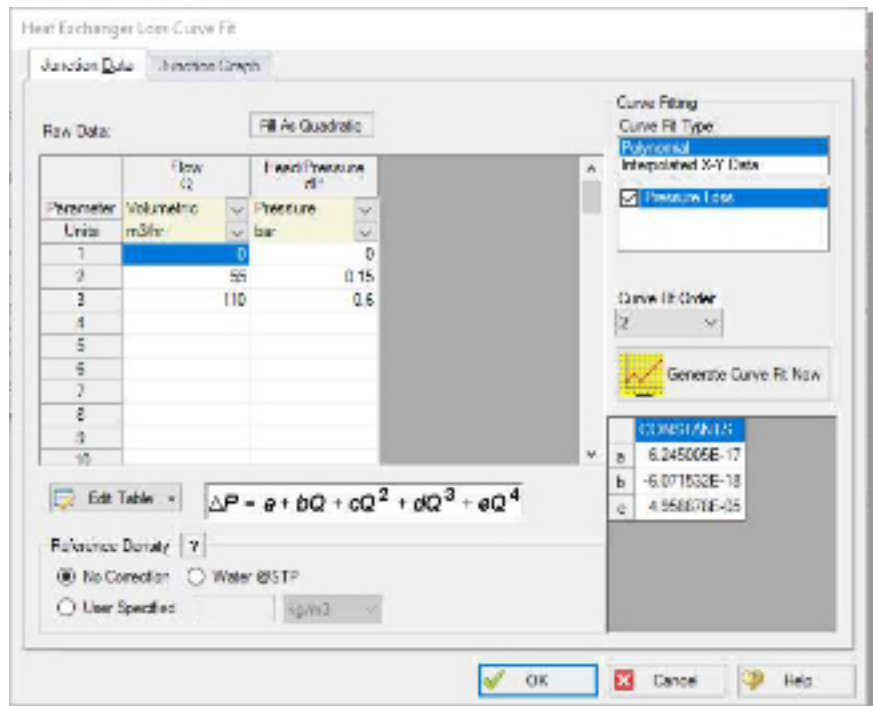   f.   Secondary Fluid Inlet Temperature = 95 deg. C

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

Start with the initial guess of 95 deg. C for the Secondary Fluid Inlet Temperature. This problem will require manual iteration to achieve the desired result. You will need to run the model, examine the results, and adjust the Secondary Fluid Inlet Temperature for both heat exchangers until you achieve the desired Freon outlet temperature of 24 deg. C.

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom-right of the Quick Access panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions, and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

## Step 4. Create a scenario for each refrigerant

To determine the best refrigerant that minimizes the temperature of the hot water used to heat the Freon to the delivery temperature, 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 differences between scenarios using the Scenario Comparison Tool

- Propagate changes from a parent scenario to its children

Create two scenarios to model these cases. Access the Scenario Manager from the Quick Access Panel, which is located on the right side of the Workspace area, or from the Tools menu. A scenario tree displays all model scenarios.

Click the Create Child button on the Quick Access Panel, or from the right-click menu on the Base Scenario. Name the child R11. A new scenario will appear below the Base Scenario. Select the Base Scenario, create another child, and name it R123. See Figure 4.8.
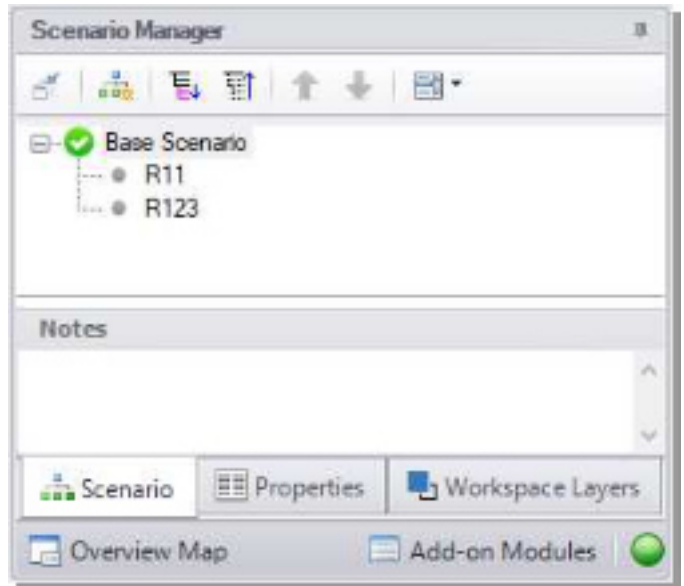
Figure 4.8 Create model variants and keep them organized using the Scenario Manager on the Quick Access Panel

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.

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

Double-click the R11 scenario in the scenario tree of the Scenario Manager on the Quick Access Panel. This loads the R11 scenario onto the Workspace as the current scenario. The currently loaded scenario is

displayed in the scenario tree with a green circle and a white check mark (Figure 4.8). Alternatively, you can use the Scenario Manager window from the Tools menu to load the scenario.

Step 6. Run the R11 case

With the initial guess of 95 deg. C for the secondary fluid inlet temperature, click Run Model from the Common Toolbar or Analysis menu. The Solution Progress window opens and begins iterating. This window displays the AFT Fathom Solver converging on the answer. After the Solver converges, view the results by clicking Output at the bottom of the Solution Progress window.

Step 7. Examine the R11 results

If the default Output Control parameters are displayed, a warning will state *HGL, EGL and head loss results may not be meaningful for variable density systems* as can be seen in Figure 4.9. Hide all head output parameters to remove this warning.

Note the R11 delivery temperature of 48.53 deg. C at the outlet of Pipe P7 by navigating to the Heat Transfer tab in the Pipe output section (Figure 4.9). This is well above the required maximum delivery temperature of 24 deg. C. The inlet temperature of the Secondary Fluid must decrease and the model must run again to determine the outlet temperature at the new value. Repeat this process until the Secondary Fluid Inlet Temperature delivers the Freon at 24 deg. C.
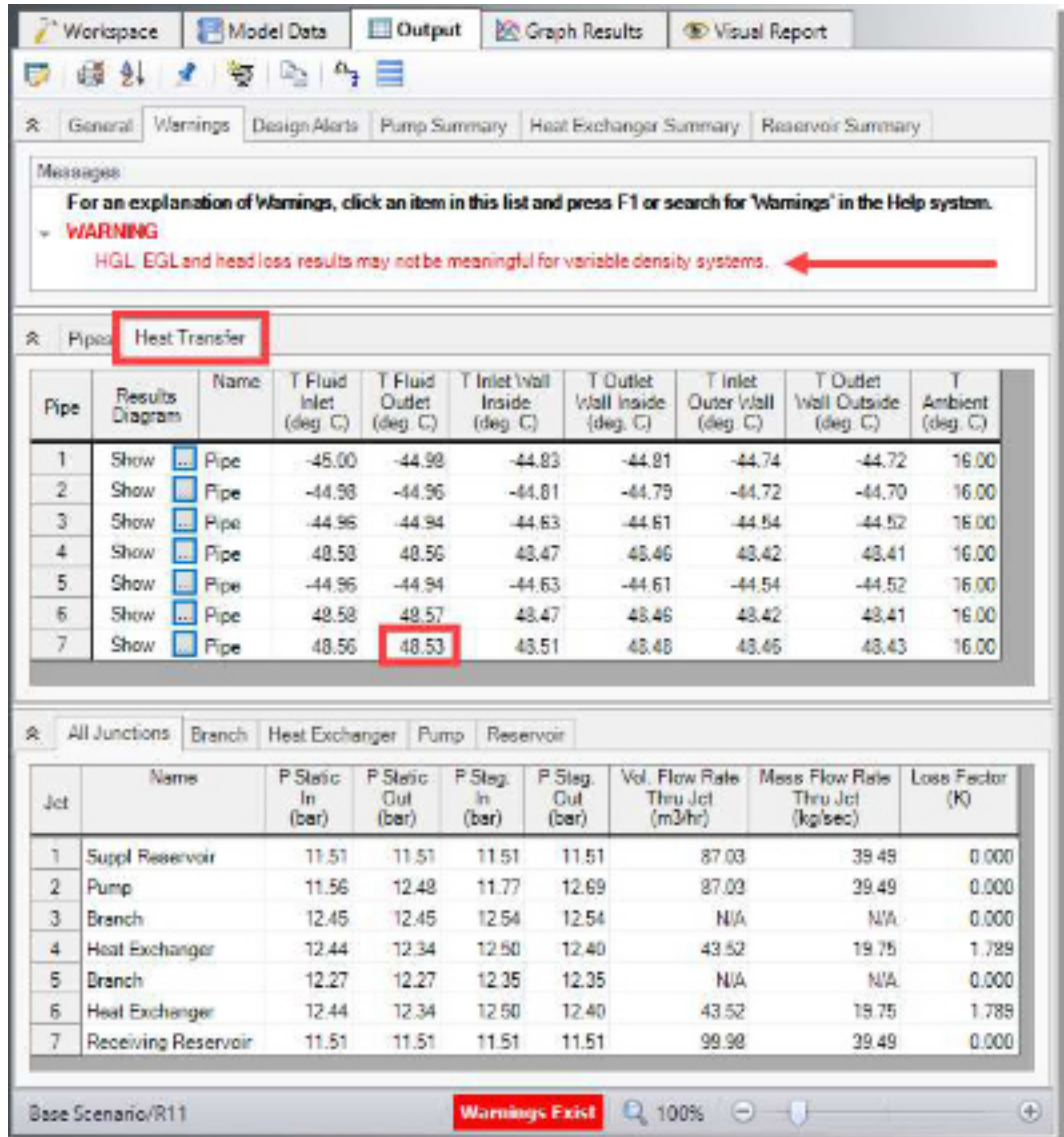
**Figure 4.9   Output for the Freon Delivery System for R11 with a
Secondary Fluid Inlet Temperature of 95 deg. C**

Open the Pipe Heat Transfer Results Diagram from the Heat Transfer
tab (see Figure 4.10) 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 Pipe Results Diagram.
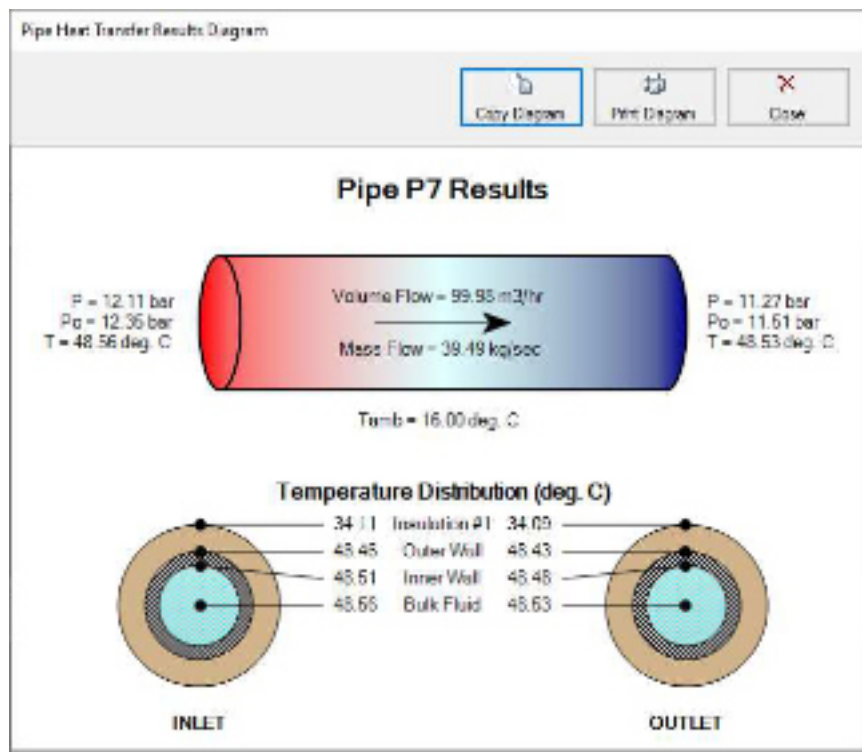

**Figure 4.10   Pipe Heat Transfer Results Diagram opened from the Heat Transfer Table (see Figure 4.9)**

Step 8. Set up the R123 case

Double-click the R123 scenario in the Scenario Manager to load it as the current scenario. Then, open Analysis Setup and open the Fluid panel. Similar to Step 2, search for and select R123, select Yes to remove R11 from the scenario when prompted, and enter same pressure and temperature as before: 10.5 barG and -45 deg. C.

Step 9. Run the R123 case and examine the results

With the initial guess of 95 deg. C for the secondary fluid inlet temperature, click Run Model. After the Solver converges, click Output.

Note the R123 delivery temperature of 41.24 deg. C at the outlet of Pipe P7 in the Heat Transfer tab (see Figure 4.11). This is above the minimum temperature of 24 deg. C. Iterate again until the Secondary Fluid Inlet Temperature delivers the Freon at 24 deg. C.
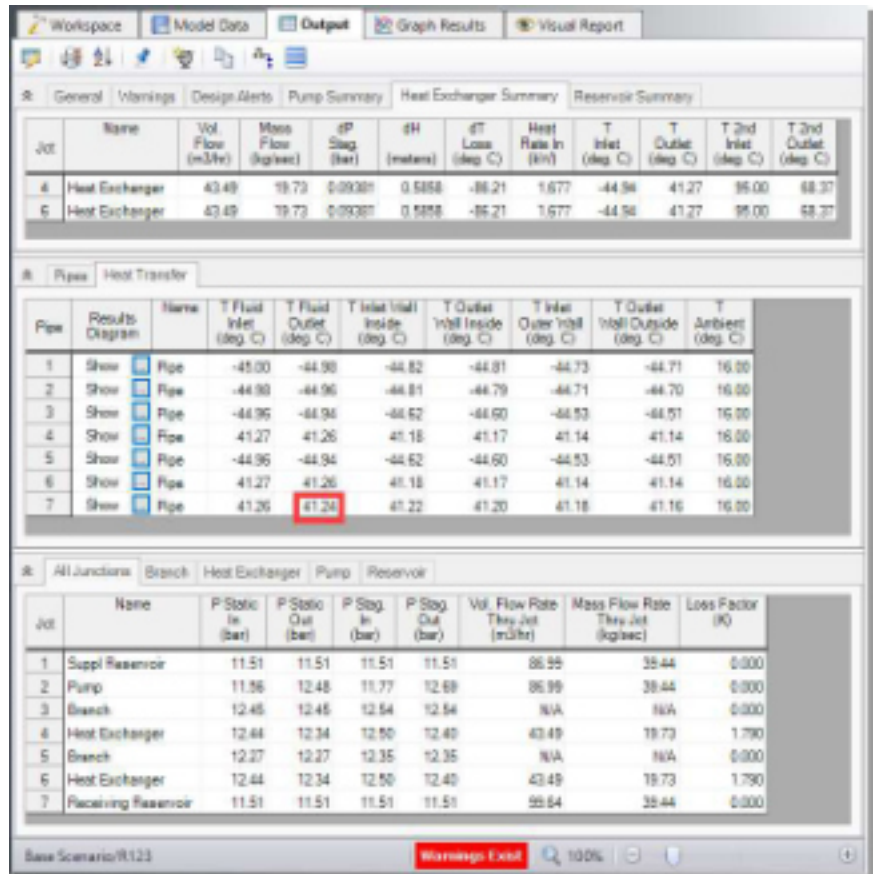
Figure 4.11   Output for the Freon Delivery System for R123 with a Secondary Fluid Inlet Temperature of 95 deg. C

Analysis summary

The results of the iterative solutions show the following:

Refrigerant	Secondary Fluid Inlet Temp (deg. C)
R11	57.76
R123	66.44

In terms of heat exchanger hot water temperature, R11 is the better choice for a refrigerant delivery temperature of 24.00 deg. C. However, consider other factors, such as environmental regulations, 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 Add-on Module solves this problem directly without manual iteration. Using both heat exchanger secondary inlet temperatures as the variables (as linked variables) and the fluid outlet temperature of Pipe P7 as the goal, the Solver can directly calculate the seconadary fluid inlet temperature.

Chapter 5 contains more discussion on the GSC Add-on Module.

C H A P T E R   5

AFT Fathom Add-on Modules Examples

This chapter covers two examples. The first example demonstrates how to use the Goal Seek & Control (GSC) and Extended Time Simulation (XTS) optional Add-on Modules for AFT Fathom. Later in the chapter, a second example uses the Settling Slurry (SSL) Add-on Module.

A separate Quick Start Guide covers the Automated Network Sizing (ANS) module.

The user can only perform these examples if access to the relevant modules are available.

Topics covered

- GSC - Using the Goal Seek and Control Manager
- GSC - Defining Variables and Goals
- XTS - Specifying the Transient Control group
- XTS - Defining system transients
- XTS - Understanding Transient Output
- SSL - 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 the example in Chapter 2 for more information on these topics.

To model the heat transfer in the GSC example, refer to Chapter 4 for more detailed explanations on modeling heat transfer.

Model files

For your reference, see the completed model files for these examples installed in the Examples folder as part of the AFT Fathom installation:

C:\AFT Products\AFT Fathom 13\Examples

- *Metric - Controlled Heat Exchanger Temperature - GSC.fth*

- *Metric - Controlled Heat Exchanger Temperature - XTS.fth*

- *Metric - Pump Sizing for Sand Transfer System - SSL.fth*

Problem statements

GSC Example
Use the Goal Seek & Control Add-on Module to determine the valve open percentage required to maintain a 50 deg. C temperature at branch junction J8 for a reservoir supply temperature of 90 deg. C.

XTS Example
Use the Extended Time Simulation Add-on Module to determine the time required to fill a finite, 6 meter high, 7.5 m^2 cross sectional area discharge reservoir and to turn off the pump when the tank is full.

SSL Example
Use the Settling Slurry Add-on Module to determine the size of the pump required when the fluid is 25% sand by volume and is being pumped from an open supply vessel with a liquid surface elevation of 1.5 meters to an open receiving tank with a liquid surface elevation of 3 meters.

Step 1. Start AFT Fathom

➢ **To start AFT Fathom,** open the all applications list in the Windows Start Menu, expand the AFT Fathom 13 folder, and choose AFT Fathom 13. This refers to the standard menu items created by a standard installation; you may have chosen to specify a different installation location.

Step 2. Specify fluid properties

1. Open Analysis Setup

2. Go to the Fluid panel, select AFT Standard, select Water (liquid), then click Add to Model

3. On the right, enter a temperature of 21 deg. C (see Figure 5.1)

4. On the Heat Transfer/Variable Fluids panel, select Heat Transfer With Energy Balance (Single Fluid)

5. Click OK
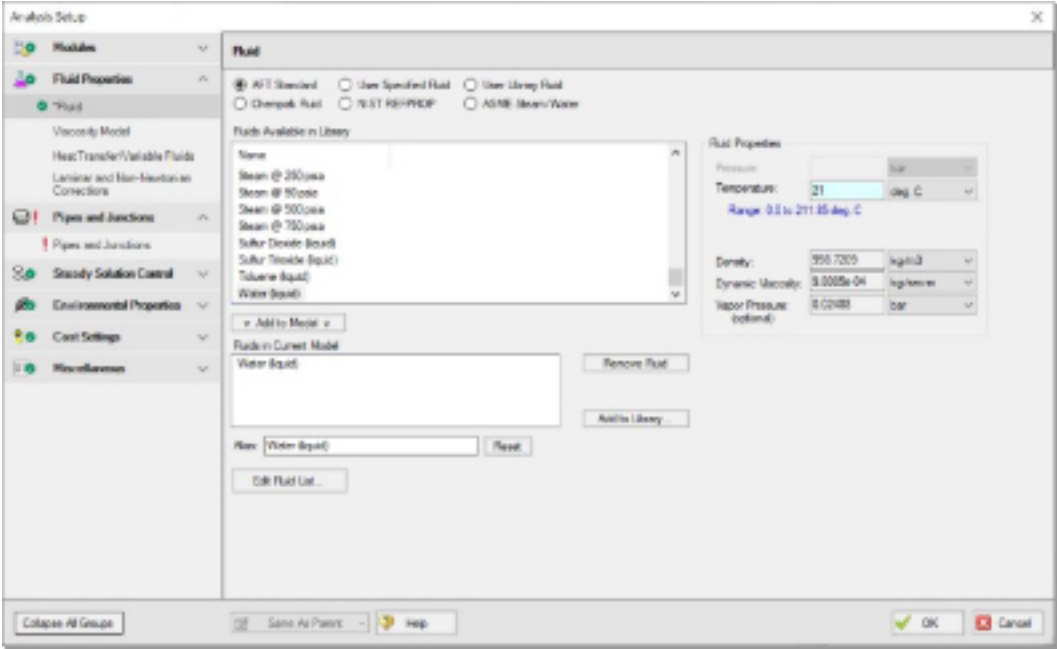
Figure 5.1   Fluid panel with Water (liquid) selected and temperature specified

Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 5.2.
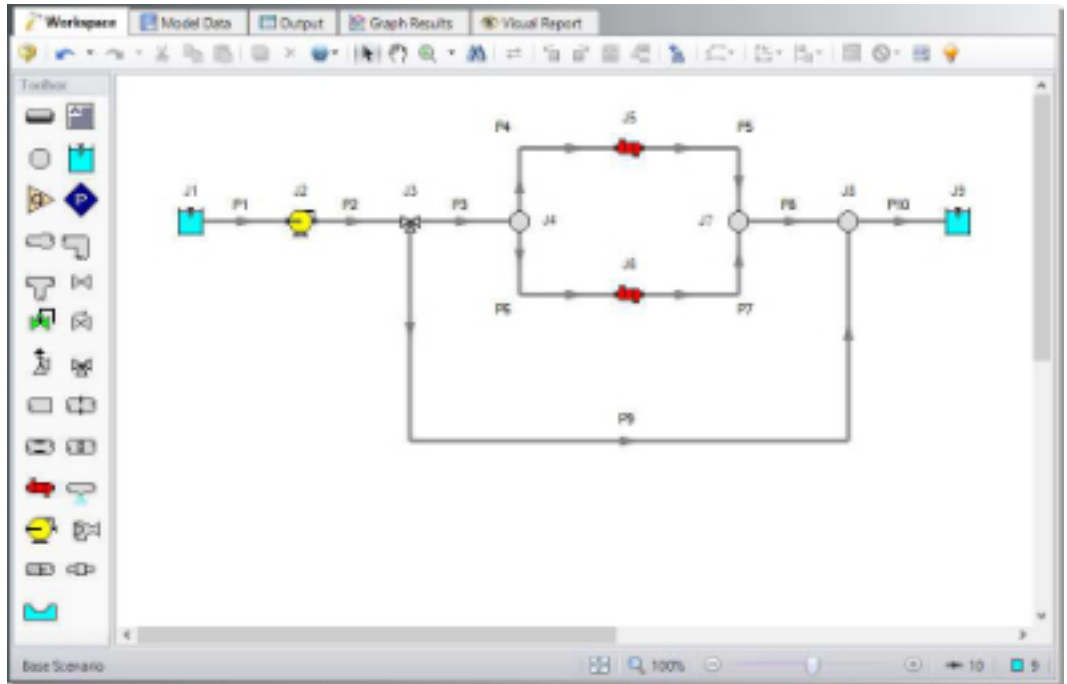
Figure 5.2   Layout of the pipe system for the Controlled Heat Exchanger Temperature Example

B. Define the pipes

With the objects in place, define the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Pipe Properties window (or use the Global Pipe Edit window, as discussed in Chapter 3).

For Pipes P1-P10, use

- Pipe Material = Steel - ANSI

- Size = 8 inch (20.3 cm ID)

- Type = STD (schedule 40)

- Friction Model = Standard

- Heat Transfer Model = Adiabatic (Perfectly Insulated)

- Length = use the table below

Pipe	Length (meters)
P1	6
P2	1.5
P3	6
P4	6
P5	6
P6	6
P7	6
P8	6
P9	60
P10	30

C. Define the junctions

J1, J9 Reservoirs

1. Liquid Surface Pressure = 0.3 barG (30 kPa (g))
2. Liquid Surface Elevation = 3 meters
3. Liquid Temperature = 90 deg. C
4. Update Density with Temperature = Checked
5. Pipe Depth = 0 meters

J2 - Pump

1. Inlet Elevation = 0 meters
2. Pump Model = Centrifugal (Rotodynamic)
3. Analysis Type = Pump Curve
4. Click Enter Curve Data then input the following and generate a second order curve fit for the data:

Volumetric	Head
m3/hr	**meters**
0	20
115	18
230	12

J3 - Three-Way Valve

1. Elevation = 0 meters

2. Combined Flow Pipe = P2

3. Split Flow Pipe #1 = P3

4. Split Flow Pipe #2 = P9

5. Open Percentage Data (ignore Flow Area data):

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

6. Actual Percent Open = 75%

J4, J7, J8 - Branches

Elevation = 0 meters

J5, J6 - Heat Exchangers

1. Inlet Elevation = 0 meters

2. Loss Model tab

    a. Loss Model = K Factor

    b. K = 1400

3. Thermal Data tab

    a.   Thermal Model = Counter-Flow

    b.   Heat Transfer Area = 148 meters2

    c.   Overall Heat Transfer Coefficient = 280 W/m2-K

    d.   Secondary Fluid Flow Rate = 90 kg/sec

    e.   Secondary Fluid Specific Heat = 4.2 kJ/kg-K

    f.   Secondary Fluid Inlet Temperature = 4.5 deg. C

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom right of the Quick Access panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions, and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

GSC problem statement

Use the Goal Seek & Control Add-on Module to determine the valve open percentage required to maintain a 50 deg. C temperature at branch junction J8 for a reservoir supply temperature of 90 deg. C.

GSC Step 1. Create a scenario for the GSC case

Create a child scenario named GSC Example from the Base Scenario by clicking Create Child in the Scenario Manager.

GSC Step 2. Activate the GSC Module

Open Analysis Setup then open the Modules panel. Under GSC (Goal Seek & Control) check the box next to Activate GSC. Make sure the radio button for Use is selected, as shown in Figure 5.3. In the GSC module, AFT Fathom modifies variables to achieve specified goals. A unique answer requires one unique variable for each specified goal. No limit exists for the number of variables and goals defined, however, keep in mind that a large number of variables and goals could result in extremely long runtimes and possible non-convergence.

With the GSC module activated, the Goal Seek and Control group appears in the Analysis Setup. The three items in the group include: Variables, Goals, and Numerical Controls. Specify the Goal Seek and Control group by entering data for variables and goals in their respective panels.
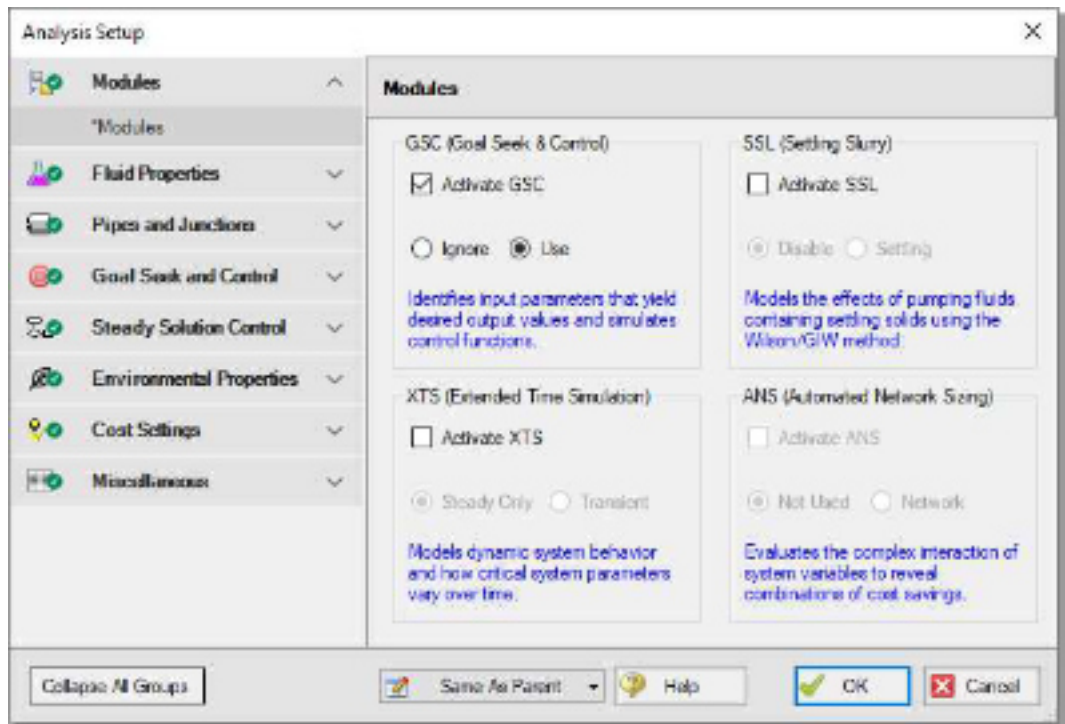
Figure 5.3   Activate GSC from the Modules panel

GSC Step 3. Open the Variables panel

Under the Goal Seek and Control group, select the Variables item to open the Variables panel, shown in Figure 5.4.
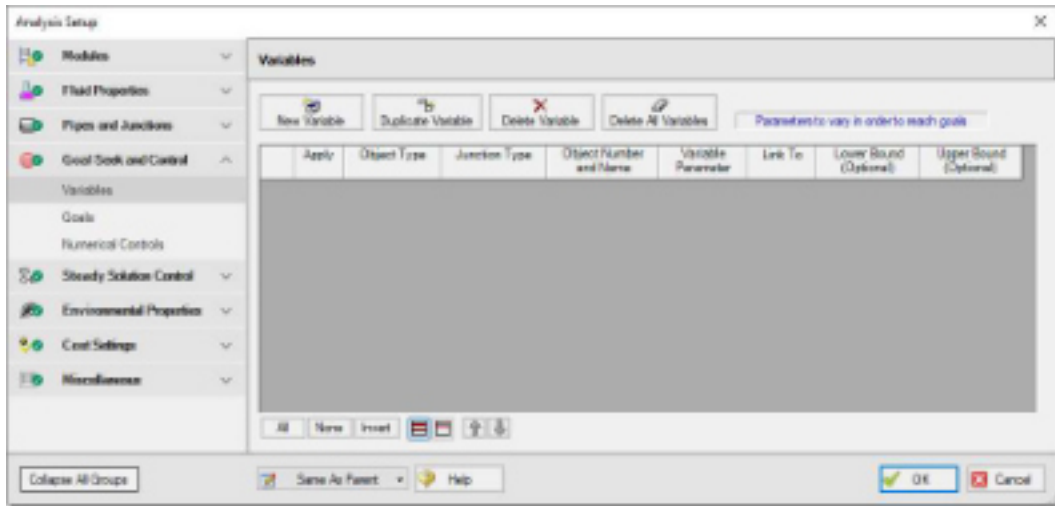
Figure 5.4   Variables panel in the Goal Seek and Control group

GSC Step 4. Add a variable

The Variables panel allows users to create and modify the system variables. Additionally, you can elect to apply certain variables while omitting others using the check box in the Apply column.

For this example, add a variable for the Three-Way Valve Open Percentage. Click New Variable and input the following data, as shown in Figure 5.5

1. Apply = Checked

2. Object Type = Junction

3. Junction Type = Three Way Valve

4. Object Number and Name = J3 (Three Way Valve)

5. Variable Parameter = Open Percentage

6. Link To = (None)

7. Lower Bound (Optional) = 0

8. Upper Bound (Optional) = 100
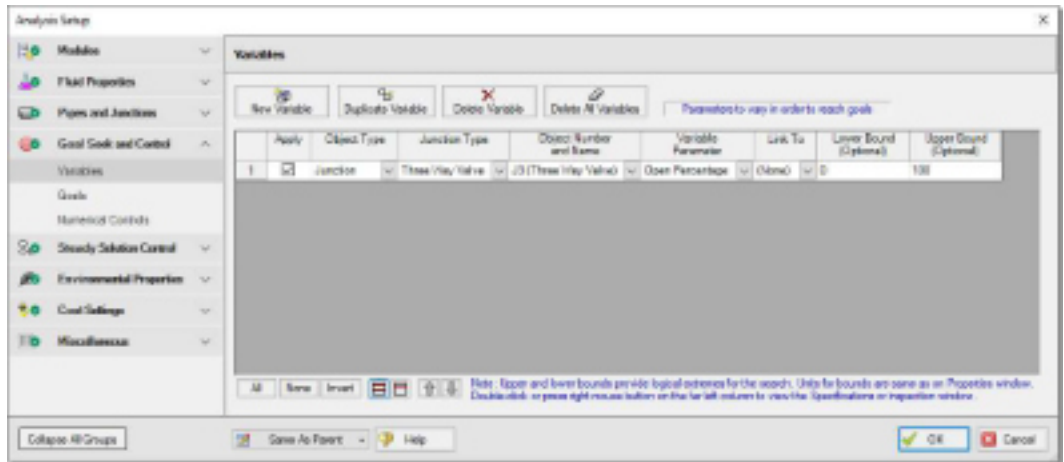
Figure 5.5   Variables panel with one variable defined for the Three Way Valve Open Percentage

Use the Apply column to specify the variables for the Solver to use. This allows the flexibility to create multiple variable cases, while only applying selected variables for any given run.

You can use the Link To column to apply the same variable to multiple objects. This will force parameters to vary identically for different objects.

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%.

GSC Step 5. Add a goal

Goals are the parameters you want to determine. AFT Fathom modifies the specified variables to achieve the goals. To obtain a unique answer there should be one goal applied for each variable.

Click on the Goals item to open the Goals panel. You create and modify system goals in the Goals panel by selecting the goal type, object type, goal parameter, criterion, value, and units. You can then select 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).

Click New Goal as defined below and shown in Figure 5.6

1. Goal Type = Point

2. Object Type = Branch

3. Goal Parameter = Temperature

4. Goal Value = 50

5. Goal Units = deg. C

6. Object ID = J8 (Branch)

7. Object Location = Outlet
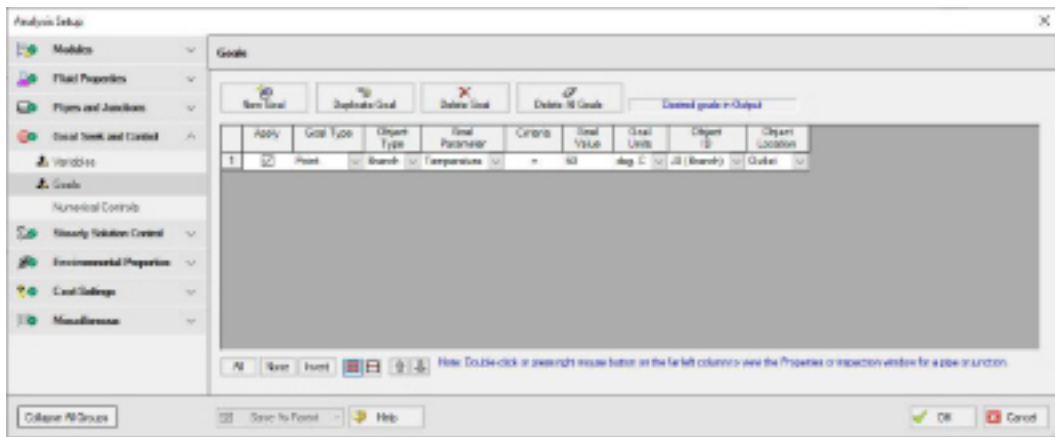
Figure 5.6   AFT Fathom adjusts Variables to achieve user Goals

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 5.7.

Note: The goal symbol is not displayed next to objects that are part of a group goal.
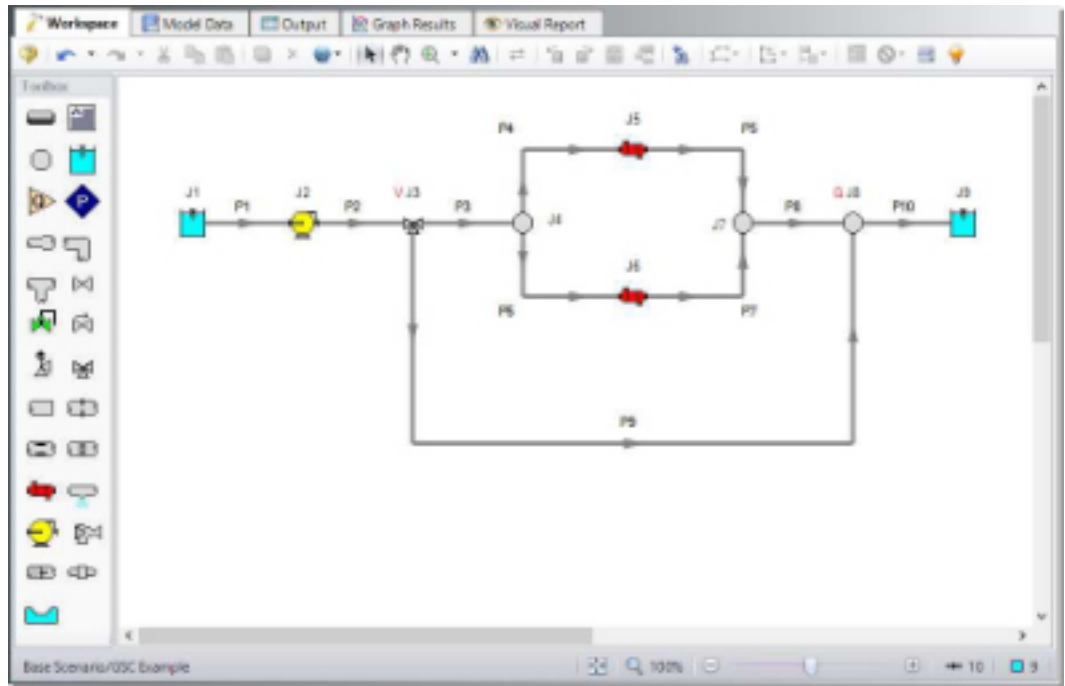
Figure 5.7   AFT Fathom displays symbols next to objects on the Workspace that have goals or variables defined

GSC Step 6. Run the model

Click Run Model from the Common Toolbar or Analysis menu. The Solution Progress window opens and begins iterating. This window displays the AFT Fathom Solver converging on the answer. After the Solver converges, view the results by clicking Output at the bottom of the Solution Progress window.
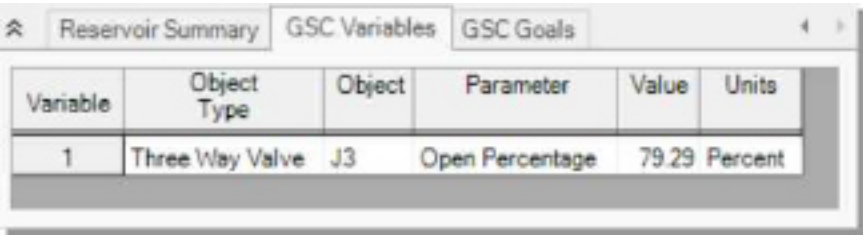
When using the GSC module, a new area appears in Solution Progress that shows the specific progress of the GSC module. As it makes progress, the Best (Lowest) value will decrease towards zero. The field in the far right displays how many complete hydraulic solutions have been run.

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

GSC Step 7. Examine the results
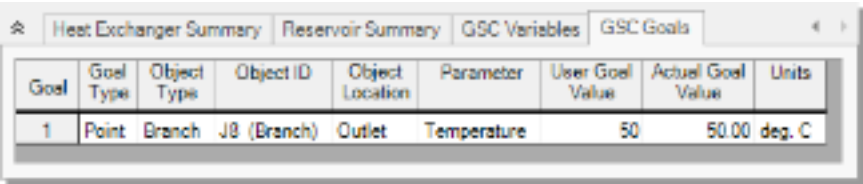
The results of the GSC analysis are shown in the General Output section.

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

Figure 5.8   **The final GSC Variable values are shown on the GSC Variables tab in the Output window General section**

Figure 5.9   **The final GSC Goal values are shown on the GSC Goals tab in the Output window General section**

Note: The Actual and User values should be close if the GSC module was successful. If not, a warning will appear.

GSC analysis summary

For this example, the goal of 50 deg. C was achieved for the temperature at branch junction J8 as shown in Figure 5.9. The open percentage required for the Three-Way Valve to achieve a temperature of 50 deg. C at Branch J8 is about 79.3%, as shown in Figure 5.8.

XTS problem statement

Use the Extended Time Simulation Add-on Module to determine the time required to fill a finite, 6 meter high, 7.5 m^2 cross sectional area discharge reservoir and to turn off the pump when the tank is full.

XTS Step 1. Create a child scenario for the XTS case

Create a child scenario from the Base Scenario for the XTS case by selecting the Base Scenario, then clicking Create Child in the Scenario Manager in the Quick Access Panel. Name the child XTS Example.

XTS Step 2. Activate the XTS Module

Open Analysis Setup then open the Modules panel. Under XTS (Extended Time Simulation) check the box next to Activate XTS. Make sure the radio button for Transient is selected.

If prompted, select Yes to disable heat transfer. The XTS module does not perform heat transfer calculations. After enabling the XTS module, the Heat Transfer/Variable Fluids panel automatically switches the selection to Constant Fluid Properties, if not already selected.

With the XTS module activated, the Transient Control group appears in the Analysis Setup.

➢ **Important:** The XTS and GSC modules can be used in the same run to simulate control functions over time, but that is not required for this example. Before proceeding, make sure the GSC module is not enabled by either unchecking the Activate GSC box or selecting the Ignore radio button in the Analysis Setup.

XTS Step 3. Modify the existing model

When using the XTS module, reservoirs may be defined as finite sized tanks. Specify the J9 Reservoir as a Finite Open Tank with the following data and as shown in Figure 5.10:

J9 - Reservoir

1. Name = Discharge Reservoir

2. Tank Model = Finite Open Tank

3. Known Parameters Initially = Both Parameters

4. Cross-Sectional Area = Constant, 7.5 meters2

5. Liquid Height = Height from Bottom, 0 meters

6. Liquid Surface Pressure = 0.3 barG (30 kPa (g))

7. Tank Height = 6 meters

8. Pipe Elevation = 6 meters

9. Tank Bottom Elevation = 6 meters
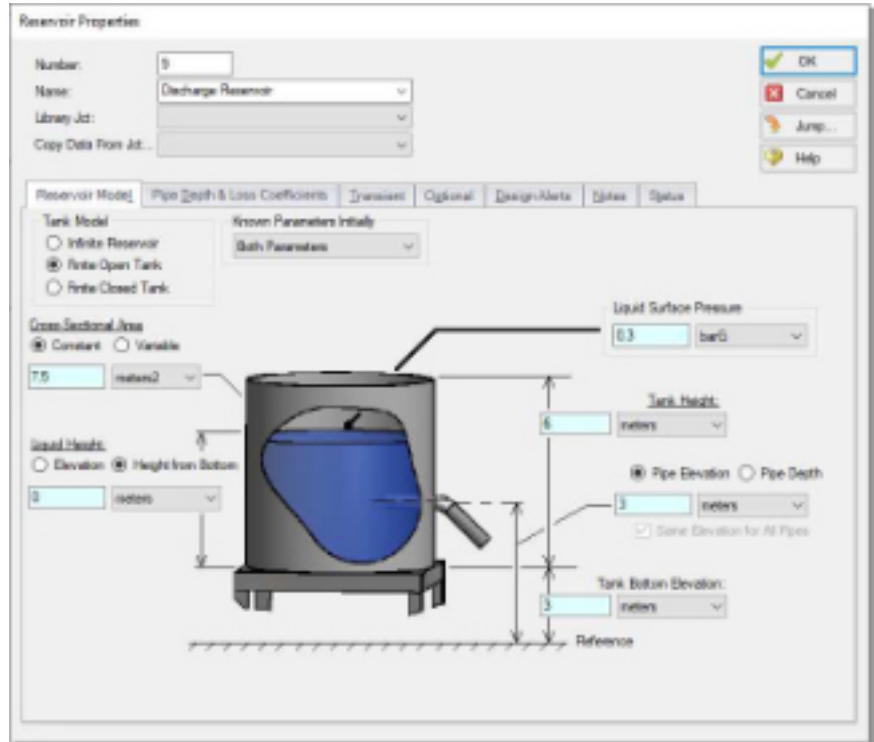
Figure 5.10   J9 Reservoir specified as a finite open tank

J3 Three-Way Valve

1.   Actual Percent Open = 79%

Check if the pipe and junction data is complete

Turn on Show Object Status from the View menu to verify that all the necessary pipe and junction data is entered. If so, the Pipes and Junctions group in the Analysis Setup will have a checkmark. If not, the incomplete pipes or junctions will have their number shown in red. If this happens, go back to the incomplete pipes or junctions and enter the missing data. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

XTS Step 4. Specify transient output time format

Open the Output Control window and navigate to the Format & Action tab. On the bottom-right, the Time Simulation Formatting section controls the units displayed in the transient output data.

Set the Time Simulation Formatting units to minutes, as shown in Figure 5.11, and then select OK to close the window.
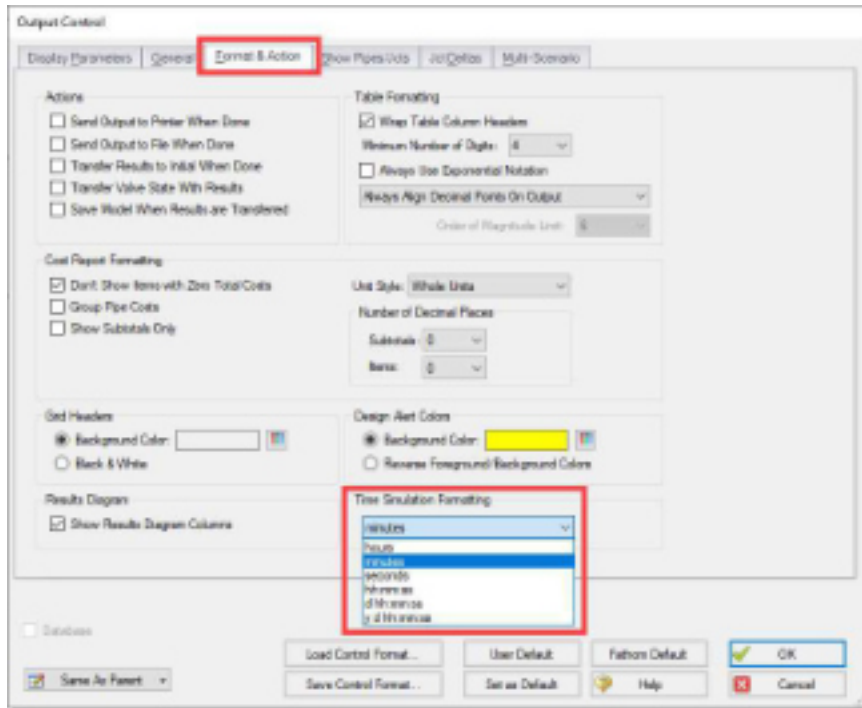
Figure 5.11   Adjusting the Time Simulation Formatting to minutes

XTS Step 5. Activate transient analysis

You can turn the XTS module on and off to run steady-state models using AFT Fathom. Use this feature to check the model steady-state solutions before attempting to run a transient case.

Toggle the time simulation mode between Steady Only and Transient from the Time Simulation option in the Analysis menu or from the

Modules panel in Analysis Setup. To run a transient analysis using XTS, select the Transient time simulation option, as shown in Figure 5.12.
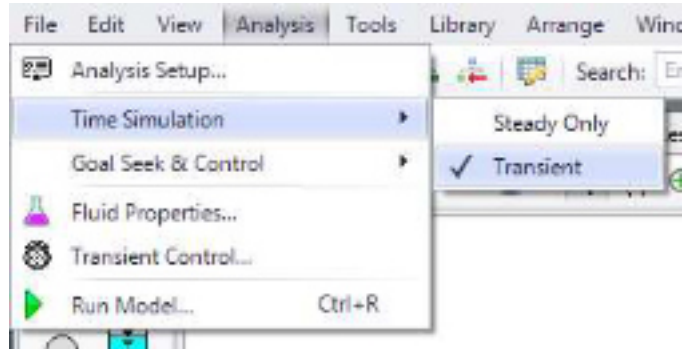
Figure 5.12   Transient Analyses are activated from the Analysis menu

XTS Step 6. Open transient control

XTS module transient simulations are controlled through the Simulation Duration panel. You can access the Simulation Duration panel by selecting Transient Control from the Analysis Menu, navigating to the Simulation Duration item in Analysis Setup, or by clicking the Simulation Duration shortcut on the Common Toolbar. When the time simulation mode is set to Steady Only, AFT Fathom will ignore the input in the Simulation Duration panel. When AFT Fathom is set to perform a transient analysis, the Transient Control becomes a required item before the model can run.

Use the Simulation Duration panel to specify transient start and stop times, set time steps, specify the number of output points that are saved to the transient output file (allows XTS to do transient calculations with a small time step while limiting the size of the transient data output file), and adjust solution parameters for finite tank liquid level calculations.

> ➢ **Set the Start Time** to 0 minutes and the Stop Time as 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 5.13.
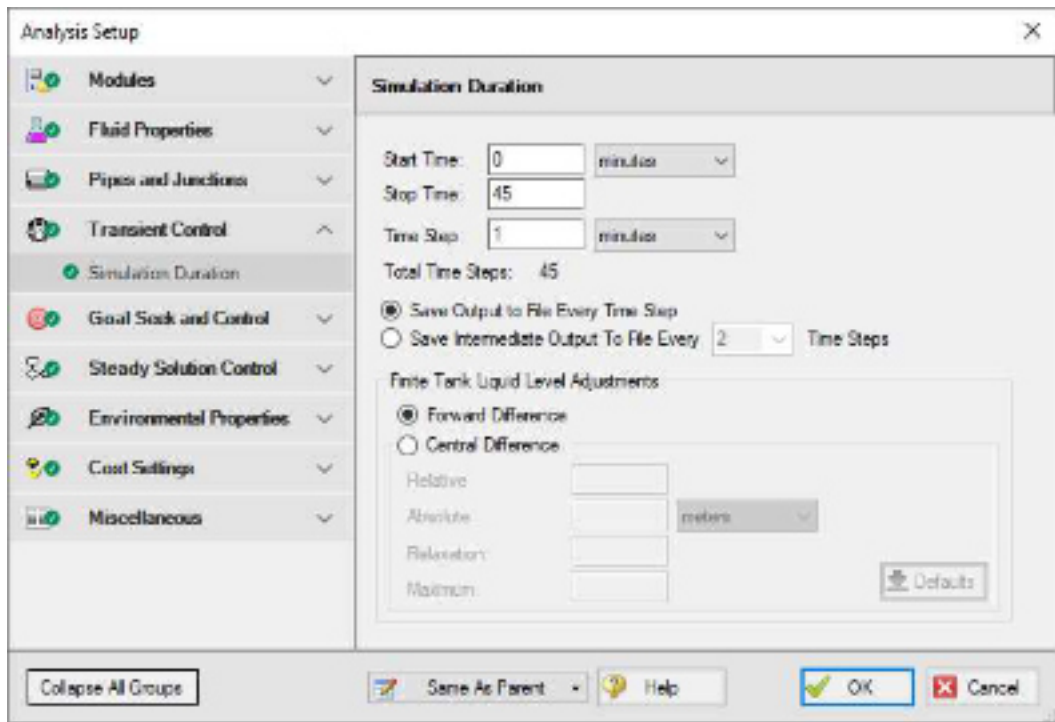
Figure 5.13   The Transient Control group is used to specify transient run times and time step sizes

XTS Step 7. Set up the pump transient

In the XTS module, you can define transients for junctions. These transients include events such as valves opening and closing and pumps starting and tripping. Transients for junctions are defined as either time-based or event-based transients. A time-based transient starts at a specified absolute time. Event based transients initiate when a defined criteria is reached (such as a specified pressure in a particular pipe).

There are three types of event transients for junctions: single event, dual cyclic, and dual sequential. 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 trigger occurs 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, set a single event-based transient on the pump so the pump speed decreases from 100% to 0% in 10 seconds when the liquid level in the discharge tank reaches 6 meters.

Note: AFT Fathom does not capture waterhammer effects. For fast transient events where waterhammer, surge pressures, acoustic wave propagation, and forces are of interest, use AFT Impulse.

Open the Pump Properties window for the J2 Pump and navigate to the Transient tab. Select the Speed vs. Time option, and under Initiation of Transient, select Single Event and enter the Transient data as shown below:

1. Event Type = Reservoir Liquid Height

2. Condition = Greater Than or Equal To

3. Value = 6 meters

4. Junction = 9 (Discharge Reservoir)

5. Transient Data = Absolute Values

Time (seconds)	Speed (Percent)
0	100
10	0

Note that after entering the transient data in the pump, a red T displays next to the junction number for the pump on the Workspace. This symbol indicates that the junction has transient data.
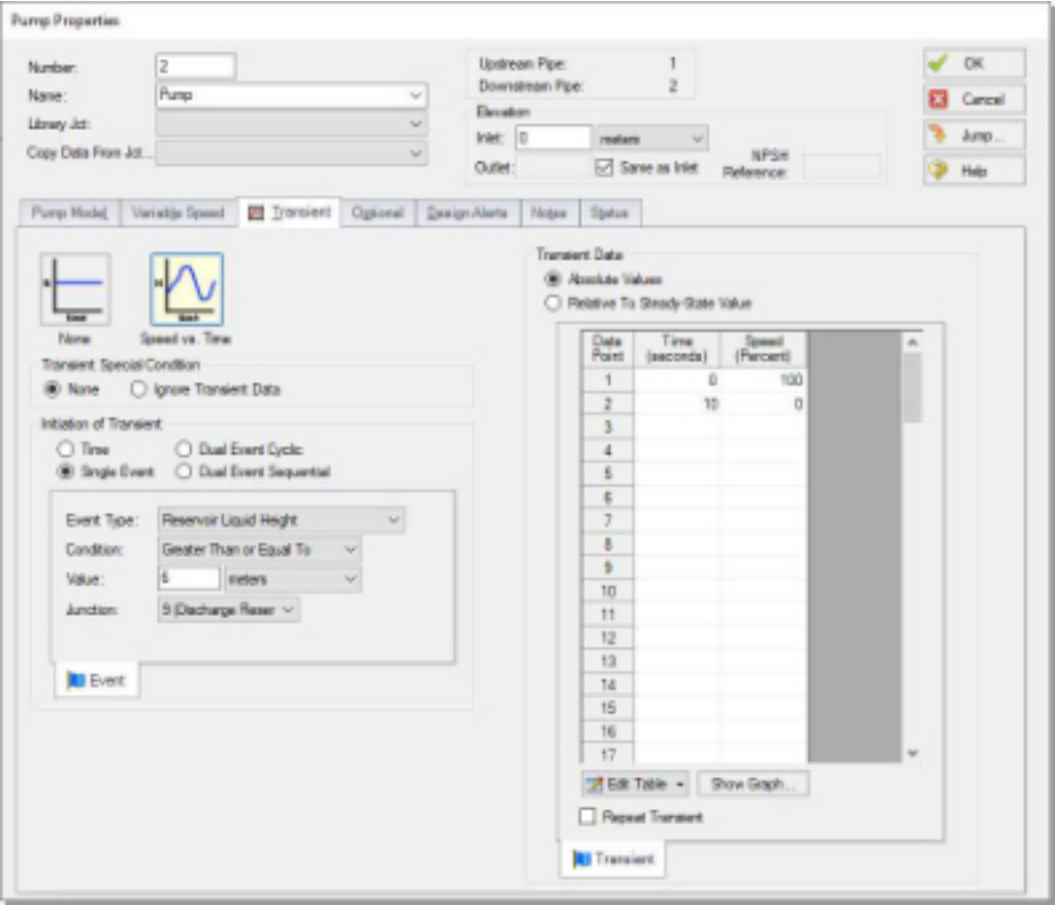
Figure 5.14   Junction transients are specified on the Transient tabs on the junction Properties windows

XTS Step 8. Run the model

After defining all of the pipe and junction transient data and transient control items, the transient model can run. Click Run Model from the Common Toolbar or Analysis menu.

When using the XTS module, the Solution Progress window displays the progress through the transient analysis. This information displays below the solution tolerance data, as shown in Figure 5.15.
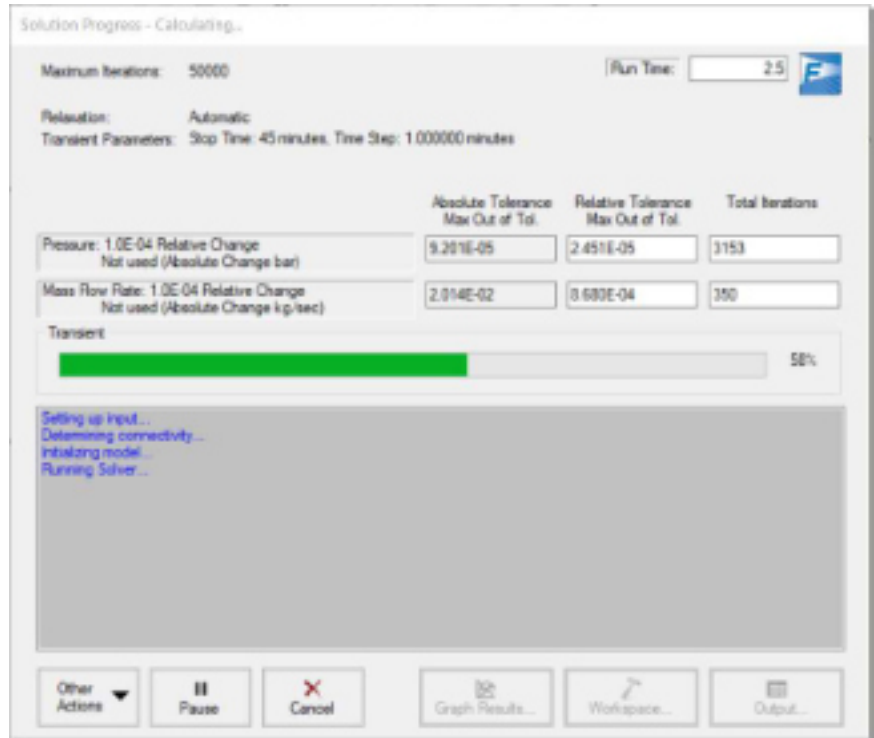
Figure 5.15   The Solution Progress window shows the progress of the transient analysis

After the run finishes, view the results by clicking the Output button.

XTS Step 9. Examine the transient results

The first thing that you should notice after navigating to the Output is that Critical Warnings Exist, as shown in Figure 5.16. For this scenario, there is a critical warning because *Junction 2 Had Reverse Flow – Pump Head Could Not Be Predicted* (32 minutes). There is also a Warning message alerting you that the *Tank Junction 9 began overflowing at 31 minutes*.

A Critical Warning is very important and should never be ignored. Examine the transient results of this scenario to understand why this warning occurs and how it affects the model.
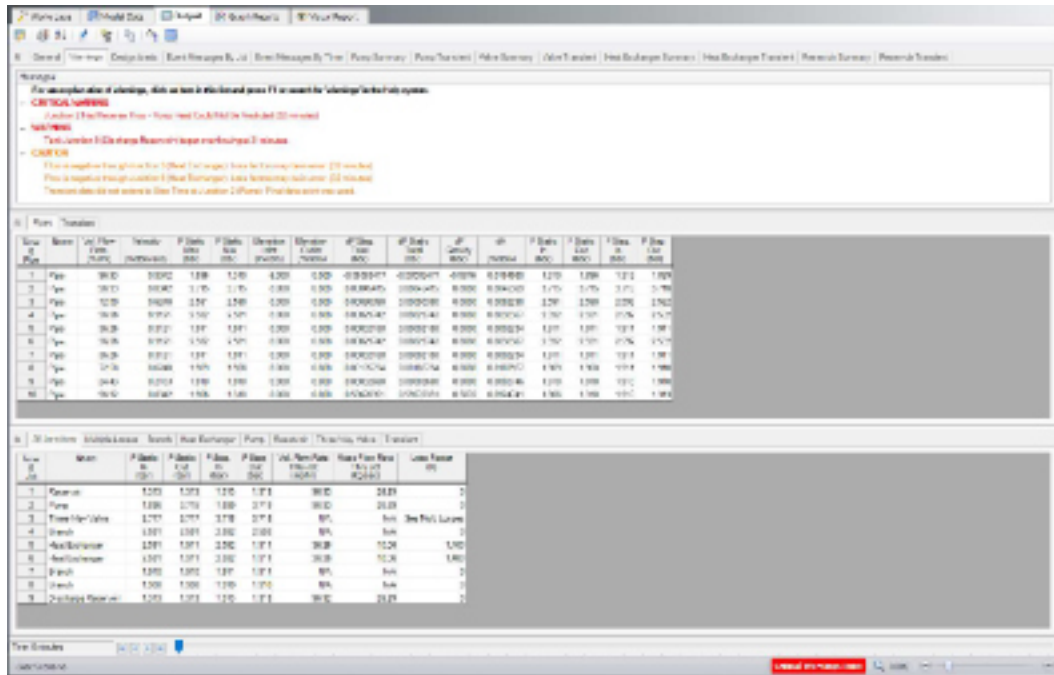
Figure 5.16   A Critical Warning Exists because there is reverse flow through the pump - the warnings and transient results need review

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. The transient summary data for each junction may be expanded or collapsed by clicking the + or – sign beside the junction data list. Figure 5.17 shows the Reservoir Transient summary tab, with the data for the J1 Reservoir collapsed, and the J9 Reservoir expanded.

The Quick Graph feature can be used to conveniently plot transient data for rapid examination in any of the Output sections (with one of the Transient tabs selected). To use the Quick Graph feature, place the mouse cursor over one of the cells of data in the column you wish to examine. Then, right-click with the mouse, and select Quick Graph from

the list of options. Figure 5.17 illustrates how to generate a Quick Graph in the General Output section for the J9 Reservoir Liquid Height vs. Time. Figure 5.18 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.
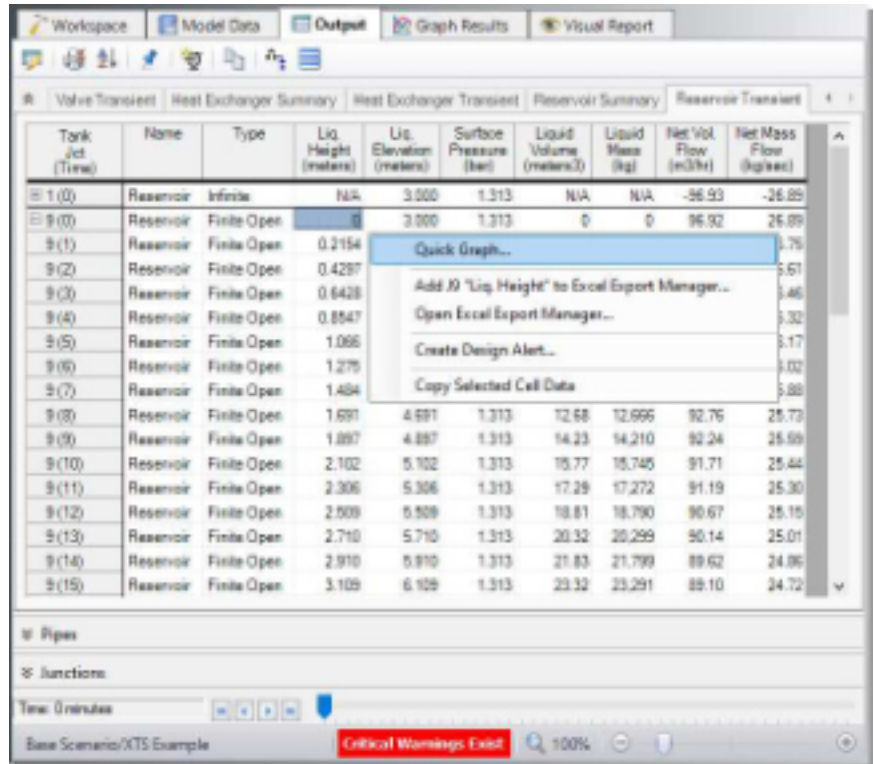
Figure 5.17   Access Quick Graphs from any transient data column
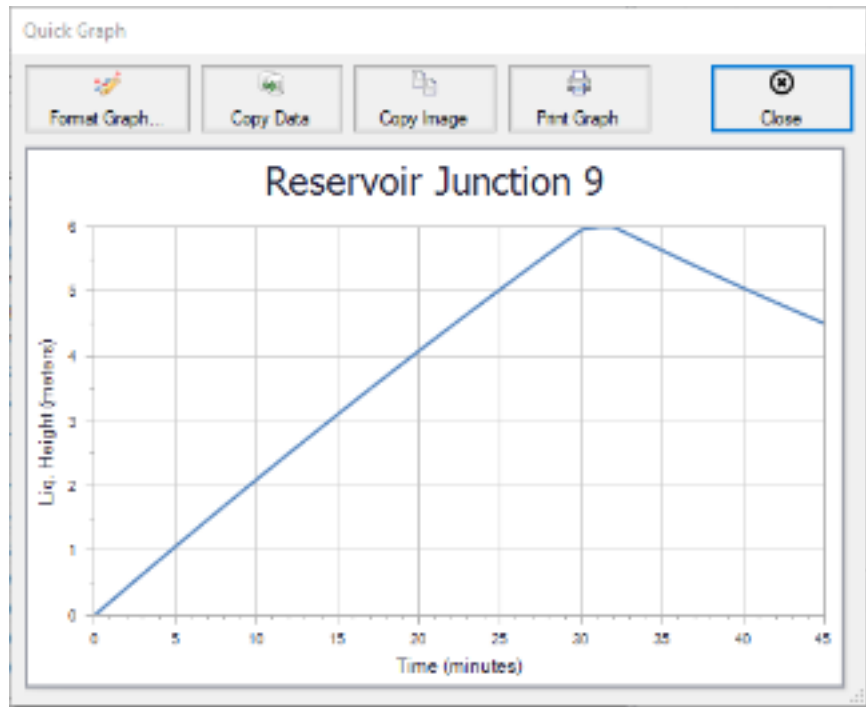
Figure 5.18   Quick Graphs plot transient data from the Output

Consider Figure 5.18 in terms of the Critical Warning, which states that pump head could not be predicted at 32 minutes due to reverse flow through the pump. The tank liquid height increases until the pump turns off. The graph then shows that there is backflow into the system, which causes the Critical Warning. Furthermore, the Warning that stated Tank Junction 9 began overflowing is due to the pump event trigger being set equal to or greater than the max tank liquid height and the timestep resolution being one minute. Note that the reservoir liquid height reported on the graph and in the Output will be capped at the tank height, but the XTS module recognizes that the tank would have overflowed and alerts the user. 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 5.19. This transient data can be expanded or collapsed in the same manner as in the General Output transient summaries.

Figure 5.20 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.

Figure 5.19   The Transient tabs show results for all time steps
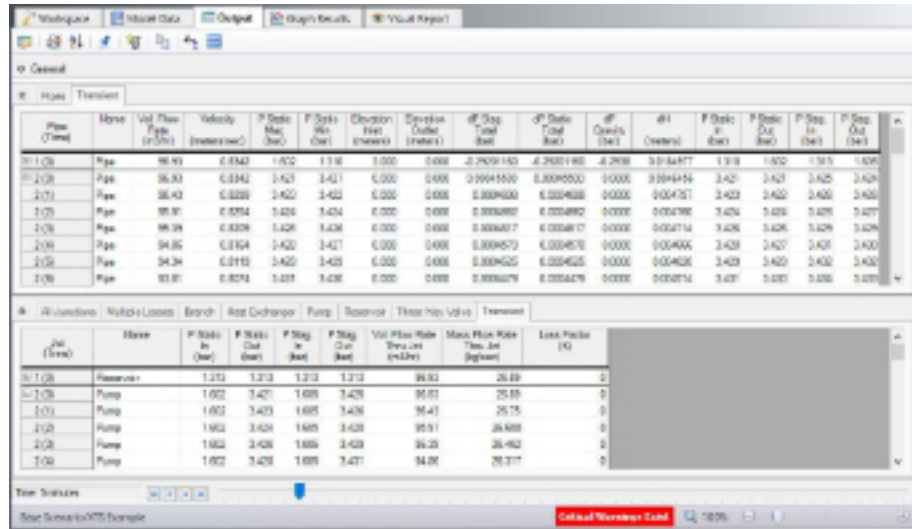
Figure 5.20   Use the slider to see the output at any time step

The general output section of the Output window displays the junction event transients on two different tabs. The first tab sorts the events by junction number, shown in Figure 5.21, and the second tab sorts the events by time in the order in which they occur.
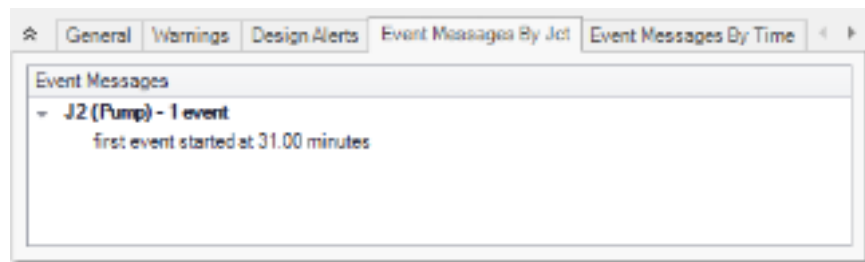
Figure 5.21   The occurrence of event transients is recorded in the General Output section

XTS analysis summary

The transient reservoir data shows that the water in the discharge tank reached a height of 6 meters in 31 minutes. The event message indicating the pump transient also initiated at this time confirms this.

The Quick Graph plot (Figure 5.18) shows the increase in the tank liquid height, until the time when the pump turns off. The XTS module gives a Warning that the tank would have overflowed due to the pump event being triggered when the liquid height reaches the maximum height of the tank. To prevent this Warning, change the event trigger to a lower value for the tank liquid height. The graph then shows that there is actually backflow through the pump, due to the hydrostatic pressure of the fluid in the tank. The pump head cannot be calculated when the pump has reverse flow, 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

Use the Settling Slurry Add-on Module to determine the size of the pump when the fluid is 25% sand by volume, and is pumped from an open supply vessel with a liquid surface elevation of 1.5 meters to an open receiving tank with a liquid surface elevation of 3 meters.

This example uses a different model than previously used for the GSC and XTS modules.

SSL Step 1. Create a new blank model and add slurry units

➢ **Select File > New to create a blank model file.**

➢ **Open Analysis Setup and disable the XTS module (the GSC module should also be disabled from the prior XTS example).**

➢ **From the Tools menu, open the User Options and select the Unit System > Preferred Units item. Ensure that the filter is set to either All Units or Common Plus Selected Industries with the checkbox next to Slurry.**

SSL Step 2. Specify fluid properties

1. Open Analysis Setup.

2. On the Modules panel, under SSL (Settling Slurry), check the box next to Activate SSL and select the Settling radio button. Note that the items in the Fluid Properties group have changed.

3. Go to the Carrier Fluid panel, select the Basic (Water) carrier fluid option, and enter a temperature of 21 deg. C.

   Note: The Advanced carrier fluid option can be used for non-water based slurries.

4. Go to the Slurry Definition panel and change the Slurry Model to Settling Slurry - Wilson, Addie, Clift.

5. Go to the Solids Definition panel, select Detailed as the Slurry Calculation Method, and select User Specified Solids Added.

6. Under M (Stratification Ratio Exponent), select the radio button for Calculated.

7. Under Terminal Velocity Parameter, select the radio button for Vt/Vts (Xi) and set it to 1.

8. Enter the following solid properties, as shown in Figure 5.22:

   a. Density = 2.9 S.G. water

   b. d50 = 0.0381 cm

   c. d85 = 0.0762 cm

9. Return to the Slurry Definition panel and enter the the following:

   a. Concentration Type = Volume Fraction

   b. Amount Solids Added = 25%

10. Click OK
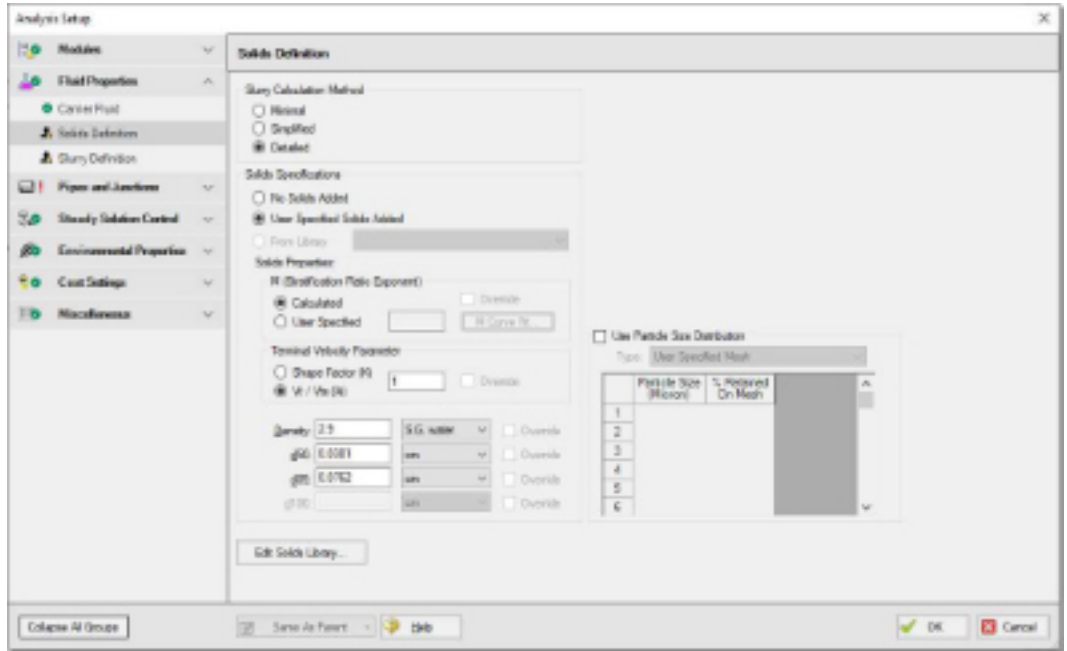
Figure 5.22   Fully Defined Solids Definition panel

SSL Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 5.23.
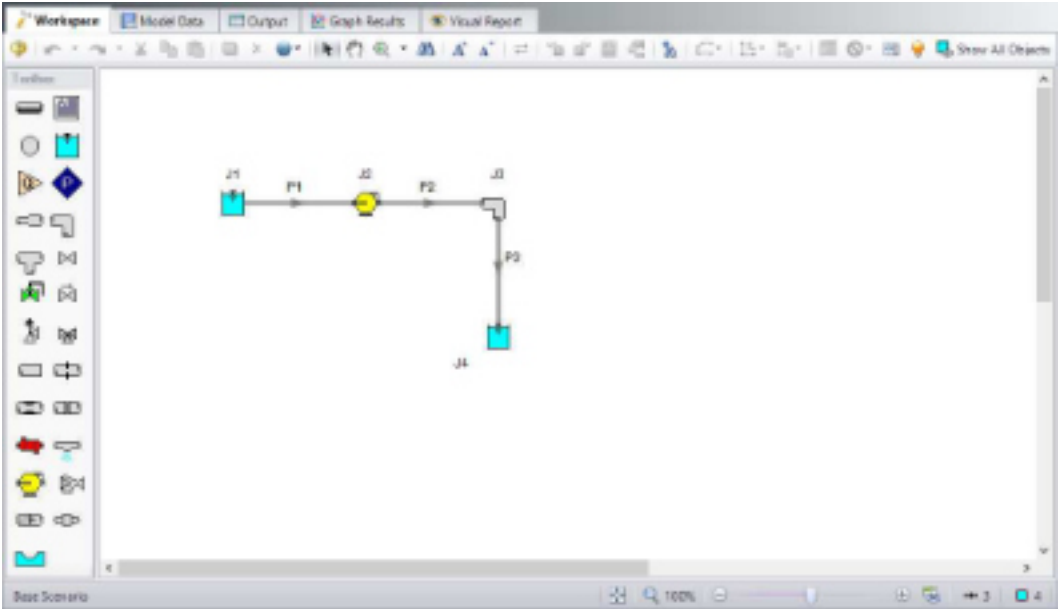
Figure 5.23   Layout of the pipe system for the Pump Sizing for Sand Transfer System Example

B. Define the pipes

With the objects in place, define the pipes and junctions according to the information below.

For all pipes, use

- Pipe Material = Steel ANSI

- Size = 12 inch (30.48 cm ID)

- Type = STD

- Friction Model = Standard

- Length = use the table below

Pipe	Length (meters)
P1	6

Pipe	Length (meters)
P2	150
P3	15

C. Define the junctions

J1 - Reservoir

1. Name = Supply Tank

2. Liquid Surface Elevation = 1.5 meters

3. Liquid Surface Pressure = 0.7 barG (70 kPa (g))

4. Pipe Depth = 1.5 meters

J2 - Pump

1. Pump Model tab

    a. Inlet Elevation = 0 meters

    b. Pump Model = Centrifugal (Rotodynamic)

    c. Analysis Type = Sizing

    d. Pump Sizing Parameter = Mass Flow Rate

    e. Fixed Flow Rate = 1270 m-tons/hr Solids

2. Slurry De-Rating tab

    a. De-Rating Method = ANSI/HI Standard 12.1-12.6-2016

    b. Head De-Rating Data, Percent Fines = 0%

    c. Head De-Rating Data = Specify Impeller Size Directly

    d. Impeller Size = 20 cm

J3 - Elbow

1. Inlet Elevation = 0 meters

2. Type = Standard Elbow (knee, threaded)

3.   Angle = 90 Degrees

J4 - Reservoir

1.   Name = Receiving Tank

2.   Liquid Surface Elevation = 3 meters

3.   Liquid Surface Pressure = 2 barG (200 kPa (g))

4.   Pipe Depth = 3 meters

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom-right of the Quick Access panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions, and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

SSL Step 4. Run the model

Click Run Model from the Common Toolbar or Analysis menu.

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

SSL Step 5. Examine the results

The Slurry tab shows the slurry results, as shown in Figure 5.24. Here, one can see that the velocity is 6.674 meters/sec, which is safely above the 3.498 meters/sec max settling velocity. Other pressure drop and relevant slurry data can be viewed here.

Also shown in Figure 5.24 is the Pump Summary in the General Section. There, the required pump head is shown as 27.81 meters, and the de-rating head correction factor (CH) is 0.7887 (the CH parameter is added via Output Control under the Pump options). Hence, a pump should be selected that can generate 27.81/0.7887 = 35.26 meters of head based on water.

Also of interest is the system curve. Navigate to the Graph Results window, and then the Graph Parameters tab on the Quick Access Panel. Then select the Slurry System Curve tab on the right. See Figure 5.25.

By default, the parameter to graph is j_m vs. velocity. Select the units of j_m as m/m (height of mixture head divided by pipe length). Check the box for cross-plotting the water curve and then click the Generate button. The curves are shown in Figure 5.26.
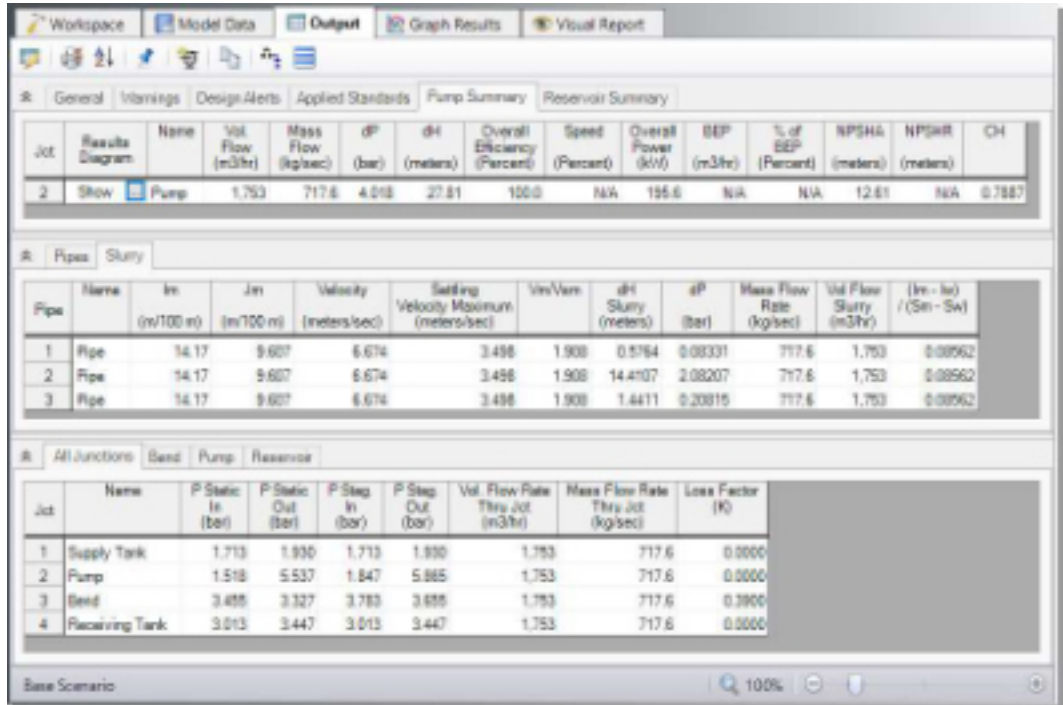
Figure 5.24   The Output window shows the Slurry results table in the center Pipes section and the pump de-rating in the Pump Summary in the upper General section
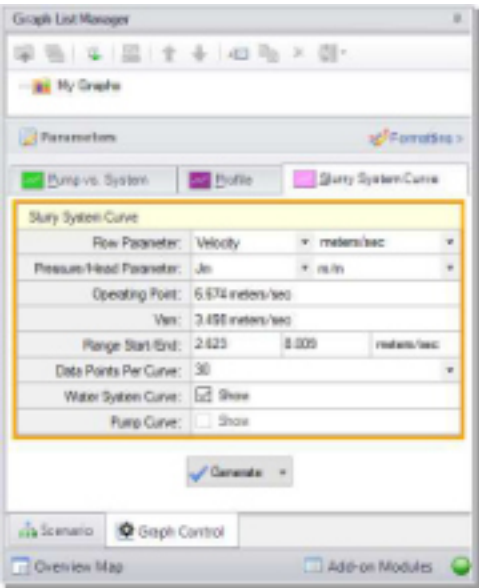
Figure 5.25   Data selection for graphing slurry system curve
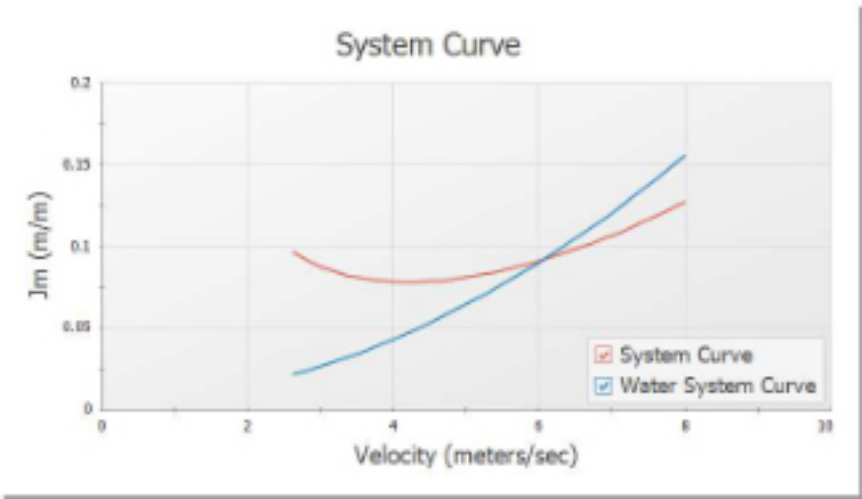
Figure 5.26   Slurry system curve and water system curve in the Graph Results window

SSL analysis summary

The pump head requirements were determined, and a de-rating factor was calculated according to the ANSI/HI Standard 12.1-12.6-2016. The de-rating factor for pump head (CH) was used to manually calculate pump head using water, such that the correct size pump could be selected when lookig at manufacturer data based on water curves. The velocity in each pipe was found to be higher than the max settling velocity (Vsm) of the sand and water slurry which ensured that no settling would occur. A slurry system curve was created to allow the user to see where the system was operating with respect to solids settling.

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.

You can find a number of other example model discussions on the AFT documentation help website. To view additional examples, open the Help menu and choose Show Examples or go to docs.aft.com, select the AFT Fathom Help Site, expand the Examples section in the sidebar, then select Walk-Through Examples.

Microsoft Excel™ data integration

AFT Fathom 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 (PRVs), Pressure Sustaining Valves (PSVs), Flow Control Valves (FCVs), and Pressure Drop Control Valves (PDCVs).

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 flow rate, pressure, or head. The pump speed is varied to meet the setpoint.

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

Compare input from different scenarios

The Scenario Comparison Tool can be used to compare pipe, junction, and general model file inputs for any number of scenarios in the model file. This allows the user to quickly identify differences between scenarios, or to generate a report on inputs for different cases.

Multi-scenario pump vs. system curve graphs

Graphs with multiple pump curves and/or system curves can be generated directly in AFT Fathom by saving multiple pump vs. system curves to a Multi-Scenario Pump vs. System Folder in the Graph List Manager. A single graph can then be generated with curves from different scenarios.

Online pump manufacturer equipment selection

AFT Fathom includes links to various pump manufacturer websites that use Intelliquip™. Operating data determined by AFT Fathom can be passed to the website, a pump model can be selected by the user, and the pump performance data can be transferred back to AFT Fathom for immediate use in the model.

Network libraries

Junction components and pipe materials can be saved to libraries for later reuse. Libraries can be located on local PCs or deployed across local or wide area networks. The Library Manager allows users to connect to relevant libraries 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 transitional flow regions.

Equivalent lengths

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

Design Alerts

Design Alerts can be created for pipes and junctions to alert the user that one of their project constraints has been violated. A common use is maximum and minimum allowed operating pressure. If the resulting value for that parameter violates the Design Alert a warning message is given in the Output.

Cost calculations

Material, Installation, and Maintenance costs can be calculated for junctions, pipes, and fittings based on Cost Libraries which are user-created. In addition, energy (operation) costs can be calculated for compressors/fans. Time value of money can be taken into consideration by entering inflation and interest rates. The results are presented in a Cost Report in the Output.

Retaining and reproducing graphs

Comprehensive graphing features allow users to graph a wide variety of parameters at single points or along flow paths within their model. Graphs added to a Graph List are saved so that the graph can be regenerated with the same styles and customization if changes are made to the model input, or if the user desires to plot the same graph in multiple scenarios. Multiple graphs can be shown at the same time.

4-Component Slurry Model

The 4-Component Slurry Model is a holistic approach to characterizing how slurry flows function by considering the combined effects of four slurry flow regimes: the carrier fluid, pseudo-homogeneous, heterogeneous, and fully stratified. The model strongly references and utilizes the method of the same name outlined by GIW Industries Inc.

Index

AFT Fathom™ Add-on Modules

ANS
Automated Network Sizing
Evaluates the complex interaction of system variables to reveal combinations of cost savings

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

AFT
Applied Flow Technology