

# **AFT Impulse<sup>TM</sup> 9**

## **Quick Start Guide**

Metric Units

Dynamic solutions for a fluid world<sup>TM</sup>

# ***AFT Impulse***<sup>™</sup>

## **Quick Start Guide**

**Metric Units**

**AFT Impulse Version 9**

**Waterhammer Modeling in Piping Systems**

**Published: May 01, 2023**



*Dynamic solutions for a fluid world* <sup>™</sup>

---

## CAUTION!

AFT Impulse is a sophisticated waterhammer and surge transient modeling program designed for qualified engineers with experience in waterhammer analysis and should not be used by untrained individuals. AFT Impulse 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 Impulse should be independently verified with other engineering methods.

AFT Impulse 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 is familiar with standard industry practice in waterhammer analysis.

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

---

Information in this document is subject to change without notice. No part of this Quick Start Guide may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose, without the express written permission of Applied Flow Technology.

© 2022 Applied Flow Technology Corporation. All rights reserved.

Printed in the United States of America.

First printing.

"AFT Impulse", "AFT Fathom", "Applied Flow Technology", "Dynamic solutions for a fluid world", and the AFT logo are trademarks of Applied Flow Technology Corporation.

Excel and Windows are trademarks of Microsoft Corporation. Intelliquip is a trademark of Intelliquip, LLC. Chempak is a trademark of Madison Technical Software, Inc. CAESAR II, SmartPlant, CADWorx, and PDS are trademarks of Intergraph Corporation. ROHR2 is a trademark of SIGMA Ingenieurgesellschaft mbH. AutoCAD Plant 3D is a trademark of Autodesk, Inc.

# Contents

<b>1. Introducing AFT Impulse</b>	<b>1</b>
Designing for waterhammer	1
Modeling capabilities	2
Add-on module capabilities (optional)	2
The steady-state solver	3
The transient solver	3
Thermophysical property data	3
Engineering assumptions in AFT Impulse	4
AFT Impulse Primary Windows	4
Input windows	5
Output windows	5
<b>2. Valve Closure Example</b>	<b>7</b>
Topics covered	7
Required knowledge	7
Model file	8
Problem statement	8
Step 1. Start AFT Impulse	8
A. The Startup window	8
B. The Workspace window	12
C. Unpinning the Quick Access Panel	13
Step 2. Lay out the model	13
A. Place a Reservoir	13
Objects and ID numbers	14
Editing on the Workspace	14
B. Place the other junctions	15
C. Draw a pipe between J1 and J3	16
D. Add the remaining pipes	18
Reference positive flow direction	18

## ii AFT Impulse™ 9 Quick Start Guide

---

Step 3. Specify Analysis Setup .....	19
A. Specify the Modules group .....	20
B. Specify the Fluid Properties group .....	21
C. Specify the Pipes and Junctions group .....	22
Object status .....	22
Showing undefined objects .....	23
Enter data for the reservoirs .....	23
Using the tabs in the properties windows .....	25
The Inspection feature .....	26
Enter data for the branch .....	27
Enter data for the valve .....	27
Enter data for Pipe P1 .....	29
The Pipe Properties window .....	30
Wavespeed .....	31
Enter data for other pipes .....	31
Check pipes and junctions .....	32
Review model data .....	32
Reviewing input in the Model Data window .....	32
Steady State groups are now fully defined .....	33
D. Specify the Pipe Sectioning and Output group .....	33
E. Specify the Transient Control group .....	36
F. Save the model .....	37
Step 4. Run the solver .....	37
A. The two solvers .....	38
B. The transient output file .....	39
Step 5. Review the Output window .....	39
Step 6. View the graph results .....	42
A. Further review .....	47
Step 7. View the Visual Report .....	49
Conclusion .....	52
 <b>3. Pump Startup with Event Transients .....</b>	<b>53</b>
Topics covered .....	53

Required knowledge .....	53
Model file .....	54
Problem statement .....	54
Step 1. Start AFT Impulse .....	54
Step 2. Define the Fluid Properties group .....	54
Step 3. Define the Pipes and Junctions group .....	55
A. Place the pipes and junctions .....	55
B. Enter the pipe and junction data .....	55
Pipes .....	56
J1 - Reservoir .....	56
J10 - Reservoir .....	57
J11 - Reservoir .....	57
J6, J7, & J8 - Branches .....	57
J2 & J4 - Pumps .....	57
J3 & J5 - Valves .....	58
J9 - Valve .....	59
C. Check if the pipe and junction data is complete .....	61
Step 4. Specify the Pipe Sectioning and Output group .....	61
Step 5. Specify the Transient Control group .....	62
Step 6. Create scenarios to model the three startup cases .....	62
A. Create scenarios .....	63
B. Set up scenarios .....	64
Two Pump Start .....	64
One Pump Start .....	64
One Pump Start with One Running .....	65
Step 7. Run the model .....	66
Step 8. Graph the results .....	66
A. Create a profile graph .....	66
B. Create a Transient Junction graph .....	68
C. Create a Transient Pipe graph .....	70
Step 9. Animate the results .....	71
Step 10. Run the other scenarios and graph the results .....	73
Conclusion .....	75

<b>4. Pump Trip Example .....</b>	<b>77</b>
Topics covered .....	77
Required knowledge .....	77
Model file .....	78
Problem statement .....	78
Step 1. Start AFT Impulse .....	78
Step 2. Define the Fluid Properties group .....	78
Step 3. Define the Pipes and Junctions group .....	79
A. Place the pipes and junctions .....	79
B. Enter the pipe and junction data .....	80
Pipes .....	80
J1 & J2 - Reservoirs .....	81
J3 - Branch .....	81
J4 - General Component .....	81
J5 & J10 - Branches .....	81
J11 - Branch .....	81
J6 & J8 - Pumps .....	82
J7 & J9 - Valves .....	84
J12 & J13 - Spray Discharges .....	84
C. Check if the pipe and junction data is complete .....	85
Step 4. Specify the Pipe Sectioning and Output group .....	85
Step 5. Define the Transient Control group .....	85
Step 6. Run the model .....	85
Step 7. Review results .....	85
A. Graph the transient pressures and flows at the pump station ....	86
B. Graph the pump speed decay .....	88
Step 8. Create additional pump scenarios .....	89
A. Four Quadrant BEP .....	90
B. Four Quadrant SSOP .....	92
Step 9. Run the four quadrant scenarios .....	92
Step 10. Adjust the valve closure .....	95
Step 11. Specify the Transient Control group .....	95



Step 12. Run the model .....	96
Step 13. Graph results .....	96
Conclusion .....	97

## **5. Valve Closure With Force Sets Example ..... 99**

Topics covered .....	99
Required knowledge .....	99
Model file .....	100
Step 1. Start AFT Impulse .....	100
Step 2. Open model file .....	100
A. Isometric Drawing Mode .....	100
Step 3. Specify the Pipe Sectioning and Output group .....	102
Step 4. Specify the Transient Control group .....	102
Step 5. Define the Forces group .....	102
Step 6. Run the model .....	103
Step 7. Graph the pipe forces .....	104
A. Final notes .....	105
Analysis summary .....	105

## **6. AFT Impulse Add-on Modules Examples ..... 107**

Topics covered .....	107
Required knowledge .....	107
Model files .....	108
SSL problem statement .....	108
SSL Step 1. Start AFT Impulse .....	108
SSL Step 2. Define the Fluid Properties group .....	108
A. Carrier Fluid panel .....	109
B. Solids Definition panel .....	109
C. Slurry Definition panel .....	110
SSL Step 3. Define the Pipes and Junctions group .....	111
A. Place the pipes and junctions .....	111
B. Enter the pipe and junction data .....	112
Pipes .....	112



J1 - Reservoir .....	113
J3 - Branch .....	113
J5 - Branch .....	113
J8 - Branch .....	113
J6 - Reservoir .....	113
J9 - Reservoir .....	114
J2 - Pump .....	114
J4 - Valve .....	114
J7 - Valve .....	115
C. Check if the pipe and junction data is complete .....	115
SSL Step 4. Specify the Pipe Sectioning and Output group .....	116
SSL Step 5. Specify the Transient Control group .....	116
SSL Step 6. Run the model .....	116
SSL Step 7. Review results .....	116
A. Check the Transient Max/Min .....	116
B. Graph the velocity ratio and pressures to Deposit #1 .....	118
C. Graph the velocity ratio over time .....	120
SSL analysis summary .....	121
PFA problem statement .....	121
PFA Step 1. Start AFT Impulse .....	122
PFA Step 2. Define the Fluid Properties group .....	122
PFA Step 3. Define the Pipes and Junctions group .....	122
A. Place the pipes and junctions .....	122
B. Enter the pipe and junction data .....	123
Pipes .....	123
J1 & J2 - Assigned Flows .....	124
J3 & J4 - Area Changes .....	124
J8 & J9 - Area Changes .....	124
J5 & J6 - Tees .....	125
J10 - Spray Nozzle .....	125
J7 - Dead End .....	125
C. Check if the pipe and junction data is complete .....	125
PFA Step 4. Define Pulsation Setup group .....	125
A. Define the Pulse Setup panel .....	126

B. Define the PD Pump Settings panel .....	127
PFA Step 5. Section the pipes .....	128
PFA Step 6. Run the model .....	129
PFA Step 7. Review results .....	132
A. Find excitation frequencies to study .....	132
B. Evaluate excitation frequencies .....	133
Determine the pressure response of the system at excited frequencies .....	135
PFA analysis summary .....	140
 <b>7. Other AFT Impulse Capabilities .....</b>	 <b>141</b>
 <b>Index .....</b>	 <b>147</b>



# Introducing AFT Impulse

Welcome to AFT Impulse™ 9, Applied Flow Technology's powerful waterhammer modeling tool. With AFT Impulse you can model transients caused by a wide range of pipe system behavior. This will allow you to understand transient pressure extremes and, when necessary, size and locate surge suppression equipment.

AFT Impulse includes a steady-state solution engine which solves for the system initial conditions. These results are used to automatically initialize the transient model. The engineer accesses these capabilities through an advanced graphical interface, which includes built-in expertise to guide the engineer through the modeling process.

AFT Impulse has two available add-on modules which extend AFT Impulse's modeling capabilities into new areas.

- The Settling Slurry (SSL) module allows the engineer to perform the complex property and system interaction calculations associated with settling slurry flows.
- The Pulsation Frequency Analysis (PFA) module allows the engineer to determine resonant frequencies that can result in excessive vibration and damage to systems.

## Designing for waterhammer

Waterhammer can cause catastrophic failure of pipe systems and damage expensive equipment. Properly addressing such issues at the design stage is essential. An AFT Impulse model allows the engineer to better understand and predict the dynamic behavior of the pipe system.

## 2 AFT Impulse™ 9 Quick Start Guide

---

When undesirable waterhammer transients are identified at the design stage, different strategies to reduce surge pressures can be evaluated. These may include using surge suppression equipment, modifying the design, or modifying the system operation.

Sometimes the undesirable transients are not discovered until after the system is built. In such cases, AFT Impulse can provide critical insight into the cause of the problem and allow the engineer to assess design and/or operational modifications to resolve the issue.

### **Modeling capabilities**

AFT Impulse provides a broad array of features to model pipe system transients. These include:

- Transients in open and closed (recirculating) systems
- Network systems that branch or loop
- Systems with valve transients
- Systems with pump transients
- Systems with turbine transients
- Systems with pressure or flow control valve transients
- Systems with transient cavitation and liquid column separation
- Systems with surge suppression devices such as accumulators, surge tanks and vacuum breaker valves
- Systems with variable density and viscosity
- Multiple design cases in a single model file
- Non-Newtonian fluid behavior

### **Add-on module capabilities (optional)**

- System performance for settling slurry calculations (SSL module)
- Identification of resonant frequencies (PFA module)
- Positive displacement pump speeds which excite resonant frequencies (PFA module)

## **The steady-state solver**

Before a waterhammer model can be run, the initial steady-state conditions are required. AFT Impulse obtains the steady-state solution using a Newton-Raphson matrix solution algorithm to obtain a mass and momentum balance. The algorithm is similar to that used in the acclaimed AFT Fathom™.

If desired, AFT Impulse can be run in “Steady Only” mode where only the steady flow pipe hydraulics are modeled. When run in “Transient” mode, the steady flow solution is used to automatically initialize the transient solution. This convenience helps the user avoid the often error-prone process of manually setting up the initial conditions, and it allows the user to quickly and safely modify and rerun the model.

## **The transient solver**

AFT Impulse employs the traditional Method of Characteristics (MOC) to solve the transient equations of pipe flow. A mass and momentum balance is performed at all computing stations in each pipe, accurately representing the propagation of transient pressure waves throughout the system.

The MOC is an explicit solution technique, where the Solver marches in time for a duration specified by the user.

## **Thermophysical property data**

AFT Impulse derives physical properties from one of five sources. The first is the standard AFT Impulse set of incompressible fluids which contains data for about 30 common fluids (called AFT Standard).

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

The fourth is the NIST REFPROP library, first introduced in AFT Impulse 7. REFPROP is licensed from the National Institute of Standards and Technology and is included in AFT Impulse. REFPROP has a library of approximately 150 fluids and supports user specified

## 4 AFT Impulse™ 9 Quick Start Guide

---

fluid mixtures. AFT Impulse is restricted to non-reacting mixture calculations.

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

### Engineering assumptions in AFT Impulse

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

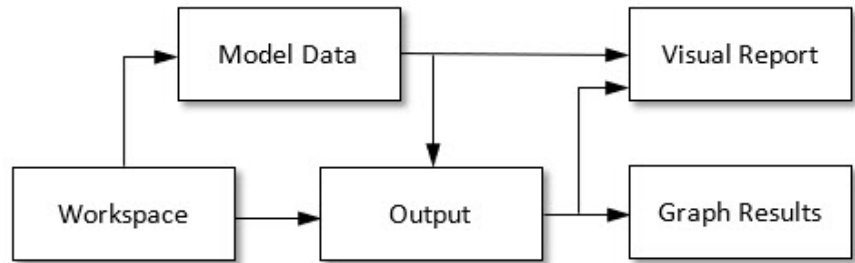
- Liquid flow (liquid full)
- One-dimensional flow
- No chemical reactions
- Wavespeed remains constant during transients
- Non-condensable gas release is negligible
- Bubbles that form during transient cavitation do not move

### AFT Impulse Primary Windows

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

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





**Figure 1.1 Primary Window workflow in AFT Impulse**

### Input windows

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

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

### Output windows

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

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

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

## 6 AFT Impulse™ 9 Quick Start Guide

---

**Note:** AFT Impulse supports multiple monitors. You can click and drag any of the five Primary Window tabs off of the main AFT Impulse window. Once you drag one of the Primary Windows off of the Impulse window, you can move it anywhere you like on your screen, including onto a second monitor in a dual monitor configuration. To add the Primary Window back to the main AFT Impulse primary tab window bar, simply click the X button in the upper right of the Primary Window.

---

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

---

# Valve Closure Example

This chapter is designed to give you the big picture of AFT Impulse's layout and structure. Some of the more basic concepts will be used to build a four-pipe, five-junction model which shows the waterhammer transients that result when a valve is closed.

A number of other example model discussions are included on the AFT website. To get there, open the Help menu and choose Show Examples or go to [www.AFT.com](http://www.AFT.com).

## Topics covered

- Building a basic model
- Entering pipe and junction data
- Defining Analysis Setup
- Entering transient data
- Sectioning pipes
- Graphing output results

## Required knowledge

No prior knowledge is required for this example.

### Model file

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

- *Metric - Beginner Valve Closure.imp*

This example is also provided in US units under file name, *US - Beginner Valve Closure.imp*.

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

### Problem statement

For this problem, water flows from two separate supply tanks and combines into a single 12 inch line before flowing to a downstream reservoir. The flow is controlled by opening and closing a valve in the 12 inch line.

This model will simulate the closure of the valve over a period of one second. The valve will stay closed for the remaining simulation duration.

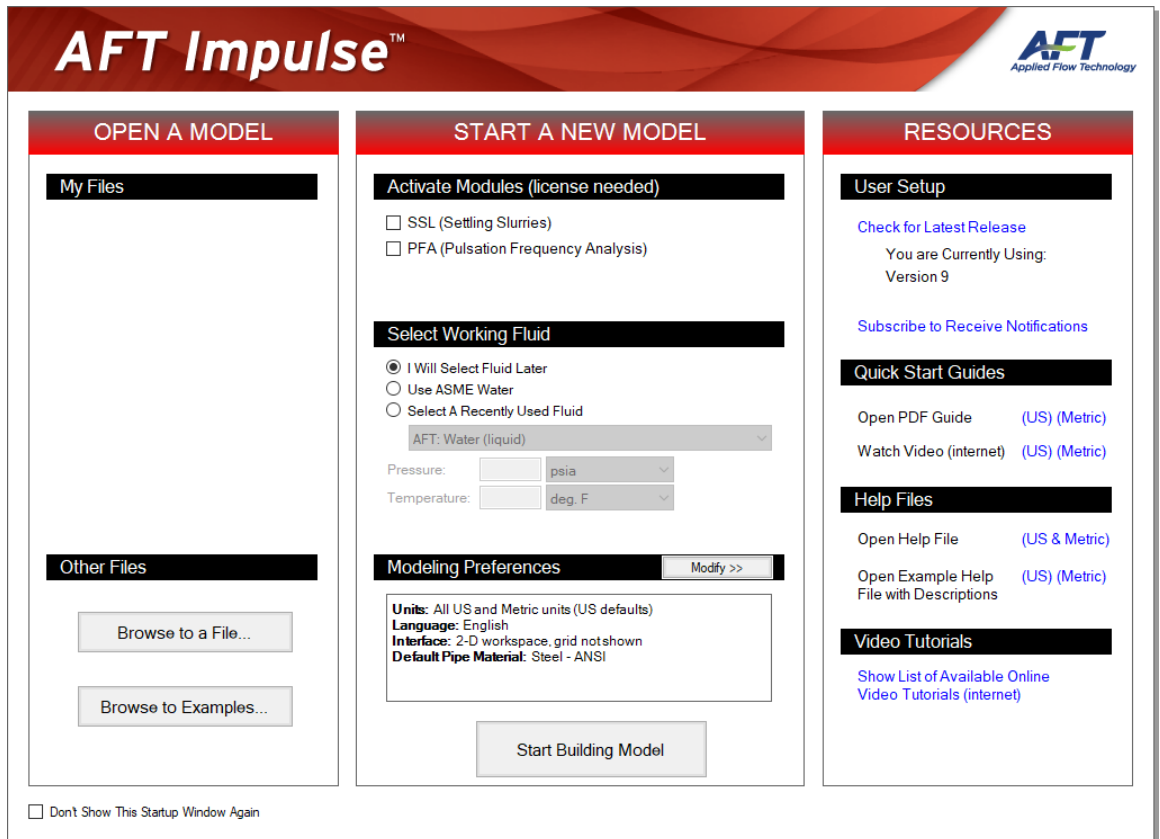
Graph the pressure at the inlet of the valve over the entire simulation duration to determine the maximum pressure that occurs at the valve during the transient.

### Step 1. Start AFT Impulse

- To start AFT Impulse, click Start on the Windows taskbar and choose AFT Impulse. (This refers to the standard menu items created by setup. You may have chosen to specify a different menu item).

#### A. The Startup window

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



**Figure 2.1 AFT Impulse Startup window**

Some of the available actions from the Startup window are:

- Open a recent model or browse to a model or an Example
- Activate an Add-on Module
- Select “ASME Water” or a recently used fluid as the Working Fluid
- Review or modify Modeling Preferences
- Select a Unit System
- Filter units to be Common Only or Common Plus Selected Industries
- Choose a Grid Style

## 10 AFT Impulse™ 9 Quick Start Guide

- Select a Default Pipe Material
- Access other Resources such as Help Files and Video Tutorials

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

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

The screenshot shows the AFT Impulse 9 Startup Window. The window has a red header with the AFT Impulse logo and the Applied Flow Technology logo. The main content area is divided into three columns: START A NEW MODEL, MODELING PREFERENCES, and RESOURCES.

**START A NEW MODEL**

- Activate Modules (license needed)**
  - ☐ SSL (Settling Slurries)
  - ☐ PFA (Pulsation Frequency Analysis)
- Select Working Fluid**
  - ☒ I Will Select Fluid Later
  - ☐ Use ASME Water
  - ☐ Select A Recently Used Fluid
- Fluid selection dropdown: AFT: Water (liquid)
- Pressure:  psia
- Temperature:  deg. F
- Modeling Preferences**
  - Units: All US and Metric units (US defaults)
  - Language: English
  - Interface: 2-D workspace, grid not shown
  - Default Pipe Material: Steel - ANSI
- Start Building Model** button

☐ Don't Show This Startup Window Again

**MODELING PREFERENCES**

- Units**
  - Unit System
    - ☐ US Only
    - ☐ Metric Only
    - ☒ Both with US Defaults
    - ☐ Both with Metric Defaults
  - Filter
    - ☒ All Units
    - ☐ Common Only
    - ☐ Common Plus Selected Industries
      - ☐ Petroleum
      - ☐ Civil
      - ☐ Slurry
      - ☐ Pipeline
      - ☐ Ducting
  - Buttons: ^^ Show Sample Units ^^, Load Units from CSV
- Language** (dropdown arrow)
- 2D or Isometric Workspace** (dropdown arrow)
- Pipe Material** (dropdown arrow)
- Note: For complete preferences, select User Options from the Tools menu
- Buttons: Remember My Preferences and Hide, Discard and Hide

**RESOURCES**

- User Setup**
  - [Check for Latest Release](#)
  - You are Currently Using: Version 9
  - [Subscribe to Receive Notifications](#)
- Quick Start Guides**
  - [Open PDF Guide](#) (US) (Metric)
  - [Watch Video \(internet\)](#) (US) (Metric)
- Help Files**
  - [Open Help File](#) (US & Metric)
  - [Open Example Help File with Descriptions](#) (US) (Metric)
- Video Tutorials**
  - [Show List of Available Online Video Tutorials \(internet\)](#)

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. Select “Common Only” under Filter to show only commonly used units, instead of “All Units”.

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

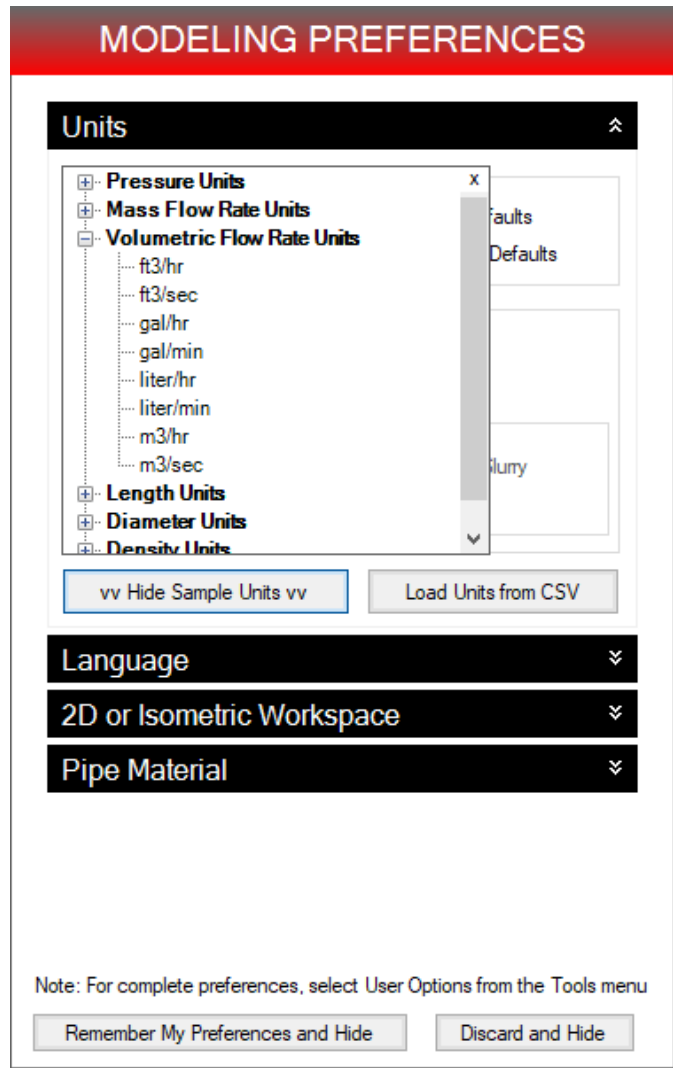


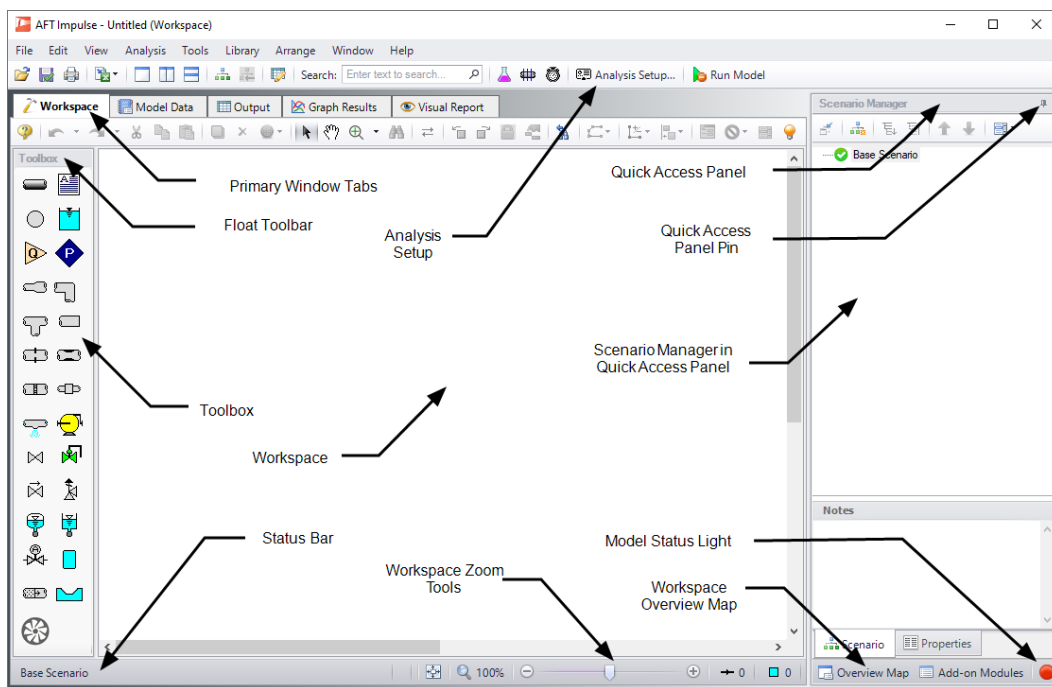
Figure 2.3 Show Sample Units with “Common Only” Selected



## 12 AFT Impulse™ 9 Quick Start Guide

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

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



**Figure 2.4** Build the model in the Workspace window. The other four Primary Windows are on the tabs along the top of the Workspace.

### B. The Workspace window

The Workspace window is the primary vehicle for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox is the bundle of tools on the far left. The Quick Access Panel is displayed on the far right. It gives easy access to a variety of features such as the Scenario Manager,

and viewing pipe and junction properties. The Workspace takes up the rest of the window.

You will build your waterhammer model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Float Toolbar. From here, you can choose the location of the Toolbox in reference to the Workspace by specifying Dock Left, Float, or Dock Right. Below the Float Toolbar are two drawing tools. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. Next to this tool is the Annotation tool. The Annotation tool allows you to create annotations and auxiliary graphics.

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

When you pass your mouse pointer over any of the Toolbox tools, a tooltip identifies the tool's function.

### C. Unpinning the Quick Access Panel

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

## Step 2. Lay out the model

To lay out the valve closure model, you will place five junctions on the Workspace. Then you will connect the junctions with pipes.

### A. Place a Reservoir

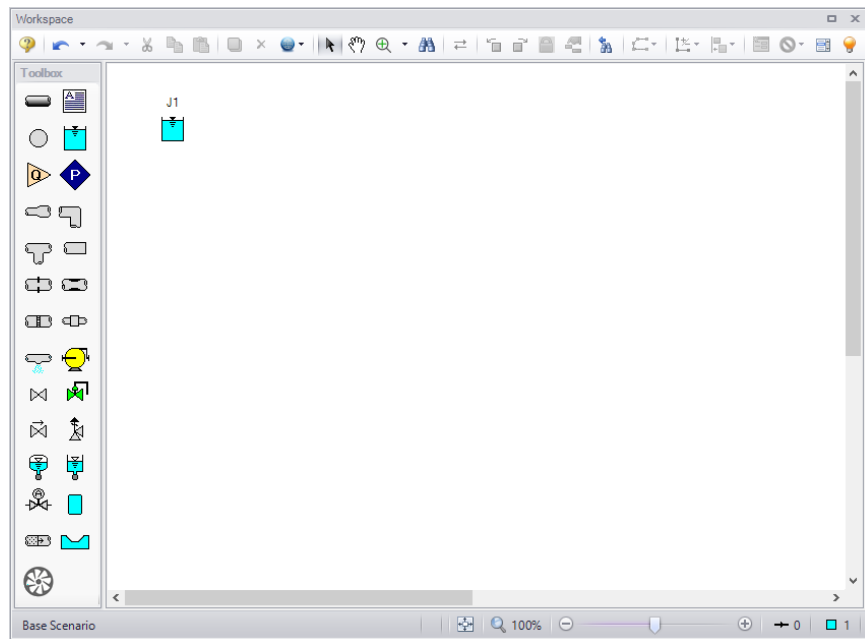
- **To start, drag a Reservoir junction from the Toolbox** and drop it on the Workspace. Figure 2.5 shows the Workspace with one Reservoir.

### ***Objects and ID numbers***

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

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

The Reservoir you have created on the Workspace will take on the default ID number of 1. You can change this to any desired number greater than zero but less than 100,000.



**Figure 2.5** Valve closure model with first junction placed

### ***Editing on the Workspace***

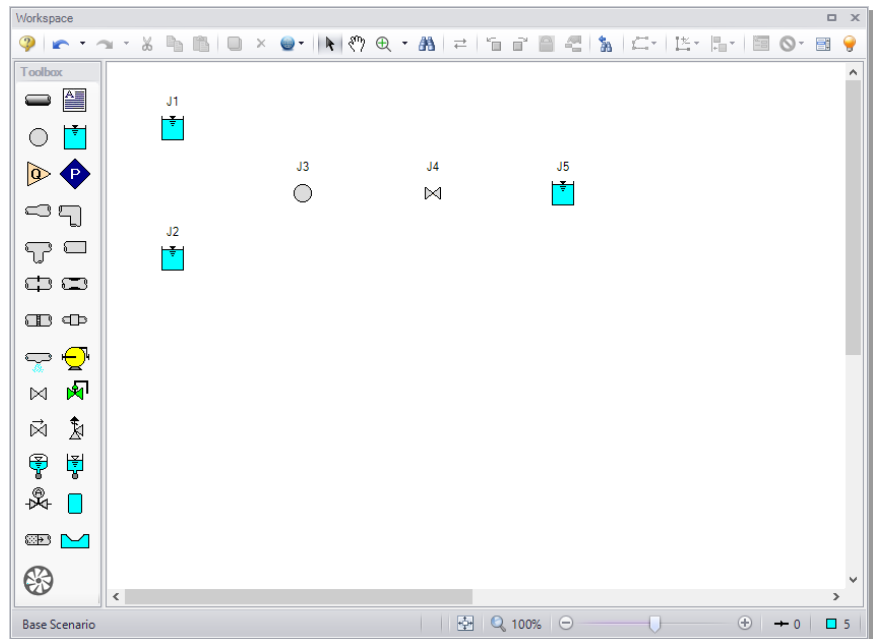
Once on the Workspace, junction objects can be moved, copied, pasted, and edited using the features on the Edit menu.

## B. Place the other junctions

If a new junction type you want to add is already on the Workspace, you have the option of duplicating that junction. You do this by choosing Duplicate from the Edit menu. Either duplicate the first Reservoir or drag a new Reservoir junction onto the Workspace. Place the new Reservoir somewhere below the first Reservoir junction (Figure 2.6).

**Note:** The relative location of objects is not important. Distances and heights are defined through dialog boxes. The relative locations on the Workspace establish the connectivity but have no bearing on the length or elevation relationships. The Isometric grid (see Chapter 5) can be used to visually represent the three-dimensional nature of a system.

Drag a Branch, Valve, and third Reservoir junction and drop them on the Workspace so that your model appears similar to Figure 2.6.



**Figure 2.6** Valve closure model with all junctions placed

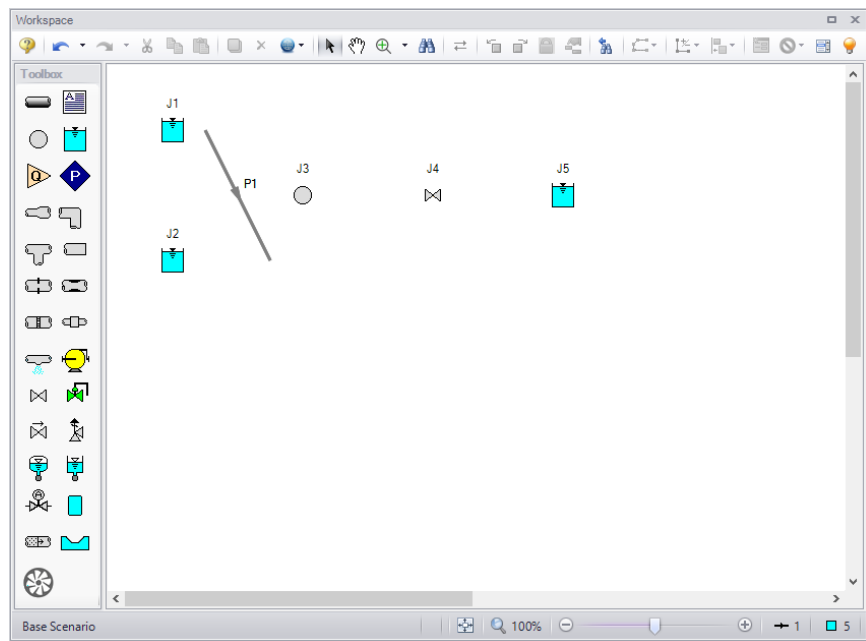
The calculations will not be affected if the icons do not line up exactly. However, your model may have a nicer appearance if they do. You can align the icons by using the Align features from the Arrange menu.

Before continuing, save the work you have done so far. Choose “Save As” from the File menu and enter a file name (Valve Closure, perhaps) and AFT Impulse will append the “.imp” extension to the file name.

### C. Draw a pipe between J1 and J3

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

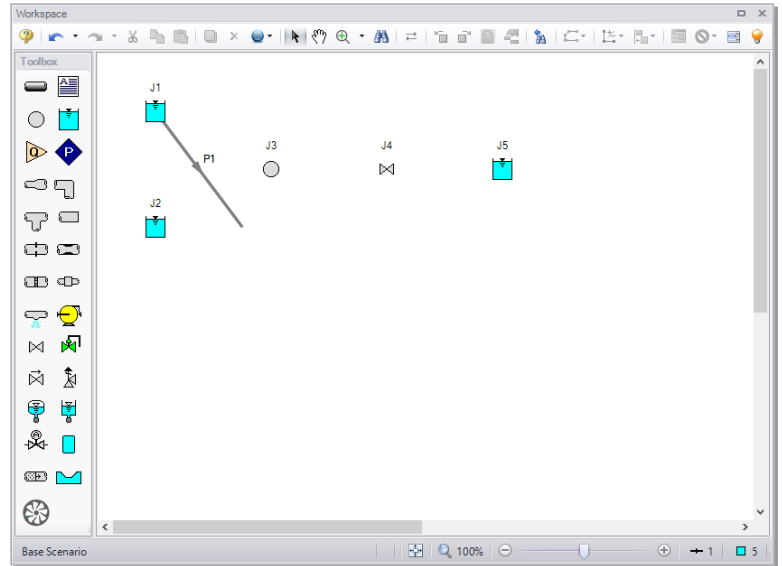
- **To create a pipe**, click the Pipe Drawing Tool icon on the Toolbox. The pointer will change to a crosshair when you move it over the Workspace. Left-click and drag to draw a pipe below the junctions, similar to that shown in Figure 2.7.



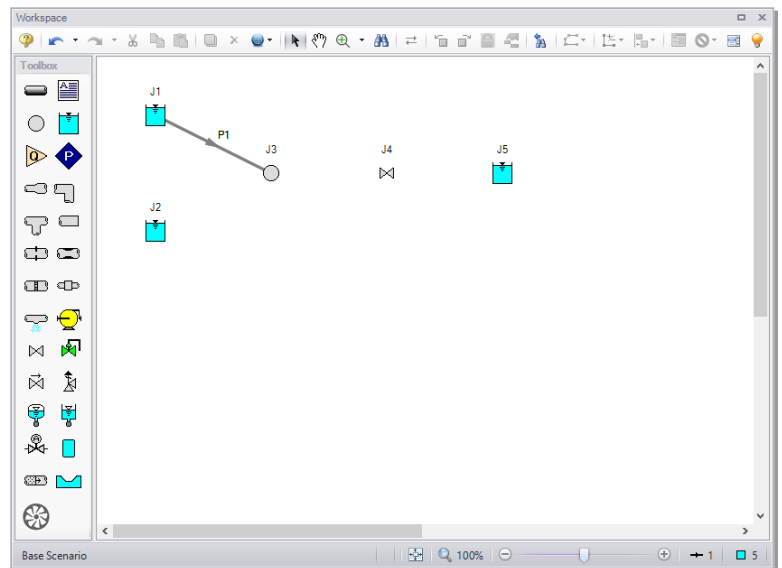
**Figure 2.7** Valve closure model with first pipe drawn

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

- **To place the pipe between J1 and J3**, use the mouse to grab the pipe in the center, drag it so that its left endpoint falls within the J1 Reservoir icon, then drop it there (see Figure 2.8). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J3 Branch icon (see Figure 2.9).



**Figure 2.8** Valve closure model with first pipe partially connected

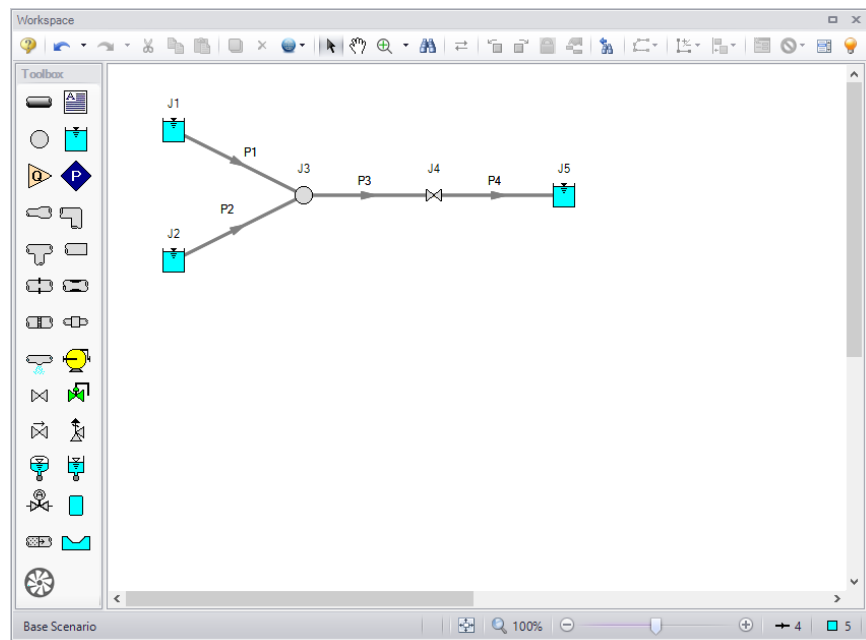


**Figure 2.9** Valve closure model with first pipe fully connected

## D. Add the remaining pipes

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

- **Activate the Pipe Drawing Tool again**, position the mouse pointer on the J2 Reservoir, then press and hold the left mouse button. Stretch the pipe across to the J3 Branch, and then release the mouse button. Continue drawing pipes P3 and P4 as indicated in Figure 2.10. After all the pipes are drawn, all the objects in the model are graphically connected. Save the model by selecting Save in the File menu or by clicking on the “save” icon on the Main Toolbar.



**Figure 2.10** Valve closure model with all pipes and junctions placed

### ***Reference positive flow direction***

There is an arrow on each pipe that indicates the reference positive flow direction for the pipe. AFT Impulse assigns a flow direction corresponding to the direction in which the pipe is drawn.



You can reverse the reference positive flow direction by choosing “Reverse Direction” from the Arrange menu or selecting the Reverse Pipe Direction button on the Workspace Toolbar.

In general, the reference positive flow direction is used for reference purposes only and need not be the actual flow direction. However, when used with pumps and certain other junction types the pipes must be in the appropriate forward flow direction because that is how AFT Impulse determines which side is suction and which is discharge. If the reference positive flow direction is the opposite of that obtained by the Solver, the output will show the flow rate as a negative number.

---

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

---

### Step 3. Specify Analysis Setup

Analysis Setup is a new feature to AFT Fathom 12 that replaces the Checklist. Select modules, define fluid properties, specify pipe and junction properties, section pipes, specify the simulation duration, adjust solution control, and more all from the Analysis Setup window.

- **Next, click Analysis Setup on the Common Toolbar** at the top of the AFT Impulse window. This opens the Analysis Setup window (Figure 2.11). The Analysis Setup window contains six or more groups depending on which 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.

---

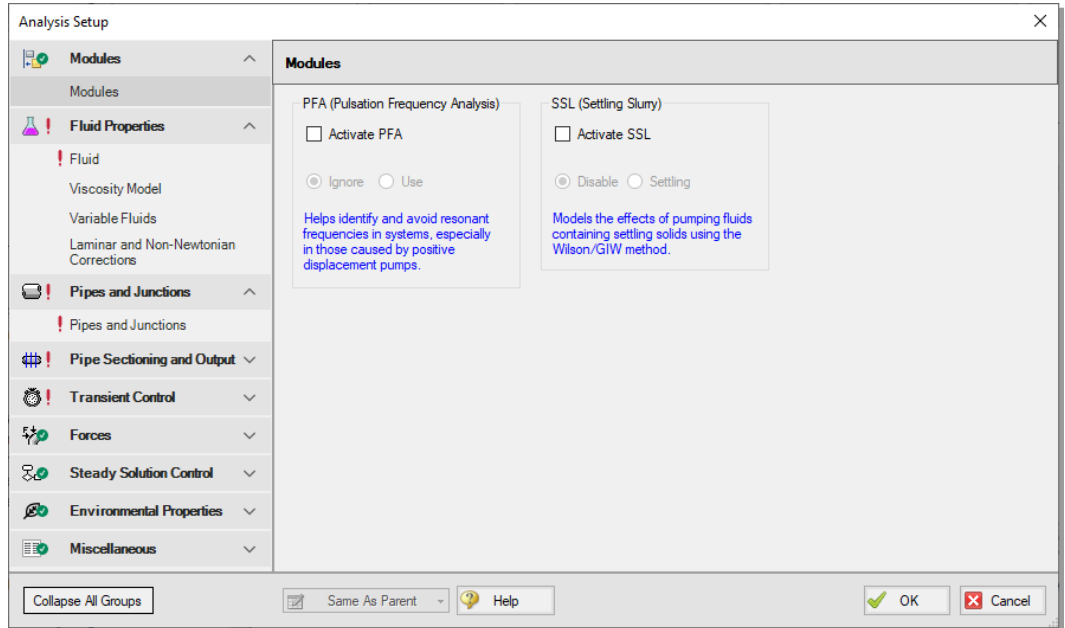
**Note:** If you are running your model in steady state mode only, the Pipe Sectioning and Output group and the Transient Control group will appear to be fully defined because they pertain to the transient analysis only. To change back and forth between the steady state and transient analyses, choose either “Steady Only” or “Transient” from the Simulation Mode/Duration panel in Analysis Setup or from the Analysis menu.

---

Analysis Setup can also be opened by clicking the Model Status light on the Status Bar at the bottom right corner of the AFT Impulse window (Figure 2.4).

### **A. Specify the Modules group**

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 2.11. Here, you can enable and disable add-on modules. These add-on modules include Pulsation Frequency Analysis and Settling Slurry. By default, the model will have all modules disabled. Therefore, make no changes to this panel for this example.



**Figure 2.11 Analysis Setup tracks the defined and undefined model input**

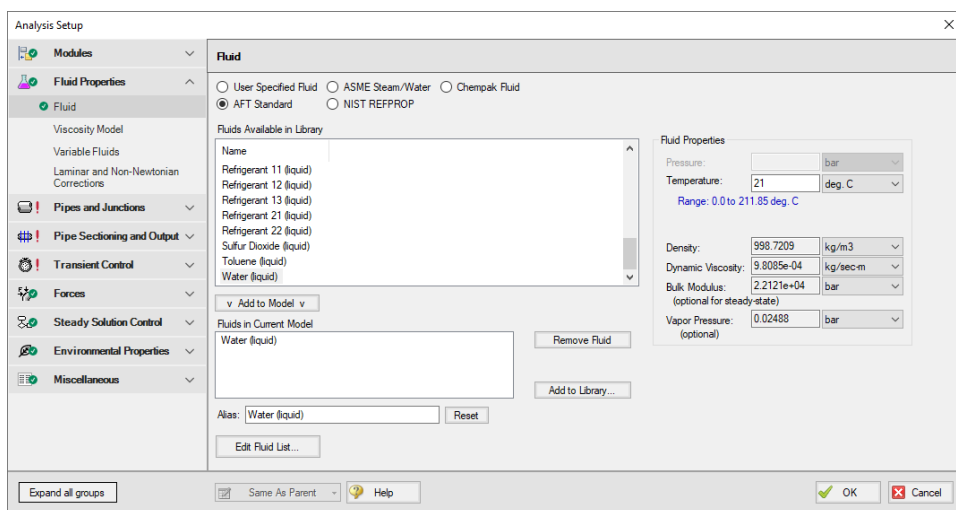
## B. Specify the Fluid Properties group

The Fluid Properties group consists of four items: Fluid, Viscosity Model, 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 select which viscosity model to use - Newtonian is used by default. On the Variable Fluids panel, you can specify constant fluids properties, the default selection, or variable fluid properties. 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 . 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 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 AFT Standard library.

- **Select the AFT Standard library and select Water (liquid) in the Fluids Available in Library list, then click the “Add to Model” button.** Enter a temperature of 21 deg. C. This satisfies the required inputs for the Fluid panel and Fluid Properties group. This Fluid panel should appear as shown in Figure 2.12.



**Figure 2.12** The Fluid panel is where you specify the fluid

### C. Specify the Pipes and Junctions group

The third group in Analysis Setup, Pipes and Junctions, is not as straightforward to satisfy as the first two. This item encompasses the proper input data and connectivity for all pipes and junctions.

#### **Object status**

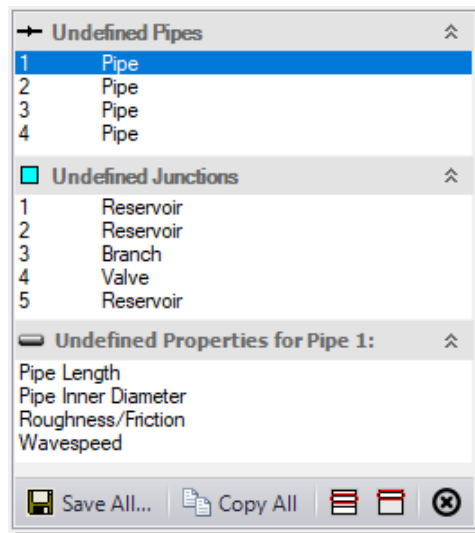
Each pipe and junction have an object status. The object status tells you whether the object is defined according to AFT Impulse’s requirements. To see the status of the objects in your model, click the light bulb icon on the Workspace Toolbar (alternatively, you could choose “Show Object Status” from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

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

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

### ***Showing undefined objects***

Another useful feature is the List Undefined Objects window (Figure 2.13). This can be opened from the View menu by clicking on “List Undefined Objects”, or from the Workspace toolbar. Here all objects with incomplete information are listed. Clicking on an undefined pipe or junction will display the property data that is missing. Click the close button in the bottom right to stop showing this window.



**Figure 2.13** The List Undefined Objects window lets you see the undefined properties for each undefined object

### ***Enter data for the reservoirs***

As shown in Figure 2.10, the J1 junction is a Reservoir junction.

- **To define the first reservoir**, open the J1 Reservoir Properties window (Figure 2.14) by double-clicking on the J1 junction.

---

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

---

- **Enter the following properties into the J1 Reservoir Properties window:**
  1. Make sure the Tank Model is set to “Infinite Reservoir”
  2. Enter 15 meters for the Liquid Surface Elevation. You can assign any unit found in the adjacent drop-down menu of units.
  3. Enter 0 barG for the Liquid Surface Pressure
  4. Enter 3 meters for the Pipe Depth (ensure that the radio button next to Pipe Depth is selected)

---

**Note:** Change default units for many parameters (such as feet for length) in User Options, found in the Tools menu under Preferred Units.

---

You can give the object 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. In Figure 2.14, the name of this Reservoir has been changed to Supply Tank A. The name can be displayed on the Workspace, Visual Report or in the Output.

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

**Reservoir Properties**

Number:

Name:

Library Jct:

Copy Data From Jct...:

OK Cancel Jump... Help

**Reservoir Model** Pipe Depth & Loss Coefficients Transient Optional Design Alerts Notes Status

**Tank Model**

☒ Infinite Reservoir

☐ Finite Open Tank

**Known Parameters Initially**

☒ Liquid Surface Level

☒ Surface Pressure

Liquid Surface Pressure:

Liquid Surface Elevation:

☐ Pipe Elevation ☒ Pipe Depth

☒ Same Depth for All Pipes

Reference

**Figure 2.14 Properties window for Reservoir J1**

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

### ***Using the tabs in the properties windows***

The information in the properties windows is grouped into categories and placed on separate tabs. Click the tab to bring its information forward. Figure 2.14 is an example of a Reservoir Properties window.

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



loss factors to be specified as zero. To change the loss factors later, click within the pipe table and enter the loss.

The Optional tab allows you to enter different types of optional data. You can select whether the junction number, name, or both are displayed on the Workspace. Some junction types also allow you to provide a guess for initial pressure or other junction-specific data.

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

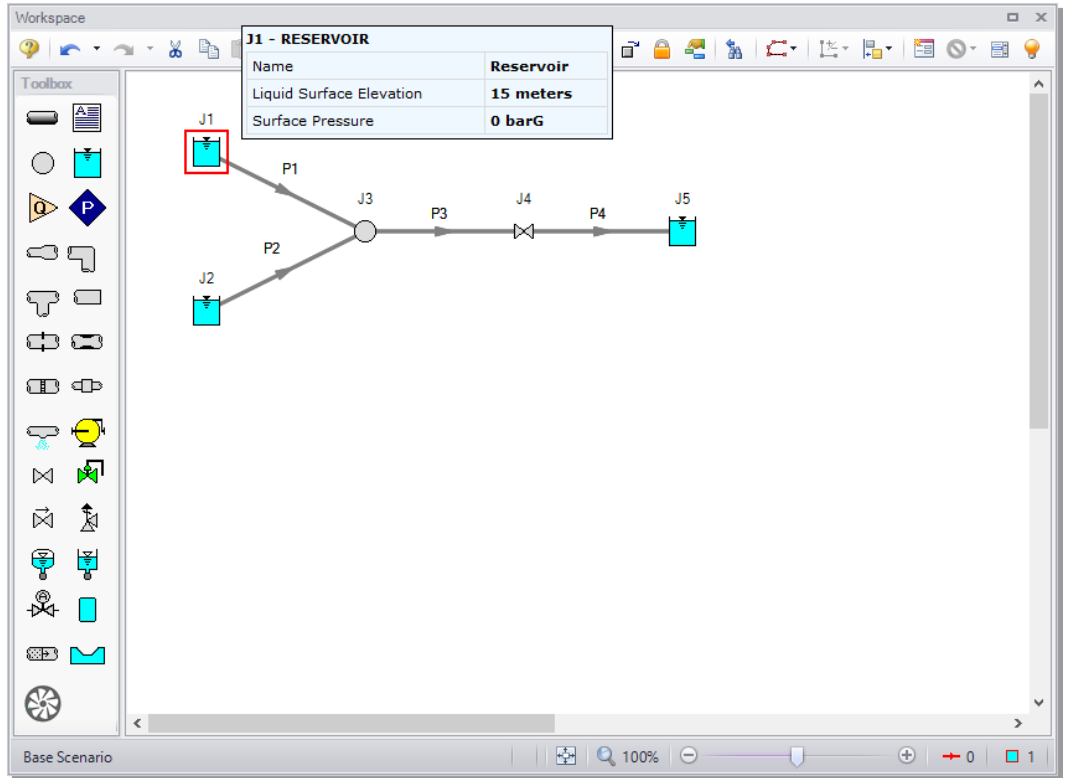
The highlight feature displays all the required information in the Properties window in light blue. The highlight feature is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window above the tabs or by pressing the F2 key. The highlight feature can also be turned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

- **Click OK.** If “Show Object Status” is turned on, you should see the J1 ID number turn black again, telling you that J1 is now defined.
- **Next, repeat this process for junctions J2 (Supply Tank B) and J5 (Discharge Tank) with the following data:**
  - For J2, use a Liquid Surface Elevation of 14 meters, a Pipe Depth of 3 meters, and a Liquid Surface Pressure of 0 barG
  - For J5, use a Liquid Surface Elevation of 1.5 meters, a Pipe Depth of 1.5 meters, and a Liquid Surface Pressure of 0 barG

### ***The Inspection feature***

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspect feature. Position the mouse pointer on the J1 Reservoir junction and hold down the right mouse button. An inspection window appears, as shown in Figure 2.15.

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



**Figure 2.15** Inspecting junction properties from the Workspace by clicking and holding the right mouse button

### ***Enter data for the branch***

- **Open the J3 properties window** and enter an elevation of 0 meters.

### ***Enter data for the valve***

The J4 Valve junction will be the initiator of the transient.

- **Double-click the J4 junction icon** to open the Valve Properties window (Figure 2.16)
  1. Enter an Inlet Elevation of 0 meters
  2. Choose Cv as the Loss Model enter a value of 1000 (this is the steady-state Cv)

Valve Properties

Number: 4

Name: Valve

Library Jct:

Copy Data From Jct...

Upstream Pipe: 3

Downstream Pipe: 4

Elevation

Inlet: 0 meters

Outlet: Same as Inlet

OK

Cancel

Jump...

Help

Loss Model

Transient

Optional

Design Alerts

Notes

Status

Valve Data Source

Handbook Data

User Specified

Loss Model

Cv

Kv

K Factor

Loss Source

Fixed Cv

From Open Percent Table (on Optional Tab)

% Open:

Cv: 1000

Basis Area for Loss Model

Upstream Pipe

Base Area from Pipe 3: Unspecified

Exit Valve (optional)

Head (HGL)

Pressure

Exit Head (HGL):

Figure 2.16 Input data for Valve junction J4

3. Click the Transient tab and select Time under “Initiation of Transient” to indicate that the valve closure will be initiated at the time you specify, then enter the following Cv vs. Time data for the valve’s Cv at various times during the simulation (Figure 2.17).

Time (seconds)	Cv
0	1000
0.4	400
0.8	100
1	0
2	0

The first data point ( $C_v = 1000$  at time zero) must match the steady-state value (entered on the Loss Model tab). The transient data represents the valve as initially open. The valve then closes over a period of one second (a  $C_v$  of zero means the valve is closed). The valve then stays closed for the rest of the simulation.

4. Click OK. There should now be a “T” symbol next to the Valve junction in the Workspace, indicating that transient data is entered for the junction.

Data Point	Time (seconds)	$C_v$
1	0	1000
2	0.4	400
3	0.8	100
4	1	0
5	2	0
6		
7		
8		
9		
10		
11		
12		
13		

**Figure 2.17 Transient input data for Valve Junction J4**

### ***Enter data for Pipe P1***

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

- **First, open the Pipe Properties window for Pipe P1.** The Pipe Model tab opens by default (Figure 2.18).

1. In the Pipe Material field, choose "Steel - ANSI"
2. In the Size and Type fields, choose "10 inch" and "STD (schedule 40)", respectively
3. Specify the length as 60 meters
4. For the Friction Model field, make sure "Standard" is selected
5. Make sure the Pipe Support is "Thick-Walled Anchored Upstream"

The screenshot shows the 'Pipe Properties' dialog box for 'Pipe P1'. The 'Pipe Model' tab is active. The 'Size' section includes fields for Pipe Material (Steel - ANSI), Size (10 inch), Type (STD (schedule 40)), Inner Diameter (25.4508 cm), Wall Thickness (0.9270999 cm), Modulus of Elasticity (2032876 bar), and Poisson Ratio (0.3). There is an option for ID Reduction (Scaling) which is currently unchecked. The 'Length' section shows a value of 60 meters and an unchecked 'Model As Zero-Length Connector' checkbox. The 'Wavespeed' section has radio buttons for 'User Specified Wavespeed' (unchecked) and 'Calculated Wavespeed' (checked), with a 'Pipe Support' dropdown set to 'Thick-Walled Anchored Upstream'. Below this, 'c1' is 0.914836 and 'Wavespeed' is 1318.918 meters/sec. The 'Friction Model' section has a 'Data Set' dropdown with 'Standard' selected. The 'Absolute Roughness' is 0.004572 cm, and there is a 'Load Default' button. On the right side of the dialog are buttons for OK, Cancel, Jump..., and Help.

**Figure 2.18 Pipe Properties window for Pipe P1**

### ***The Pipe Properties window***

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

The Inspect feature can be accessed not only from pipes and junctions located on the Workspace, but also from within the Properties window of each pipe (and certain junctions). This is helpful when you want to

quickly check the properties of objects that connect to a pipe or junction whose Properties window you already have open.

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

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

### ***Wavespeed***

The wavespeed is a very important parameter in waterhammer analysis. The wavespeed can be calculated with reasonable accuracy from fluid and pipe data, or it may be available from test data or industry publications. If the wavespeed is not known (which is typical), then the Calculated Wavespeed option is the preferred option. In this case, data is required for pipe wall thickness, modulus of elasticity, Poisson Ratio, and pipe support details. Data for pipe wall thickness, modulus of elasticity, and Poisson Ratio are built into the pipe material libraries supplied with AFT Impulse, and was automatically obtained when the Steel - ANSI, 10-inch, STD (schedule 40) option was chosen. The calculated wavespeed is 1319 meters/sec (see Figure 2.18). Click OK.

### ***Enter data for other pipes***

Similar to Pipe P1, enter the following data for the other pipes. Note that all pipes will use the Thick-Walled Anchored Upstream Pipe Support and the Standard Friction Model.

Pipe	Size (inches)	Length (meters)
1	10	60
2	10	45

Pipe	Size (inches)	Length (meters)
3	12	15
4	12	12

### ***Check pipes and junctions***

- **Check if all the pipes and junctions are defined.** If all the pipes and junctions are defined the Pipe and Junctions group in the Analysis Setup window will have a green checkmark. If not, turn on the “Show Object Status” from the View menu or Workspace Toolbar icon, and open each undefined pipe and junction. The Status tab on each Properties window will indicate what information is missing.

### ***Review model data***

- **Save the model one more time.** It is also a good idea to review the input using the Model Data window.

### ***Reviewing input in the Model Data window***

The Model Data window is shown in Figure 2.19. To change to this window, you can click on the Model Data tab, select it from the Window menu, or press CTRL+M. The Model Data window gives you a text-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, exported into Excel, saved to a formatted file, printed to an Adobe™ PDF, or printed out for review.

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

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

**Model Data**

General Notes

Title: AFT Impulse Model  
Input File: Untitled

Number Of Pipes= 4  
Number Of Junctions= 5

Output File:

Pressure/Head Tolerance= 1E-05 relative change  
Flow Rate Tolerance= 1E-05 relative change

Pipes Pipe Fittings & Losses Pipe Detail Summary

Pipe	Name	Pipe Defined	Zero-Length Connector	Length	Length Units	Hydraulic Diameter	Hydraulic Diam. Units	Friction Data Set	Roughness	Roughness Units	Losses (K)	Initial Flow	Initial Flow Units	Junctions (Up,Down)
1	Pipe	Yes	No	60 meters		25.4508 cm		Standard	0.004572 cm		0			1.
2	Pipe	Yes	No	45 meters		25.4508 cm		Standard	0.004572 cm		0			2.
3	Pipe	Yes	No	15 meters		30.48 cm		Standard	0.004572 cm		0			3.
4	Pipe	Yes	No	12 meters		30.48 cm		Standard	0.004572 cm		0			4.

Branch Reservoir Valve Transient Data

Reservoir	Name	Object Defined	Inlet Elevation	Elevation Units	Initial Pressure	Initial Pressure Units	Library Source	Liquid Elev.	Liquid Elev. Units	Surface Pressure	Surface Pressure Units
1	Reservoir	Yes						15 meters		0 barG	
2	Reservoir	Yes						14 meters		0 barG	
5	Reservoir	Yes						1.5 meters		0 barG	

Base Scenario 100%

**Figure 2.19** The Model Data window shows all input in text form

### ***Steady State groups are now fully defined***

The first three groups in Analysis Setup are now fully defined. If desired, the model could be run in steady-state. The final two groups that need to be defined are for transient modeling. To run the model in steady-state, go to the Simulation Mode/Duration panel in Analysis Setup and change the Time Simulation selection to Steady Only.

In general, it is a good idea to always run your model in steady-state first before running the full transient analysis to make sure the model is giving reasonable results.

## **D. Specify the Pipe Sectioning and Output group**

The fourth group in Analysis Setup is the Pipe Sectioning and Output group. The pipes in the model cannot be sectioned until sufficient data in the Fluid Properties group and Pipe and Junctions group have been specified. These required data include the fluid specification (this was



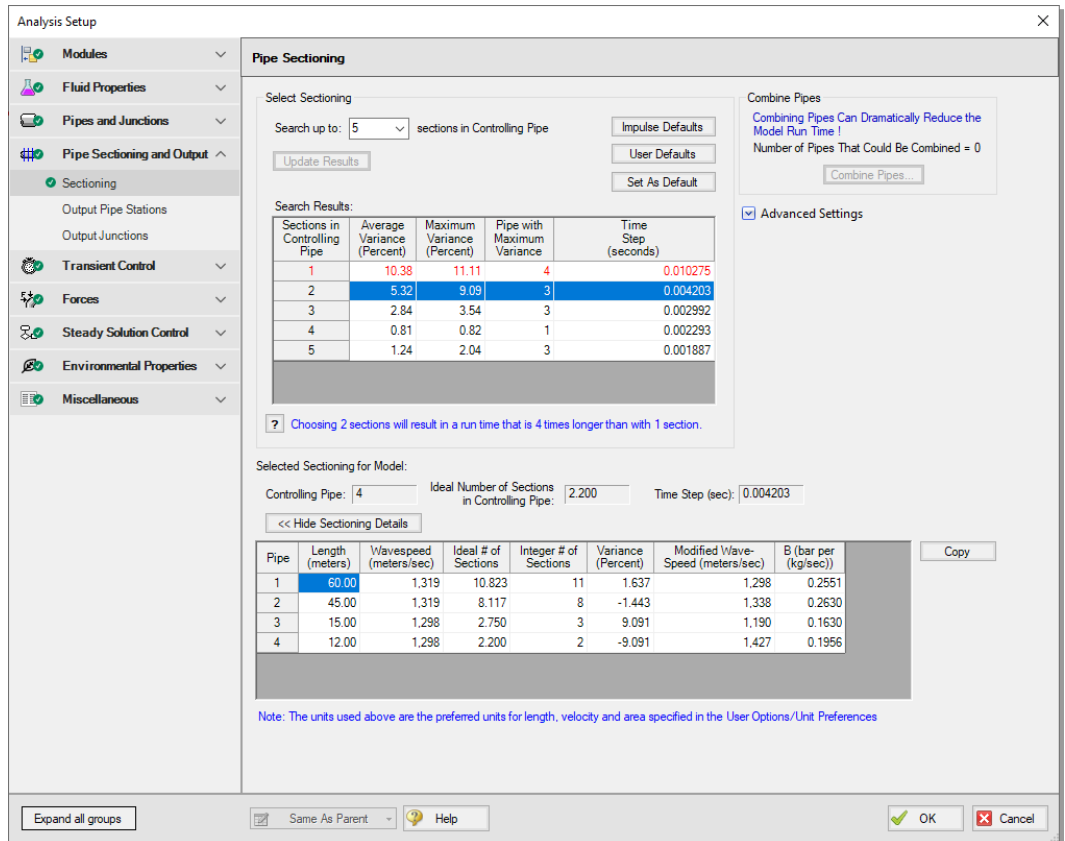
done in Step 3B when the Fluid panel was defined) and all pipes and junctions definitions. After these data have been entered, the pipes can be sectioned. The Sectioning panel divides the pipes into computation sections in a manner which is consistent with the Method of Characteristics (MOC).

Go to the Sectioning panel in Analysis Setup (Figure 2.20). The pipes will automatically be sectioned and a controlling pipe will be selected. For this model, the controlling pipe is P4. This is the pipe with the shortest end-to-end communication time (i.e.,  $L/a$  – the length divided by the wavespeed). To satisfy the MOC, the following equation must be applied:

$$n_i = \frac{L_i}{a_i \Delta t}$$

where  $n$  is the number of sections in pipe  $i$ ,  $L$  is the length, and  $a$  is the wavespeed. The  $\Delta t$  is the time step. Since all pipes in the network must be solved together, the same time step must be used for each pipe. With a given length and wavespeed for each pipe, it can be seen from the above equation that it is unlikely that the number of required sections,  $n$ , for each pipe will be a whole number.

To address this situation, it is helpful to recognize that the wavespeed,  $a$ , is the least certain input parameter. It is therefore generally considered acceptable to allow up to a 15% uncertainty in wavespeed. By adjusting the wavespeed for each pipe within this tolerance, the sectioning can be made to come out as whole numbers for each pipe. The Sectioning panel automates this process by searching for sectioning which satisfies the required tolerance. While a 15% variance in wavespeed is often acceptable, AFT Impulse uses 10% by default.



**Figure 2.20** The Pipe Sectioning panel automates the sectioning process and calculates the time step

➤ **Open Analysis Setup and go to the Sectioning panel.**

When the Sectioning panel is first opened, it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. If more sections are needed, you can change the “Search up to” quantity and click “Update Results”.

In this example model, the results when the controlling pipe has only one section show a maximum variance of 11%. Since this is greater than the default 10% threshold (shown when the Advanced Settings section is expanded), the first row is colored red and was not automatically selected. The option to have two sections in the controlling pipe is

selected since that has the fewest sections with an acceptable maximum variance. The time step is shown to be 0.004203 seconds.

Note that if the number of sections in the controlling pipe is doubled, the runtime increases roughly by a factor of four (twice the number of computations per time step and twice the number of time steps for the run). In this example, selecting four sections in the controlling pipe would result in the smallest variance in wavespeed modifications out of the options displayed, but the runtime would be four times that compared to the case where two sections are selected (16 times longer than with one section in the controlling pipe). Remembering that it is acceptable to have up to a 15% uncertainty in wavespeed, and that there are other uncertainties in the model, one would typically pick the option with the fewest sections with an acceptable maximum variance.

### E. Specify the Transient Control group

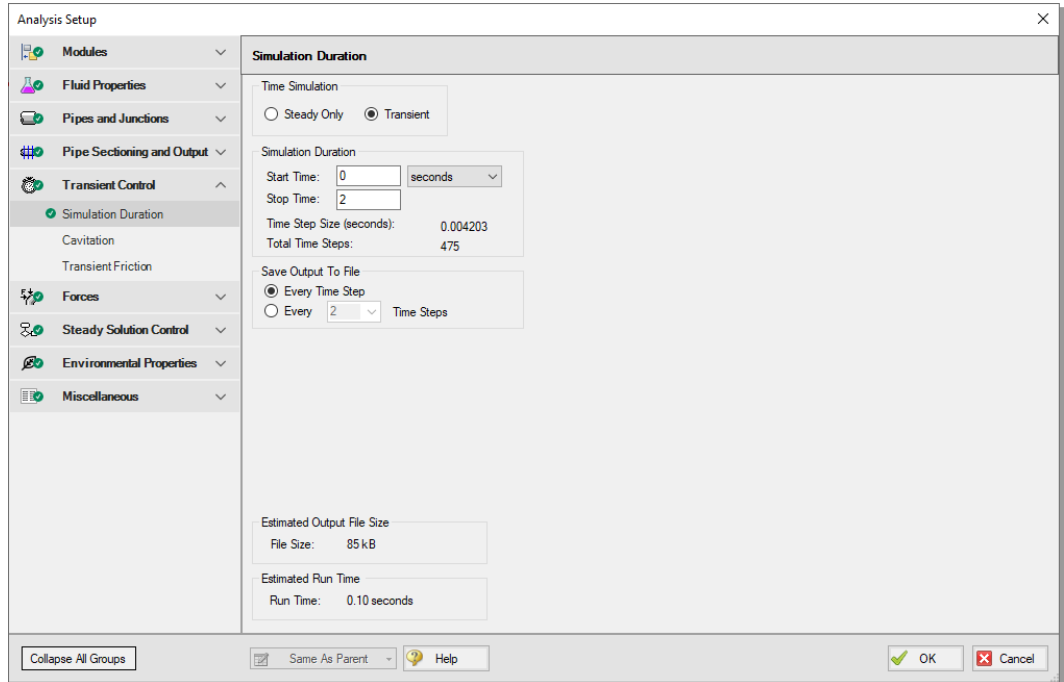
The final group to be defined in Analysis Setup before the transient solver can run is the Transient Control group. This group allows you to specify the time at which the transient starts and ends, as well as the cavitation and friction model to use in the calculations.

- **Go to the Simulation Mode/Duration panel in Analysis Setup.** Enter 2 seconds in the Stop Time field, as shown in Figure 2.21.

The Transient Cavitation panel allows you to enable or disable transient cavitation modeling and select a model to use.

At the bottom of the window, the projected output file size is shown. You should pay attention to this number, as the output file size can grow very large. In this case, the output file will be 85 kB. If the output file does become excessively large, you will want to limit the number of time steps and pipe output written to disk. Note that the estimated file size and computational time may vary between machines.

- **Click OK to accept the current settings.** All groups in Analysis Setup should now be defined. The model is ready to be solved.



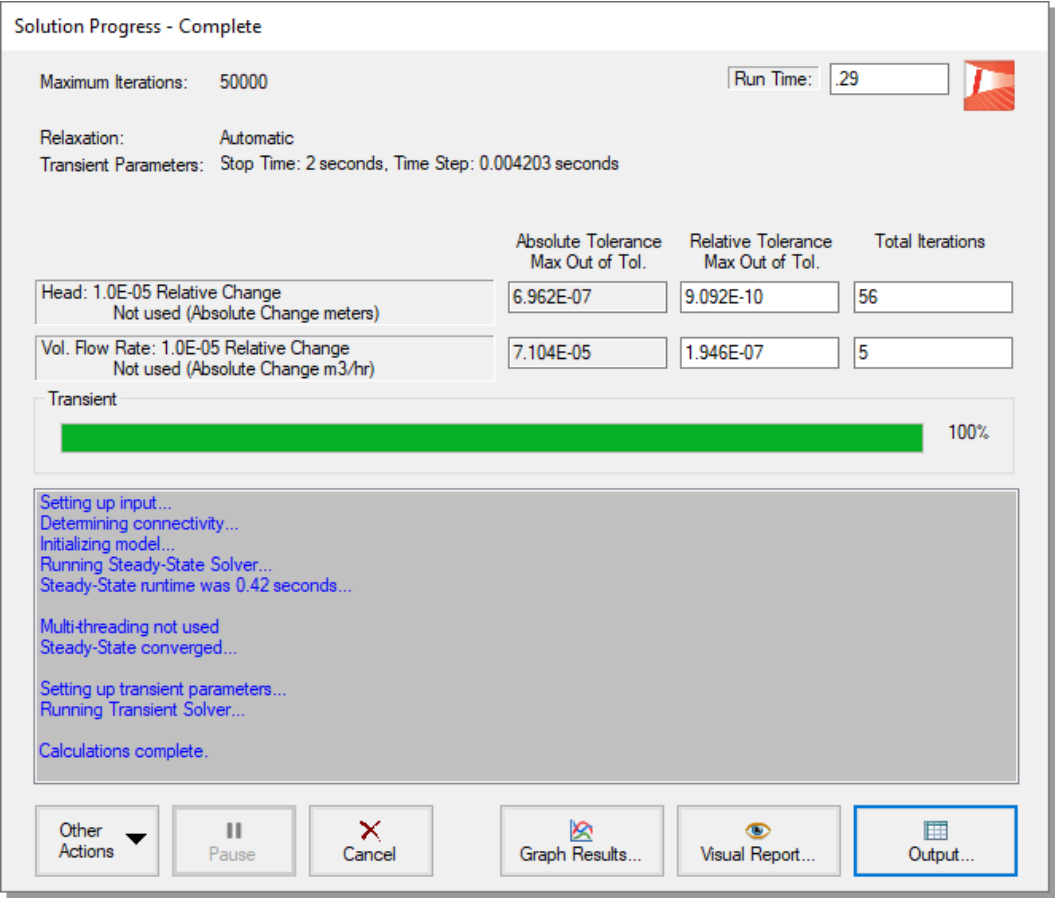
**Figure 2.21** The Simulation Mode/Duration panel offers features to specify the time span for the transient

## F. Save the model

Saving the model immediately before running it is always a good idea. Ensure that you save your model now.

## Step 4. Run the solver

- **Choose “Run Model...” from the Analysis menu** or click the Run Model button on the Toolbar. During execution, the Solution Progress window is displayed (Figure 2.22). You can use this window to pause or cancel the Solver's activity.



**Figure 2.22    The Solution Progress window displays the state of the simulation**

**A. The two solvers**

AFT Impulse has two Solvers. The first is called the Steady-State Solver, which as its name suggests, obtains a steady-state solution to the pipe network. The second Solver is called the Transient Solver. This solves the waterhammer equations.

Before a transient simulation can be initiated, the initial conditions are required. These initial conditions are the steady-state solution to the system. After the steady-state solution is obtained by the Steady-State

Solver, AFT Impulse uses the results to automatically initialize the Transient Solver and then run it.

When the solution is obtained, click the Output button on the Solution Progress window to display the text-based Output window. The information in the Output window can be reviewed visually on the screen, saved to file, exported to a spreadsheet-ready format, copied to the clipboard, and printed out.

### **B. The transient output file**

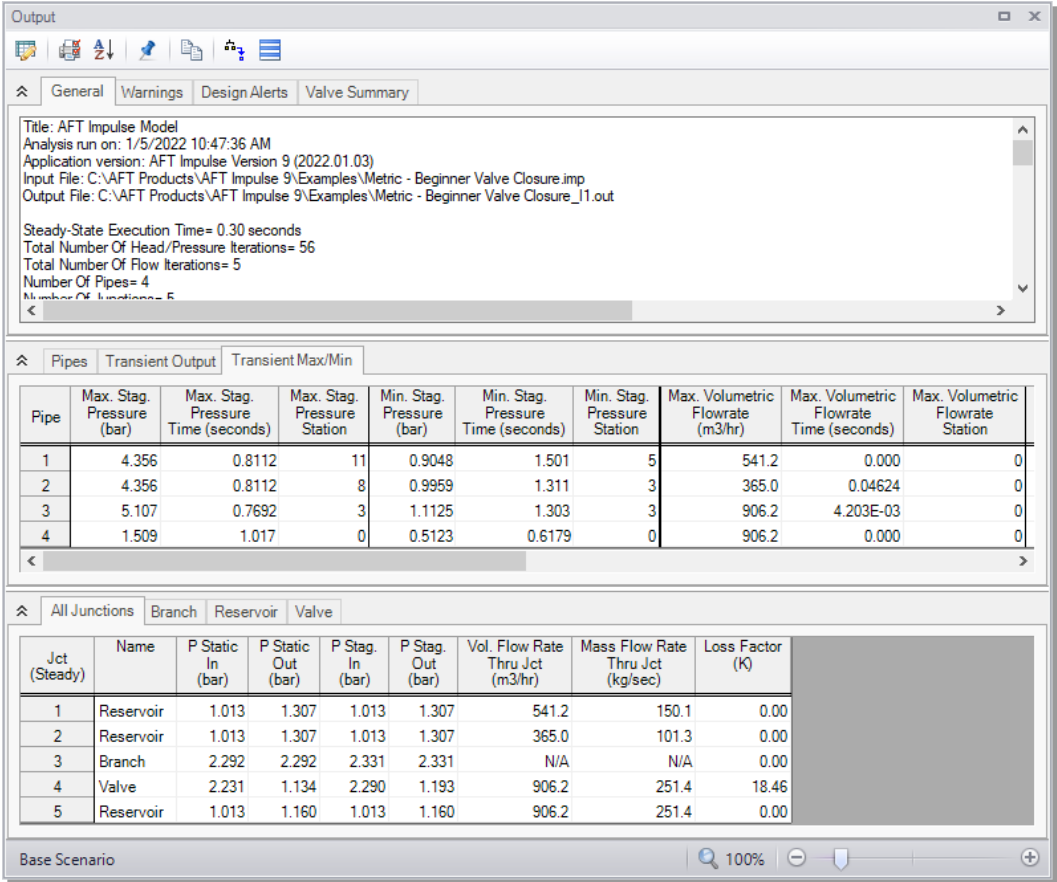
When the Transient Solver runs, the transient output data is written to a file. This file is given the same name as the model itself with a number appended to the name, and with an “.out” extension appended to the end. For all transient data processing, graphing, etc., the data is extracted from this file. The number is appended because AFT Impulse allows the user 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 remain on disk until the user erases it or the input model is modified. 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 5. Review the Output window**

The Output window (Figure 2.23) is similar in structure to the Model Data window. Three areas are shown (including the General section, Pipe Section, and Junction section), and 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 items displayed in the tables are those items you select in the Output Control window.

The Output window allows you to review both the steady-state and transient results. The Pipes tab, All Junctions tab, and any specific junction tabs in the Junction section (such as Branch, Reservoir, Valve, etc.) show the steady-state results. A summary of the maximum and minimum transient results for each pipe is given on the Transient Max/Min tab in the Pipe section and is the active tab displayed in the center section of Figure 2.23.



**Figure 2.23 The Output window displays steady and transient output in text form**

You can also review the solutions for each time step (i.e., a time history) for which data was written to file on the Transient Output tab in the Pipe section (Figure 2.24). Moving the slider along the bottom will change the time step shown in the table.

The Output Control window (Figure 2.25) allows you to select the specific output parameters you want to display in your output. You also can choose the units for the output. If you do not change any of the Output Control settings, default Output Control parameters and a default title are assigned.

Step: 0	Time: 0.00000E+00	Pipe -- Station	Stag. Pressure (bar)	Volumetric Flowrate (m3/hr)	Vapor Volume (meters3)
+	1 -- 0		1.307	541.2	0
+	2 -- 0		1.307	365.0	0
+	3 -- 0		2.331	906.2	0
+	4 -- 0		1.193	906.2	0

Figure 2.24 The Output window displays transient data for each time step in the Transient Output table

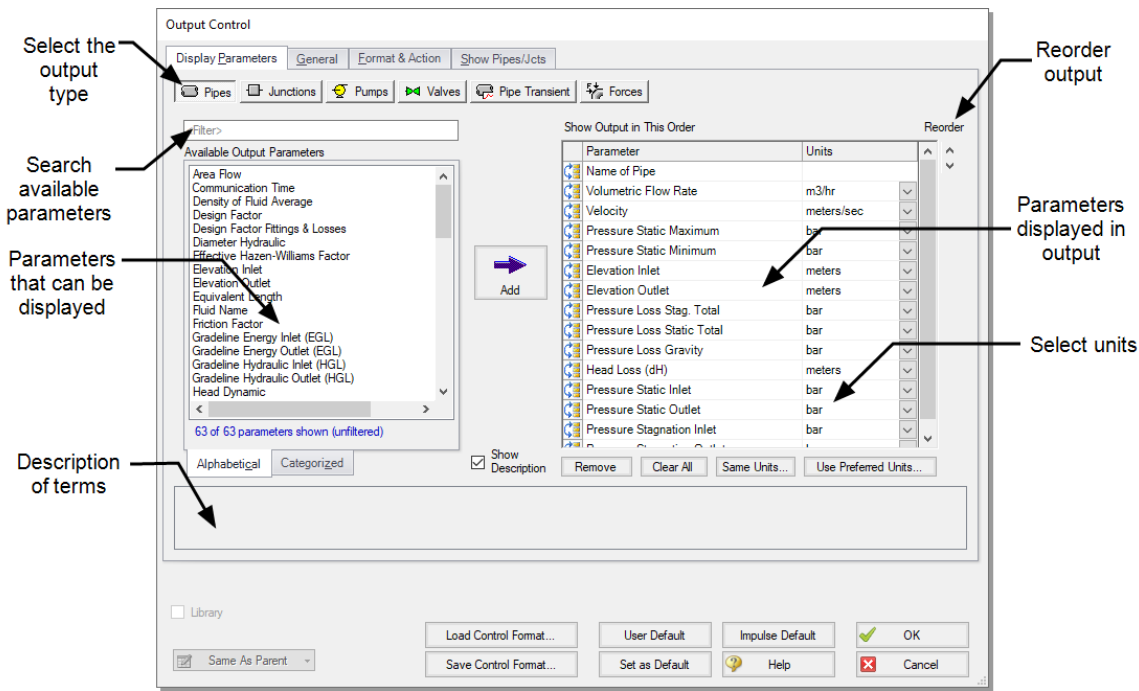
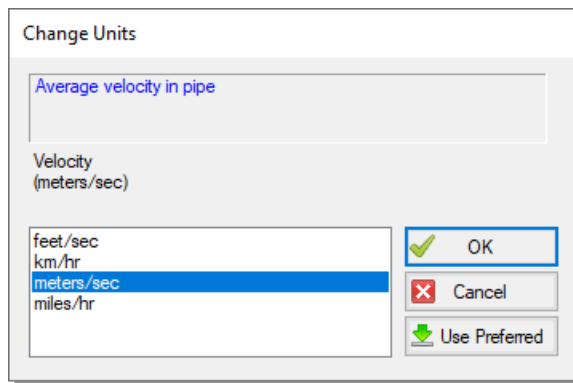


Figure 2.25 Output Control lets you customize the output



Units for each column in the Output section can also be changed by double-clicking the column header. This will open a window in which you can select the units again if you prefer (Figure 2.26). These changes are extended to the Output Control parameter data that is set.



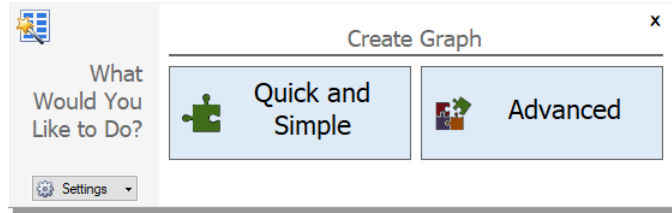
**Figure 2.26** The Change Units window is opened from the Output window tables by double-clicking the column header

## Step 6. View the graph results

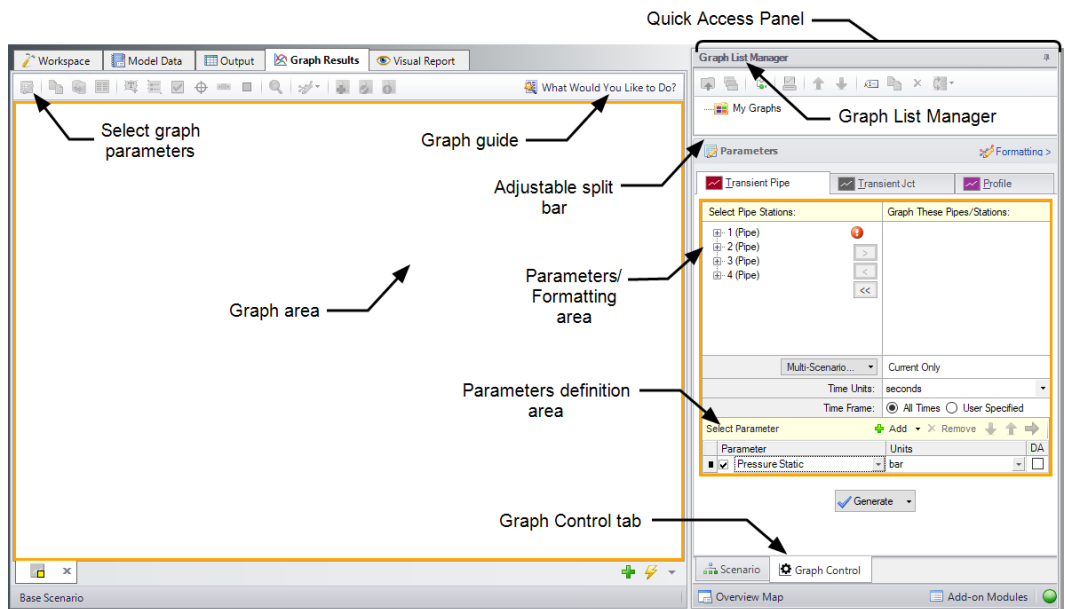
For transient analyses, the Graph Results window will usually be more helpful than the Output window because of the more voluminous data.

AFT Impulse’s Graph Guide feature, 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 2.27). You can create a graph by following the prompts on the Graph Guide, or 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 2.28 shows the Graph Results window with the Graph Control tab enabled on the Quick Access Panel.



**Figure 2.27** 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 2.28** The Graph Results window and Quick Access Panel Graph Control tab is where system parameters (both steady-state and transient) can be graphed

AFT Impulse gives you the ability to create “stacked graphs”. These are graphs that are displayed on top of each other with the same X-axis but with different parameters on the Y-axis. This feature is helpful to see how multiple parameters change in the same location of your model at the same time without having to create separate graphs.

In this example, you will create stacked graphs of the pressures and flows at the inlet and outlet of Valve J4 to see how these parameters change as the valve closes.

- **First, change to the Graph Results window** by clicking on the Graph Results tab, by clicking “Graph Results” from the Solution Progress window after running the model, or by pressing CTRL+G. The Graph Results window offers full-featured Windows plot preparation.
- **Access the graphing parameters by opening the Graph Control tab on the Quick Access Panel (this tab is automatically selected when the Graph Results window is opened).** Because you are interested in seeing how the pressures and flows in specific pipe sections respond over time, ensure that the Transient Pipe tab is selected in the Parameters/Formatting area on the Quick Access Panel. Alternatively, you can open the Select Graph Parameters window by clicking on the corresponding icon on the Graph Results Toolbar (this is the first icon located above the top left corner of the Graph area).
- **Under “Select Pipe Stations” on the Transient Pipe tab, expand the P3 pipe stations and double-click on Outlet**, which is the pipe computing station at the valve inlet. Alternatively, you can click on the right arrow button after selecting the pipe station you want to graph to add it to the “Graph These Pipes/Stations” list. Also add the inlet of Pipe P4 which is the valve exit.

---

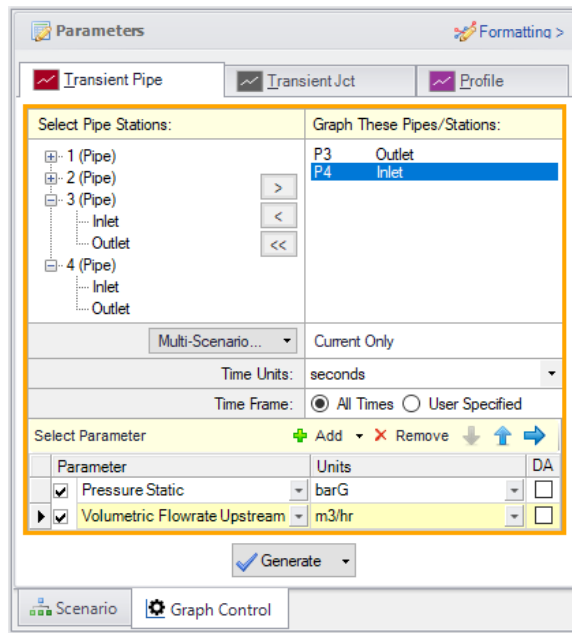
**Note:** Once a pipe is broken into sections, computation takes place where the sections join together, these are called "stations".

---

- **Select seconds for the Time Units** and select “All Times” next to “Time Frame”.
- **In the Parameters definition area, select “Pressure Static” and choose “barG” in the dropdown box under “Units”** to graph the static pressure in psig at the valve inlet and outlet over time.
- **In order to add the stacked graph under the static pressure graph showing the volumetric flowrate at the inlet and outlet of Valve J4, click the “Add” button**, which has the green “+” icon next to “Select Parameter”. A new row under the Parameter definition area will appear (See Figure 2.29).

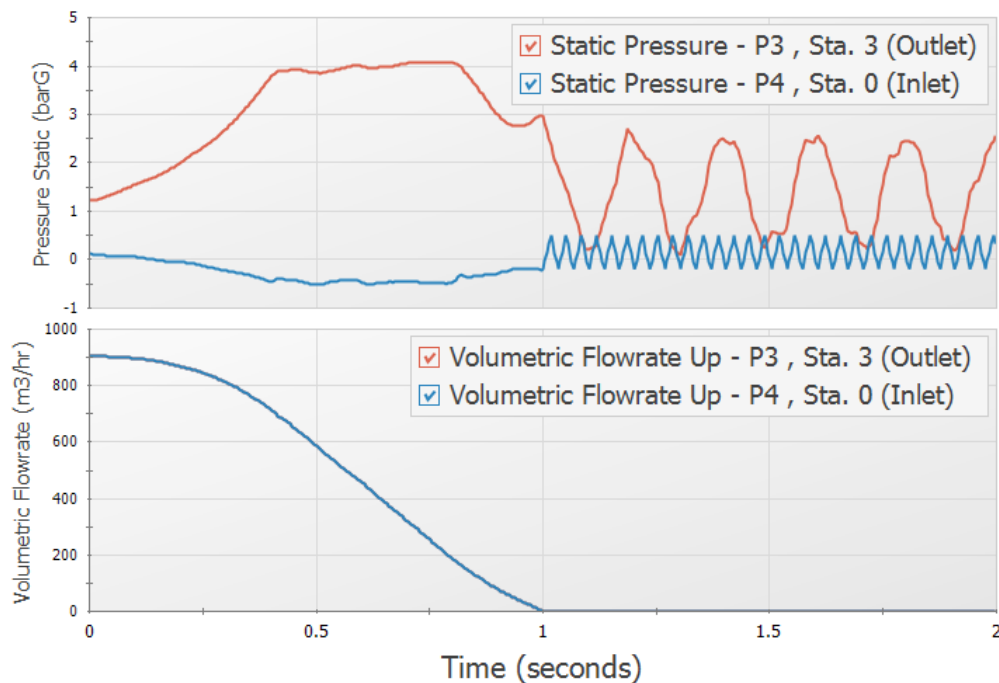
- Choose “Volumetric Flowrate Upstream” in this new parameter row and select units of “m3/hr”.
- Click **Generate** at the bottom of the Quick Access Panel to create the static pressure and volumetric flowrate graphs at the inlet and outlet of the valve over the duration of the simulation.
- To format the legend font size, right-click on each legend and use the scroll bar to decrease the font size to 14 for both graphs. Drag the legend to the upper right corner of the graph.
- To format each axis font size, right click on each axis title and use the scroll bar to decrease the size until the font size on each axis appears as you want it.

Figure 2.29 shows the input in the Parameters/ Formatting area on the Quick Access Panel.



**Figure 2.29** 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

Figure 2.30 shows the stacked graphs detailing the static pressures and volumetric flowrates at the Valve J4 inlet and outlet over two seconds. Here you can see that the maximum pressure at the valve inlet is about 4 barG. It should be noted that these graphs could also be made from the Transient Jct tab. In this case, the inlet and outlet pressures at the valve would be selected separately.



**Figure 2.30** The Graph Results window offers full-featured plot generation. Here the static pressure and volumetric flow at the Valve J4 inlet and outlet are shown over 2 seconds.

The graph colors, fonts and other elements can be modified using the Formatting area on the Graph Control tab on the Quick Access Panel. The Graph Results window can be printed, saved to file, copied to the clipboard, or printed to an Adobe PDF file. The graph's x-y data can be exported to file or copied to the clipboard.

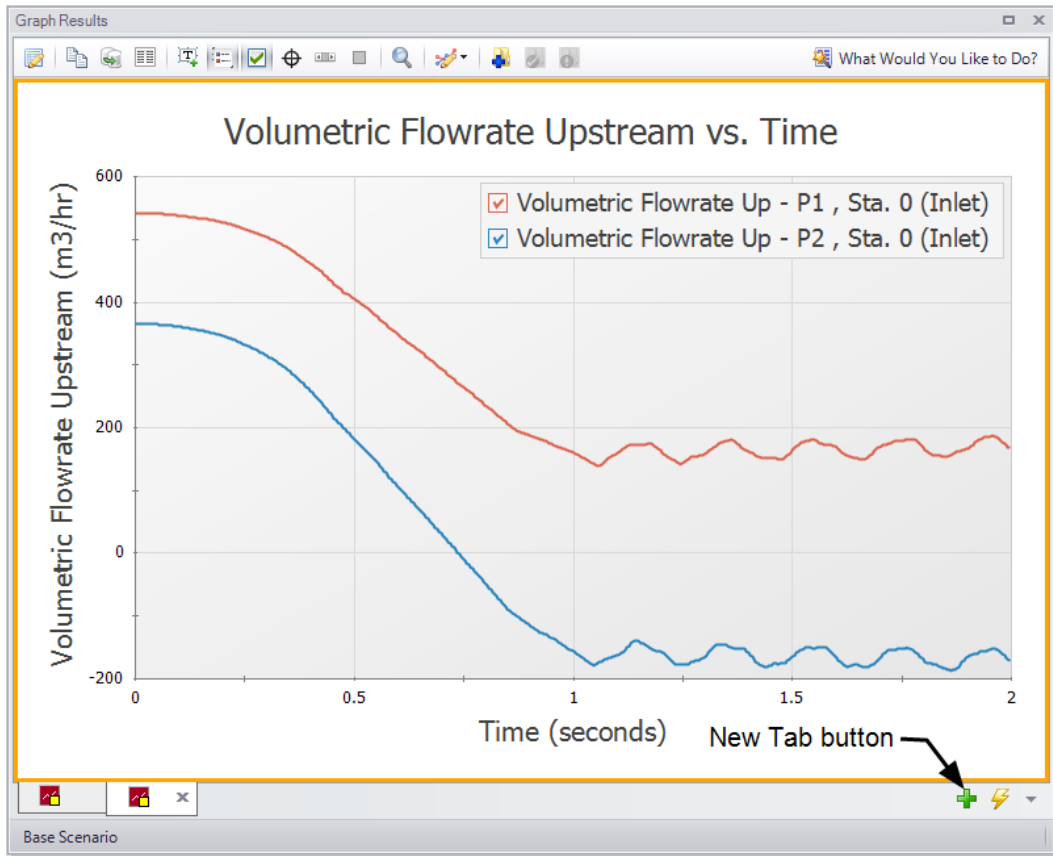
### A. Further review

Further review of the valve graph results in Figure 2.30 shows that at time zero, the pressure difference between the top and bottom curves of the pressure graph is about 1.1 barG. This is the steady-state pressure difference across the valve, which can also be found in the Output window to be 1.096 barG.

As time increases, one sees that the two curves move further apart, which represents the increased pressure drop across the valve as it closes. At one second, the valve closes entirely and the pipes upstream and downstream of the valve are isolated from each other and will decay to the steady-state conditions which exist for a closed valve.

For the purpose of analysis, it would also be useful to view the volumetric flow rate from each supply tank over time.

- **Create a new graph tab** by clicking the “New Tab” button which is the green plus icon located on the bottom right, immediately below the graph area (Figure 2.31)
- **Open the Transient Pipe tab** on the Graph Control tab of the Quick Access Panel. Add the P1 and P2 inlet pipe stations to the “Graph These Pipes/Stations” list and remove P3 outlet and P4 inlet pipe stations.
- **Select “seconds” in the Time Units section and “All Times” in the Time Frame section** to graph the volumetric flowrate over the entire simulation duration.
- **Select Volumetric Flowrate Upstream** and specify the units as “m3/hr”.
- **Click the Generate button** to display the graph. A graph of these stations shows the flowrates out of the two supply reservoirs. Note that after the valve closes, the fluid in Reservoir J1 flows to Reservoir J2 (Figure 2.31).



**Figure 2.31** Transient flowrates out of the reservoirs

**Note:** AFT Impulse allows you to quickly generate several types of graphs by right-clicking on pipes or junctions on the Workspace after your model has output. To quickly generate the static pressure plot at the Valve J4 inlet and outlet, you can right-click on Valve J4 on the Workspace, then select “Generate Transient Graph for Pipes”, then select Pressure Static. To add the volumetric flowrate stacked graph, or to modify any units or other graph parameters, access the Graph Control tab on the Quick Access Panel and adjust the graph parameters as desired.

## Step 7. View the Visual Report

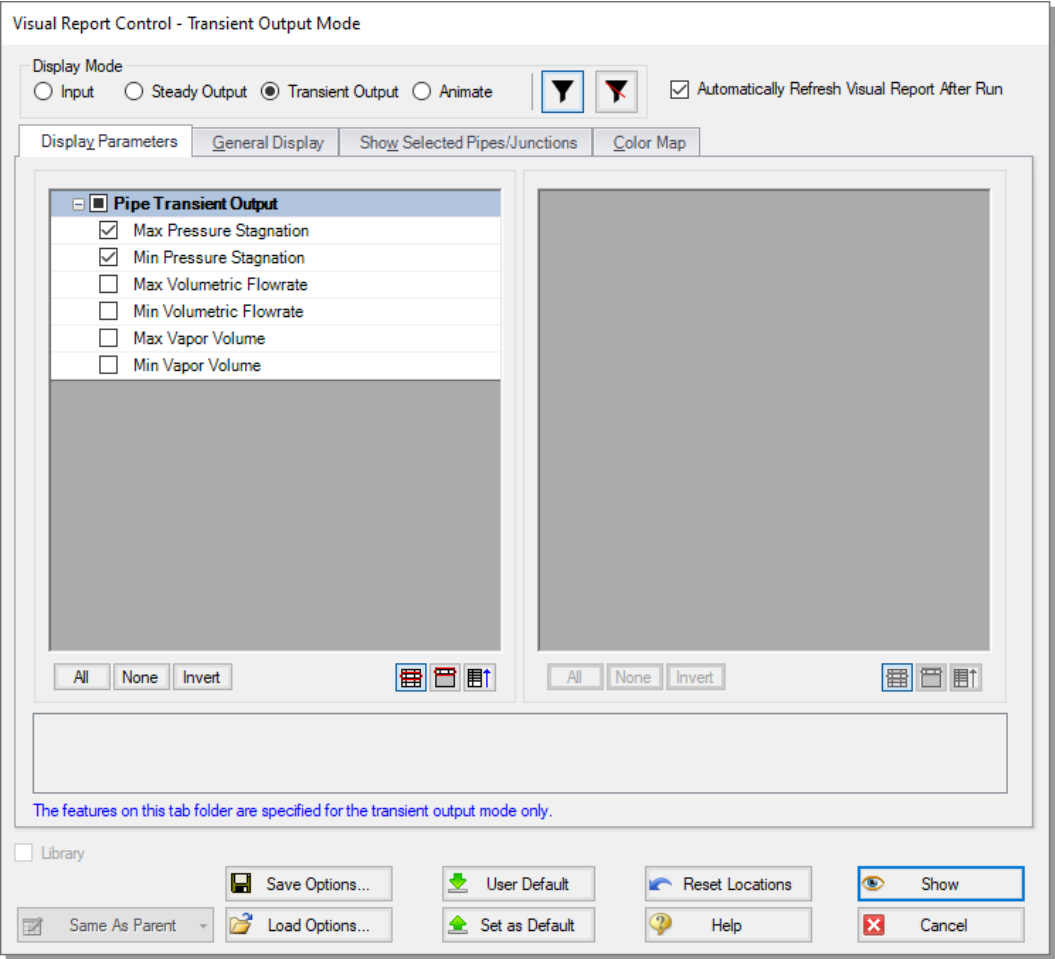
The Visual Report allows you to show text output alongside the model schematic. This is useful to quickly show pertinent information in relation to the location in the model. The Visual Report can also animate the transient pipe results in a color animation overlaid on the model.

- **Change to the Visual Report window** by clicking on the Visual Report tab, choosing it from the Window menu, or pressing CTRL+I. This window allows you to integrate your text results with the graphic layout of your pipe network. The Visual Report Control window will automatically open when accessing the Visual Report window for the first time. Visual Report Control can also be opened from the Toolbar or the Tools menu.
- **Select “Max Pressure Stagnation” and “Min Pressure Stagnation”** in the Pipe Results area as shown in Figure 2.32. Click the Show button. The Visual Report window graphic is generated (Figure 2.33).

It is common for the text in the Visual Report window to overlap when first generated. You can change this by selecting smaller fonts or by dragging the text to a new area to increase clarity. You can also use the Visual Report Control window to display units in a legend to increase the clarity of the display. These adjustments have already been done in Figure 2.33. This window can be printed or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.

The Visual Report window also provides the ability to animate the transient results as a qualitative tool to visualize the behavior in a network. This option can also be accessed from the Visual Report Control by selecting the Animate Display Mode at the top of the window (see Figure 2.32). When using the animation mode, text results cannot be displayed for the report. Instead, one parameter can be animated using absolute values, values relative to the steady state, or a static map of all maximum/minimum transient values can be generated. The animation can be recorded to a file. The model setup for animating static pressure can be seen in Figure 2.34.





**Figure 2.32** The Visual Report Control window selects content for the Visual Report window

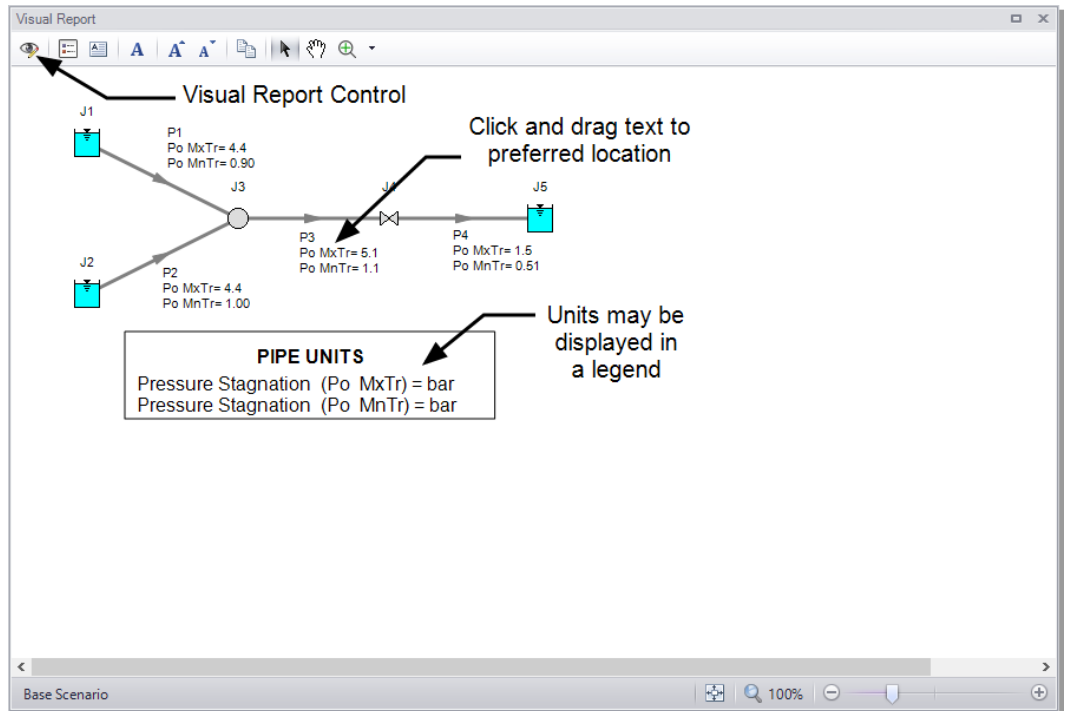


Figure 2.33 The Visual Report integrates results with model layout

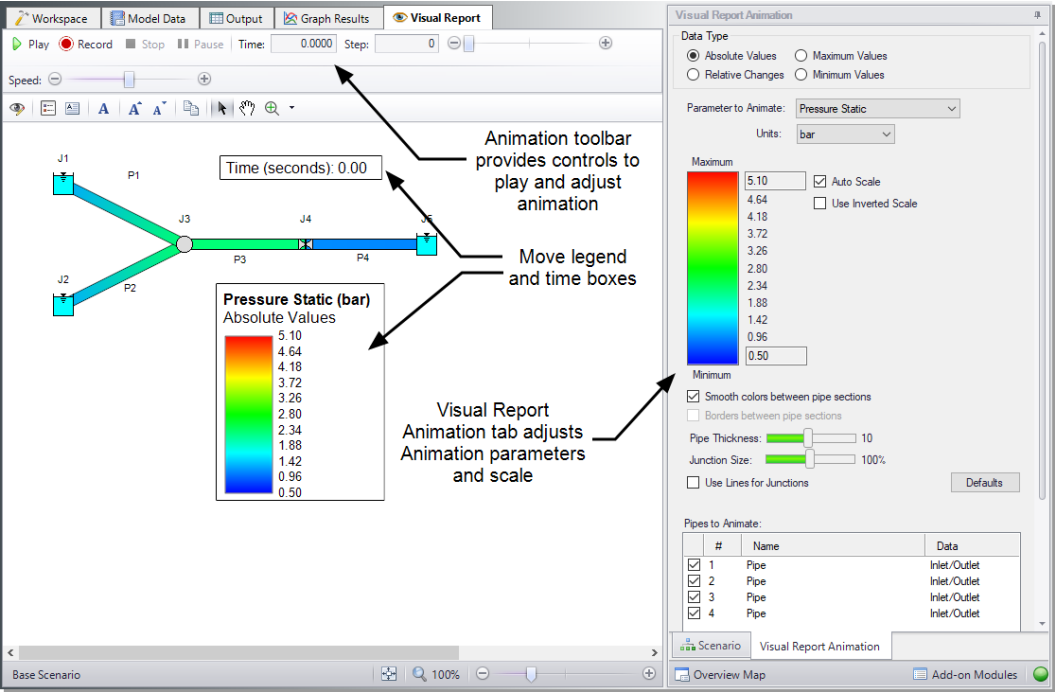


Figure 2.34 The Visual Report provides an animation feature as an alternate way to view the transient results

Conclusion

The maximum pressure at the valve’s inlet during this valve closure is approximately 5.1 bar. You have now used AFT Impulse's five Primary Windows to build a simple waterhammer model.

# Pump Startup with Event Transients

This example looks at three pump startup cases for a water transfer system. The objective is to determine the maximum pressures in the system and to assess whether the system cavitates or experiences sub-atmospheric pressure.

## Topics covered

- Starting one or more pumps
- Using steady-state and transient special condition features
- Using event transients
- Using Scenario Manager
- Adding Graphs to the Graph List Manager
- Using Graph Animation

## Required knowledge

These examples assume that the user has some familiarity with AFT Impulse such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Valve Closure Example in Chapter 2 for more information on these topics.

### Model file

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

- *Metric - Pump Startup with Event Transients.imp*

### Problem statement

This problem contains two pumps in parallel that transfer water from a supply reservoir to two process tanks downstream. The following three different pump startup scenarios are to be modeled and evaluated:

1. Both pumps starting
2. One pump starting
3. One pump starting with the other pump already running

As the design engineer, you will evaluate these three scenarios to determine the maximum and minimum system pressures, as well as use the animation features in AFT Impulse to understand how the system will respond in each case.

### Step 1. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

### Step 2. Define the Fluid Properties group

1. Open Analysis Setup using the button on the toolbar or from the Analysis menu.
2. Open the Fluid panel, select the AFT Standard library, then select Water (liquid) from the “Fluids Available in Library” list.
3. Click "Add to Model" to select water for use in this model.
4. Enter 16 deg. C in the Temperature field. This calculates the fluid properties to use in the model.
5. Click OK.

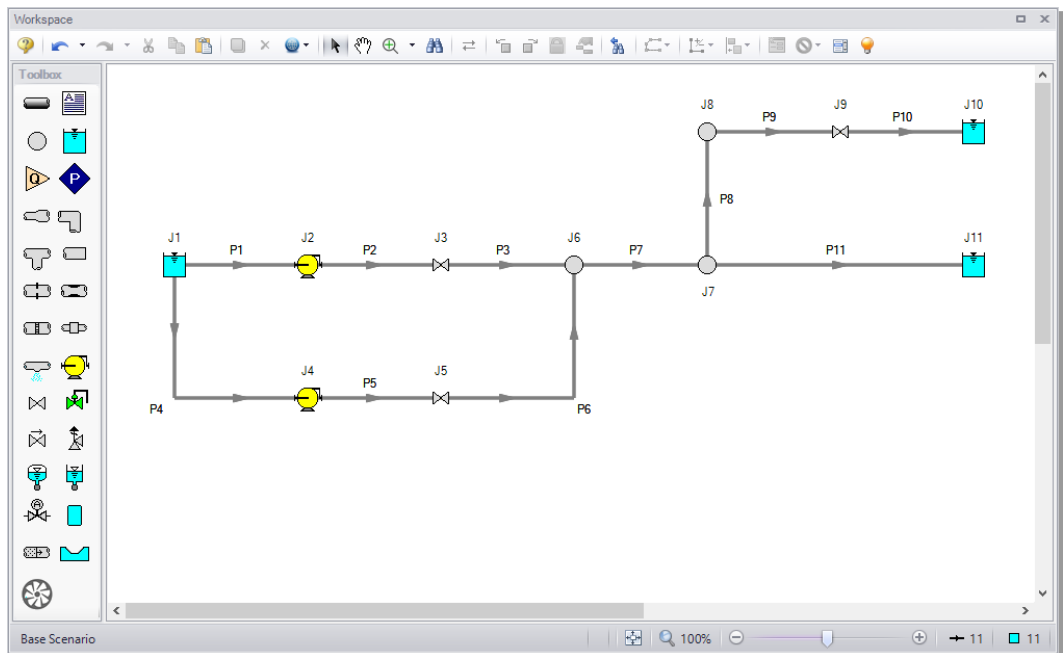
**Note:** The fluid can also be specified in the Startup window when AFT Impulse is first opened.

### Step 3. Define the Pipes and Junctions group

At this point, the first two groups are completed in Analysis Setup. The next group is the Pipes and Junctions group.

#### A. Place the pipes and junctions

In the Workspace window, assemble the model as shown in Figure 3.1. The Add/Remove Segments button on the Toolbar can be used to create the bend in pipes P4 and P6. Segments are only visual, and do not affect calculations.



**Figure 3.1** Layout of pipe system for pump startup example

#### B. Enter the pipe and junction data

The system is in place, but now the input data for the pipes and junctions need to be entered. Double-click each pipe and junction and

## 56 AFT Impulse™ 9 Quick Start Guide

---

enter the following data in each Pipe and Junction Properties window.

### ***Pipes***

All pipes are Steel – ANSI, STD, Thick-Walled Anchored Upstream, and Standard Friction Model with the following properties:

Pipe	Name	Size (inches)	Length (meters)
1	Suction Pipe Pump #1	10	15
2	Discharge Pipe Pump #1	10	6
3	Pipe	10	6
4	Suction Pipe Pump #2	10	18
5	Discharge Pipe Pump #2	10	6
6	Pipe	10	6
7	Pipe	16	30
8	Line to Process #1 Tank	12	76
9	Line to Process #1 Tank	12	61
10	Line to Process #1 Tank	12	366
11	Line to Process #2 Tank	12	1524

### ***J1 - Reservoir***

1. Name = Supply Reservoir
2. Tank Model = Infinite Reservoir
3. Liquid Surface Elevation = 6 meters
4. Liquid Surface Pressure = 0 barG
5. Pipe Depth = 6 meters

***J10 - Reservoir***

1. Name = Process #1 Tank
2. Tank Model = Infinite Reservoir
3. Liquid Surface Elevation = 30 meters
4. Liquid Surface Pressure = 0 barG
5. Pipe Depth = 3 meters

***J11 - Reservoir***

1. Name = Process #2 Tank
2. Tank Model = Infinite Reservoir
3. Liquid Surface Elevation = 3 meters
4. Liquid Surface Pressure = 0 barG
5. Pipe Depth = 3 meters

***J6, J7, & J8 - Branches***

1. Elevation = 0 meters

***J2 & J4 - Pumps***

1. J2 Name = Transfer Pump #1
2. J4 Name= Transfer Pump #2
3. Inlet Elevation = 0 meters
4. Pump Model = Centrifugal (Rotodynamic)
5. Enter Curve Data =

<b>Volumetric</b>	<b>Head</b>
<b>m3/hr</b>	<b>meters</b>
0	45.7
227	42.7
454	36.6



6. Curve Fit Order = 2
7. Click Generate Curve Fit Now then click OK
8. Click the Transient tab
9. Transient = Speed vs. Time
10. Transient Data = Absolute Values

Time (seconds)	Pump Speed (Percent)
0	0
2	100
10	100

11. Click the Optional tab
12. Special Condition = Pump Off With Flow Through

---

**Note:** The pump special condition indicates that initially the pump is turned off, but flow can go through it during steady-state. Valve junctions will be used to prevent any steady-state flow, as mentioned earlier. The pump junction in the Workspace will now have a red 'X' next to it indicating that the special condition is set to closed.

---

### **J3 & J5 - Valves**

1. J3 Name = Valve #1
2. J5 Name = Valve #2
3. Inlet Elevation = 0 meters
4.  $C_v = 1000$
5. Click the Transient tab
6. Transient Data = Absolute Values

Time (seconds)	Cv
0	0
1	800
2	1000
10	1000

7. Click the Optional tab
8. Special Condition = Closed

---

**Note:** When you close the valves, the adjacent pipes are displayed with dashed lines to denote that the pipes will have no flow.

---

### **J9 - Valve**

This valve will use an event transient. This means that the time zero in the Transient Data table is with respect to a condition occurring in the system. If the condition is never reached, the valve transient never initiates. Here, the valve should open when the pressure at Branch J7 is sufficient for water to flow into Reservoir J10. There is no "Junction Pressure" for the "Initiation of Transient", but there is one for pipes. Use one of the pipes connected to Branch J7 as a basis for the event.

1. Name = Valve to Process #1 Tank
2. Inlet Elevation = 0 meters
3. Cv = 500
4. Click the Transient tab (Figure 3.2)
5. Initiation of Transient = Single Event
6. Event Type = Pressure Stagnation at Pipe
7. Condition = Greater Than
8. Value = 3.1 barG
9. Pipe = 7 (Pipe) Outlet
10. Transient Data = Absolute Values

Time (seconds)	Cv
0	0
2	400
3	500
10	500

Valve Properties

Number: 9

Name: Valve to Process #1 Tank

Library Jct:

Copy Data From Jct...

Upstream Pipe: 9

Downstream Pipe: 10

Elevation

Inlet: 0 meters

Outlet: ☒ Same as Inlet

OK

Cancel

Jump...

Help

Loss Model

Transient

Optional

Design Alerts

Notes

Status

Transient Special Condition

☒ None

☐ Ignore Transient Data

Initiation of Transient

☐ Time

☐ Dual Event Cyclic

☒ Single Event

☐ Dual Event Sequential

Event Type: Pressure Stagnation at Pipe

Condition: Greater Than

Value: 3.1 barG

Pipe: 7 (Pipe) Outlet

Transient Data

☒ Absolute Values

☐ Relative To Steady-State Value

☐ Open Percentage

Data Point	Time (seconds)	Cv
1	0	0
2	2	400
3	3	500
4	10	500
5		
6		
7		
8		
9		
10		
11		
12		
13		

Edit Table

Show Graph...

☐ Repeat Transient

Event

Transient

Figure 3.2 Specifying an event transient for Valve J9

11. Click the Optional tab
12. Special Condition = Closed

---

**Note:** Transients which have multiple event initiators can be defined using Multi-Condition events, which are defined by selecting Multi-Condition for the Event Type, then using the Multi-Condition Event Manager to group the events.

---

### **C. Check if the pipe and junction data is complete**

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

---

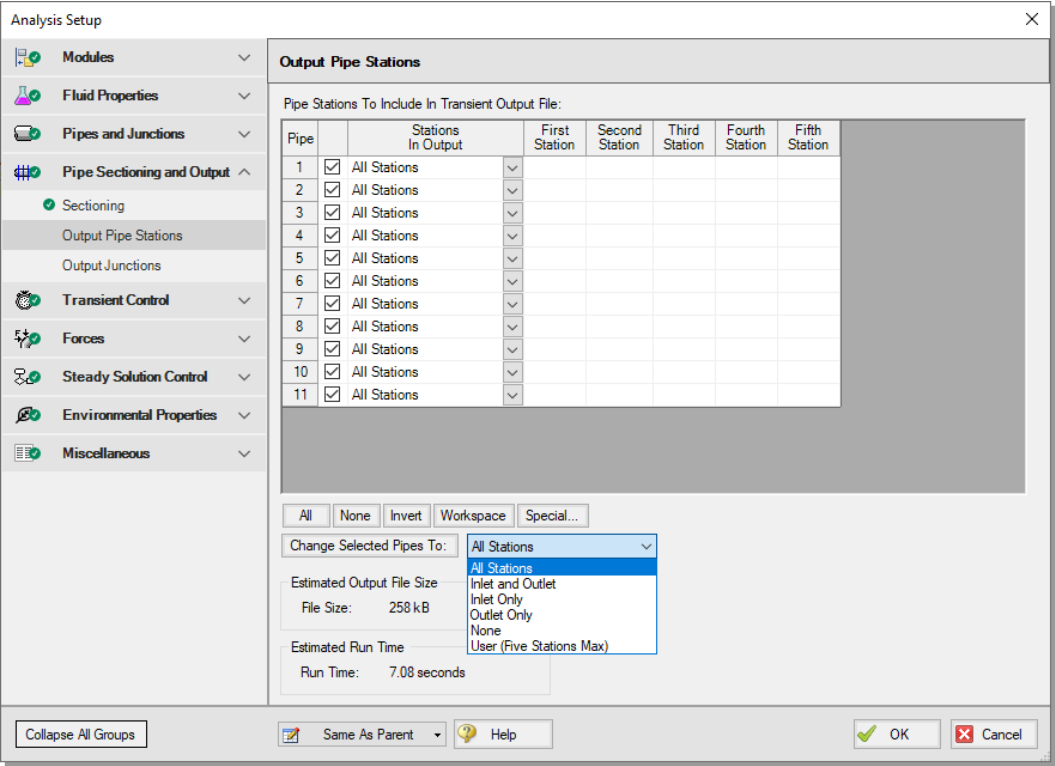
**Note:** Red is the default color that shows what objects are undefined in your model when Show Object Status is active. This color is configurable in the User Options.

---

## **Step 4. Specify the Pipe Sectioning and Output group**

Open Analysis Setup and go to the Sectioning panel. When this panel is first opened, it will automatically search for the best (least number of sections within 10% variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. No input is required for this step.

To minimize run time and output file size, the table on the Output Pipe Station panel uses “Inlet and Outlet” for all pipes by default. This will cause the output for only the inlet and outlet station of each pipe to be saved to file. Data can be saved for all pipe stations or only selected stations. Click the All button to select all the pipes in the table. Drop down the list next to the “Change Selected Pipes To” button and select “All Stations”. Then click the “Change Selected Pipes To” button (Figure 3.3). This will save all pipe station data for all pipes, which will be useful later for animation purposes.



**Figure 3.3** Output Pipe Sections panel with All Stations selected for all pipes

## Step 5. Specify the Transient Control group

Go to the Simulation Mode/Duration panel in the Transient Control group and enter a Stop Time of 10 seconds.

Next, go to the Transient Cavitation panel and clear the checkbox for Model Transient Cavitation (cavitation will be ignored until it becomes clear that it should be modeled)

## Step 6. Create scenarios to model the three startup cases

In this model, the following three pump startup cases are to be evaluated:

1. Both pumps starting
2. One pump starts while the other pump stays off
3. One pump starts while the other is already running

Three scenarios need to be created to model the three cases.

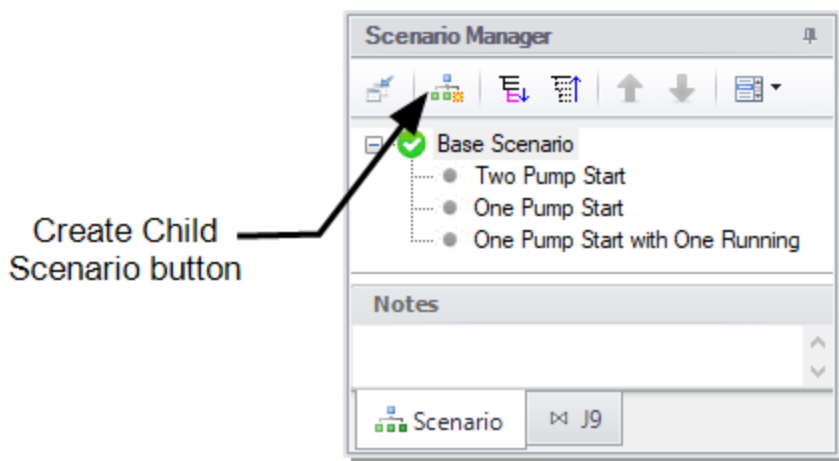
### **A. Create scenarios**

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

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

You will create three scenarios to model these cases. Scenarios are created using the Scenario Manager from the Tools menu or the Scenario Manager on the Quick Access Panel. The Quick Access Panel, located on the right side of the Workspace area, contains the Scenario Manager on the Scenario tab. A scenario tree displays all model scenarios.

Click the "Create Child" button on the Scenario Manager on the Quick Access Panel. Name the child “Two Pump Start.” A new scenario will appear below the Base Scenario in the scenario tree. Select the Base Scenario, create another child, and call it “One Pump Start.” Finally, create a third child called “One Pump Start With One Running” (see Figure 3.4). Note that the new child scenario is loaded when it is created using the Scenario Manager on the Quick Access Panel.



**Figure 3.4** The Scenario Manager on the Quick Access Panel displays the scenario tree that allows you to create model variants (scenarios) and keep them organized within the same model file

## B. Set up scenarios

Child scenarios “inherit” data from their ancestors. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent.

### ***Two Pump Start***

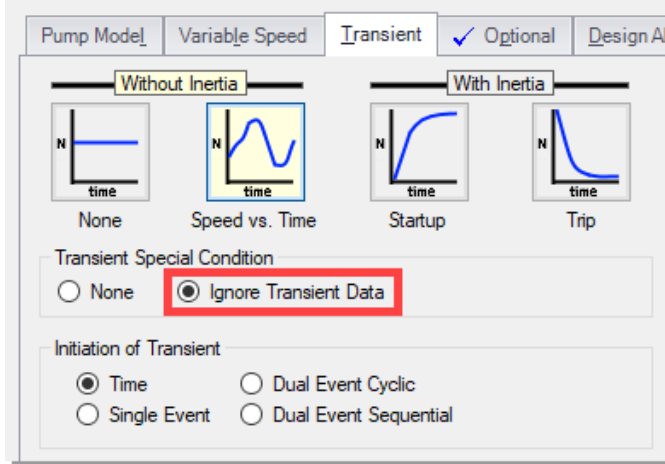
Since the Base Scenario already has been setup with two pumps starting, the “Two Pump Start” child does not need to be modified.

### ***One Pump Start***

Double-click the “One Pump Start” scenario in the scenario tree of the Scenario Manager on the Quick Access Panel. This will load the “One Pump Start” scenario onto the Workspace as the current scenario. The currently loaded scenario is displayed in the scenario tree with a green check mark (Figure 3.4).

Here, only Pump J2 will be started while keeping Pump J4 off. Deleting the transient input data for Pump J4 and Valve J5 will keep these two junctions turned off because their Special Conditions specify that they are off. Alternatively, their transient data can be set to be ignored. The second option is what will be done here.

- **Open the J4 Pump Properties window.** Click on the Transient tab, then choose the “Ignore Transient Data” option in the “Transient Special Condition” area as shown in Figure 3.5 then click OK. Note that a red ‘#’ will now be shown next to junction J4 indicating that the transient data is ignored. Do the same with the J5 Valve. The required input for this scenario is now complete.



**Figure 3.5** Transient Special Condition section on the Transient tab of the Pump Properties window

### ***One Pump Start with One Running***

Double-click the “One Pump Start With One Running” scenario from the scenario tree in the Scenario Manager on the Quick Access Panel. This will load the “One Pump Start With One Running” scenario onto the Workspace as the current scenario. This step can also be accomplished by opening the Scenario Manager from the Tools menu, selecting the “One Pump Start With One Running” scenario in the scenario tree and then clicking the “Load As Current Scenario” button.

Here, Pump J2 will be running at 100% speed during the steady-state and transient, and Pump J4 will be started.

- **Open the J2 Pump window** and select the Optional tab. Set the Special Condition to “None”. This specifies that during the steady-state the pump will be on and operate on its curve.



- **Select the Transient tab** and set the Transient Special Condition to “Ignore Transient Data”, since this pump will run at 100% speed during the transient. Alternatively, the speed data could be deleted.

The Special Condition for Valve J3 also needs to be set to “None”, and the Transient Special Condition on the Transient tab set to “Ignore Transient Data” by following the two steps above in the J3 Valve Properties window. The required input for this scenario is now complete.

## **Step 7. Run the model**

Load the "Two Pump Start" scenario. All groups in Analysis Setup should now be complete and the model ready to run. Run the model by pressing the “Run Model” icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. The Waterhammer Assistant window may pop up when the model is run. This window is a tool which provides advice to help simplify and improve the model before it is run. Click OK to dismiss the window. This will open the Solution Progress window. When complete, press the “Graph Results” button.

## **Step 8. Graph the results**

First a profile graph will be created to show the static pressure across the length of a series of pipes.

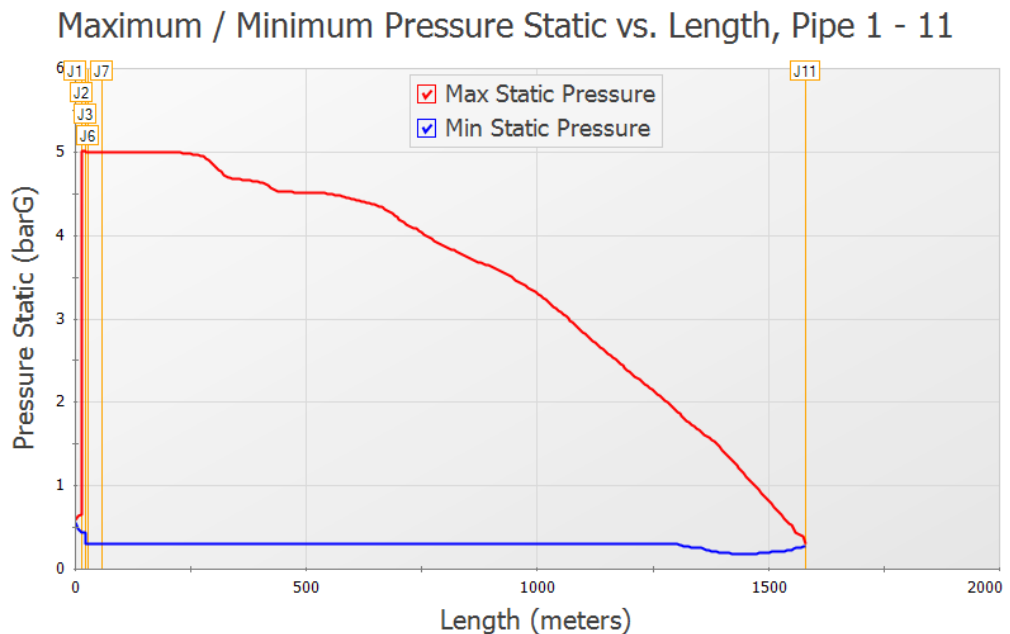
### **A. Create a profile graph**

1. From the Graph Control tab on the Quick Access Panel, select the Profile tab.
2. Select pipes 1, 2, 3, 7 and 11 to specify the path from the supply reservoir to Reservoir J11. Alternatively, you can select all these pipes on the Workspace by holding SHIFT while clicking on all pipes in the path, then return to the Graph Results tab and select “Workspace” to automatically choose all pipes currently selected on the Workspace.
3. Use “meters” for the Length Units.

4. In the Parameters definition area, use “Pressure Static” and specify “barG” in the dropdown box under “Units”.
5. Ensure that the boxes in the Mx and Mn columns are selected (these indicate that your graph will display the maximum and minimum pressures, respectively).
6. Click Generate.

Results (shown in Figure 3.6) indicate that the peak pressure occurs at the pump discharge, and that the minimum pressure is above atmospheric pressure at all times.

The “Show Junction Locations” icon shows the locations of junctions along the selected flow path.



**Figure 3.6** Profile of the maximum and minimum pressures through Pump #1 to Process #2 Tank for Two Pump Start scenario

To easily recreate this graph in other scenarios or in the future, it can be added to the Graph List. This can be done using the Graph List Manager on the Quick Access Panel. After generating the graph, click the “Add Graph To List” button on the Graph Results Toolbar (this icon is a blue “+” sign on top of a file folder image). Give the graph a name like “Pressure Profile Pump #1 to Process #2 Tank”. The graph will be added to the currently selected folder in the Graph List Manager. Note that graphs added to the Graph List Manager are not permanently saved until the model itself is saved again.

The profile through the other flow paths (there are four paths altogether) can also be plotted, and similar conclusions are obtained. Add the other three paths to the Graph List.

## **B. Create a Transient Junction graph**

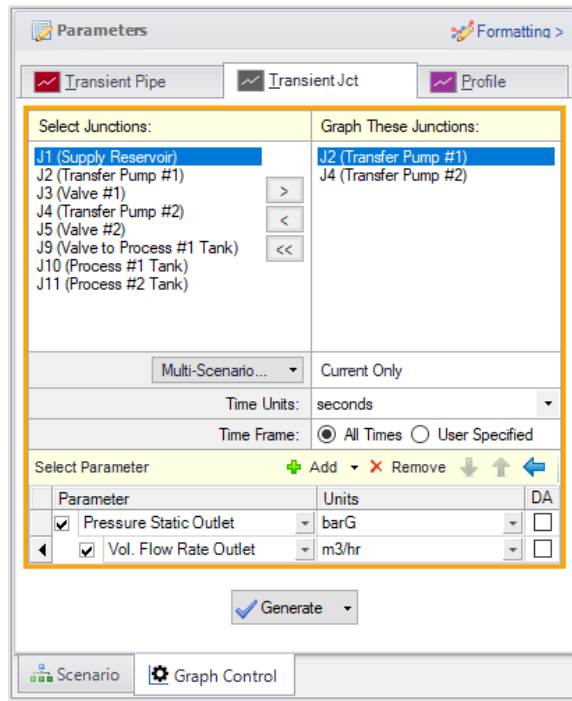
Since the maximum pressure is at the pump discharge, it is of interest here to plot the pressure vs. time. The volumetric flowrate vs. time at the pump discharge is also of interest. You will use AFT Impulse’s dual Y-axis graphing abilities to plot both the pressure and volumetric flow rate vs. time on the same graph to observe how the pressure and flow change at the same time.

1. From the Graph Control tab, choose the Transient Jct tab in the Parameters/Formatting area and add pumps J2 and J4 to the “Graph These Junctions” list.
2. Ensure that the Time Units is set to “seconds” and the Time Frame is set to “All Times”.

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 pressure on the primary Y-axis and volumetric flowrate on the secondary Y-axis.

3. To do this, select “Pressure Static Outlet” in the Parameters definition area and choose “barG” as the units.
4. Add a parameter by clicking on the “Add” button next to “Select Parameter” in the Parameter definition area.
5. Change this added parameter to “Vol. Flow Rate Outlet” and choose “m3/hr” as the units.

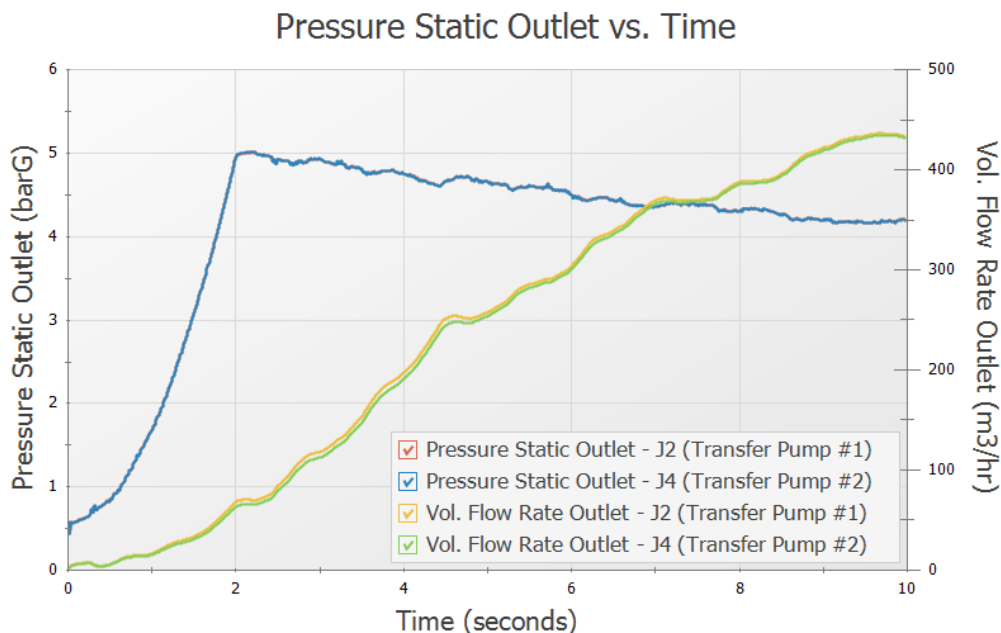
6. To specify that the volumetric flowrate will be plotted on the secondary Y-axis, click the “Make Secondary Y-Axis” button, which is the blue, right arrow icon located to the top right over the selected parameters. Alternatively, you can double-click on the column in front of “Volumetric Flowrate” in the Parameter definition area. Note that “Vol. Flow Rate Outlet” becomes indented under “Pressure Static Outlet”, and that the black arrow next to “Vol. Flow Rate Outlet” turns to point towards the graph. See Figure 3.7 to see how these graphing parameters are defined.



**Figure 3.7** Parameter definitions for dual Y-axis graph of static pressure (primary Y-axis) and volumetric flowrate (secondary Y-axis) on the Graph Control tab on the Quick Access Panel

Click Generate. You will see that the transient pressures at the pump discharge are plotted versus time on the primary Y-axis, and the volumetric flowrate vs. time is plotted on the same graph on the secondary Y-axis. As would be expected, the pressure and volumetric

flowrate transients are very similar for both pumps. The resulting graph is shown in Figure 3.8. Add this graph to the Graph List.



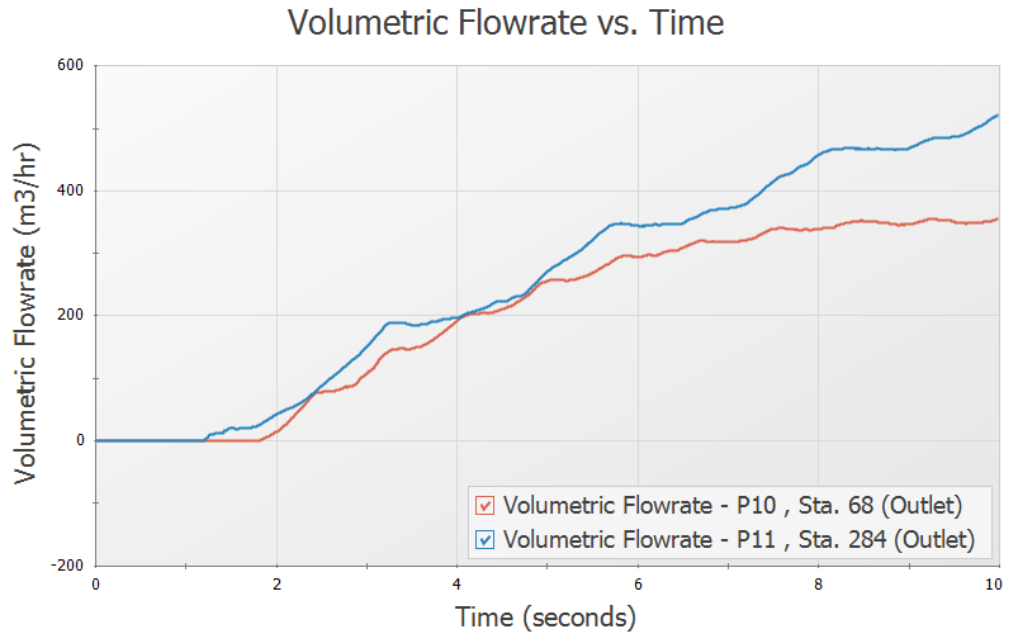
**Figure 3.8** Pressure and volumetric flowrate transients at both pump discharge locations for Two Pump Start scenario

### C. Create a Transient Pipe graph

The flowrates into the two process tanks are of interest.

1. Go to the Transient Pipe tab on the Graph Control tab.
2. Add the outlet of Pipes P10 and P11 from the "Select Pipe Stations" list to the "Graph These Pipes/Stations" list.
3. Select "Volumetric Flowrate" from the Parameter list.
4. Choose units of m3/hr.
5. Click Generate.

Results are shown in Figure 3.9. Add this plot to the Graph List.



**Figure 3.9** Flowrate transient at both process tanks for Two Pump Start scenario

## Step 9. Animate the results

AFT Impulse can animate hydraulic parameters along pipeline paths in your model in the Graph tab. This feature is very helpful when trying to understand how your system will respond when a transient event occurs. You can also animate multiple graphs at the same time.

Here, you will animate the pressure and volumetric flowrate responses along the path you previously graphed to gain an understanding of how these hydraulic parameters respond as the pumps start up. To animate these parameters:

1. Select the Profile tab under the Graph Control tab on the Quick Access Panel. Alternatively, you can access the Select Graph Parameters window by clicking on the “Select Graph Parameters” icon located at the top left of the Graph Results window.

2. Make sure pipes 1, 2, 3, 7, and 11 are selected
3. Add a second parameter in the Parameters definition area, then specify one parameter to be “Pressure Static” and the other to be “Volumetric Flowrate”.
4. As was done in Step 8, specify that the volumetric flowrate will be plotted on the secondary Y-axis by double-clicking on the column in front of “Volumetric Flowrate” in the Parameters definition area. Note that “Volumetric Flowrate” becomes indented under “Pressure Static”, and that the black arrow next to “Volumetric Flowrate” turns to point towards the graph (Figure 3.10)

---

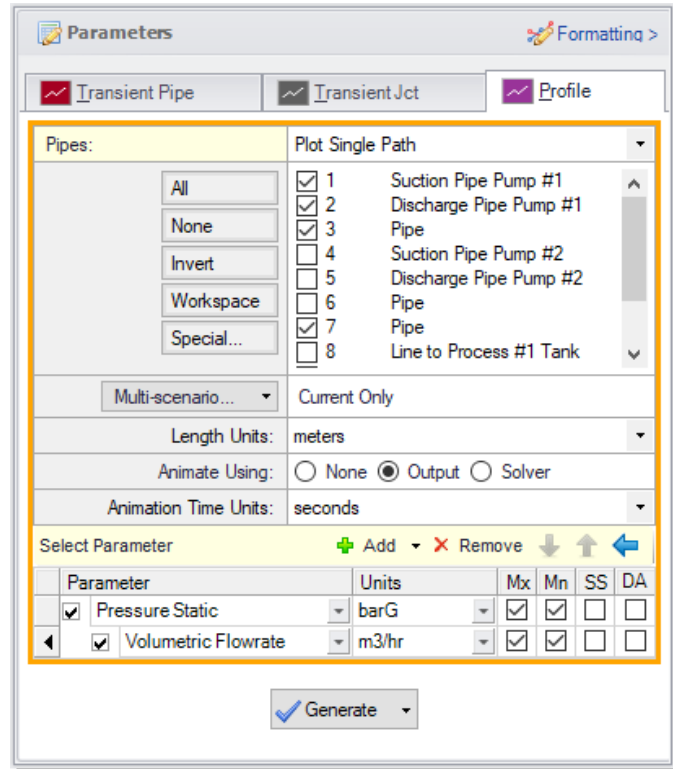
**Note:** AFT Impulse provides the flexibility to simultaneously animate stacked graphs, as well as graphs with dual Y-axes plotted. In this example, you will animate the static pressure and volumetric flowrate displayed on dual Y-axes.

---

5. Use the Length Units of meters
6. Next to Animate Using, specify “Output”, and specify “seconds” next to Animation Time Units (see the following paragraph for an explanation about this option)
7. Change the pressure units to barG and the volumetric flowrate units to m<sup>3</sup>/hr
8. Specify for both the static pressure and volumetric flowrate graphs that you want to see the maximum and minimum values plotted along the length of the graphed path by placing a checkmark in the boxes in the Mx and Mn columns next to each parameter (See Figure 3.10)

To use the “Animate Using: Output” option for animation, all pipe stations need to be specified to be saved in the Output Pipe Stations panel in the Pipe Sectioning and Output group (this was done previously in Step 4). This animation option has the most flexibility to start, pause, and restart animation.

Another animation option in Figure 3.10 is “Animate Using: Solver”. This option reruns the Transient Solver to generate the data for animation, and thus does not need to read it from file. Therefore, it does not require all of the pipe station data to be saved in Transient Control.



**Figure 3.10** Selecting animation in Graph Results for Two Pump Start scenario

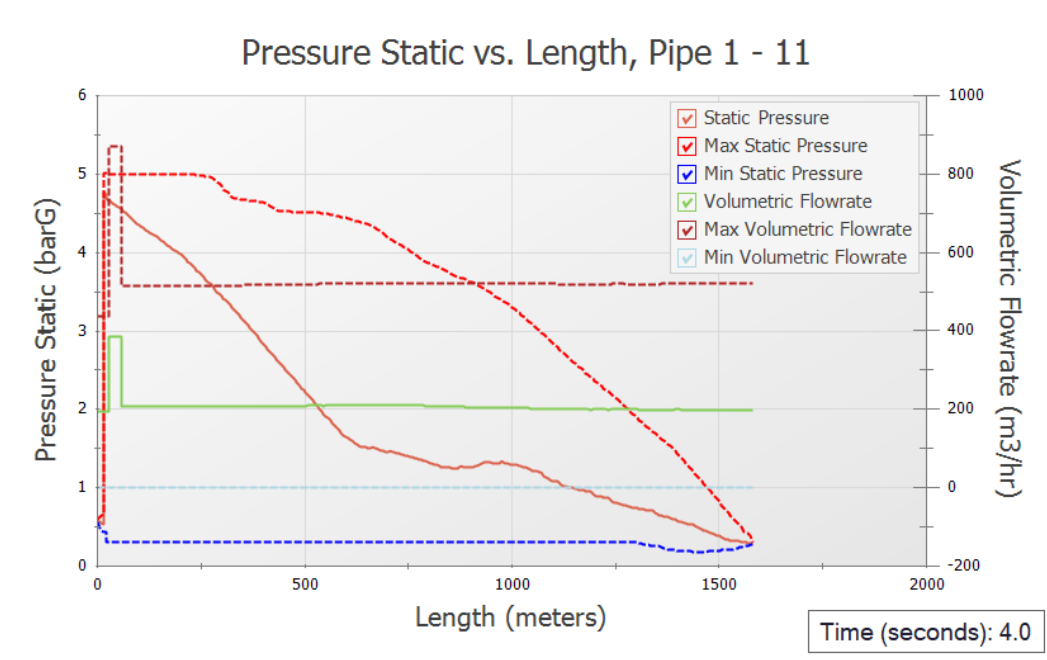
Click Generate. Additional animation control features appear on the Graph Results window (Figure 3.11). Press the Play button and watch the pressure waves and volumetric flowrate responses move between the maximum and minimum values. This animation can also be recorded to a file.

## Step 10. Run the other scenarios and graph the results

Using Scenario Manager, load the other two scenarios and run them. Reload the graph settings saved in Step 8 for the new scenarios by double-clicking the graph names in the Graph List Manager. This will reveal that the maximum pressure for the “One Pump Start” scenario occurs at the pump discharge location, similar to the first scenario, but



in the “One Pump Start With One Running” scenario, the maximum pressure occurs in Pipe P11. Also, the pressure drops below atmospheric pressure, but does not reach the vapor pressure of water (-0.995 barG).



**Figure 3.11 Animating output in Graph Results for Two Pump Start scenario**

In the “One Pump Start With One Running” scenario, it should also be noted that there is a warning given for the pump. This is indicated by the red message in the status bar. The warning can be viewed in the top third section of the Output window on the Warnings tab, and states that there was flow at pump junction J5. This warning message indicates that there was insufficient information available to accurately calculate the pump head with backwards flow. It is recommended to adjust the model to better account for this flow.

Depending on the system, there are several actions that may be appropriate. One change may be to adjust model inputs such as the discharge valve opening profile to prevent reverse flow to the pump. This is only acceptable if the changes are physically accurate for the system. Otherwise, it will be necessary to choose an appropriate four

quadrant data set for the pump to accurately predict the head at reverse flow conditions. This will be discussed further in the next chapter.

## Conclusion

The numerical maximum pressure can be found at the bottom of the General data section in the Output window of each scenario and is summarized in Table 3.1 for all three scenarios. For these cases, the maximum pressures do not significantly differ from each other.

**Table 3.1      Summary of maximum static pressure for the three cases**

<b>Case</b>	<b>Max. Static Pressure (barG)</b>
Two Pumps Start	5.0156
One Pump Start	4.9306
One Pump Start With One On	5.0646



# Pump Trip Example

This example evaluates two parallel pumps tripping in a cooling tower system. Various discharge valve closure times are analyzed to examine the impact on the system when the valves close quickly enough to prevent reverse flow at the pump, and when the valves do not close completely before reverse flow occurs in the pump.

## Topics covered

- Specifying a pump and power curve
- Using one of the transient pump inertia models
- Using Four Quadrant Data to model reverse flow and reverse rotation in the pump.

## Required knowledge

These examples assume that the user has some familiarity with AFT Impulse such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Valve Closure Example in Chapter 2 for more information on these topics.

### Model file

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

- *Metric - Pump Trip With Backflow - Four Quadrant Modeling.imp*

### Problem statement

In this problem, two parallel pumps in a cooling tower system lose power, causing the pumps to trip. It is desired for the valves at the discharge of the pumps to close quickly enough to prevent reverse flow and rotation in the pumps. You will evaluate the maximum and minimum pressures experienced in the system during this pump trip event.

Furthermore, you will look at the case where the minimum valve closure time is 12 seconds, in which case reverse flow and rotation will occur at the pump.

### Step 1. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

### Step 2. Define the Fluid Properties group

1. Open Analysis Setup using the button on the toolbar or from the Analysis menu.
2. Open the Fluid panel, select the AFT Standard library, then select Water (liquid) from the “Fluids Available in Library” list.
3. Click "Add to Model" to select water for use in this model.
4. Enter 38 deg. C in the Temperature field. This calculates the fluid properties to use in the model.
5. Click OK.

### Step 3. Define the Pipes and Junctions group

At this point, the first two groups are completed in Analysis Setup. The next group is the Pipes and Junctions group.

#### A. Place the pipes and junctions

In the Workspace window, assemble the model as shown in Figure 4.1. Note that the spray discharge junctions can be rotated as shown by selecting the junctions in the Workspace and clicking the “Rotate Icons...” button on the Workspace Toolbar.

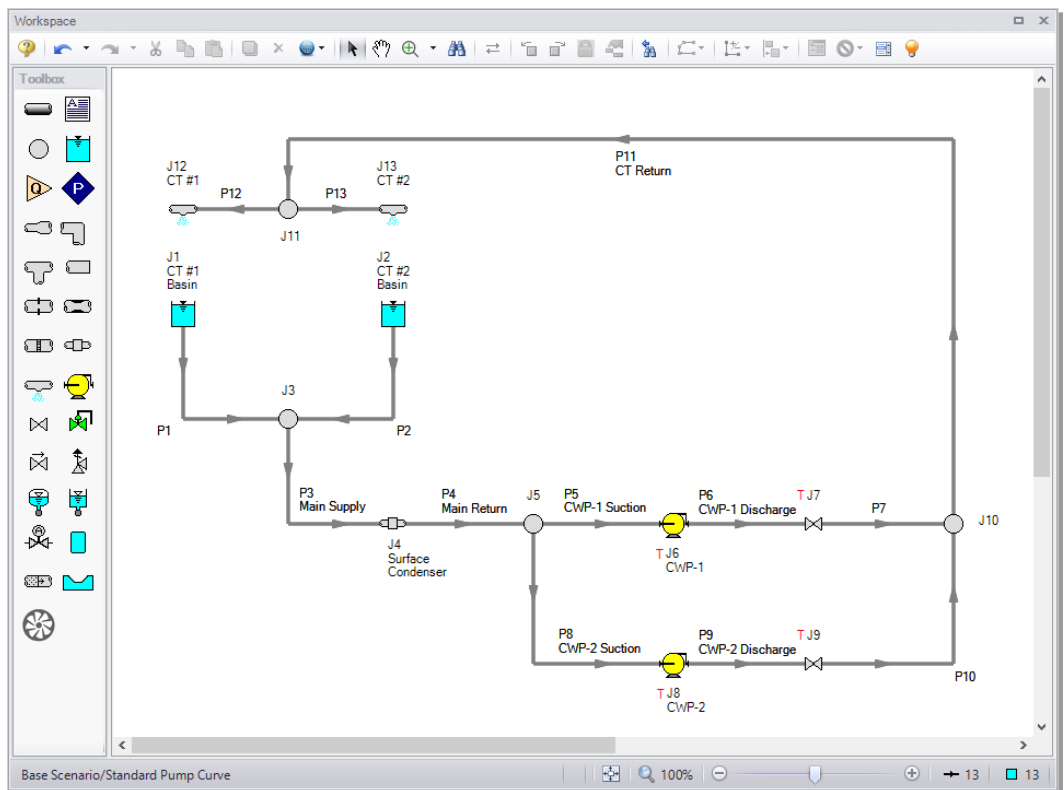


Figure 4.1 Workspace for pump trip with backflow example

## B. Enter the pipe and junction data

The system is in place, but now the input data for the pipes and junctions need to be entered. Double-click each pipe and junction and enter the following data in each Pipe and Junction Properties window.

### ***Pipes***

All pipes are Steel - ANSI, STD Type, Standard Friction Model, with Thick-Walled Anchored Upstream Pipe Support.

Pipe	Name	Size (inches)	Length (meters)
1	Pipe	18	12
2	Pipe	18	12
3	Main Supply	30	99
4	Main Return	30	11
5	CWP-1 Suction	20	3
6	CWP-1 Discharge	20	3
7	Pipe	20	3
8	CWP-2 Suction	20	3
9	CWP-2 Discharge	20	3
10	Pipe	20	3
11	CT Return	30	122
12	Pipe	20	12
13	Pipe	20	12

**J1 & J2 - Reservoirs**

1. J1 Name = CT #1 Basin
2. J2 Name = CT #2 Basin
3. Tank Model = Infinite Reservoir
4. Liquid Surface Elevation = 12 meters
5. Liquid Surface Pressure = 0 barG
6. Pipe Depth = 1 meters

**J3 - Branch**

Elevation = 11 meters

**J4 - General Component**

1. Name = Surface Condenser
2. Inlet Elevation = 0 meters
3. Loss Model = Resistance Curve
4. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	0
4542	6.7
9085	26.8

**J5 & J10 - Branches**

Elevation = 0 meters

**J11 - Branch**

Elevation = 15 meters



***J6 & J8 - Pumps***

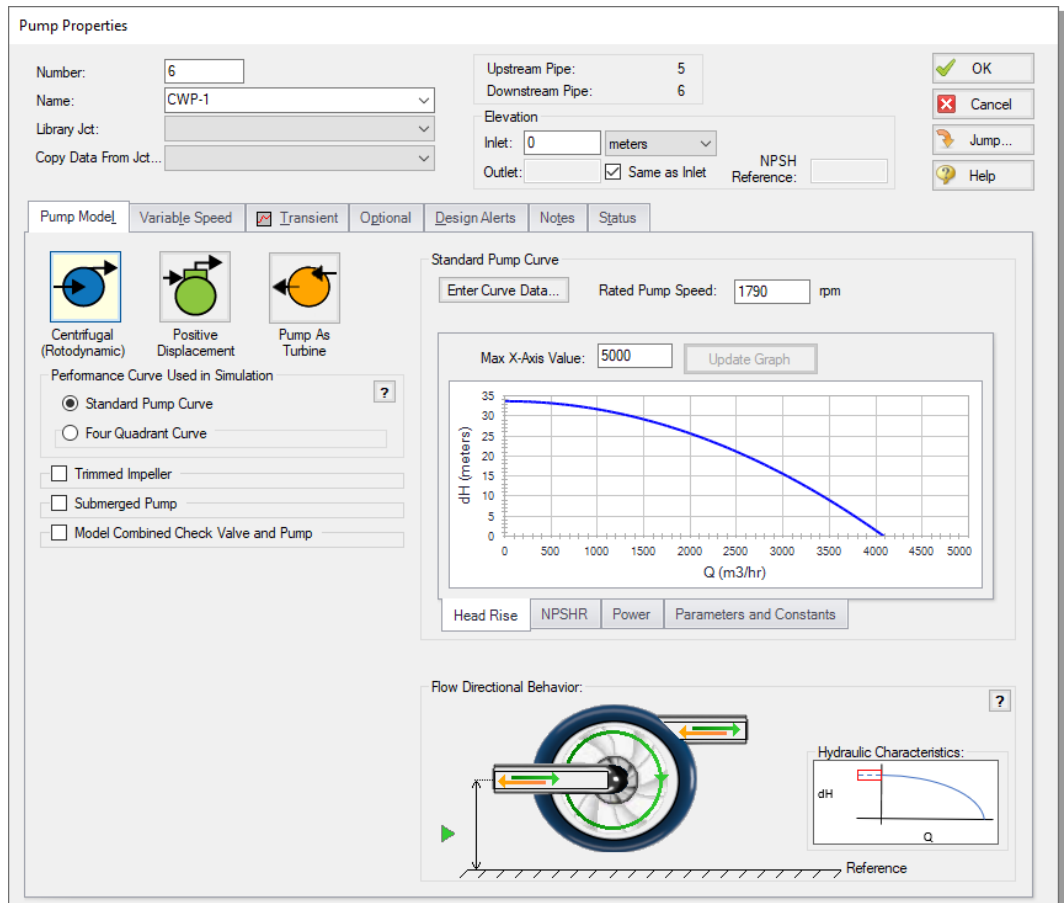
1. J6 Name = CWP-2
2. J8 Name = CWP-2
3. Inlet Elevation = 0 meters
4. Pump Model = Centrifugal (Rotodynamic)
5. Performance Curve Used in Simulation = Standard Pump Curve
6. Enter Curve Data =

<b>Volumetric</b>	<b>Head</b>	<b>Power</b>
<b>m3/hr</b>	<b>meters</b>	<b>kW</b>
0	34.1	78.3
454	32.9	96.9
908	31.7	119.3
1363	29.9	141.6
1817	27.4	156.5
2271	23.8	164
2825	18.3	156.5

7. Curve Fit Order = 2
8. Rated Pump Speed = 1790 rpm
9. Click the Transient tab
10. Transient = Trip
11. Initiation of Transient = Single Event
12. Event Type = Time Absolute
13. Condition = Greater Than or Equal To
14. Value = 0 seconds

15. Total Rotating Inertia = User Specified
16. Value = 5.9 kg-m<sup>2</sup>

**Note:** When reverse flow occurs at the pump, the Four Quadrant Model is recommended to obtain a better estimate for the head at reverse flow. To simplify the input, we will define the pump with the Standard Pump curve for now (Figure 4.2). We will come back later and compare these results to the Four Quadrant option.



**Figure 4.2** Pump Model tab for Pump J6. The pump behavior with forwards and backwards flow can be seen in the Flow Directional Behavior section.

***J7 & J9 - Valves***

1. Inlet Elevation = 0 meters
2.  $C_v = 10,000$
3. Click the Transient tab
4. Transient Data =

Time (seconds)	$C_v$
0	10,000
3	1,000
5	0
15	0

***J12 & J13 - Spray Discharges***

These junctions each represent numerous spray nozzles that are modeled as a single nozzle for simplicity.

1. J12 Name = CT #1
2. J13 Name = CT #2
3. Elevation = 24 meters
4. Loss Model = Cd Spray (Discharge Coefficient)
5. Geometry = Spray Nozzle
6. Exit Pressure = 0 barG
7.  $C_d = 0.6$
8. Discharge Flow Area = 0.93 meters<sup>2</sup>

### **C. Check if the pipe and junction data is complete**

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

## **Step 4. Specify the Pipe Sectioning and Output group**

Open Analysis Setup and go to the Sectioning panel. When this panel is first opened, it will automatically search for the best (least number of sections within 10% variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. No input is required for this step.

## **Step 5. Define the Transient Control group**

Go to the Simulation Mode/Duration panel in the Transient Control group and enter a Stop Time of 15 seconds. Next, in the Save Output To File section, change the selection to Every 10 Time Steps to reduce runtime.

## **Step 6. Run the model**

All groups in Analysis Setup should now be complete and the model ready to run. Run the model by pressing the “Run Model” icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. This will open the Solution Progress window. When complete, press the “Output” button.

## **Step 7. Review results**

Clicking “Output” from the Solution Progress window, selecting the Output tab, or pressing CTRL+U on the keyboard will take you to the Output window, which will display any warnings

A warning is given for each pump stating that the pump experienced reverse flow and was using a Standard Pump Curve, meaning that the zero-flow head value was used to predict head and power at reverse flow. If the reverse flow is low in magnitude and duration, using the zero-flow head may be a good assumption. We will examine the impact of using a four quadrant curve as suggested by this warning later in this example.

In addition, a caution is given for each pump that the end of curve flow was exceeded. In this model we did not explicitly define the end of curve flow, so the highest flowrate that was entered was assumed to be the end of curve flow. The pump head and power curve fits will only be reliable within the range of data entered by the user, thus predictions at flow rates above this range will be inaccurate, particularly when the pump is operating at full speed. Operating beyond the end of curve is less of a concern when the pump is operating at lower speeds, such as occurs for the pump in this case since the end of curve is only exceeded after the pump trip beings. However, it is still prudent to verify that the effect of operating past the end of curve is low by using a four quadrant curve.

The Graph Results window will be useful in evaluating the results. We will graph the static pressure and volumetric flowrate at the pump suction and valve discharge as a dual y-axis graph to see how these parameters vary with time.

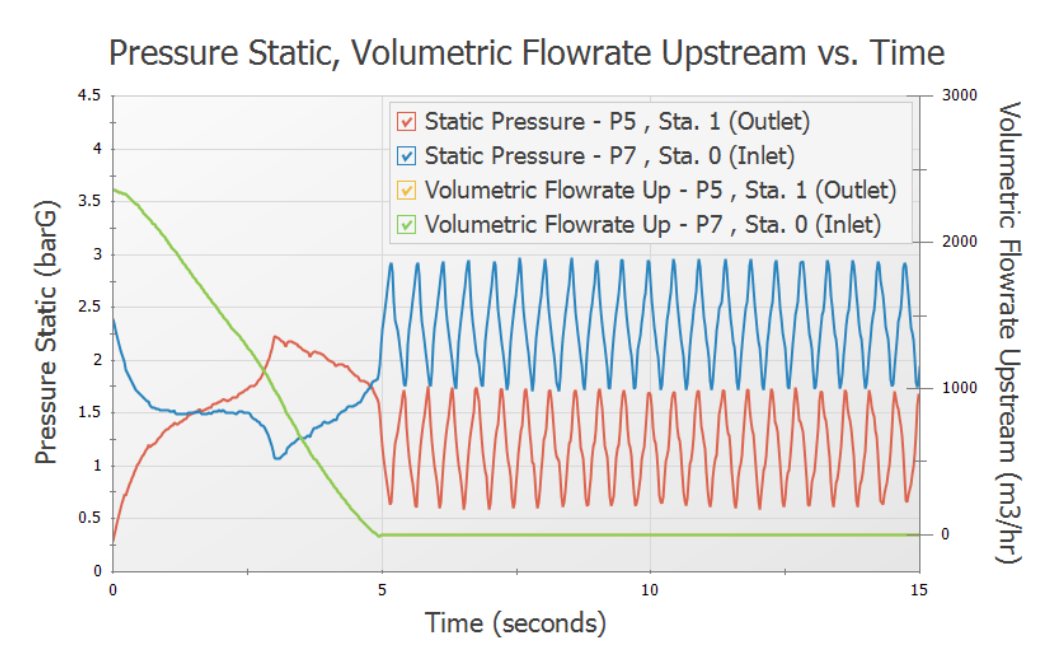
### **A. Graph the transient pressures and flows at the pump station**

1. Change to the Graph Results window from the Window menu, by clicking on the Graph Results tab, or by pressing CTRL+G.
2. Go to the Transient Pipe tab on the Graph Control tab and add Pipe P5 Outlet and Pipe P7 Inlet under “Graph These Pipes/Stations”. These locations represent the pump suction and valve discharge piping, respectively.
3. Select the “Time Units” as seconds and ensure that the “Time Frame” is showing “All Times”.
4. Add a second parameter to be graphed by clicking on the “Add” button, represented with the green “+” icon. Select “Pressure

Static” as one parameter to be graphed and “Volumetric Flowrate Upstream” as the second parameter to be graphed.

5. Move the Volumetric Flowrate Upstream to the secondary Y-axis by clicking the “Make Secondary Y-axis” button, which is the blue, right arrow icon. This parameter should now be indented.
6. Set the units to “barG” for the static pressure and “m3/hr” as the volumetric flowrate.
7. Click Generate. The resulting pressure and flow transients are shown in Figure 4.3.
8. Click the Add Graph to List button from the Toolbar to save this graph to the Graph List Manager so that it can be regenerated later, and name it “Pressure & Flowrate vs. Time”.

In Figure 4.3 you can see that the pressure increases at the pump inlet and decreases at the valve outlet until a maximum pressure of 2.24 barG at the pump inlet and a minimum pressure of 1.06 barG at the valve outlet is reached. The pressures then begin to oscillate once the discharge valve becomes fully closed.



**Figure 4.3** Pressure and volumetric flow rate transients using the standard pump curve data.

## B. Graph the pump speed decay

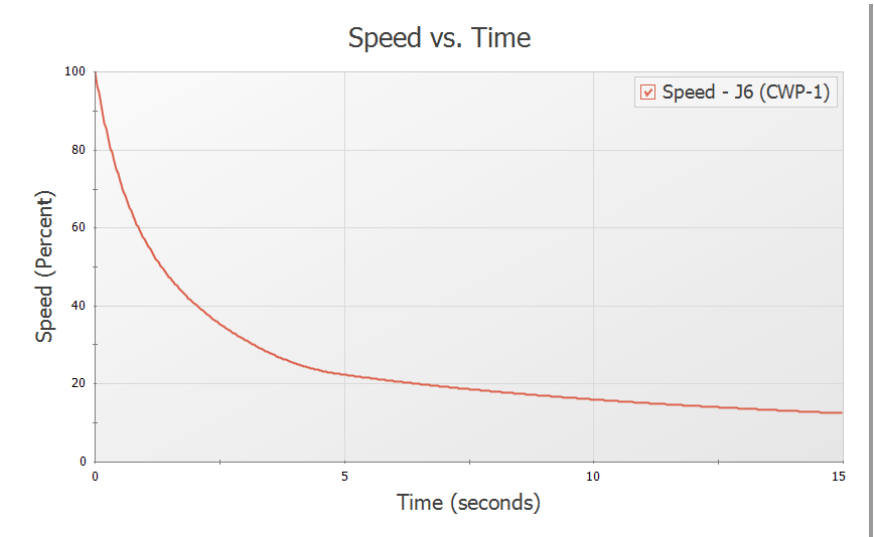
You can graph certain junction properties for junctions in your model. To generate these graphs, the junction output data will need to be saved to file. You can specify which junctions you wish to save output data for by checking the box next to the junctions of interest in the Output Junctions panel in the Pipe Sectioning and Output group of Analysis Setup.

To graph the pump speed decay,

1. Return to the Graph Control tab on the Quick Access Panel.
2. On the Transient Junction tab, add Pump J6.
3. In the Parameter definition area, choose “Pump Speed” as the parameter to be graphed.

4. Choose “Percent” in the Units column.
5. Click Generate.

The predicted pump speed is shown in Figure 4.4. Click the Add Graph to List button from the Toolbar to save this graph to the Graph List Manager so that it can be regenerated later, and name it “Speed vs. Time”.



**Figure 4.4** The pump speed decay is shown. The speed decays quickly at first, and then slows down after the check valve closes and the flow goes to zero.

## Step 8. Create additional pump scenarios

To check the validity of our results when using the standard pump curve to perform the pump trip with inertia, we should now go back and run the model with the pumps specified using four quadrant data for comparison. We will create three child scenarios, one for the current pump configuration using the Standard Pump Curve, and two using a Four Quadrant Data Set.

To do this, click the "Create Child" button on the Scenario Manager on the Quick Access Panel. Name the child “Standard Pump Curve”. A new scenario will appear below the Base Scenario in the scenario tree.



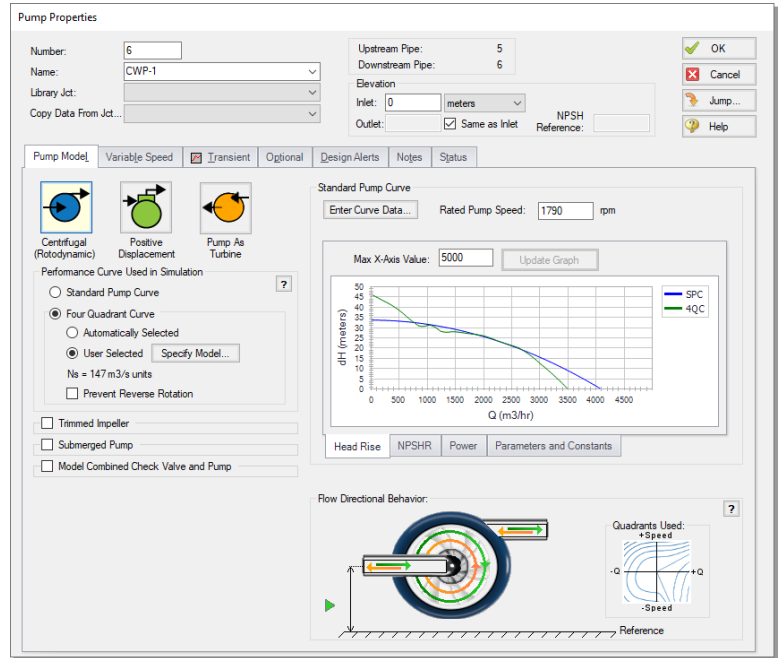
Select the Base Scenario, create another child, and call it “Four Quadrant BEP”. We will later clone this scenario to create our third child scenario.

The “Standard Pump Curve” Scenario will use the pump settings that were already defined, so no changes will be needed for this scenario.

### **A. Four Quadrant BEP**

Load the “Four Quadrant BEP” scenario by double-clicking the name in the Scenario Manager. Open the Pump Properties window for pump J6.

1. Select the option for “Four Quadrant Curve”. (Figure 4.5)
2. Under the “Four Quadrant Curve” option click “Specify Model...”
3. AFT Impulse provides multiple pre-defined Four Quadrant Data sets which are characterized by specific speed. To choose a data set to use, we will first estimate the specific speed for the pump, which AFT Impulse will use to recommend a data set. To do this, click “Suggest a Data Set” under “Four Quadrant Data Set”.
4. By default, AFT Impulse will select "Calculated from Standard Pump Curve", which allows most of the data to be automatically populated from the Pump Properties window. In this case the predefined data is accurate for the pump, so click Calculate.



**Figure 4.5 The Pump Model tab defined using the Four Quadrant Curve. The Flow Directional Behavior section shows the pump behavior for different flow and speed cases.**

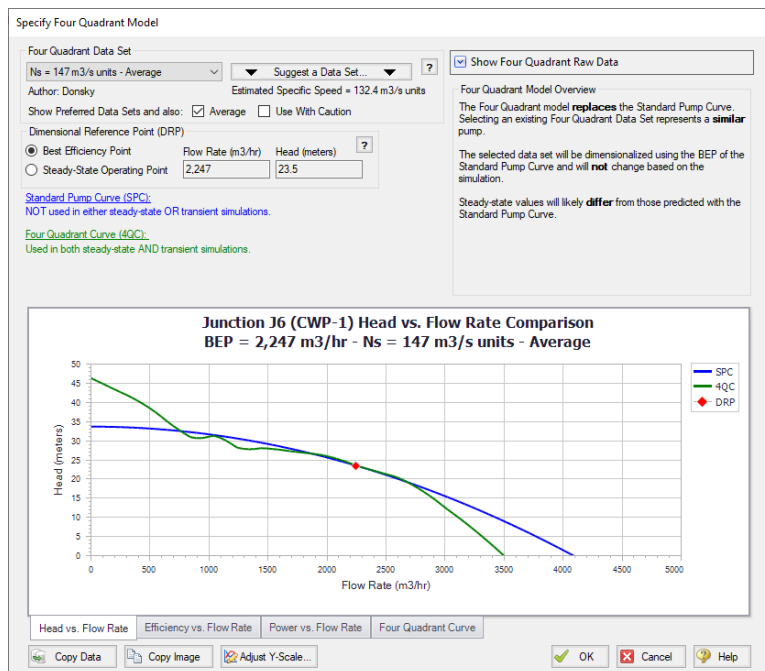
5. The specific speed is estimated to be 132.4 m³/s units, suggested data set  $N_s = 147$  m³/s units. Click "Suggested Data Set and Close". The  $N_s = 147$  m³/s units data set will now be selected, and the dimensionalized curve will be displayed against the previously entered Standard Pump Curve. (Figure 4.6)

---

**Note:** It is always advised to select a "Preferred" data set close to the pump's specific speed when possible. An "Average" data set may be selected if there are no close "Preferred" data sets, as was done here.

---

6. Make sure that the Four Quadrant Curve Dimensional Reference Point is set to use the Best Efficiency Point. We will select the Steady-State Operating Point option later for the third scenario. Click OK on this window and the Pump Properties window.
7. Repeat steps 1 through 6 for junction J8. The required input for this scenario is now complete.



**Figure 4.6** The Specify Four Quadrant Model window can be used to select and view options for the dimensionalized four quadrant data set

## B. Four Quadrant SSOP

Clone the “Four Quadrant BEP” scenario by right-clicking the scenario name in the Scenario Manager and choosing “Clone Without Children”. Name this cloned scenario “Four Quadrant SSOP”. For pump J6 and J8, open the Pump Properties window and click the Specify Model... button under Four Quadrant Curve. Change the Four Quadrant Curve Dimensional Reference Point to Steady-State Operating Point, then click OK on all open windows to accept the changes. The required input for this scenario is now complete.

## Step 9. Run the four quadrant scenarios

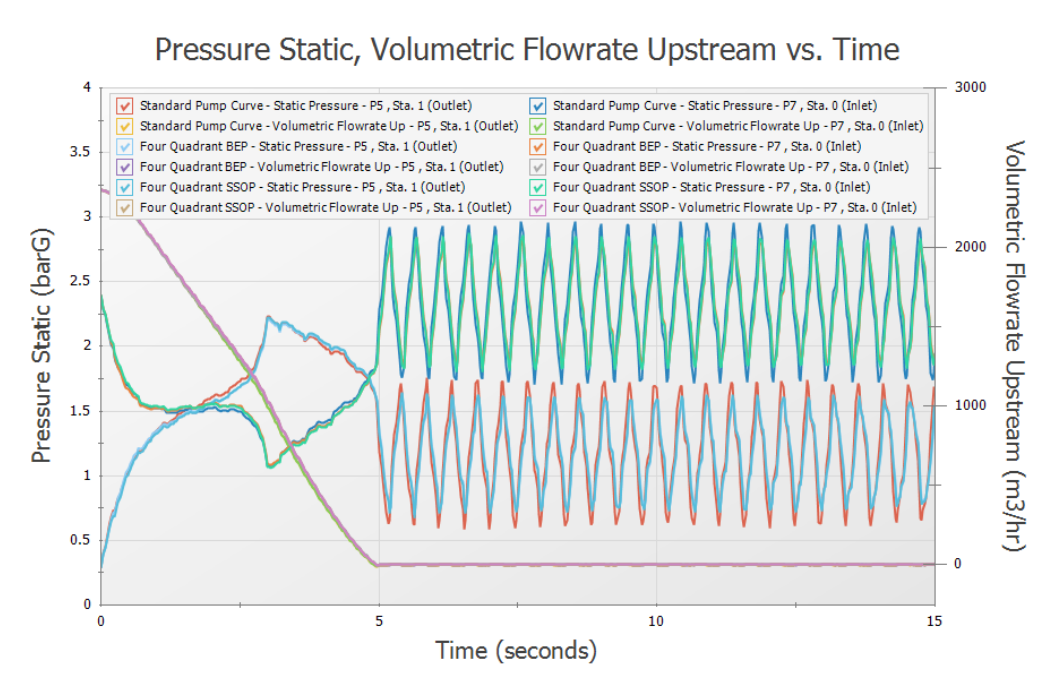
Using the Scenario Manager load each of the four quadrant scenarios and run them. Once all of the scenarios have been run, go to the Graph Results tab. Multi-scenario graphing can be used to compare the results

for all of these scenarios. To create the multi-scenario graph, do the following.

1. Double-click the “Pressure & Flowrate vs. Time” graph in the Graph List Manager on the Quick Access Panel. This will load the saved graph for the current scenario.
2. Select the Multi-Scenario button in the Parameters section of the Quick Access Panel. Check the box next to each of the child scenarios so that they will be included in the multi-scenario graph.
3. Click OK to confirm your selections, then click Generate to update the graph to include the additional scenarios. (Figure 4.7)

As shown in Figure 4.7, the colored curves of the Standard Pump Curve scenario and the two four quadrant scenarios produce results that are practically indistinguishable.

Comparison between the four quadrant graphs and standard pump curve graph reveals that the results are similar, though there is a visible difference of about 0.1 barG between the maximum and minimum points in the four quadrant scenario graphs and the standard pump curve scenario graph after the discharge valves close fully at five seconds. These differences can be attributed to the larger flow oscillations that occur when the pump head is fixed to the zero flow head for all values of reverse flow. Overall the effects of reverse flow and exceeding the end of curve flow are minor in the standard pump curve scenario.



**Figure 4.7** Valve discharge pressure, pump suction pressure, and volumetric flow rate for the Standard Pump Curve, Four Quadrant BEP, and Four Quadrant SSOP scenarios

Now navigate to the Output window for each of the scenarios, then select the Pump Summary tab in the top third section. Notice that for the Four Quadrant SSOP and standard pump curve scenarios, the steady state flow rate is about 2,362 m³/hr, while the steady state flow rate for the BEP scenario is 2,365 m³/hr. This difference in the Four Quadrant BEP scenario results from the fact that the pump is not operating exactly at BEP, as is the case for most pumps.

While the difference in steady state and transient results for the two dimensional points is minor in this case, it may become more pronounced in other cases, and may lead to a trade-off between the accuracy of steady-state and transient data, especially if the pump operates further from BEP. Further details on this topic can be found in the main Help file for AFT Impulse.

## Step 10. Adjust the valve closure

Now, consider the case where the valves at the discharge of the pumps cannot be closed in five seconds, but instead will close fully at 12 seconds. In this case the reverse flow at the pump will be significant due to the delay in the valve closure profile. We will need to use a four quadrant data set to analyze the effects of the possible reverse flow and reverse rotation in the pump, since the Standard Pump Curve does not have sufficient data. We will use the default selection to use the pump BEP as the dimensional reference point, though using the steady state operating point as the reference point would produce similar results.

In the Scenario Manager, select the Four Quadrant BEP scenario and create a child scenario named “12 Sec Valve Closure”. The new child scenario should now be active. Pumps J6 and J8 will already inherit the four quadrant model from the Four Quadrant BEP scenario, but we will need to adjust the valves. For Valves J7 and J9 open the Valve Properties window, then edit the Transient Data table found on the Transient tab to reflect the following closure profile. This scenario should now be complete.

Time (seconds)	Cv
0	10,000
5	1,000
12	0
30	0

## Step 11. Specify the Transient Control group

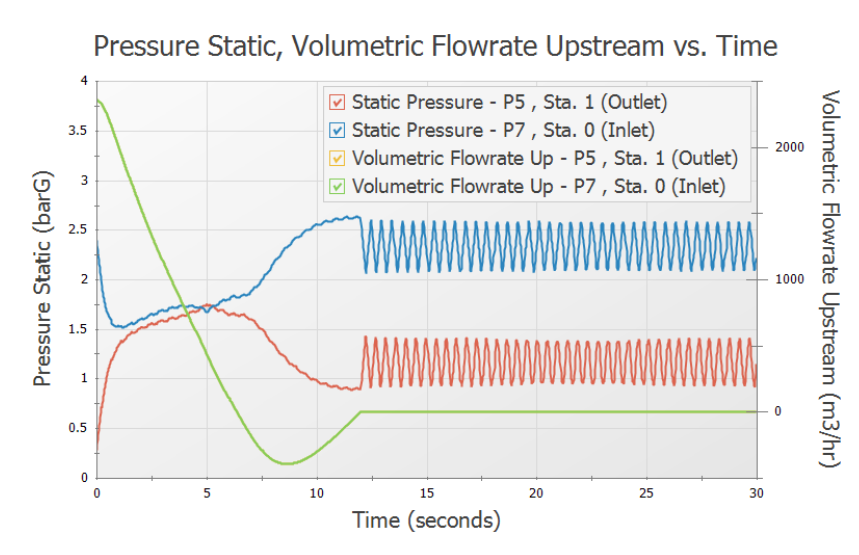
Since the valve closure takes a longer time, it will be useful to extend the run time. Open the Simulation Mode/Duration panel from the toolbar or Analysis menu and change the stop time to 30 seconds, then click OK.

## Step 12. Run the model

The model should now be fully defined and ready to run. Select “Run Model” in the Analysis menu. View the results by pressing “Graph Results” at the bottom of the Solution Progress window.

## Step 13. Graph results

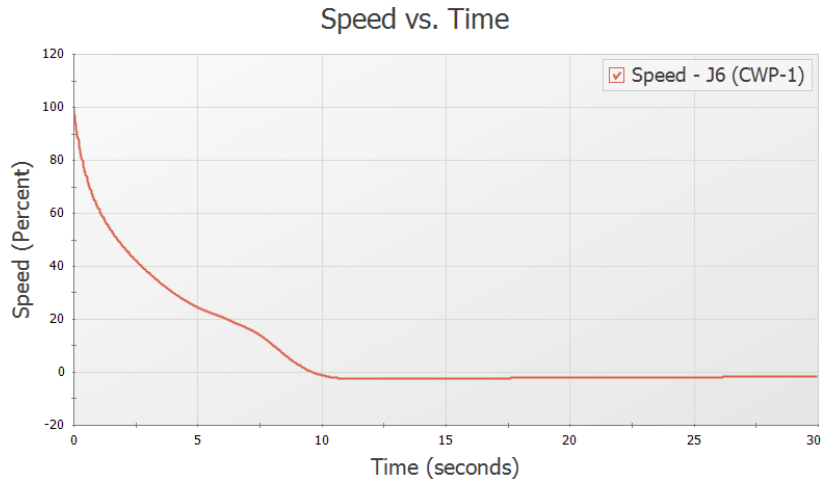
Load the “Pressure & Flowrate vs. Time” graph from the Graph List Manager, as seen in Figure 4.8. Note that as expected, a significant amount of reverse flow occurs at the pump from about 6 to 12 seconds, requiring data beyond the standard data which is typically used for forward flow with forward rotation at the pump. The transient results show that the maximum pressures at the valve outlet and the pump suction have been reduced.



**Figure 4.8** Transient pump suction and discharge pressures and volumetric flowrates with a 12 second valve closure

The pump speed decay is shown in Figure 4.9. Note how the pump speed decreases at a faster rate than Figure 4.4. Additionally, at about 10 seconds the pump begins to experience reverse rotation. This means that from 6 to 10 seconds, the pump will have forward rotation with reverse

flow, then from 10 to 12 seconds, the pump will experience reverse flow with reverse rotation. Both of these situations can be accounted for using the additional information provided by the Four Quadrant Curve.



**Figure 4.9** Pump Speed decay when the valves are modified to close fully at 12 seconds

## Conclusion

For the valve closure of five seconds a minimal amount of reverse flow was experienced at the pump. As a result, the standard pump curve was sufficient, since use of four quadrant data to accurately model this reverse flow had only a small impact on the results. Additionally, the choices of using the BEP or Steady State Operating Point to dimensionalize the four quadrant data were shown to be very similar in the transient and the steady state operating point, since the pump was operating close to its BEP. This is not always the case, especially when the pump operates far from BEP. For an example which shows larger sensitivity to the selection of four quadrant data set and choice of dimensional reference point see the “Selecting a Pump Four Quadrant Curve” example accessed from “Show Examples...” in the Help menu.

Changing the valve to close fully at 12 seconds resulted in significant reverse flow at the pump which required four quadrant data to be used.



It was then found that increasing the valve closure time reduced pressures at the valve and pump.

# Valve Closure With Force Sets Example

This example examines a gravity drain system which transfers water from one tank to another through a filter and a valve in the connecting pipeline. It is necessary to determine the transient forces that occur when the valve is partially closed in order to analyze pipe stresses and loads on the pipe supports.

## Topics covered

- Defining force sets
- Graphing transient forces
- Evaluating the effect of frictional losses on transient pipe forces
- Using Isometric Pipe Drawing Mode

## Required knowledge

These examples assume that the user has some familiarity with AFT Impulse such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Valve Closure Example in Chapter 2 for more information on these topics.

### Model file

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

- *Metric - Valve Closure with Pipe Forces - Initial.imp*

The model file *Metric - Valve Closure with Pipe Forces.imp* contains the completed model, for reference if needed.

### Step 1. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

### Step 2. Open model file

For this example, we will be starting from a pre-built model file which has the fluid, pipes, and junction properties completed to save time. From either the startup screen or the File menu, browse to the model file name shown above. The workspace should appear as shown in Figure 5.1.

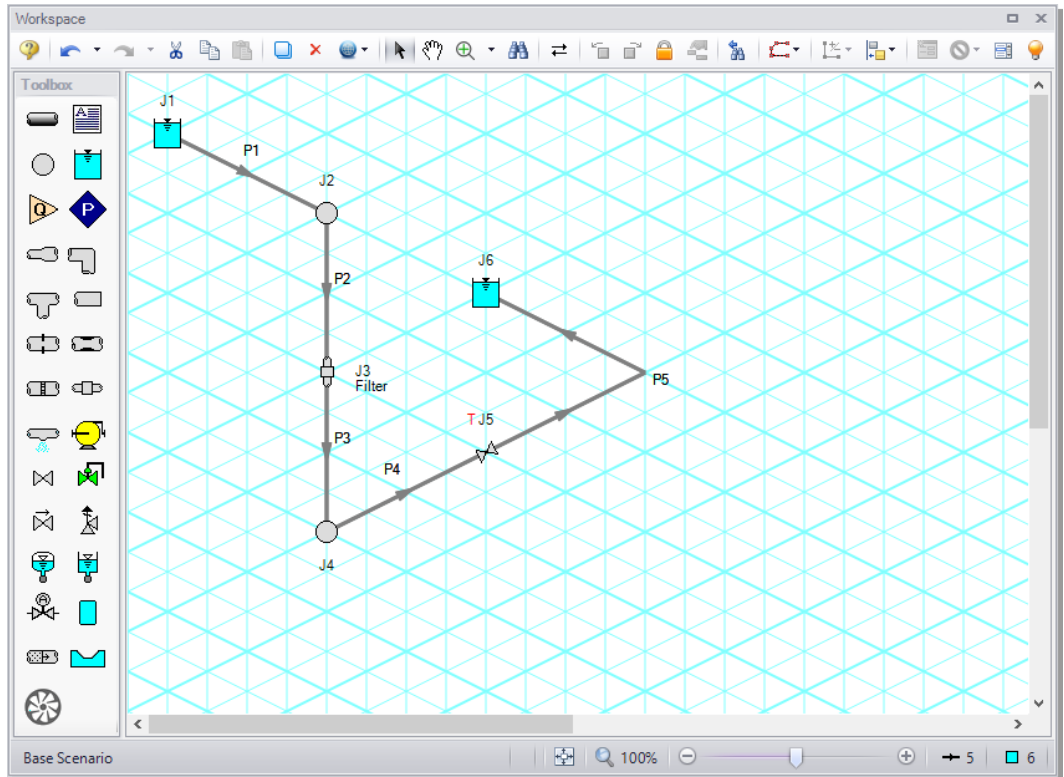
#### A. Isometric Drawing Mode

The previous examples' models were drawn using the default Pipe Drawing Mode, 2D Freeform. However, AFT Impulse has two additional available drawing modes, 2D Orthogonal and Isometric. Due to the nature of this model the Isometric mode was used to visually interpret the pipe layout and provide a better understanding of the system when analyzing calculated forces.

For future reference when using this feature, the Isometric drawing mode can be turned on in the Arrange menu under Pipe Drawing Mode. The isometric grid can also be turned on or off from the Arrange menu. When drawing segmented pipes such as P5, a red-dashed preview line will show how the pipe will be drawn on the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse. 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.

For junctions such as J5, it may be useful to rotate the icon to be aligned with the isometric grid. This can be accomplished by right-clicking on the junction and choosing “Customize Icon”.



**Figure 5.1** Layout of gravity drain system with isometric grid on

**Note:** Pipe forces are typically calculated around pairs of pipe direction changes or at single points where the pipe is interrupted (for example, at an untied or non-pressure compensated expansion joint). These locations must be determined from piping drawings showing the physical arrangement of the piping, then related to the corresponding pipe in the AFT Impulse model along with the length along the pipe where the direction change or pipe interruption is located. Intermediate elevations are not necessary but are included here for later discussion.

Step 3. Specify the Pipe Sectioning and Output group

Open Analysis Setup and go to the Sectioning panel. When this panel is first opened, it will automatically search for the best (least number of sections within 10% variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. No input is required for this step.

**Note:** The closer the computing stations are to the desired force calculation location, the more accurate the force calculation will be.

Step 4. Specify the Transient Control group

Go to the Simulation Mode/Duration panel in the Transient Control group and enter a Stop Time of 10 seconds.

Step 5. Define the Forces group

Go to the Forces Definitions panel in the Forces group. Define a force set by clicking New, then entering the data as shown in Figure 5.2. Repeat this process for the second force set.

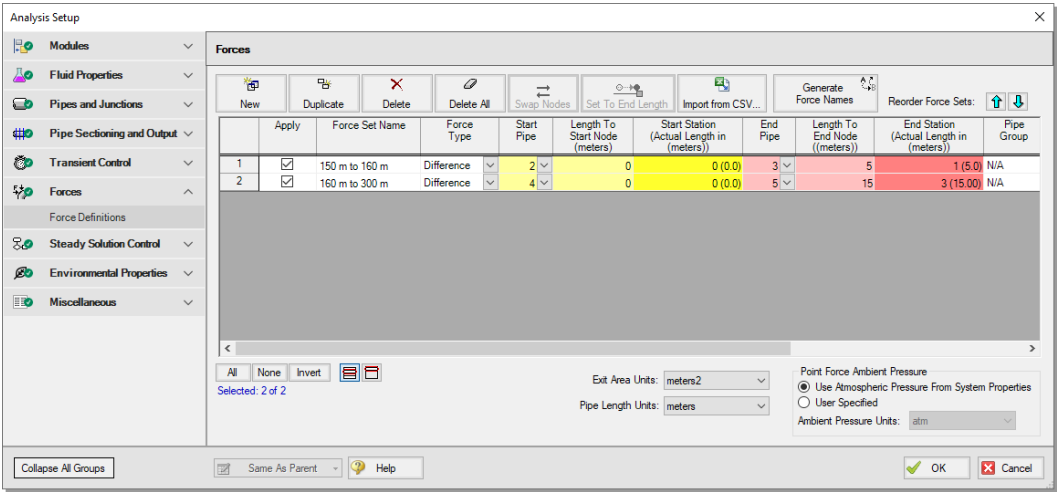


Figure 5.2 The completed Force Definitions panel

Since the force sets in this example calculate the force imbalance between adjacent pipe direction changes, the default force type of ‘Difference’ is used. The ‘Point’ type would be used for calculating the force at a location where the fluid is leaving the system, such as an exit valve. The additional difference methods are available to analyze forces across more than two pipes, or for cases where a user defined exit area would be useful, such as for forces across a nozzle.

For a specified length to start node and length to end node, AFT Impulse will determine and display the nearest computing station number and actual length from inlet of the start and end pipe to the node within the pipe.

Sectioning has produced one section each in pipes P2 and P3 for the 150 m to 160 m force set and then 25 sections and 6 sections in pipes P4 and P5, respectively for the 160 m to 300 m force set. Section length is 5 meters for each pipe.

In our example, this sectioning conveniently results in computing stations at the locations where we want to calculate forces. Where computing stations do not coincide with the desired force calculation locations, some loss in accuracy will occur. By increasing the number of sections in the controlling pipe during sectioning, a greater number of computing stations will exist, thus reducing the distance to the desired force calculation locations, at the expense of longer run time.

---

**Note:** AFT Impulse will set Pipe Station Output automatically to include only those stations required by the force set nodes up to five stations, or all stations if more than five stations are required.

---

Click OK to close Analysis Setup.

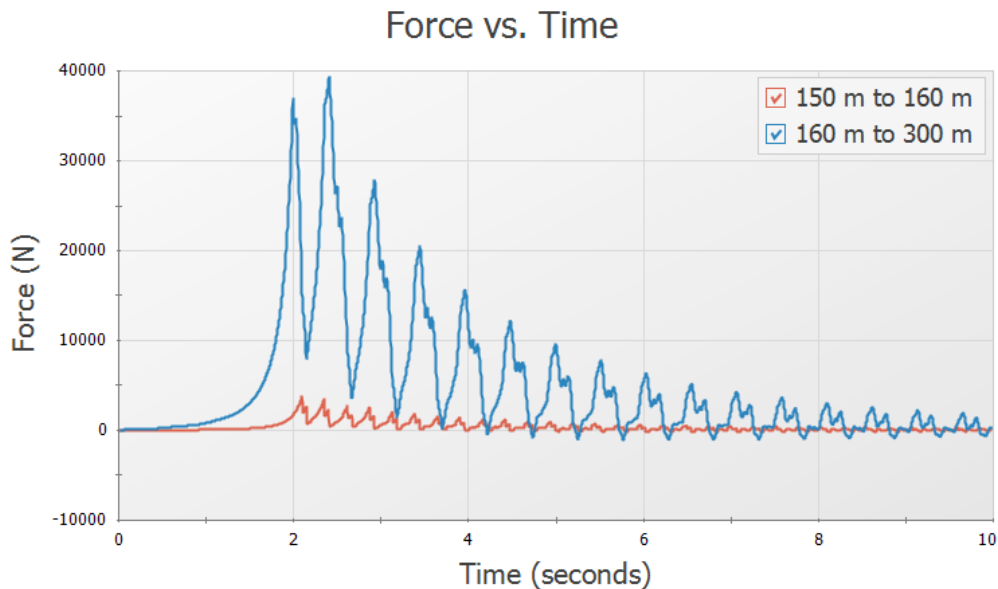
## **Step 6. Run the model**

All groups in Analysis Setup should now be complete and the model ready to run. Run the model by pressing the “Run Model” icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. This will open the Solution Progress window. When complete, press the “Output” button.

## Step 7. Graph the pipe forces

Go to the Graph Results window and select the Forces tab from the Graph Parameters section on the Quick Access Panel. The force sets defined on the Force Definitions panel will appear as available forces to graph.

Check the box next to the "150 m to 160 m" force set. This corresponds to the elevation change between the branch junctions of 10 meters and 0 meters specified from pipes P2 and P3. Also check the box next to the "160 m to 300 m" force set, which corresponds to the force over the closing valve. Click Generate, and the following force vs time graph is displayed (Figure 5.3).



**Figure 5.3 Force vs. Time for both force sets**

Note that at time zero, which represents the initial, steady state results, there is no force imbalance. This is the expected results for a system at steady state.

Some traditional methods of analyzing force sets will not have this same result, since they do not include the effects of friction or momentum in

their force balances. If the graph from Figure 5.3 created with friction and momentum ignored, the steady state values calculated as approximately -5,100 N and -11,940 N. Due to this effect, AFT Impulse will always include both friction and momentum when graphing force sets.

In summary, there are two important points to be observed here:

1. AFT Impulse calculates transient, or time varying hydraulic forces. This does not include constant loads from fluid, piping and component weight. A comprehensive analysis of pipe loading must separately include these items.
2. In some cases, ignoring friction and momentum force balance components will result in force imbalances that do not exist in reality, since the frictional forces on the pipe exactly counterbalance the force calculated from the pressure differential at the selected locations.

### **A. Final notes**

An AFT Impulse model does not contain directional data with regard to the forces, it knows only pipe length and elevation. Since forces are vectors with both magnitude and direction, the user must identify the direction of the calculated forces using data in the pipe arrangement drawing that defines the geometry of the pipe routing.

## **Analysis summary**

In this example it can be seen that AFT Impulse is capable of predicting transient hydraulic forces which occur in the system due to surge. Including frictional results is important to avoid force imbalance in the steady state. With the data that is obtained from the force sets in AFT Impulse the engineer can then export this information to CAESAR II, ROHR2, TRIFLEX or other pipe stress analysis software for further analysis on the impact of forces on the structural support.





# AFT Impulse Add-on Modules Examples

This chapter covers the Settling Slurry (SSL) and Pulsation Frequency Analysis (PFA) modules. The user can only perform these examples if access to the relevant modules is available.

## Topics covered

- Defining settling slurry properties
- Reviewing slurry output
- “Ringing” systems to find natural acoustic frequencies
- Finding system excitation frequencies
- Calculating pressure response of system at ‘worst case’ positive displacement pump speeds

## Required knowledge

These examples assume that the user has some familiarity with AFT Impulse such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to Valve Closure Example in Chapter 2 for more information on these topics.

## Model files

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

- *Metric - Pipe Sizing for Sand Transfer - SSL.imp*
- *Metric - PD Pulsation Study - PFA.imp*

## SSL problem statement

Sand is being transferred from a quarry sand pit to receiving deposits in two different locations. There are two valves that control the flow to each of these deposits by changing their opening position (modeled as Cv). The ratio of the mixture velocity to the velocity at which the sand will settle out of the liquid and form a stationary bed on the bottom of the pipe ( $V_m/V_{sm}$ ) must be kept above 1.2 to avoid potentially plugging the pipe. Determine if the pipe size and velocity ratio is acceptable during the transient where one valve starts to close while the other opens further.

## SSL Step 1. Start AFT Impulse

From the Start menu, choose AFT Products and AFT Impulse. On the Startup window, select to activate the SSL module. If AFT Impulse is already running, activate the SSL module from the Modules panel in Analysis Setup.

## SSL Step 2. Define the Fluid Properties group

With the SSL module enabled, the four items in the Fluid Properties group in Analysis setup change to the three items required for defining a slurry: the Carrier Fluid panel, Solids Definition panel, and the Slurry Definition panel. Open Analysis Setup to define these items.

### A. Carrier Fluid panel

1. Go to the Carrier Fluid panel (Figure 6.1)
2. Carrier Fluid = Basic (Water)
3. Temperature = 21 deg. C

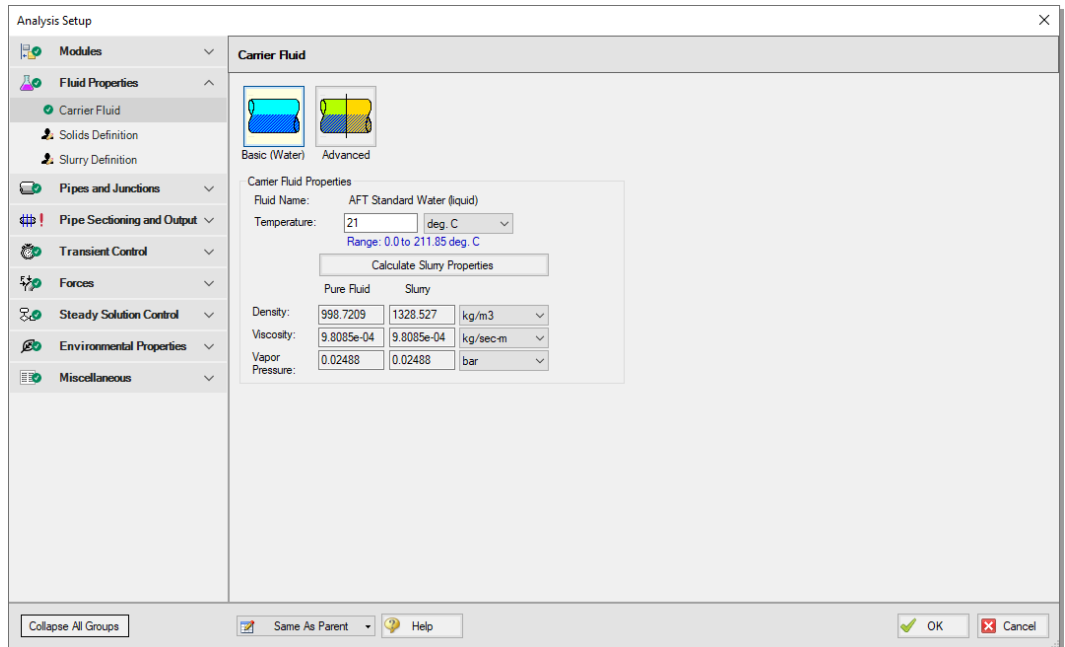
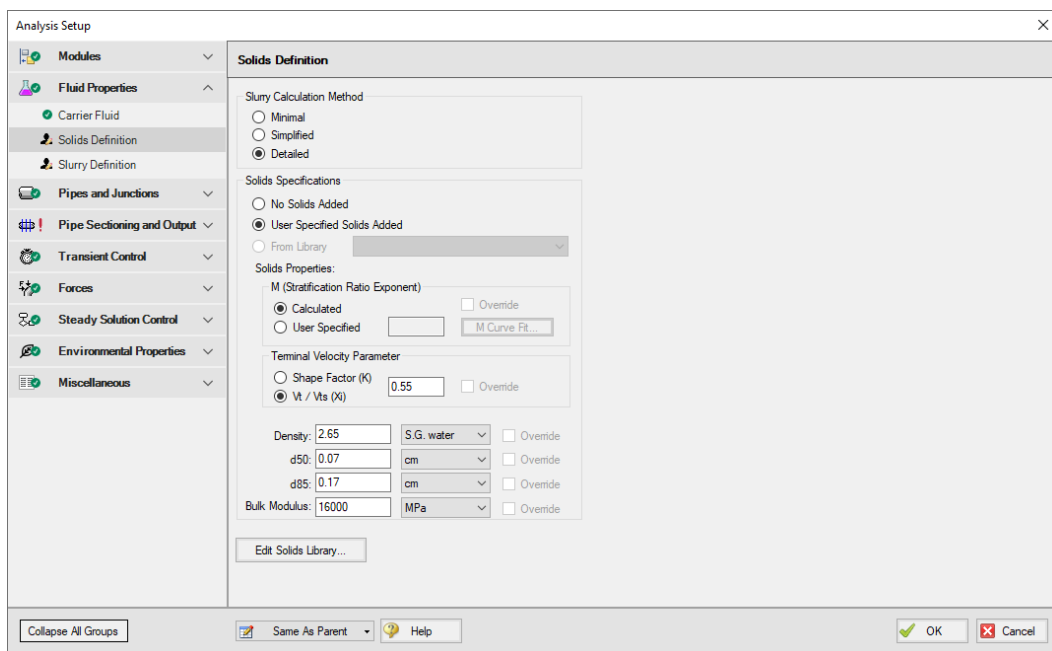


Figure 6.1 Completed Carrier Fluid panel

### B. Solids Definition panel

1. Go to the Solids Definition panel (Figure 6.2)
2. Slurry Calculation Method = Detailed
3. Solids Specifications = User Specified Solids Added
4. Terminal Velocity Parameter =  $V_t/V_{ts}$
5. Value = 0.55
6. Density = 2.65 S.G. water
7.  $d_{50} = 0.07$  cm

8.  $d_{85} = 0.17 \text{ cm}$
9. Bulk Modulus = 16000 MPa



**Figure 6.2 Completed Solids Definition panel**

### C. Slurry Definition panel

1. Go to the Slurry Definition panel (Figure 6.3)
2. Concentration Type = Volume Fraction
3. Amount Solids Added = 20 Percent
4. Click OK to save and close Analysis Setup

Analysis Setup

**Modules**

- Fluid Properties
  - Carrier Fluid
  - Solids Definition
  - Slurry Definition**
- Pipes and Junctions
- Pipe Sectioning and Output
- Transient Control
- Forces
- Steady Solution Control
- Environmental Properties
- Miscellaneous

**Slurry Definition**

Slurry Calculation Method

☐ Minimal  
☐ Simplified  
☒ Detailed

Slurry Definition

Concentration Type: Volume Fraction

Amount Solids Added: 20 Percent

Slurry Model: Settling Slurry - Wilson, Addie, Clift

Sliding Friction Coefficient (µs):

Bed Concentration: Percent

Name Alias: Water slurry Reset

	Pure Fluid	Slurry	
Density:	998.7209	1328.527	kg/m3
Viscosity:	9.8085e-04	9.8085e-04	kg/sec-m
Vapor Pressure:	0.02488	0.02488	bar
Bulk Modulus:	2.2121e+04	2.6728e+04	bar

Deposition Velocity (Vsm)

Nomograph - Equation Limited meters/sec

Edit Solids Library...

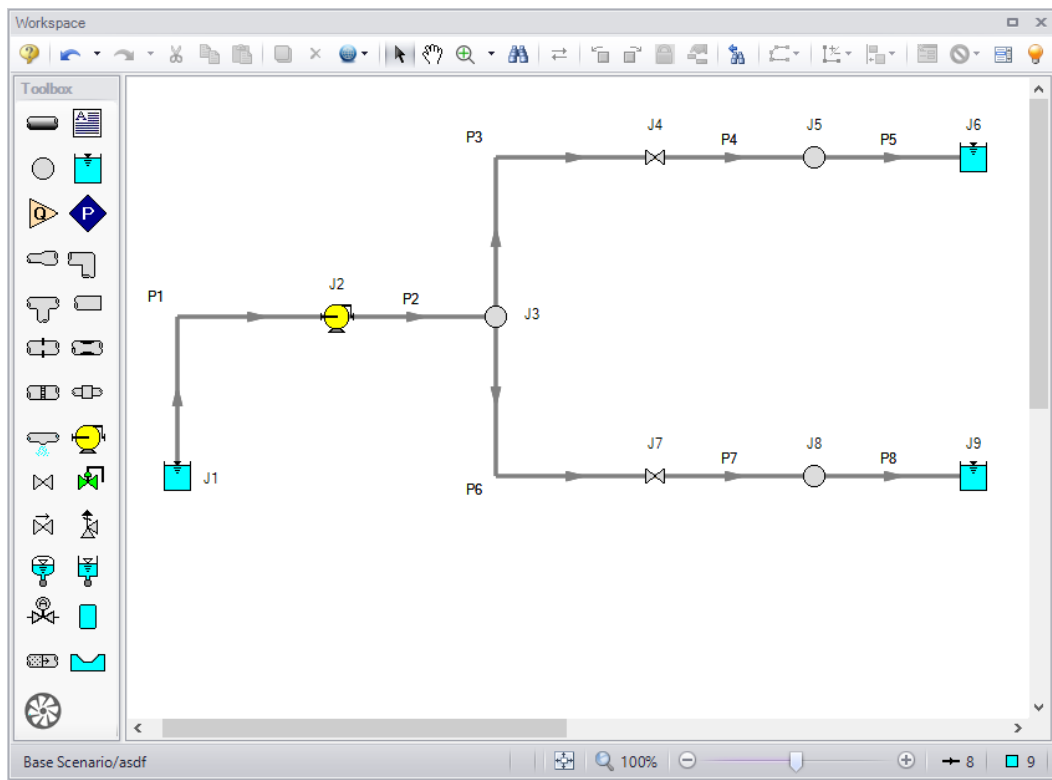
Collapse All Groups Same As Parent Help OK Cancel

Figure 6.3 Completed Slurry Definition panel

## SSL Step 3. Define the Pipes and Junctions group

### A. Place the pipes and junctions

In the Workspace window, assemble the model as shown in Figure 6.4.



**Figure 6.4** Layout of pipe system for Sand Transfer to Two Locations example

## B. Enter the pipe and junction data

The system is in place, but now the input data for the pipes and junctions need to be entered. Double-click each pipe and junction and enter the following data in each Pipe and Junction Properties window.

### *Pipes*

All pipes are Steel - ANSI, 10 inch Size, STD Type, Standard Friction Model, Thick-Walled Anchored Upstream Pipe Support, with the following lengths:

Pipe	Length (meters)
1	3
2	122
3	1.5
4	26
5	122
6	1.5
7	30.5
8	213

***J1 - Reservoir***

1. Name = Sand Pit
2. Liquid Surface Elevation = 0 meters
3. Liquid Surface Pressure = 0 barG
4. Pipe Depth = 0 meters

***J3 - Branch***

Elevation = 1.2 meters

***J5 - Branch***

Elevation = 6 meters

***J8 - Branch***

Elevation = 9 meters

***J6 - Reservoir***

1. Name = Deposit #1
2. Liquid Surface Elevation = 6 meters



3. Liquid Surface Pressure = 0 barG
4. Pipe Depth = 0 meters

**J9 - Reservoir**

1. Name = Deposit #2
2. Liquid Surface Elevation = 9 meters
3. Liquid Surface Pressure = 0 barG
4. Pipe Depth = 0 meters

**J2 - Pump**

1. Name = Main Slurry Pump
2. Inlet Elevation = 1.2 meters
3. Pump Model = Centrifugal (Rotodynamic)
4. Enter Curve Data =

<b>Volumetric</b>	<b>Head</b>	<b>Power</b>
<b>m3/hr</b>	<b>meters</b>	<b>kW</b>
0	152	764
910	145	820
1595	122	894
2050	91	1006

5. Curve Fit Order = 2

**J4 - Valve**

1. Name = Valve to Deposit #1
2. Inlet Elevation = 1.2 meters
3.  $C_v$  = 500

4. Click the Transient tab
5. Transient Data =

Time (seconds)	Cv
0	500
1	500
6	300
10	300

### **J7 - Valve**

1. Name = Valve to Deposit #2
2. Elevation = 1.2 meters
3. Cv = 400
4. Click the Transient tab
5. Transient Data =

Time (seconds)	Cv
0	400
5	500
10	500

### **C. Check if the pipe and junction data is complete**

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

## SSL Step 4. Specify the Pipe Sectioning and Output group

Open Analysis Setup and go to the Sectioning panel. When this panel is first opened, it will automatically search for the best (least number of sections within 10% variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. No input is required for this step.

## SSL Step 5. Specify the Transient Control group

Go to the Simulation Mode/Duration panel in the Transient Control group and enter a Stop Time of 10 seconds.

## SSL Step 6. Run the model

All groups in Analysis Setup should now be complete and the model ready to run. Run the model by pressing the “Run Model” icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. This will open the Solution Progress window. When complete, press the “Output” button.

## SSL Step 7. Review results

Click “Output” on the Solution Progress window will take you to the Output window.

### A. Check the Transient Max/Min

As a design objective, the system is required to have at minimum a  $V_m/V_{sm}$  of 1.2 to avoid the potential of the sand falling out of the slurry and plugging the pipe.

By inspecting the Min.  $V_m/V_{sm}$  column in the Transient Max/Min tab, as shown in Figure 6.5, it can be seen that the minimum  $V_m/V_{sm}$  drops below this value in pipe P4.

**Note:** A Design Alert could be defined in the Tools menu to automatically evaluate if this design objective is met, and issue a warning if the condition is not satisfied. A predefined Design Alert for this model can be seen by opening the *Metric - Pipe Sizing for Sand Transfer - SSL.imp* model file which is installed in the Examples folder in the AFT Impulse 9 folder.

<span>⌵</span> Pipes Transient Output Transient Max/Min Slurry						
Pipe	Max. Vm/Vsm	Max. Vm/Vsm Time (seconds)	Max. Vm/Vsm Station	Min. Vm/Vsm	Min. Vm/Vsm Time (seconds)	Min. Vm/Vsm Station
1	2.661	1.345	0	2.487	7.438	0
2	3.361	1.342	0	3.142	7.436	0
3	1.919	1.231E-03	1	1.275	9.994	1
4	1.707	0.01970	15	1.134	9.999	14
5	1.919	0.1084	71	1.275	9.506	81
6	1.882	9.999	0	1.441	1.231E-03	0
7	1.605	9.999	0	1.229	0.02340	20
8	1.882	9.998	0	1.441	0.1859	131

**Figure 6.5 Design objective violation in pipe P4 station 17**

There are several factors that may be contributing to this low Vm/Vsm value. Pipe P4 is sloped upward which increases the settling velocity and decreases the velocity ratio of concern. Also, all pipes in the model have a 10 inch diameter. However, the main Pipes P1 and P2 are carrying much more flow than the others so the pressure drop in these is greater. This can be reduced by increasing the size. The size cannot be increased too much because this would reduce the velocity ratio below the 1.2 minimum.

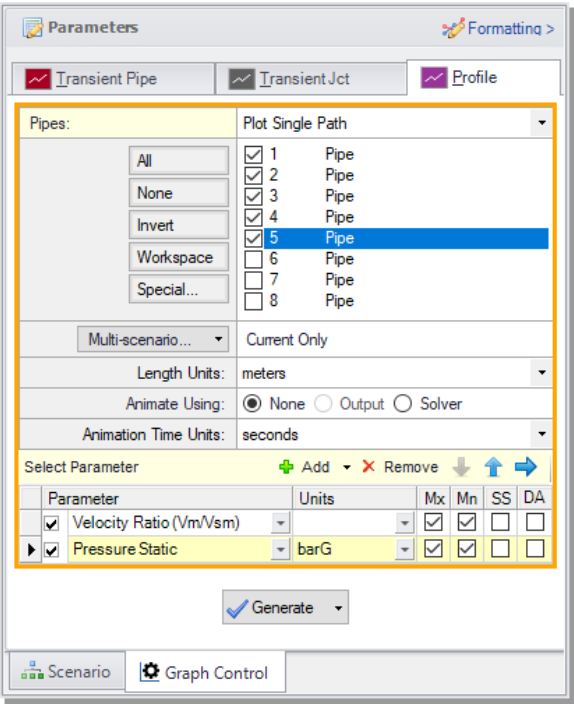
Return to the Workspace and change Pipes P1 and P2 to have a 14 inch diameter. Since the size of the pipes was changed, the pipes need to be re-sectioned by opening the Sectioning panel in the Analysis Setup window. Click OK to accept the results.

Re-run the model and view the Output. All pipes should meet the requirements now that the pipes were changed. The Graph Results window will be more useful in understanding the results.

**B. Graph the velocity ratio and pressures to Deposit #1**

You will use stacked graphs to simultaneously view the maximum and minimum velocity ratio and pressures in the system during the transient.

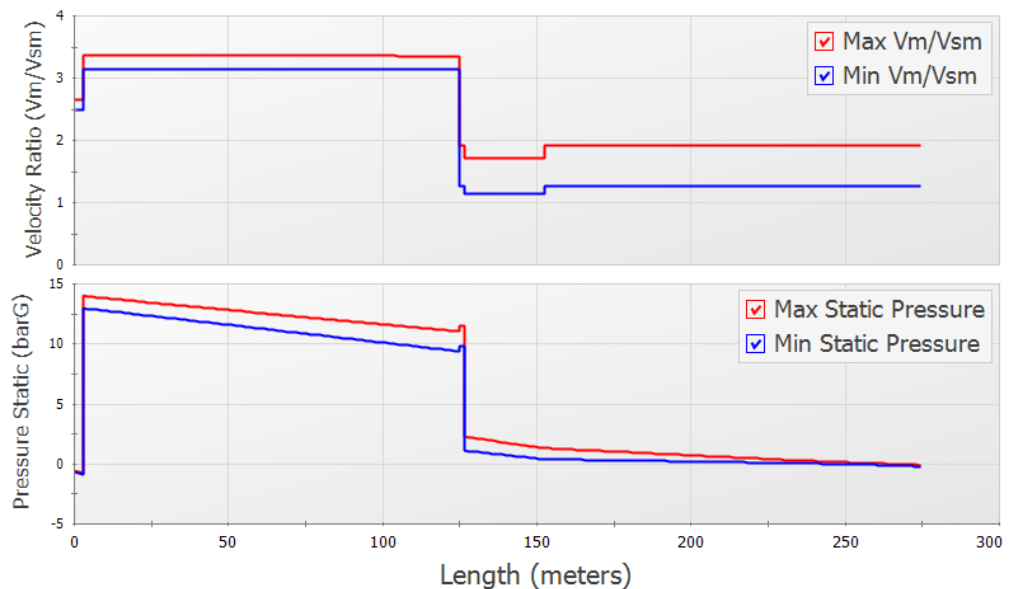
- 1. Change to the Graph Results window from the Window menu, by clicking on the Graph Results tab, or by pressing CTRL+G.
- 2. Open the Profile tab on the Graph Control tab.
- 3. Select pipes P1, P2, P3, P4 and P5 (Figure 6.6). These represent the flow path to Deposit #1.
- 4. In the Parameters definition area, ensure that there are two parameters by selecting the green “+” icon to add a parameter to graph. Choose “Velocity Ratio (Vm/Vsm)” for one parameter and “Pressure Static” for the other.



**Figure 6.6** Data selection to view the maximum and minimum values of the velocity ratio and static pressure along the flow path to Deposit #1

5. Specify “barG” as the Units for the Pressure Static parameter.
6. Make sure both of the boxes under the “Mx” and “Mn” columns for both parameters are checked to ensure that the maximum and minimum values are being displayed for both graphs. If a design alert has been created, this can be added to the plot as well by checking the box under “DA”.
7. Click Generate. The resulting graphs are shown in Figure 6.7.

Recall that, before modifying pipes P1 and P2 to 14 inch pipe, the minimum velocity ratio was below the design alert limit of 1.2 for a portion of the path. Now with 14 inch pipe, the minimum is above the 1.2 limit along the entire path. You can also check the same parameters for the flow path leading to Deposit #2.



**Figure 6.7** Maximum and minimum velocity ratio and static pressure values in the flow path to Deposit #1. Note that the velocity ratio is above the minimum Design Alert of 1.2 at all locations.

C. Graph the velocity ratio over time

A time comparison of the velocity ratio in the critical pipes along the two flow paths is important to consider. To create this graph:

- 1. Return to the Graph Control tab on the Quick Access Panel.
- 2. On the Transient Pipe tab, select Pipes P4 Inlet and P14 Inlet and add them to the pipe stations to graph (Figure 6.8). These are the two sloped pipes which have the lowest velocity ratio.
- 3. In the Parameters area, choose Velocity Ratio ( $V_m/V_{sm}$ ).
- 4. Click Generate.

The resulting graph is shown in Figure 6.9. This shows that, as Valve J4 closes and Valve J14 opens, the velocity ratios in the pipe immediately downstream change but stay in the acceptable range.

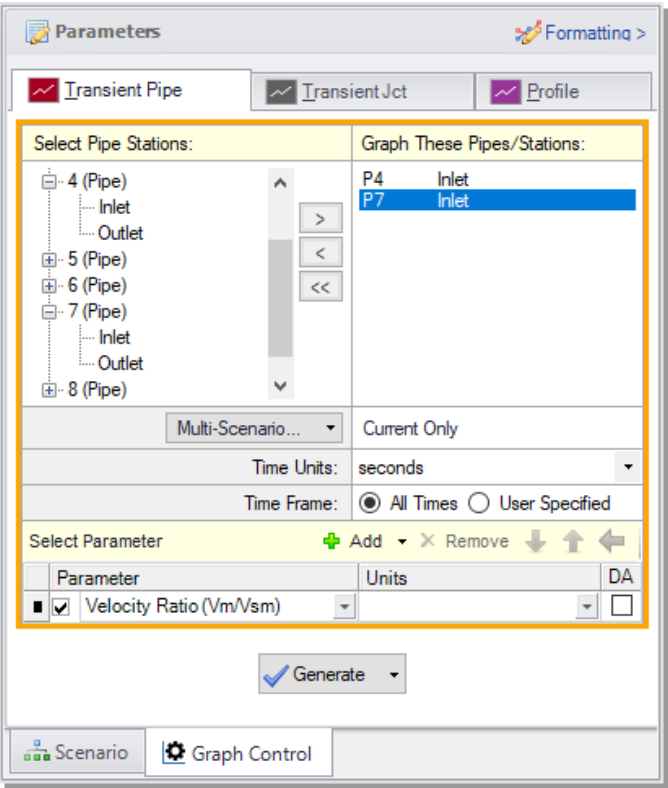
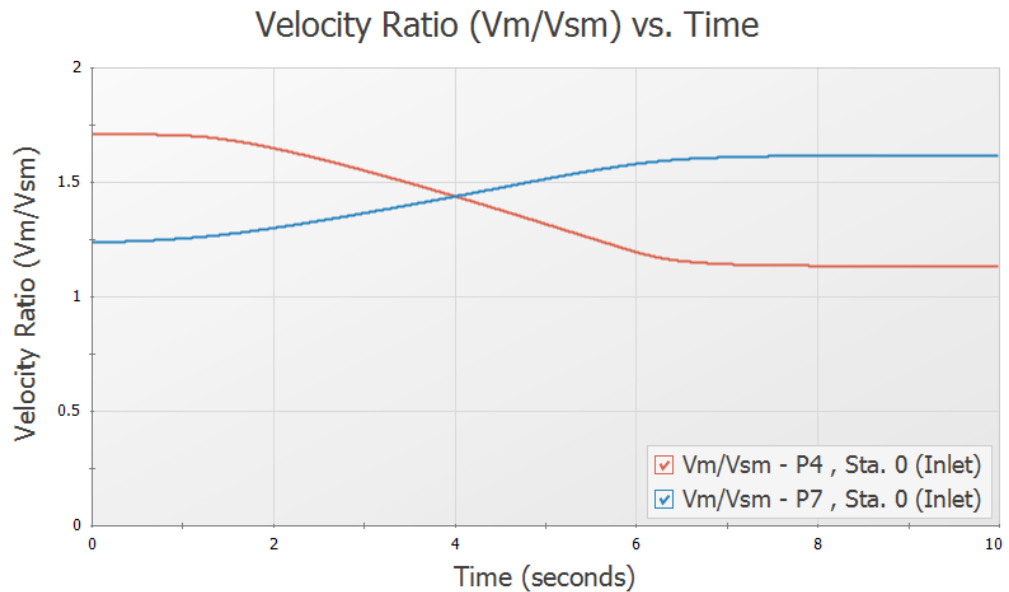


Figure 6.8 Data selection to view the velocity ratio over time



**Figure 6.9** The velocity ratio in the sloped pipes as the valves change position

## SSL analysis summary

A slurry of sand and water flowing to two deposits was modeled. The velocity ratio was examined to ensure a minimum value was maintained throughout the transient to avoid the danger of the sand settling out of the slurry and plugging the pipe. The analysis showed that the initial pipe diameter selected was not adequate. A larger size was selected to meet the system requirements over the simulation time.

## PFA problem statement

A system with positive displacement pumps has a known pulsation issue in a section of piping with a closed valve. This issue is caused by the system's natural frequency resonating with certain PD pump frequencies, resulting in severe pressure oscillations at the closed valve. A forcing function will be placed on the PD Pump to discover the



natural acoustic frequencies, which will then be used to find the worst-case scenarios for the system.

## **PFA Step 1. Start AFT Impulse**

From the Start Menu choose AFT Products and AFT Impulse. From the Startup window under the "Activate Modules" section in the middle panel, select the PFA (Pulsation Frequency Analysis) option to activate the PFA module. If AFT Impulse is already running, activate the PFA module from the Modules panel in Analysis Setup.

## **PFA Step 2. Define the Fluid Properties group**

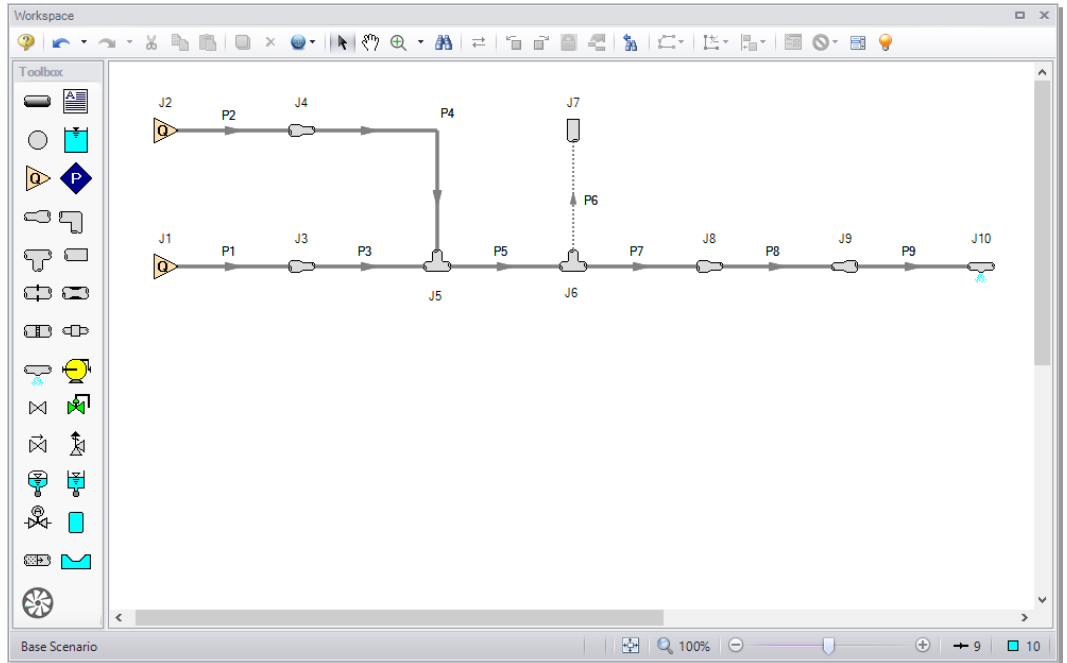
1. Open Analysis Setup using the button on the toolbar or from the Analysis menu.
2. Open the Fluid panel and make sure User Specific Fluid is selected
3. Name = PFA Fluid
4. Density = 0.92 gram/cm<sup>3</sup>
5. Dynamic Viscosity = 0.95 centipoise
6. Bulk Modulus = 9653 bar

## **PFA Step 3. Define the Pipes and Junctions group**

At this point, the first two groups are completed in Analysis Setup. The next group is the Pipes and Junctions group.

### **A. Place the pipes and junctions**

In the Workspace window, assemble the model as shown in Figure 6.10.



**Figure 6.10** Layout of PD Plunger Pulsation Study example

## B. Enter the pipe and junction data

The system is in place, but now the input data for the pipes and junctions need to be entered. Double-click each pipe and junction and enter the following data in each Pipe and Junction Properties window.

### ***Pipes***

All pipes use user-specified material with the following inputs. Throughout the model a User Specified Wavespeed of 915 m/s is used, and the absolute roughness is 0.00152 cm.

Pipe	Inner Diameter (cm)	Length (meters)
1	30.5	1
2	30.5	1
3	5.5	12
4	5.5	17
5	8.3	10
6	4.3	6
7	8.3	15
8	4.3	1
9	8.3	9

***J1 & J2 - Assigned Flows***

For the sake of simplicity, assigned flow junctions are being used to represent the PD pumps with the information listed below. The Pulsation Setup group in Analysis Setup will be used later to place the necessary transient data in the assigned flow junction for the pump being analyzed.

1. Elevation = 0 meters
2. Flow Specification = Volumetric Flow Rate
3. Flow Rate = 8 m<sup>3</sup>/hr

***J3 & J4 - Area Changes***

1. Inlet Elevation = 0 meters
2. Type = Abrupt Transition (Cylindrical)

***J8 & J9 - Area Changes***

1. Inlet Elevation = 0 meters
2. Type = Conical Transition

3. Angle ( $\mu$ ) = 90 Degrees

**J5 & J6 - Tees**

1. Elevation = 0 meters
2. Loss Model = Simple

**J10 - Spray Nozzle**

1. Elevation = 0 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) = 1
7. Discharge Flow Area = 0.148 cm<sup>2</sup>

**J7 - Dead End**

Elevation = 0 meters

**C. Check if the pipe and junction data is complete**

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

**PFA Step 4. Define Pulsation Setup group**

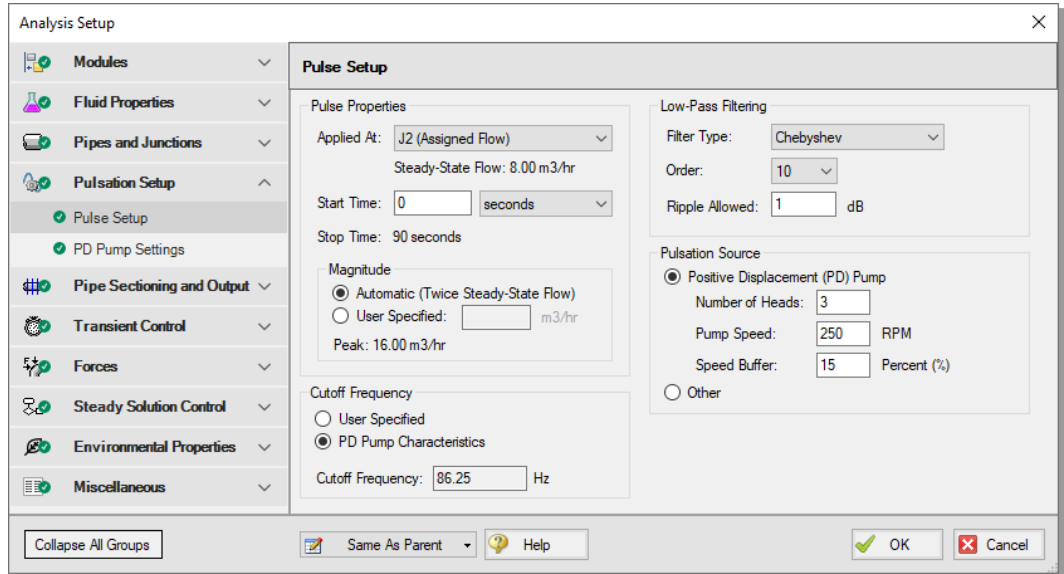
Defining the initial pulse or “ring” is an important step in determining the natural frequencies that different stations in the model will respond to. This is the numerical equivalent of hitting the system with a hammer to see how it responds at different frequencies. This first step does not represent a realistic operating condition, but rather reveals what frequencies of real-world operation may present operational problems.

Open Analysis Setup and go to the Pulse Setup panel in the Pulsation Setup group. The Pulsation Setup group is the main interface for specifying input regarding the initial pulse and other information about the pulsation source. There are two items in the Pulsation Setup group: the Pulse Setup panel and the PD Pump Settings panel. The PD Pump Settings panel only requires input if the source of pulsation is a positive displacement pump. Since the source of pulsation in our model is a positive displacement pump, we will need to complete the input on both of these panels.

### **A. Define the Pulse Setup panel**

Define the Pulse Setup panel (Figure 6.11) using the following values:

1. Applied At = J2 (Assigned Flow)
2. Start Time = 0 seconds
3. Magnitude = Automatic (Twice Steady-State Flow)
4. Peak = 16.00 m<sup>3</sup>/hr
5. Cutoff Frequency = PD Pump Characteristics
6. Value = 86.25 Hz
7. Filter Typer = Chebyshev
8. Order = 10
9. Ripple Allowed = 1 dB
10. Pulsation Source = Positive Displacement (PD) Pump
11. Number of Heads = 3
12. Pump Speed = 250 RPM
13. Speed Buffer = 15 Percent (%)



**Figure 6.11 Completed Pulse Setup panel**

## **B. Define the PD Pump Settings panel**

Define the PD Pump Settings panel (Figure 6.12) using the following values:

1. Minimum Pump Speed = 125 RPM
2. Bore = 7.6 cm
3. Stroke = 5.2 cm
4. Rod Length = 25.4 cm
5. Clearance = 70 Percent (%)
6. Suction Pressure = 4.5 bar
7. Discharge Pressure = 415 bar

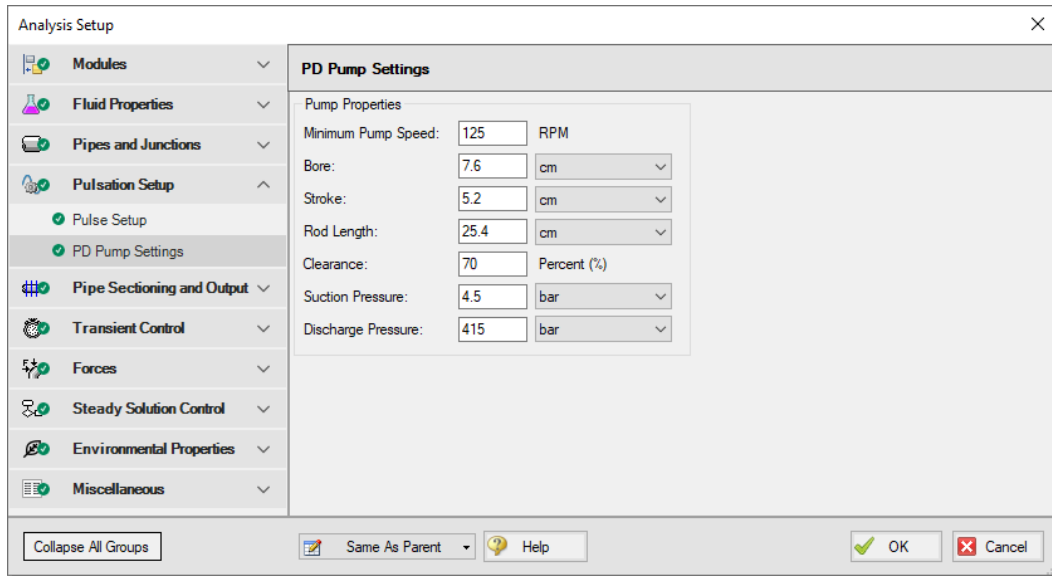


Figure 6.12 Completed PD Pump Settings panel

## PFA Step 5. Section the pipes

Go to the Sectioning panel in the Pipe Sectioning and Output group of Analysis Setup. While using the AFT Impulse PFA module, the pulsation setup needs to be completed prior to sectioning the pipes in the model. To section the model's pipes for the simulation, complete the following:

1. Select the significant pipes that should consider variance. In our example, we will consider the variance in all modeled pipes, except for pipe P8. You can see that this pipe has been excluded in Figure 6.13. Pipe P8 is excluded because it represents a flow meter and not an actual pipe. Units such as flow meters and dampeners may be better modelled with a pipe than with a junction if they have significant length and associated acoustic interaction aspects in the physical system. Ensure that the box next to each pipe is selected under Consider Variance, except for pipe P8.

Selected Sectioning for Model:

Controlling Pipe:  Ideal Number of Sections in Controlling Pipe:  Time Step (sec):

Pipe	Consider Variance	Length (meters)	Wavespeed (meters/sec)	Ideal # of Sections	Integer # of Sections	Variance (Percent)	Modified Wave-Speed (meters/sec)	B (bar per (kg/sec))
4	<input checked="" type="checkbox"/>	17.000	915.0	17.000	17	0.000	915.0	3.8513
5	<input checked="" type="checkbox"/>	10.000	915.0	10.000	10	0.000	915.0	1.6911
6	<input checked="" type="checkbox"/>	6.000	915.0	6.000	6	0.000	915.0	6.3008
7	<input checked="" type="checkbox"/>	15.000	915.0	15.000	15	0.000	915.0	1.6911
8	<input type="checkbox"/>	1.000	915.0	1.000	1	0.000	915.0	6.3008
9	<input checked="" type="checkbox"/>	9.000	915.0	9.000	9	0.000	915.0	1.6911

Note: The units used above are the preferred units for length, velocity and area specified in the User Options/Unit Preferences

Estimated Output File Size: File Size: 21.0 MB Estimated Run Time: Run Time: 4.16 minutes

**Figure 6.13 Sectioning panel excluding the variance at P8**

2. Select 1 section in the controlling pipe by clicking on 1 section in the table under Search Results.
3. Select Show Pulsation Graphs located at the bottom left of the Sectioning panel. A graph of the pulse will appear (see Figure 6.14). You can also view graphs for the FFT of the Pulse, the LPF of the Pulse FFT, and the FFT of the LPF in the Graph Parameter graph that appears. After you are done viewing these graphs, click Close.
4. Click OK to close the Analysis Setup window, and click OK for the pop-up window notifying you that the forcing function has been placed on J2 (the junction that was selected in the Pulse Setup panel) as transient data.
5. Open J2 to verify that the numerical “ring” has been applied to the transient (see Figure 6.15 for the transient input). This transient is a brief jump in flow rate at junction J2.

## PFA Step 6. Run the model

All groups in Analysis Setup should now be complete and the model ready to run. Run the model by pressing the “Run Model” icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. This will open the Solution Progress window. When complete, press the “Output” button.



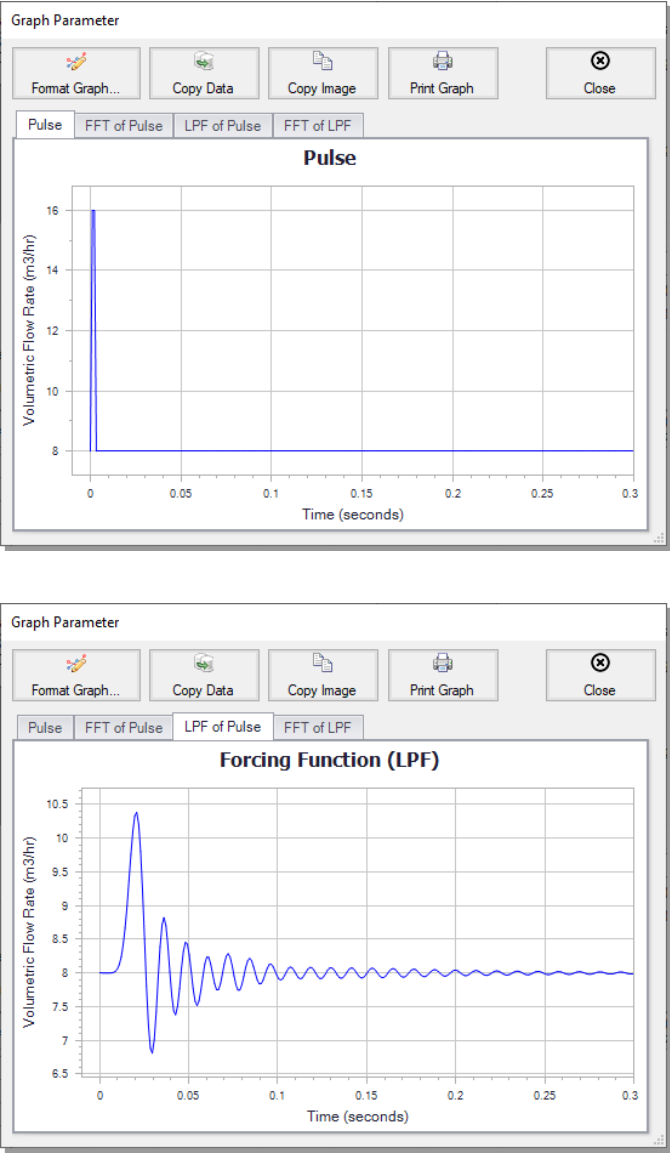


Figure 6.14    Graphs of Pulse and LPF of Pulse in this example

Assigned Flow Properties

Number:  Upstream Pipe:  Downstream Pipe:

Name:

Library Jct:

Copy Data From Jct...

Elevation:

OK Cancel Jump... Help

Flow Model ☒ Transient Optional Design Alerts Notes Status

Transient Special Condition

☒ None

☐ Ignore Transient Data

☐ No Reflections (Infinite Pipe)

☐ Periodic Flow

☐ Chopped Sine Wave

Frequency (1/s):

Flow Amplitude:

Initiation of Transient

☒ Time ☐ Dual Event Cyclic

☐ Single Event ☐ Dual Event Sequential

Transient Data

☒ Absolute Values

☐ Relative To Steady-State Value

Data Point	Time (seconds)	Vol. Flow (m3/hr)
1	0	8
2	0.00109289	8
3	0.00218579	8.000003
4	0.00327869	8.000025
5	0.00437159	8.000158
6	0.00546449	8.000754
7	0.00655738	8.002859
8	0.00765028	8.009021
9	0.00874318	8.024422
10	0.00983608	8.058013
11	0.01092898	8.122982
12	0.01202188	8.235703

Edit Table Show Graph...

☐ Repeat Transient

Transient

**Figure 6.15** Assigned Flow Properties with forcing function applied to J2. Choosing “Show Graph...” under the data shows this in graphical form.

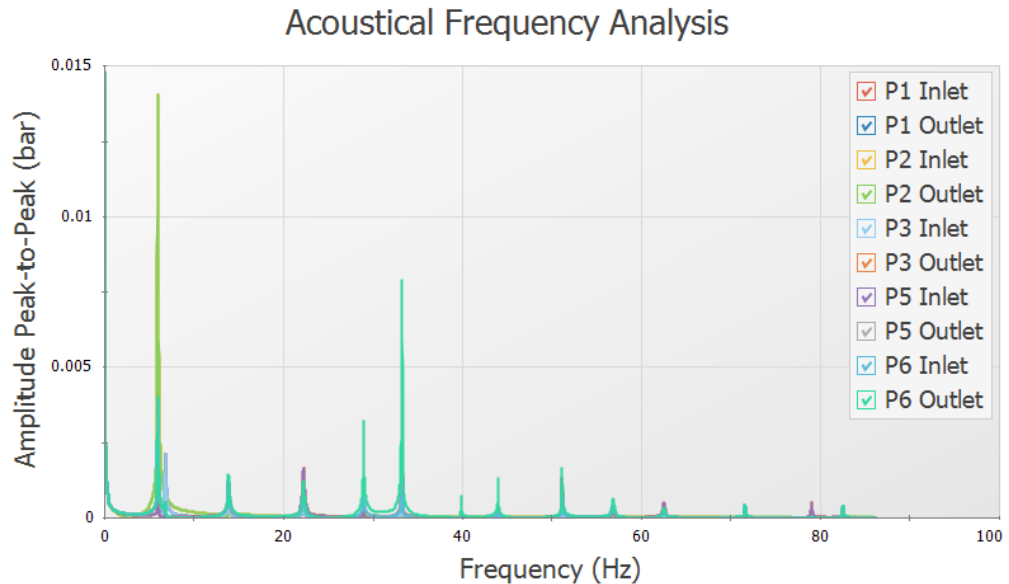
## PFA Step 7. Review results

### A. Find excitation frequencies to study

1. On the Graph Control tab on the Quick Access Panel, click the Frequency tab to begin generating an Excitation Frequency Analysis graph.
2. We need to determine which frequencies excite the pump speeds within the pump speed range we have specified. For the purposes of this example, we will graph the Excitation Frequency graph for 10 different pipe stations. Note that it is the engineer's responsibility to evaluate all pipe stations that could be excited by various frequencies and to perform the necessary analysis.
3. On the Graph Results tab, select the Frequency tab and select the inlet and outlet stations for the following pipes: 1, 2, 3, 5, and 6.
4. Click Generate. The resulting graph (shown in Figure 6.16) represents the frequencies that are excited by the pulse defined in the Pulsation Setup group.

From the graph in Figure 6.16, we see that there are several frequencies that produce a large pressure magnitude. Points on the plot with a large response magnitude are easily identified by finding local maxima on the graph. These maxima are the acoustic natural frequencies that are potentially the most damaging when excited by the identified pump speeds.

Note that the graph in Figure 6.16 can quickly be changed to display the magnitude on a logarithmic scale in order to allow for easier viewing of the magnitude behavior changes with a changing frequency. To view the Excitation Frequency Analysis graph with the magnitude displayed on a logarithmic scale, generate the frequency graph as discussed above. Right-click on either axis and select the box next to Logarithmic.

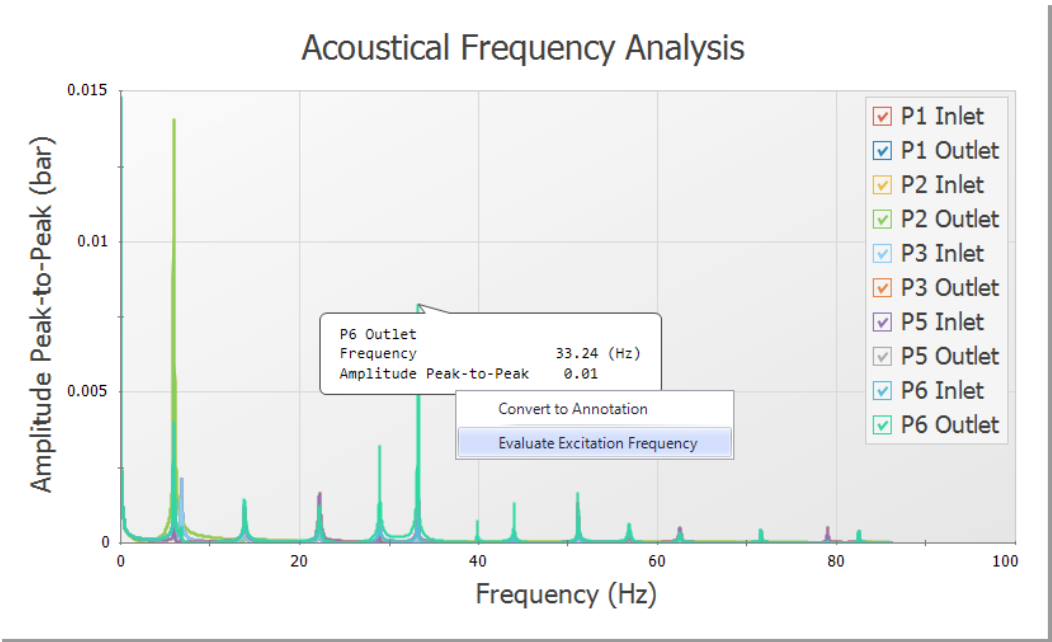


**Figure 6.16 Excitation Frequency Analysis graph**

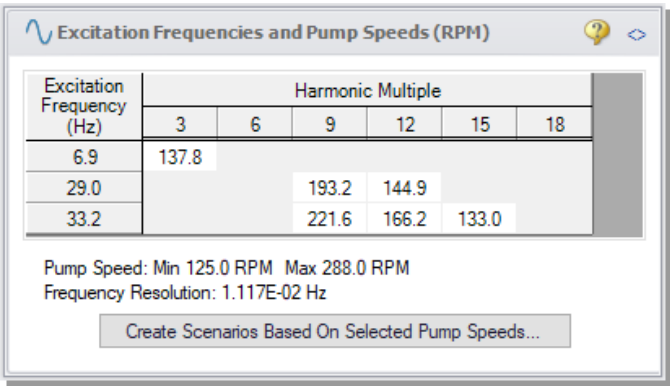
## B. Evaluate excitation frequencies

The identified frequencies can now be evaluated to determine pump speeds that would cause them to be excited. These natural frequencies need to be evaluated to examine potentially problematic pump speeds. Follow the steps below to select the excitation frequencies:

1. With the crosshair cursor off, left-click near the frequency of around 33.2 Hz and drag the mouse over the peak on the graph near this frequency. This will highlight and flag the local maximum.
2. Right-click on the dialog flag that appears on the peak of this frequency, as demonstrated in Figure 6.17.
3. Select "Evaluate Excitation Frequency" to add this frequency to the Pump RPM Evaluation panel, which is labeled with "Excitation Frequencies and Pump Speeds (RPM)". Repeat steps 1 through 3 for the local maxima around the frequencies of 29.0 Hz and 6.9 Hz. The panel should now appear as shown in Figure 6.18.

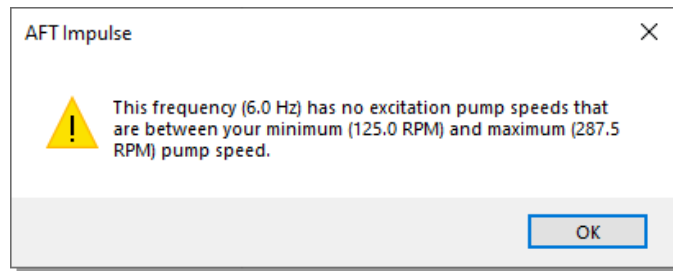


**Figure 6.17** Excitation Frequency Analysis with Dialog Flag. Right-clicking on the Dialog Flag provides the option to Evaluate Excitation Frequency.



**Figure 6.18** Pump RPM Evaluation Panel with Excitation Frequencies and their Harmonic Multiples Displayed in the bottom of the Quick Access Panel

When choosing low or high frequencies on the graph, it is possible to select a frequency which is excited in the system but does not have a pump speed within the minimum and maximum pump speeds for the analysis. The minimum pump speed is defined directly in the PD Pump Settings panel, while the maximum is calculated using the Pump Speed and Speed Buffer defined under “Pulsation Source” in Pulse Setup panel. For example, try selecting the largest maxima, which occurs at 6.0 Hz. The message in Figure 6.19 appears.



**Figure 6.19** Message Indicating No Excitation Pump Speeds Between the Min and Max Pump Speeds Exist

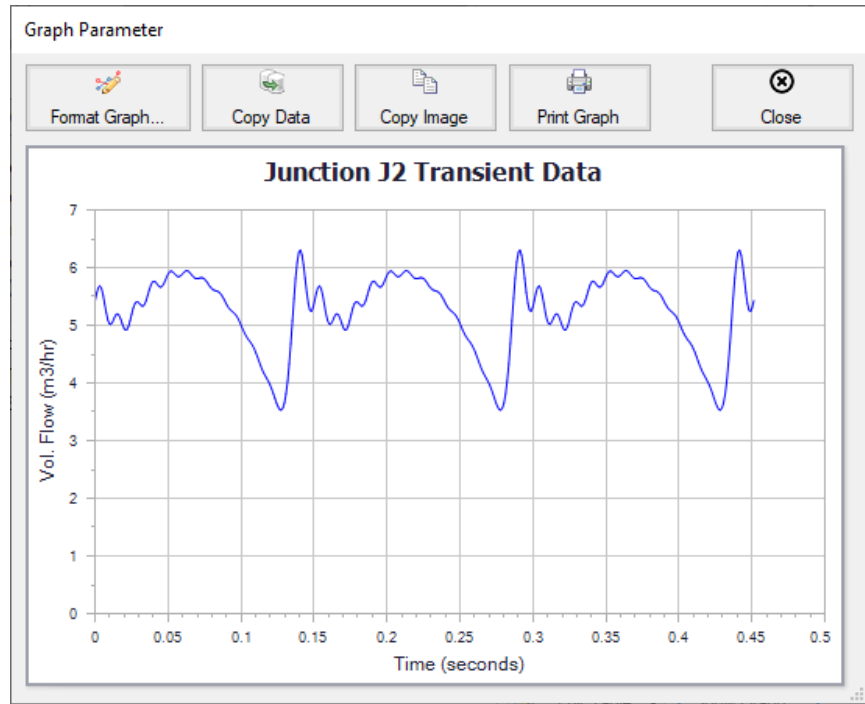
***Determine the pressure response of the system at excited frequencies***

In the Pump RPM Evaluation panel, corresponding pump speeds can be selected for study. At these speeds, the pressure response is important to characterize. By default, all speeds in the Pump RPM Evaluation panel are pre-selected and must be clicked to be turned off. Child Pump Speed scenarios can then be created that will have the Flow vs. Time profile at the selected speed (RPM) that will excite the evaluated frequencies.

We will create child scenarios for all in the Pump RPM Evaluation panel. To do this, ensure that pump speeds are selected (not italicized).

1. Click “Create Scenarios Based On Selected Pump Speeds...”
2. A dialog box will appear stating that six scenarios will be created based on the selected speeds in the Pump RPM Evaluation panel. Click Yes to continue creating the child scenarios (this may take a few minutes).
3. Double-click on the "133 RPM (33.2 Hz)" child scenario to load it as the current scenario.

The scenario will now have different transient information in J2, modeling the pump performance at 133 RPM. Go to the Workspace and open the Properties window for J2. On the Transient tab click Show Graph under the Transient Data table to verify the Pump Flow Forcing Function has been applied, as shown in Figure 6.20.



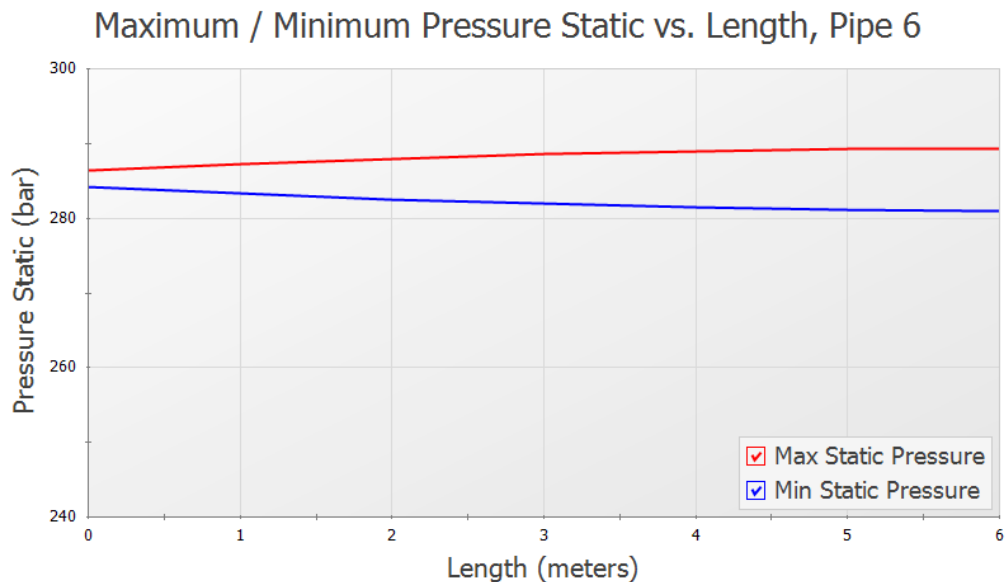
**Figure 6.20 New Assigned Flow transient representing the PD Pump operating at RPM**

Run the "133 RPM (33.2 Hz)" scenario to see the pressure response. Often, the most noticeable pressure response will occur in stagnant branched lines. For the purposes of this example, we will evaluate the pressure responses through pipe P6. However, engineers must ensure that all pipes that could experience significant pressure responses are analyzed.

1. To create a pressure profile graph in pipe P6, go to the Graph Results tab and in the Graph List Manager, select Profile for the graph type

2. Check the box for Pipe P6
3. Use Pressure Static for the Parameter
4. Use bar for the units
5. Click Generate
6. Right-click on the y-axis to adjust the y-axis scale by unchecking the Auto Scale box, then changing the Minimum to 240 and the Major Val to 20.

Figure 6.21 shows the profile plot for Pipe P6 at a pump speed of 133 RPM.



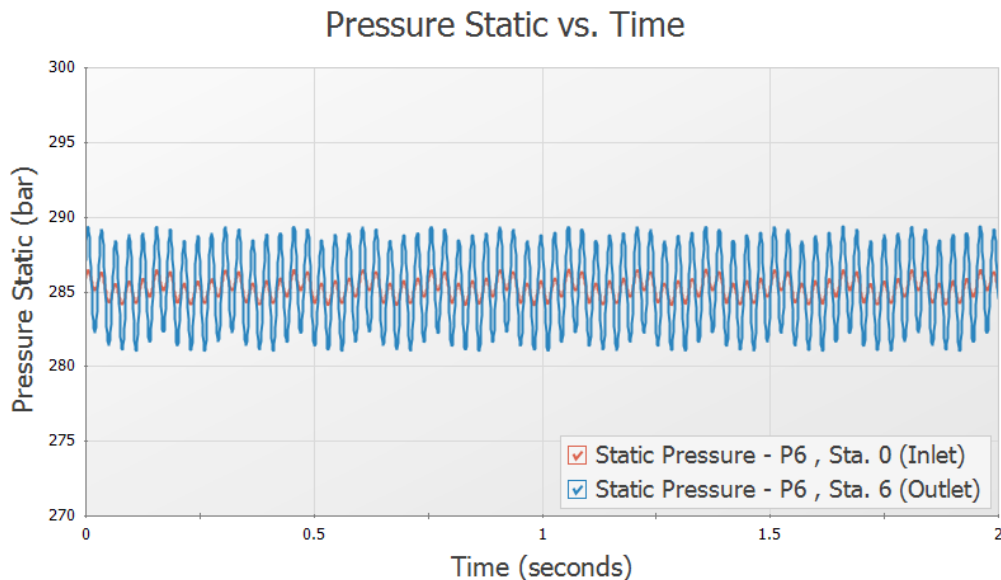
**Figure 6.21 Max/Min Pressure Profile of Pipe P6 for 133 RPM**

The pressure vs. time can be displayed for any pipe station. Plot the pressure vs. time for pipe P6 by following the directions below:

1. In the Quick Access Panel, select Transient Pipe for the graph type
2. Add the outlet and inlet of Pipe P6 to the “Graph These Pipes/Stations:” list



3. Use Pressure Static for the Parameter
4. Use bar for the units
5. Set the Time Frame to User Specified with a Start Time of 0 and a Stop Time of 2 seconds
6. Click Generate (see Figure 6.22)



**Figure 6.22 Pressure Oscillations at Inlet and Outlet of Pipe P6 for 133 RPM**

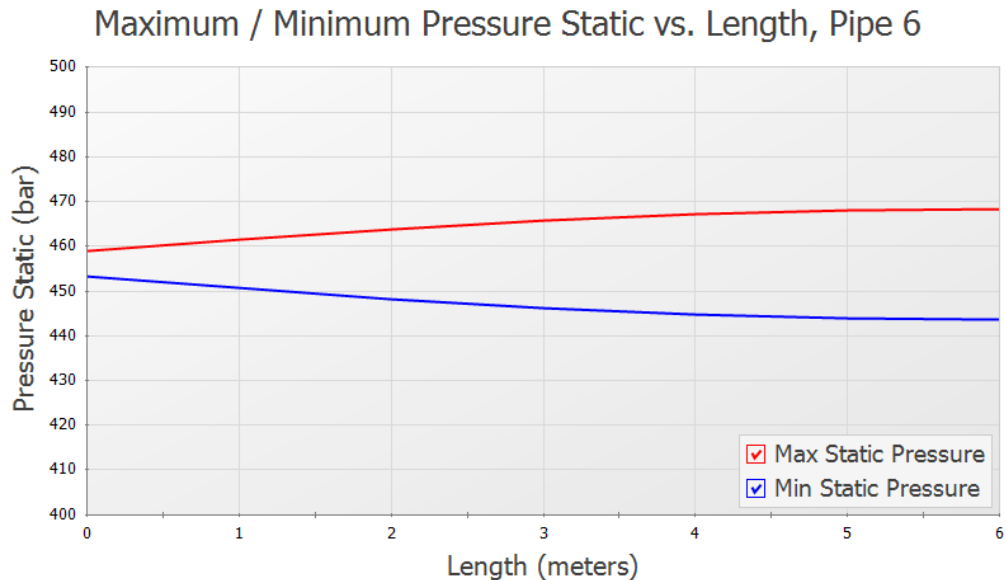
After looking at these graphs, return to the Output tab and make note of the message in red at the bottom right of the screen that states that Warnings Exist. These warnings are reported in the upper third of the output window and warn the user of potential issues with the model.

The PFA module will check the Peak-to-Peak pressures and compare them to the permissible values in the API-674 standard. If the limits from the standard are violated in the model, a warning will appear like in this case, where the Peak-to-Peak pressures exceeded the allowable limits. Additionally, the vapor pressure margin is checked. This is the margin between the minimum pressure and 10% over the vapor

pressure, per the API-674 standard. More information on this can be seen on the Applied Standards and Pulsation Summary tabs.

For the purposes of this example, we will repeat these steps for the pump speed of 221.6 RPM because this pump speed causes the largest pressure oscillations. When performing an analysis on your system, all pump speeds that excite the system should be evaluated.

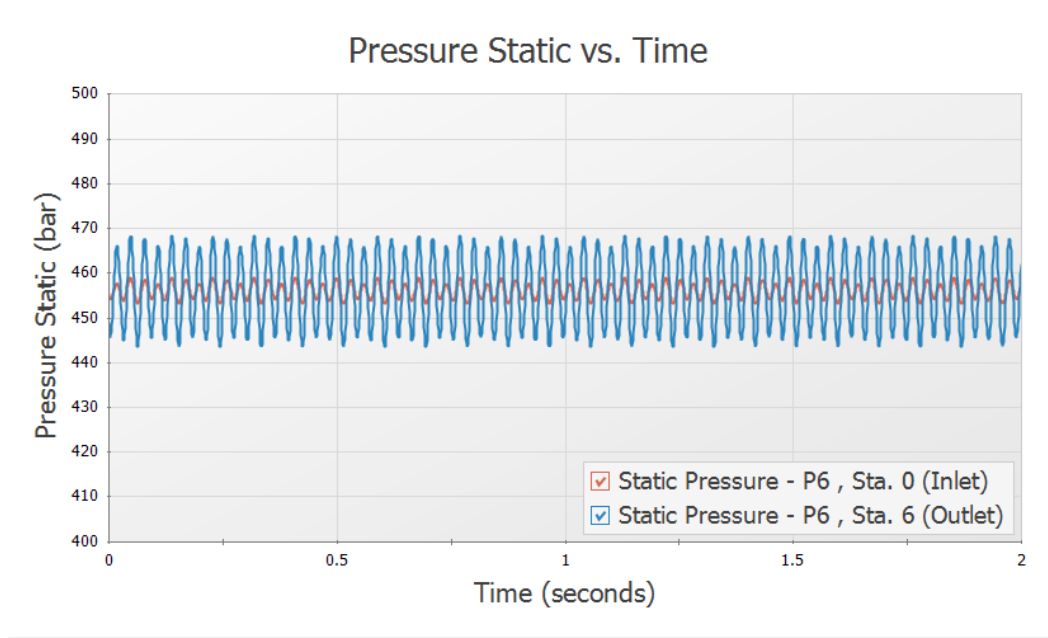
Figure 6.23 shows the Max/Min Pressure Profile for P6 for the pump speed of 221.6 RPM.



**Figure 6.23 Max/Min Pressure Profile of Pipe P6 for 221.6 RPM**

Notice the large difference between the maximum and minimum pressures at the end of the pipe, located next to the Dead End junction.

The large pressure oscillation in Pipe P6 can be seen in Figure 6.24. It is important to note that the inlet of Pipe P6 does not exhibit as large of a response as the outlet does. This oscillation, if allowed to continue, can lead to mechanical and support fatigue and thus failure. It should be noted in the Output that this scenario also violates the API-674 standard.



**Figure 6.24 Pressure Oscillations at Inlet and Outlet of Pipe P6 for 221.6 RPM**

## PFA analysis summary

AFT Impulse and the PFA (Pulsation Frequency Analysis) module were used to discover the problematic pump frequencies occurring in a system with a known pulsation issue. The software was used to identify these frequencies, and after the frequencies were identified, a corresponding pump operating speed was examined to find the pressure response of the system. Using graphing features, the pressure response was plotted and studied.

# Other AFT Impulse Capabilities

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

## Microsoft Excel™ data integration

AFT Impulse 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 Impulse includes a number of importing and exporting capabilities. Piping layouts and dimensional data can be imported from GIS Shapefiles (SHP) to build a model. CAESAR II Neutral Files (CII) can be imported into AFT Impulse and EPANET Files (INP) can be both imported and exported. Additionally, force files can be exported for use in CAESAR II, TRIFLEX, ROHR2

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

### **Variable density and viscosity modeling**

Systems that have variable density and viscosity can be modeled. The density and viscosity can be assigned on a pipe by pipe base.

### **Transient cavitation and liquid column separation**

AFT Impulse offers the Discrete Gas Cavity model (DGCM) and the Discrete Vapor Cavity model (DVCM) for modeling transient cavitation (also known as liquid column separation). These models calculate vapor volume size over time and account for pressure spikes when cavities collapse. Vapor volume can be plotted in the Graph Results window.

### **Positive displacement pumps**

AFT Impulse can model positive displacement pumps. Typically, this is done by modeling the pump as a known flowrate over time.

Complicated periodic flowrates from multiple pistons and/or multiple out of phase pumps can also be modeled.

### **Pumps with viscosity corrections**

When the liquid is sufficiently viscous, centrifugal pump performance will be degraded. AFT Impulse models corrections to manufacturer performance curves during steady and transient flow.

### **Pumps with variable speed controllers**

Pumps with speed controllers can be modeled during steady and transient flow. Transients caused by flow or pressure control level changes with time can be modeled.

### **Pumps as turbines (PAT)**

Pumps being operated as turbines can be modeled during steady and transient flow using the Pump as Turbine model in the Pump junction.

## **Flow and pressure control valve transients**

AFT Impulse can model flow and pressure control valves. AFT Impulse accounts for control valves which lose and regain control during the transient. Transients caused by changes in flow or pressure control setpoints can also be modeled

## **Air valves**

Air valves (also known as vacuum breaker valves) are used to protect against low pressure conditions. AFT Impulse can model air valves, including different inflow and outflow geometries.

## **Surge tanks**

Surge tanks are a surge suppression device used in low pressure systems with non-volatile fluids. AFT Impulse can model surge tanks, including transient surface pressure for enclosed tanks.

## **Relief valves**

The Relief Valve junction allows user-defined combinations of opening and closing profiles based on parameters such as time or pressure. Users can define a blowdown pressure for relief valves that close at pressures below the set pressure. Relief valves can have constant backpressure acting on the valve stem (hydraulically balanced) so the set points can be defined as absolute upstream pressures, or the set points can be entered as a pressure difference across the valve. For valves with pressure profiles, the Cv profile for the valve can be user-defined or set based on the valve set points. Users can also add opening and closing rate limits.

## **Infinite pipe boundaries**

Infinite pipe boundaries can be very useful for long, non-reflecting pipes. Model run times can be significantly reduced with these elements. Both Assigned Flow and Assigned Pressure junctions can be specified to behave as infinite pipe boundaries during transients.

## **Non-Newtonian fluid modeling**

AFT Impulse can model steady and transient flow of non-Newtonian fluids which behave as Power Law, Bingham Plastic, Herschel-Bulkley, or Homogeneous Scale-up viscosity models.

## **Pulp and paper modeling**

AFT Impulse can model steady and transient flow of pulp and paper stock using the Duffy method or Brecht & Heller method.

## **Repeat transient feature for periodic transient behavior**

For elements which have periodic behavior (such as positive displacement pumps), one cycle of data can be entered and then specified to repeat once the cycle is finished.

## **Intermediate elevations for pipes**

Typically, elevation changes along pipes do not affect the steady or transient system. Exceptions are high points which cavitate. If desired, AFT Impulse can model varying elevations along a pipe.

## **Fitting library**

AFT Impulse offers a library of about 400 fitting losses which can be added to pipes.

## **Design alerts**

Design Alerts can be entered for pipes then cross-plotted vs. system behavior. A common use is maximum allowed operating pressure.

## **Network libraries**

Junction components and pipe materials can be saved to libraries for later reuse. Libraries can be located on local PC's or deployed across

local or wide area networks. The Library Manager allows users to connect to relevant libraries for their specific pipe system design.

### **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.





# Index

## A

- AFT Fathom 3
- AFT Impulse
  - Engineering assumptions 4
  - Overview 4
  - Summary of capabilities 2
- AFT Standard fluid
  - Library 3
- Air Valve junction 2
- Animation
  - Graph Animation 71
  - Visual Report Animation 49
- Annotation tool 13
- AutoCAD Plant 3D 141

## B

- Bingham Plastic 144
- Brecht & Heller method 144

## C

- CADWorx 141
- CAESAR II 141
- Cavitation 2
- Change Units window 42
- Chempak database 4
- Controlling pipe 34

## D

- Defining objects
  - Undefined Objects window 23
- Design Alerts 117, 144
- Duffy method 144

## E

- EPANET 141
- Event transient specification 60
- Excel 141
- Exporting 141

## F

- Fitting Library 144
- Fluid Properties 78
- Force sets 99

## G

- Graph Guide 42
- Graph List 68
- Graph Results window 5, 42, 44, 66, 86, 118
  - Design Alerts 144
  - Forces tab 104
  - Secondary Y-axis 68

## H

- Highlight feature 26

## I

- ID numbers 14, 23

Importing 141

Inspection feature 26

Intermediate elevations 144

Isometric 15, 100

### J

Junctions

Air Valve 2, 143

Area Change 124

Assigned Flow 124

Gas Accumulator 2

Pump 57, 114

Relief Valve 143

Reservoir 23, 25, 56, 113

Spray Discharge 84, 125

Surge Tank 2, 143

Valve 27, 114

### L

Libraries 144

Library Manager 144

Liquid column separation 2

### M

Method of Characteristics 3, 33

Mixtures 4

Model Data window 5, 32

Modulus of elasticity 31

### N

Newton-Raphson method 3

NIST REFPROP 3

Non-Newtonian fluid 2, 144

### O

Open Pipe/Jct Window 24

Output file 39

Output window 5, 39, 85

SSL output 116

Overview of AFT Impulse 4

### P

PCF 141

PDS 141

Periodic transient behavior 144

PFA module 121

Excitation Frequency Analysis 132

Pulsation Setup group 126

Pipe Drawing tool 16

Pipe forces 99

Pipe Object 14

Pipe Properties window 29-31

Design Alerts 144

Intermediate elevations 144

Poisson Ratio 31

Power Law 144

Preferred Units 24

Properties windows

Using tabs 25

Pulp and paper modeling 144

Pump junction 57

    Entering pump curves 57

    Four quadrant modeling 90

    Positive displacement pump  
        modeling 142

    Pump as turbine (PAT) 142

    Variable speed controllers 142

    Viscosity corrections 142

## **Q**

Quick Access Panel 12

    Pinning 13

## **R**

Reference positive flow direction 18

Relief Valve junction 143

Reservoir junction 56

Reverse flow 19

Reverse Pipe Direction 19

Run Model 37

## **S**

Scenario Manager 12, 63

Sectioning panel 33, 61, 85, 102, 116

Show Object Status 22-23, 26, 32

SmartPlant 141

Solution Progress window 37, 66, 85,  
    103, 116, 129

SSL module 107

Stacked Graphs 43

Status Bar 12, 20

Steady-state solver 3, 38

Steady Solution Control 20

Surge Tank junction 2

## **T**

Toolbars 19

Toolbox 12-13, 16

Transient cavitation 2, 142

Transient Control group 36, 62, 85, 95,  
    102, 116

Transient output file 39

Transient solver 3

## **U**

Undefined Objects window 23

User Options 23

## **V**

Valve junction 27

    Event transient specification 60

Viscosity model 22

Visual Report Control window 49

Visual Report window 5, 49

## **W**

Wavespeed 31, 34

    Assumption that it remains  
        constant 4

Workspace window 5, 13

# AFT Impulse™ Add-on Modules



## Settling Slurries

The only commercial software solution that can model both non-settling and settling slurry waterhammer effects.



## Pulsation Frequency Analysis

Helps identify and avoid resonant frequencies in systems, especially in those caused by positive displacement pumps.



For Product Support:

Call: (719) 686-1000

Email: [support@aft.com](mailto:support@aft.com)

---



2955 Professional Place, Suite 301  
Colorado Springs, CO 80904 USA  
(719) 686 1000  
[info@aft.com](mailto:info@aft.com)  
[www.aft.com](http://www.aft.com)

