



# **Quick Start Guide**

**Metric Units**

**Impulse Version 11**

**Waterhammer Modeling in Piping Systems**

**Published: June 27, 2025**



**Distribuito in Italia da Prode**

**tel. 02-680291**

**[www.prode.com](http://www.prode.com)**

## 2 Impulse 11 Quick Start Guide

---

### **CAUTION!**

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. Impulse is intended solely as an aid 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 Impulse should be independently verified with other engineering methods.

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.

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

© 2025 Applied Flow Technology LLC. All rights reserved.

“AFT Impulse”, “AFT Fathom”, “Applied Flow Technology”, “Dynamic solutions for a fluid world”, “Pulsation Frequency Analysis” and the AFT logo are trademarks of Applied Flow Technology LLC.

Excel and Windows are trademarks of Microsoft Corporation. Intelliquip is a trademark of FPX, LLC. Chempak is a trademark of Madison Technical Software, Inc. REFPROP is a trademark of the National Institute of Standards and Technology (NIST). CAESAR II, SmartPlant, CADWorx, and PDS are trademarks of Intergraph Corporation / Hexagon AB. ROHR2 is a trademark of SIGMA Ingenieurgesellschaft mbH. AutoCAD Plant 3D is a trademark of Autodesk, Inc.

# Contents

<b>1. Introducing Impulse</b>	<b>1</b>
Designing for waterhammer	1
Modeling capabilities	2
Standard version capabilities	2
Add-on module capabilities (optional)	2
Engineering assumptions in Impulse	2
The Steady-State Solver	3
The Transient Solver	3
Thermophysical property data	3
Impulse Primary Windows	4
Workspace	4
Model Data	4
Output	4
Graph Results	4
Input vs. Output	4
 <b>2. Valve Closure Example</b>	 <b>6</b>
Topics covered	6
Required knowledge	6
Model file	6
Problem statement	7
Step 1. Start Impulse	7
A. The Startup window	7
B. The Workspace window	9
C. Hiding the Quick Access Panel	9
Step 2. Build the model	10
A. Place a reservoir	10
Objects and ID numbers	10
Editing on the Workspace	10
B. Place the other junctions	11
C. Draw a pipe between J1 and J3	12
D. Add the remaining pipes	13
Reference positive flow direction	14
Step 3. Specify Analysis Setup	15
A. Modules	15
B. Fluid Properties	16
C. Pipes and Junctions	16
Object status	17

Showing undefined objects . . . . .	17
Enter data for the reservoirs . . . . .	17
Using the tabs in the properties windows . . . . .	19
The Inspection feature . . . . .	19
Enter data for the branch . . . . .	20
Enter data for the valve . . . . .	20
Enter data for Pipe P1 . . . . .	22
The Pipe Properties window . . . . .	23
Wavespeed . . . . .	23
Enter data for the other pipes . . . . .	24
Check pipes and junctions . . . . .	24
Review Model Data . . . . .	24
Reviewing input in the Model Data window . . . . .	24
Steady-state groups are now fully defined . . . . .	25
D. Pipe Sectioning and Output . . . . .	25
E. Transient Control . . . . .	27
F. Save the model . . . . .	28
Step 4. Run the model . . . . .	28
A. The two Solvers . . . . .	28
B. The output file . . . . .	29
Step 5. Review the Output window . . . . .	29
A. Modify the output format . . . . .	32
Step 6. View the graph results . . . . .	33
A. Further review . . . . .	36
Step 7. View the Workspace Layers . . . . .	38
Conclusion . . . . .	41

### **3. Pump Startup with Event Transients . . . . . 42**

Topics covered . . . . .	42
Required knowledge . . . . .	42
Model file . . . . .	42
Problem statement . . . . .	42
Step 1. Define the Fluid Properties group . . . . .	43
Step 2. Define the Pipes and Junctions group . . . . .	43
A. Place the pipes and junctions . . . . .	43
B. Enter the pipe and junction data . . . . .	44
Pipes . . . . .	44
J1 - Reservoir . . . . .	44
J10 - Reservoir . . . . .	44
J11 - Reservoir . . . . .	45
J6, J7, & J8 - Branches . . . . .	45
J2 & J4 - Pumps . . . . .	45

J3 & J5 - Valves .....	46
J9 - Valve .....	46
C. Check if the pipe and junction data is complete .....	48
Step 3. Review the Pipe Sectioning and Output group .....	48
Step 4. Specify the Transient Control group .....	49
Step 5. Create scenarios to model the three startup cases .....	49
A. Create scenarios .....	49
B. Set up scenarios .....	50
Two Pump Start .....	50
One Pump Start .....	50
One Pump Start with One Running .....	51
Step 6. Run the model .....	52
Step 7. Graph the results .....	52
A. Create a profile graph .....	52
B. Create a Transient Junction graph .....	53
C. Create a Transient Pipe graph .....	55
Step 8. Animate the results .....	56
Step 10. Run the other scenarios and graph the results .....	59
Conclusion .....	60

#### 4. Pump Trip Example .....

61

Topics covered .....	61
Required knowledge .....	61
Model file .....	61
Problem statement .....	61
Step 1. Define the Fluid Properties group .....	62
Step 2. Define the Pipes and Junctions group .....	62
A. Place the pipes and junctions .....	62
B. Enter the pipe and junction data .....	63
Pipes .....	63
J1 & J2 - Reservoirs .....	63
J3 - Branch .....	63
J4 - General Component .....	64
J5 & J10 - Branches .....	64
J11 - Branch .....	64
J6 & J8 - Pumps .....	64
J7 & J9 - Valves .....	66
J12 & J13 - Spray Discharges .....	67
C. Check if the pipe and junction data is complete .....	67
Step 3. Define the Transient Control group .....	67
Step 4. Run the model .....	67
Step 5. Review results .....	67

A. Graph the pump speed decay . . . . .	68
B. Graph the transient pressures and flows at the pump station . . . . .	68
Step 6. Create additional pump scenarios . . . . .	69
A. Four Quadrant BEP . . . . .	70
B. Four Quadrant SSOP . . . . .	71
Step 7. Run the four quadrant scenarios . . . . .	71
Step 8. Adjust the valve closure speed . . . . .	73
Step 9. Specify the Transient Control group . . . . .	73
Step 10. Run the model . . . . .	73
Step 11. Graph results . . . . .	73
Conclusion . . . . .	74

## **5. Pipe Forces from Valve Closure Example . . . . . 76**

Topics covered . . . . .	76
Required knowledge . . . . .	76
Model file . . . . .	76
Step 1. Open the model file . . . . .	76
A. Isometric Pipe Drawing Mode . . . . .	77
Step 2. Specify the Transient Control group . . . . .	78
Step 3. Define the Forces group . . . . .	78
A. Force Types . . . . .	78
B. Defining Force Sets . . . . .	79
Step 4. Run the model . . . . .	80
Step 5. Graph the pipe forces . . . . .	81
Conclusion . . . . .	82

## **6. Impulse Add-on Modules Examples . . . . . 83**

Topics covered . . . . .	83
Required knowledge . . . . .	83
Model files . . . . .	83
SSL problem statement . . . . .	83
SSL Step 1. Activate the SSL module . . . . .	84
SSL Step 2. Define the Fluid Properties group . . . . .	84
A. Carrier Fluid panel . . . . .	84
B. Slurry Definition panel . . . . .	85
C. Solids Definition panel . . . . .	85
SSL Step 3. Define the Pipes and Junctions group . . . . .	86
A. Place the pipes and junctions . . . . .	86
B. Enter the pipe and junction data . . . . .	87
Pipes . . . . .	87
J1 - Reservoir . . . . .	87
J2 - Pump . . . . .	87

## v Impulse 11 Quick Start Guide

---

J3 - Branch .....	88
J4 - Valve .....	88
J5 - Branch .....	88
J6 - Reservoir .....	88
J7 - Valve .....	89
J8 - Branch .....	89
J9 - Reservoir .....	89
C. Create a Design Alert .....	89
D. Check if the pipe and junction data is complete .....	91
SSL Step 4. Specify the Pipe Sectioning and Output group .....	91
SSL Step 5. Specify the Transient Control group .....	91
SSL Step 6. Run the model .....	91
SSL Step 7. Review results .....	91
A. Check the Transient Max/Min .....	91
B. Increase the Pipe Diameter and Re-run the Model .....	92
C. Graph the velocity ratio and pressures to Deposit #1 .....	92
D. Graph the velocity ratio over time .....	94
SSL Conclusion .....	96
PFA problem statement .....	96
PFA Step 1. Activate the PFA module .....	96
PFA Step 2. Define the Fluid Properties group .....	96
PFA Step 3. Define the Pipes and Junctions group .....	96
A. Place the pipes and junctions .....	96
B. Enter the pipe and junction data .....	97
Pipes .....	97
J1 & J2 - Assigned Flows .....	98
J3 & J4 - Area Changes .....	98
J8 & J9 - Area Changes .....	98
J5 & J6 - Tees .....	98
J10 - Spray Nozzle .....	98
J7 - Dead End .....	98
C. Check if the pipe and junction data is complete .....	98
PFA Step 4. Define Pulsation Setup group .....	99
A. Define Pulse Location and Frequency Domain .....	99
B. Define Excitation RPM Scenarios .....	100
PFA Step 5. Define the Transient Control Group .....	101
PFA Step 6. Run the model .....	103
PFA Step 7. Review results .....	104
A. View the Frequency Response Function .....	104
B. Choose excitation frequencies to evaluate .....	105
C. Create excitation RPM child scenarios and determine the pressure response of the system at excited frequencies .....	107
PFA Conclusion .....	112

7. Other Impulse Capabilities . . . . . 113

Index ..... 119



### C H A P T E R 1

# Introducing Impulse

Welcome to Impulse, Datacor's powerful waterhammer modeling tool. With 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.

Impulse includes a steady-state solution engine which solves for the system's initial conditions. These results are used to automatically initialize the transient model.

Impulse has two available Add-on Modules for purchase which extend the 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 Impulse model allows the engineer to better understand and predict the dynamic behavior of the pipe system. 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, 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

#### Standard version capabilities

- 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 limited 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 calculations for settling slurries (SSL)
- Identification of natural acoustic frequencies (PFA)
- Positive displacement pump speeds which excite resonant frequencies (PFA)

Impulse provides hundreds of standard loss values from handbook sources for pipe system components, and also allows the engineer to enter their own loss data. Variable loss models that depend on flow are supported.

### Engineering assumptions in Impulse

Impulse is based on the following fundamental fluid mechanics assumptions:

- Liquid flow (system is fluid-full and single phase)
- One-dimensional flow
- No chemical reactions
- Wavespeed remains constant during transients
- Non-condensable gas release is negligible
- Fluid density is not affected by pressure waves in the system
- Bubbles that form during transient cavitation do not move

## The Steady-State Solver

Before a waterhammer model can be run, the initial steady-state conditions are required. 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 Fathom.

If desired, 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

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 Transient Solver marches in time for a duration specified by the user.

## Thermophysical property data

Impulse offers several sources from which thermophysical properties can be derived:

- ASME Steam/Water - Properties are calculated with the equations of state from the IAPWS Industrial Formulation (1997), also used to generate the ASME steam/water tables.
- Chempak™ - Optional add-on for purchase which contains data for approximately 700 fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.
- NIST REFPROP - The Reference Fluid Thermodynamic and Transport Properties Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via equations of state. REFPROP supports user-specified fluid mixtures.
- Impulse Standard - Common fluids from various industry standard sources have been curve fit to optimize calculation speed.
- User Specified Fluid - Custom fluid properties can be entered if the desired working fluid is not available via the other sources. Impulse requires density, viscosity, and bulk modulus at a minimum. Vapor pressure can be optionally entered to check for cavitation.
- User Library Fluid - A custom fluid can be saved to the User Library for repeated use in the future.

# Impulse Primary Windows

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

## Workspace

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

The Workspace Layers are also located on this Primary Window and allow the user to display both input and output data superimposed on the pipe system schematic.

## Model Data

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

## Output

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

## Graph Results

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

## Input vs. Output

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

## 5 Impulse 11 Quick Start Guide

---

**Table 1.1      Primary Window Input vs. Output Data**

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

---

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

---

---

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

---

## CHAPTER 2

# Valve Closure Example

This chapter is designed to give you the big picture of 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.

### Topics covered

- Model building basics
- Entering Pipe and Junction data
- Defining Analysis Setup
- Entering transient data
- Sectioning pipes
- Graphing output results
- Using Workspace Layers

### Required knowledge

No prior knowledge is required for this example.

### Model file

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

C:\Users\[username]\Documents\Datacor\Impulse 11\Examples

- *Metric - Beginner - Valve Closure.imp*

### 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.

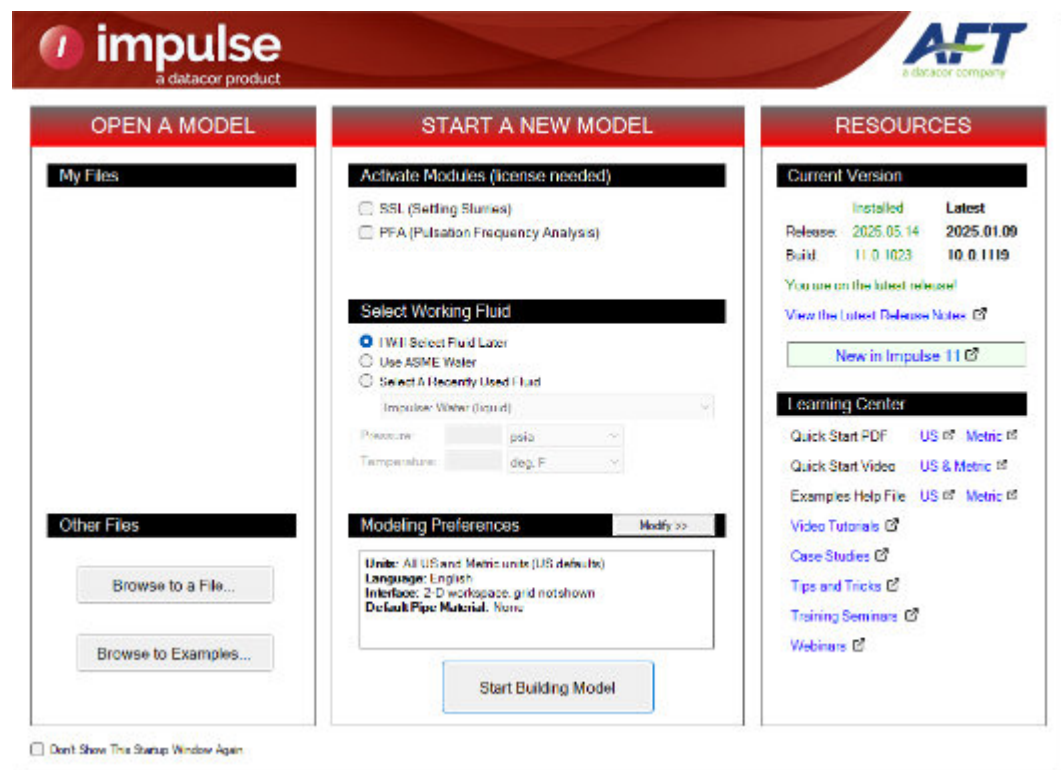
Graph the pressure at the inlet of the valve over the entire simulation duration to determine the maximum pressure that occurs at the valve during the transient.

### Step 1. Start Impulse

- **To start Impulse**, open the all applications list in the Windows Start Menu, expand the Impulse 11 folder, and choose Impulse 11 to launch the application. This refers to the standard menu items created by a standard installation.

#### A. The Startup window

As Impulse is started, the 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 Startup Window with Modeling Preferences collapsed**

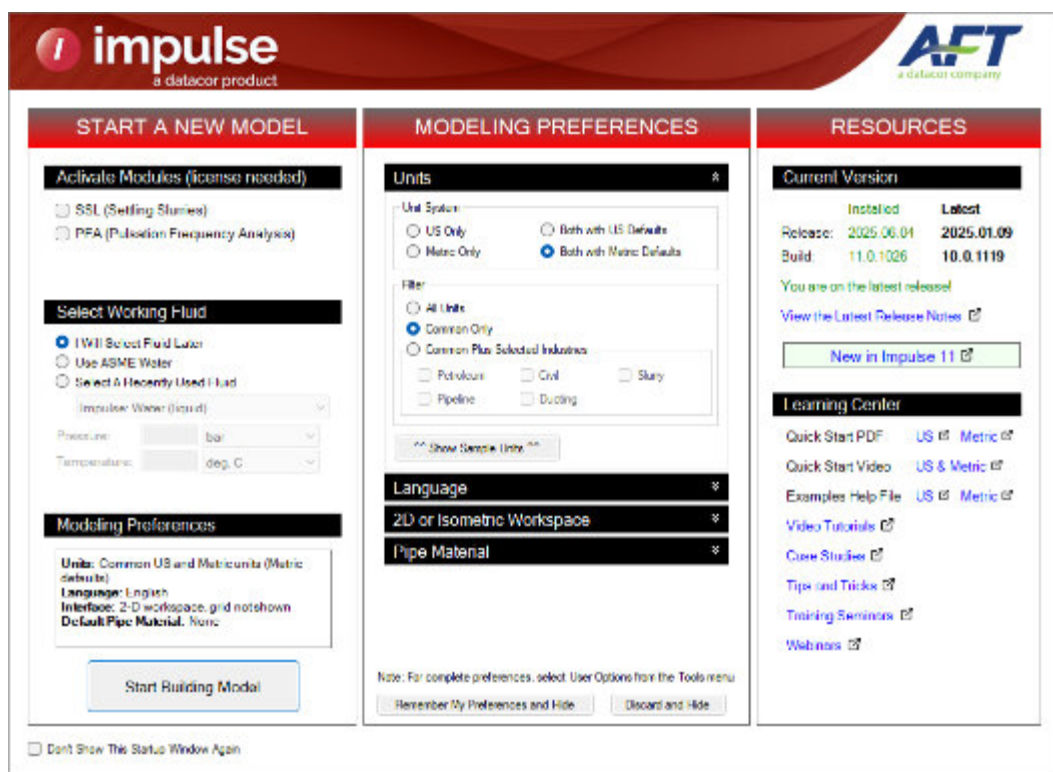
Some of the available actions from the Startup window are:

## 8 Impulse 11 Quick Start Guide

- Open a recent model, browse to a model, or browse to an example
- Activate an Add-on Module
- Select ASME Water or a recently used fluid
- Review or modify Modeling Preferences: select a Unit System, filter units, choose a Grid Style, select a Default Pipe Material, and more
- Access other Resources such as Help Files and Video Tutorials

If this is the first time that you have started 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 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 the Start a New Model section. To further review or adjust your preferences, click the Modify button (see Figure 2.1).



**Figure 2.2 Startup Window with Modeling Preferences expanded**

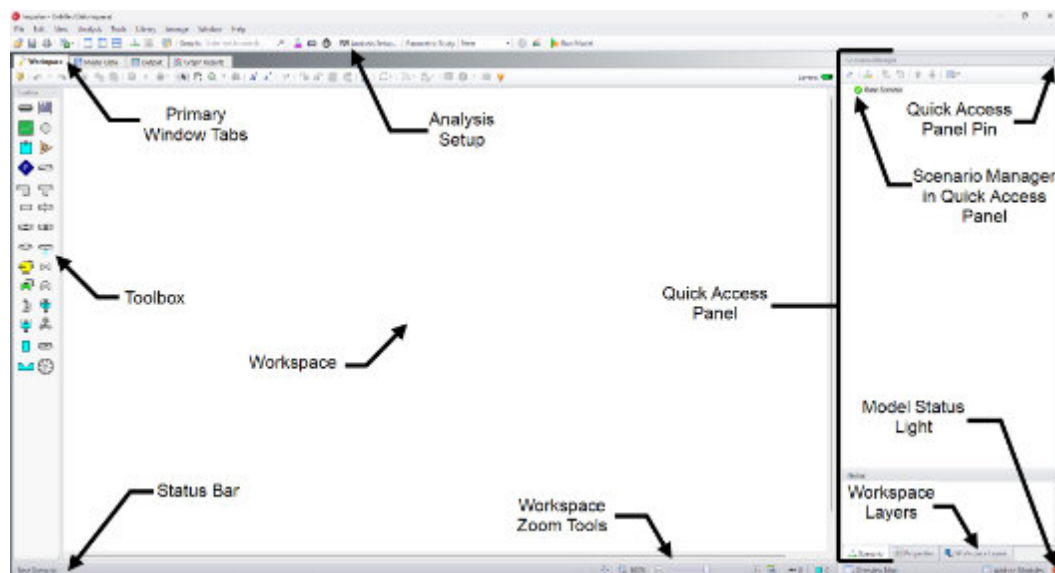
- ▶ **With Modeling Preferences expanded, as in Figure 2.2, select Both with Metric Defaults under Unit System.** The Filter section can be used to limit the number of units that are shown in the interface.

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



### ► Click Start Building Model.

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



**Figure 2.3 The Workspace Primary Window**

### B. The Workspace window

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

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

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

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

### C. Hiding the Quick Access Panel

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

## Step 2. Build the model

To lay out this example model, you will place several junctions on the Workspace. Then, you will connect the junctions with pipes.

### A. Place a reservoir

- To start, drag a Reservoir junction from the Toolbox and drop it on the Workspace. Figure 2.4 shows the Workspace with one reservoir.

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

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

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

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

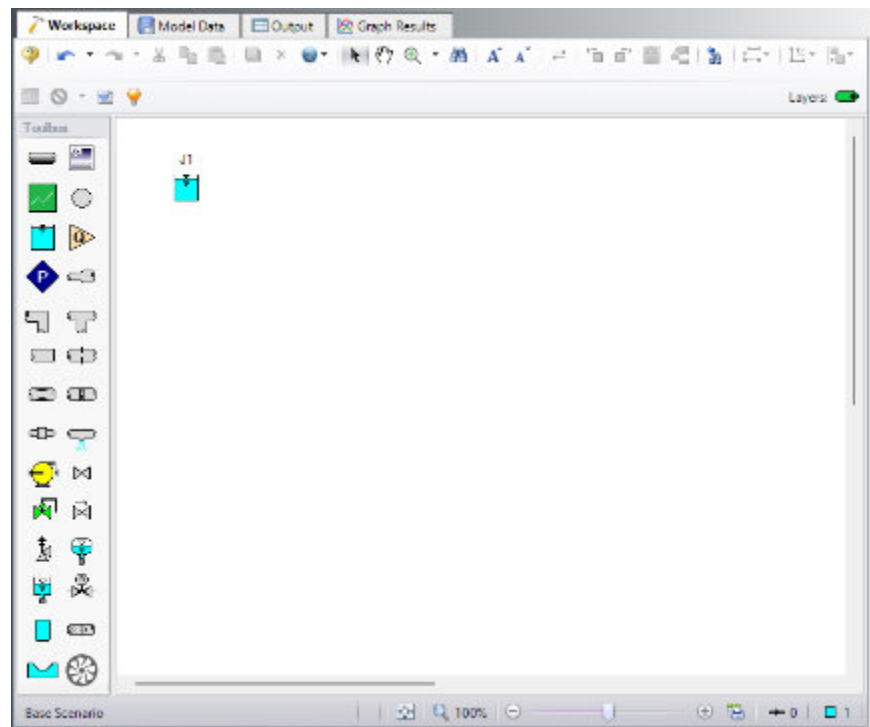


Figure 2.4 Valve closure model with the first junction placed

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

You can move junctions on the Workspace to new locations. You can also cut, copy, paste, and duplicate junctions from the Edit menu or using keyboard

shortcuts. Undo is available for most arrangement actions on the Workspace.

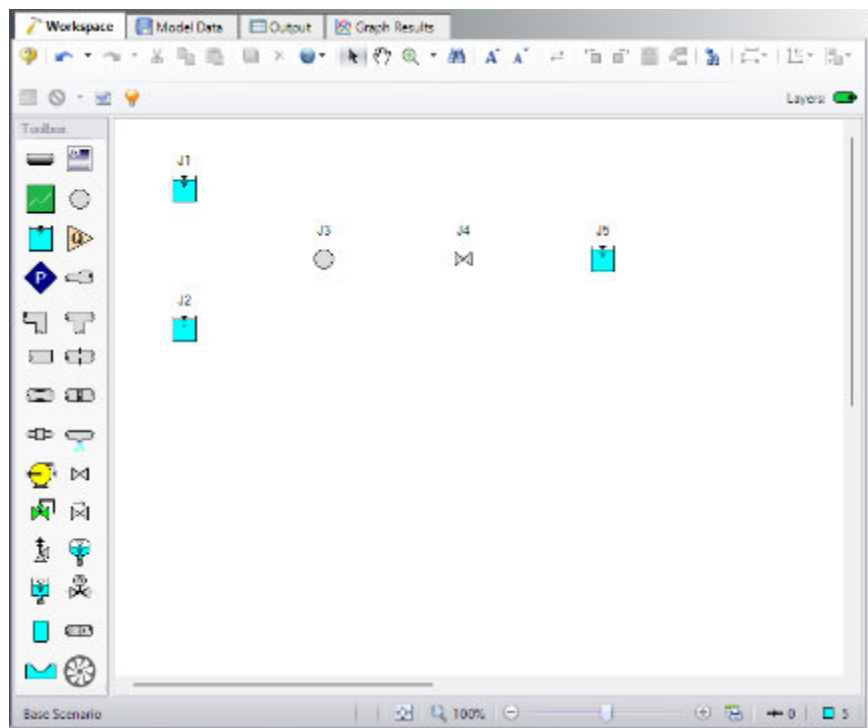
### B. Place the other junctions

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 2.5).

Drag a Branch, Valve, and third Reservoir junction and drop them on the Workspace so that your model appears similar to Figure 2.5.

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

---



**Figure 2.5** Valve closure model with all junctions placed

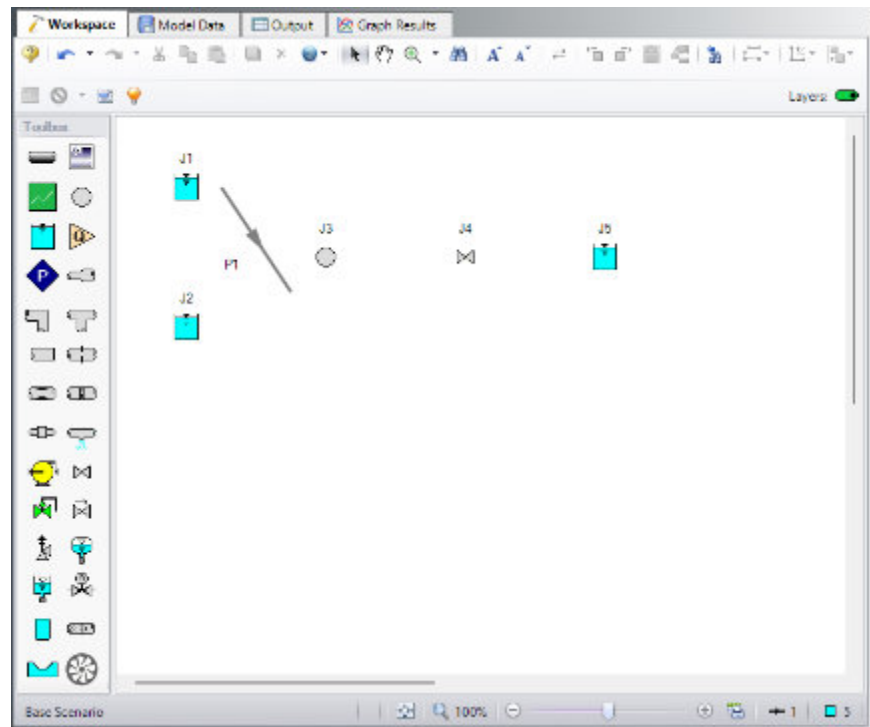
The calculations will not be affected if the icons do not line up exactly. However, your model may have a nicer appearance if they do. You can align the icons by using the Align features from the Arrange menu.

- **Before continuing, save your work.** Press CTRL+S, use the Save button on the Common Toolbar, or go to the File menu and select Save. Impulse models use the \*.imp file extension.

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

Now that you have all of the junctions placed, 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 near the junctions, similar to that shown in Figure 2.6.



**Figure 2.6** Valve closure model with the 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**, click and drag the pipe so that the left endpoint falls within the J1 Reservoir junction (see Figure 2.7). Next, grab the right endpoint of the pipe and drag it until the endpoint falls within the J3 Branch icon (see Figure 2.8).

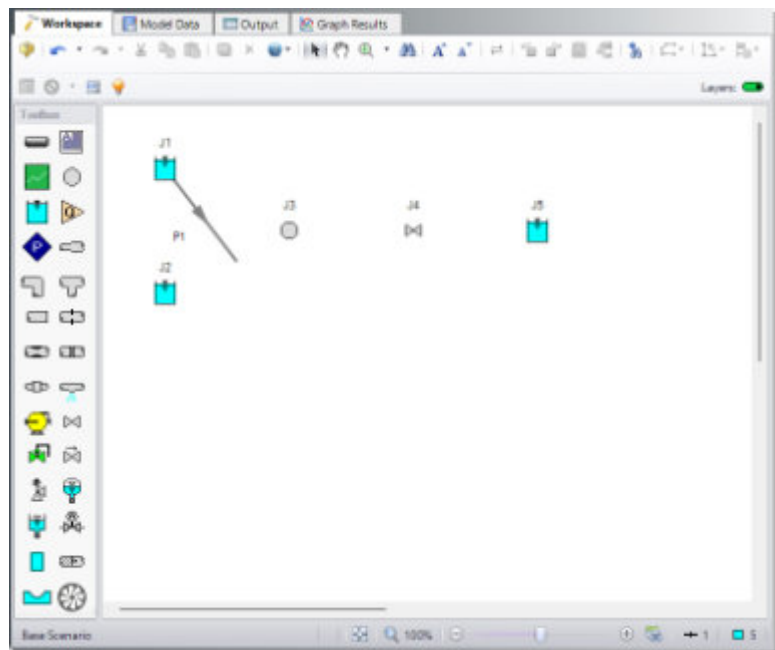


Figure 2.7 Valve closure model with the first pipe partially connected

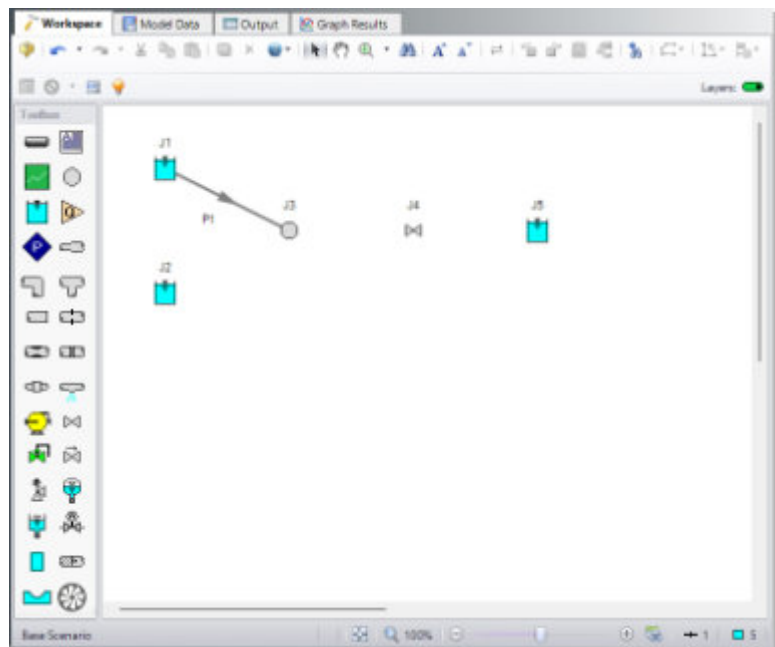
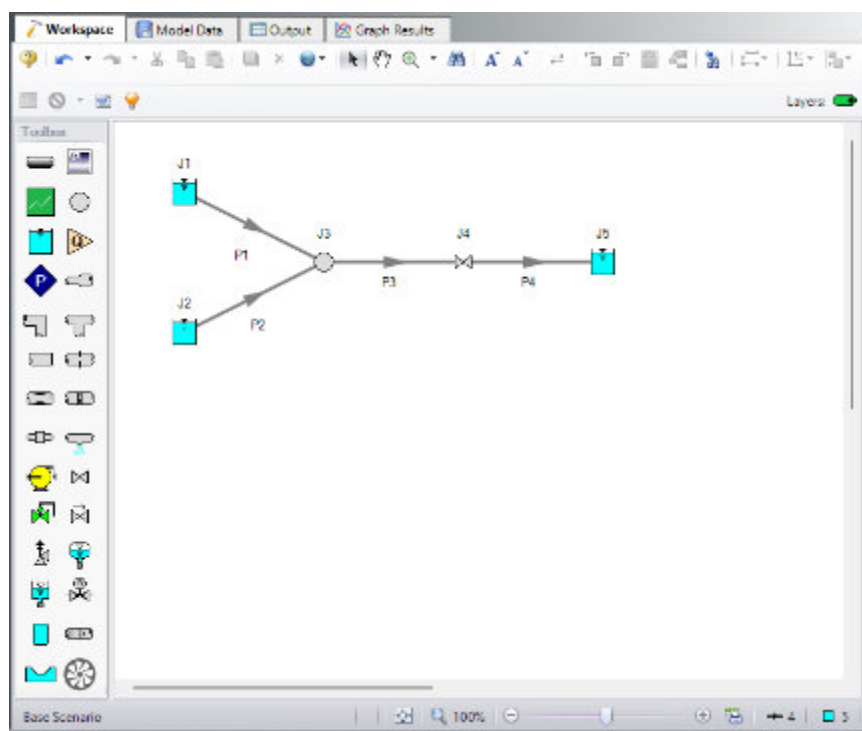


Figure 2.8 Valve closure model with the first pipe fully connected

### 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 2.9. After all the pipes are drawn, all the objects in the model are graphically connected. Save the model again.



**Figure 2.9** Valve closure model with all pipes and junctions placed

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

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

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

Although the reference positive flow direction does not need to point in the direction of the actual flow, when used with pumps, the reference direction determines the suction side and discharge side of the pump.

---

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

---



---

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

---

## Step 3. Specify Analysis Setup

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

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

**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 analysis, choose either “Steady Only” or “Transient” from the Simulation Mode/Duration panel in Analysis Setup or from the Analysis menu.

### A. Modules

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 2.10. Here, you can enable and disable Add-on Modules if they are available on the license. These Add-on Modules include Pulsation Frequency Analysis (PFA) and Settling Slurry (SSL). By default, the model will have all modules disabled. Make no changes to this panel for this example.

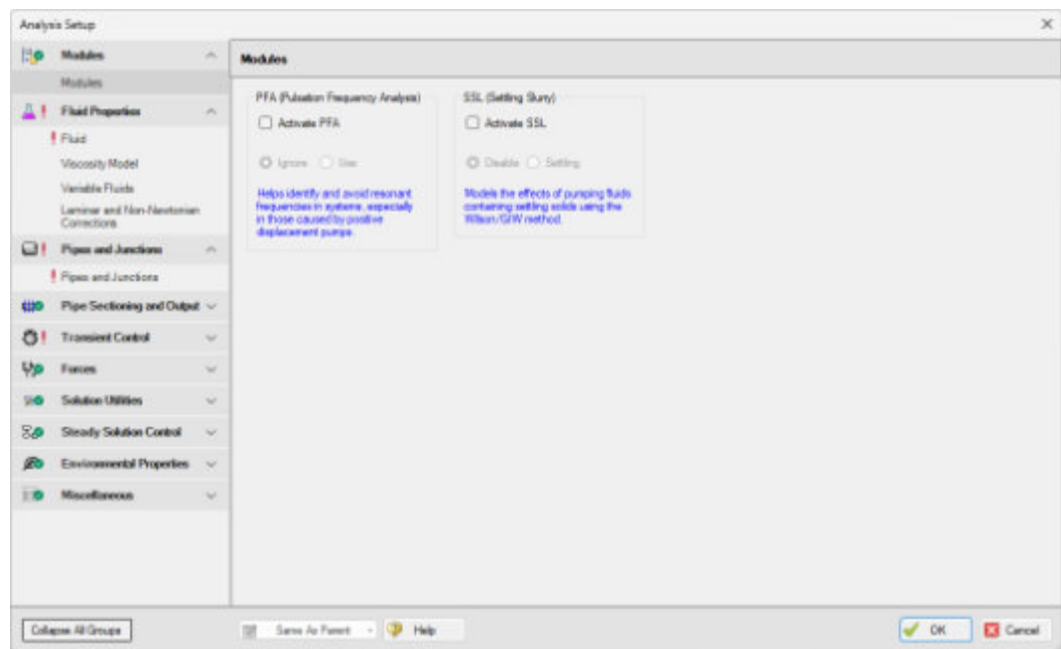


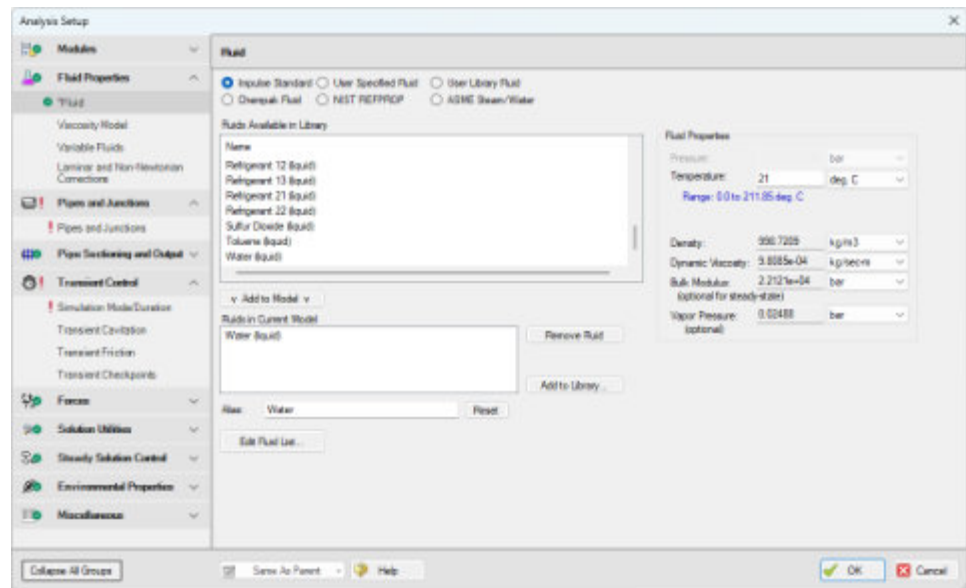
Figure 2.10 On the Modules panel, you can enable or disable any module

## B. Fluid Properties

The Fluid Properties group consists of several 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. You can directly enter density and viscosity data (using User Specified Fluid), select a fluid from the Impulse Standard fluid library, or select Water Data from ASME Steam/Water Tables. Additionally, you can select from fluids and/or create mixtures using the NIST REFPROP library (included in Impulse) or the Chempak library (an optional add-on to Impulse). You can also add your own custom fluids to the User Library.

- **Select the Impulse 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 2.11.**



**Figure 2.11 The Fluid panel is where you specify the fluid**

## C. Pipes and Junctions

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***

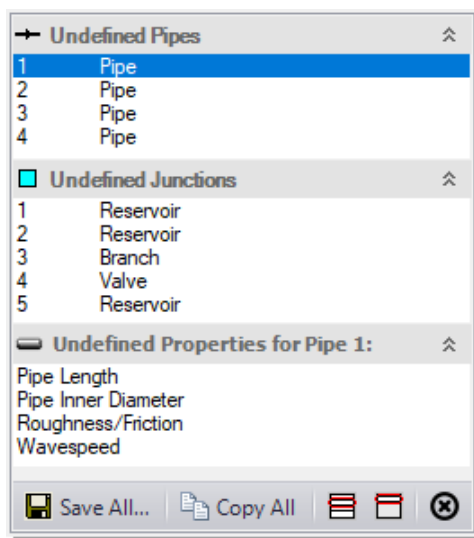
Each pipe and junction have an object status. The object status tells you whether the object is defined according to Impulse's requirements. To see the status of the objects in your model, click the light bulb icon on the Workspace Toolbar (alternatively, you could choose "Show Object Status" from the 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 object 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 2.12). 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 (black X icon) in the bottom-right to stop showing this window.



**Figure 2.12** The List Undefined Objects window lets you see the undefined properties for each undefined object

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

- **To define the first reservoir**, open the J1 Reservoir Properties window (Figure 2.13) by double-clicking on the J1 icon.

---

**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.

---

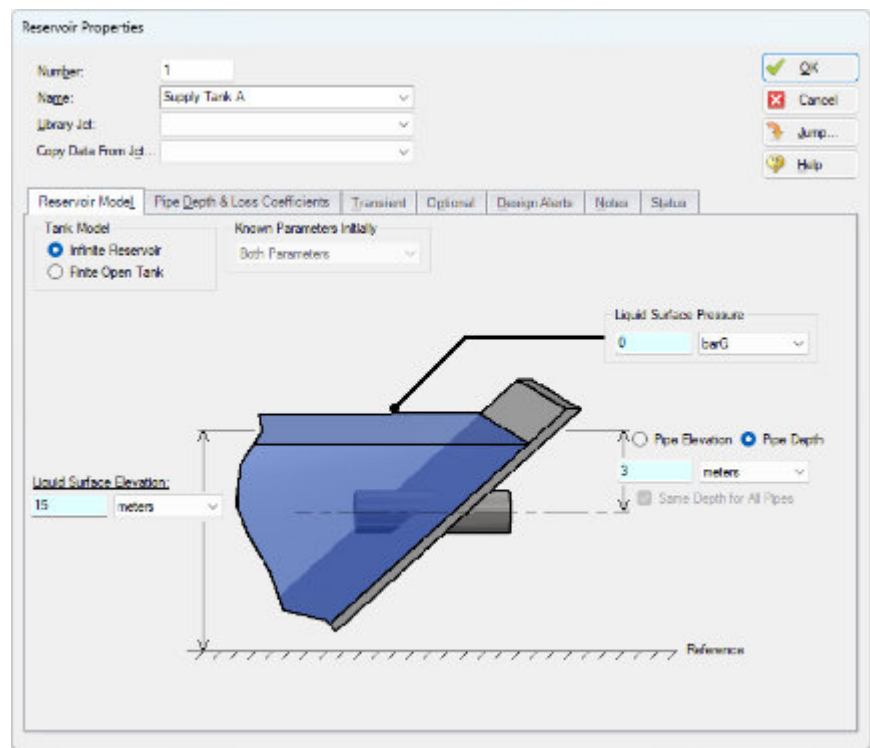
► **Enter the following properties into the J1 Reservoir Properties window:**

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 meters for length) in User Options, found in the Tools menu, under Preferred Units.

You can give the junction a name, if desired, by entering it in the Name field at the top of the window. By default, the junction's name is not specified. If a junction name is entered, it will be displayed in the Model Data, Output, and in Workspace Layers. In Figure 2.13, the name of this Reservoir has been changed to Supply Tank A.

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.



**Figure 2.13 Properties window for the J1 Reservoir**

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 2.13 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.

Alternatively, enter the depth, regardless of the number of pipes, on the Pipe Depth & Loss Coefficients tab (also called the pipe table). The pipe table allows you to specify entrance and exit loss factors for each pipe connected to the reservoir (in this case there is only one). The default selection is for the loss factors to be specified as zero. You can change the loss factors by clicking within the pipe table and entering the loss.

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

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

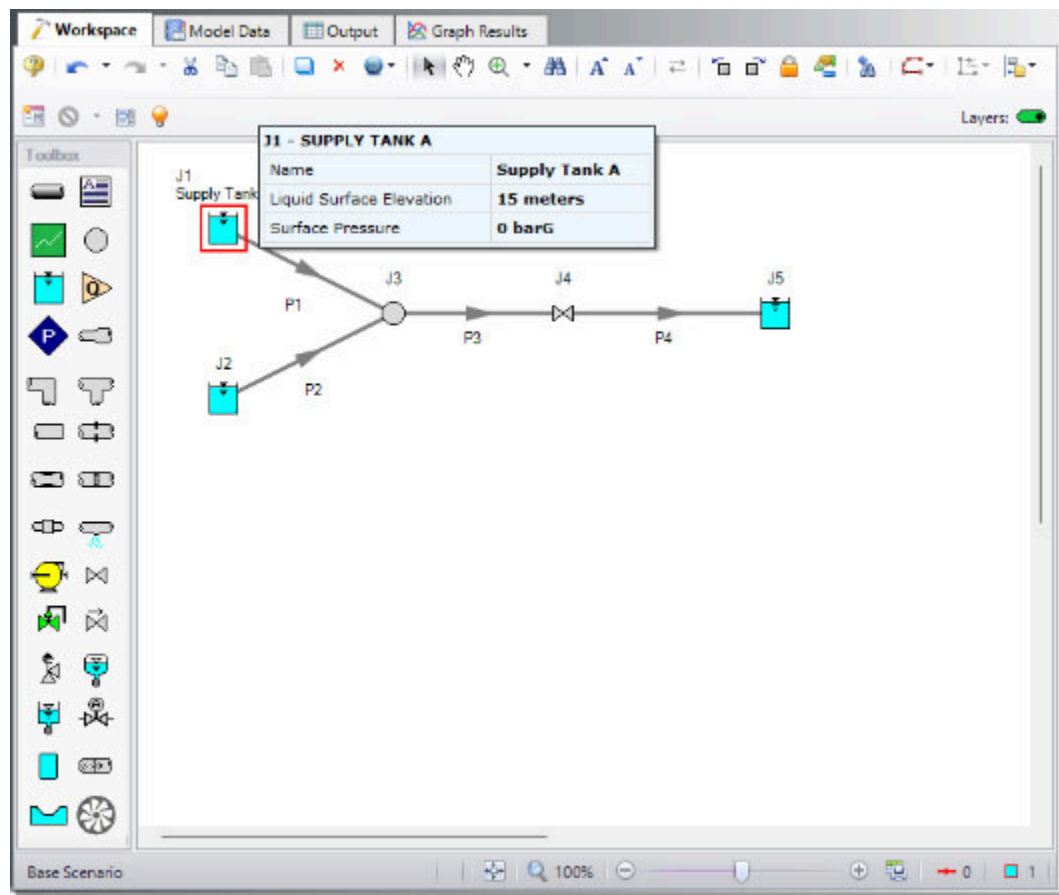
The highlight feature (on by default) displays all the required information in the Properties window in light blue. You can toggle the highlight on and off by pressing F2 with the properties window open or by double-clicking an empty area above the tabs of the properties window. The highlight feature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

- ▶ **Click OK and click Yes** to display the Reservoir name if prompted. With “Show Object Status” turned on, you should see the J1 ID number and name turn black again, informing you that the junction is fully defined.
- ▶ **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 Inspection feature. Position the mouse pointer on the J1 Reservoir icon and hold down the right mouse button. An inspection window appears, as shown in Figure 2.14.

Inspecting is a faster way of examining the input (and output if results are available) than opening the Properties window.



**Figure 2.14** Inspect junction properties from the Workspace by clicking and holding the right mouse button

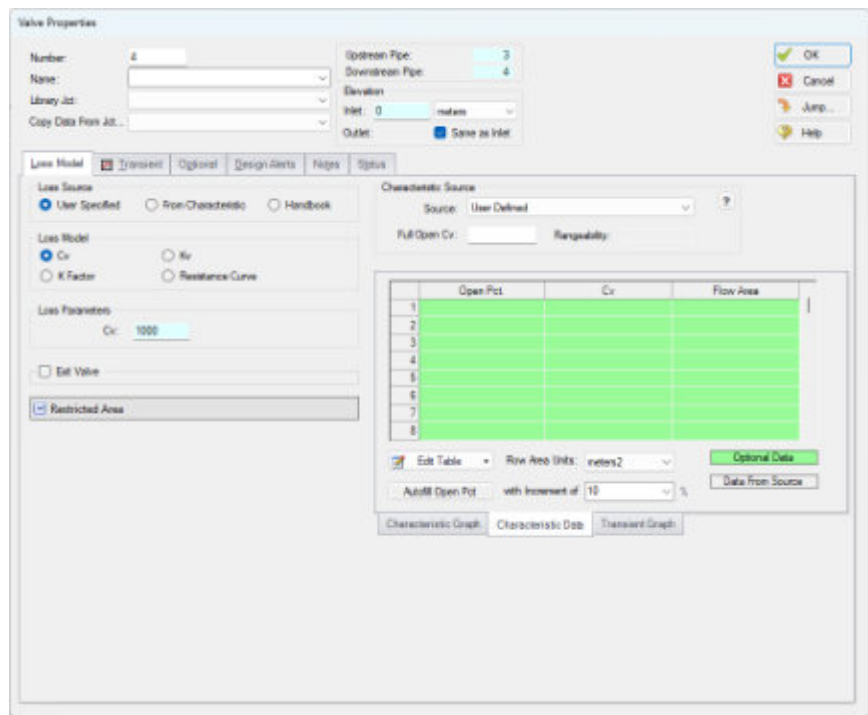
### ***Enter data for the branch***

- **Open the J3 Branch 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.

- **Open the J4 Valve Properties window** and enter the following data (Figure 2.15):
  1. Enter an Inlet Elevation of 0 meters.
  2. Choose the User Specified Loss Source, Cv as the Loss Model, and enter a value of 1000 (this is the steady-state Cv).



**Figure 2.15 Input data for the J4 Valve junction**

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 2.16).

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

Note that the first data point (Cv = 1000 at time zero) must match the steady-state value (entered on the Loss Model tab) and will be automatically populated. The transient data represents the valve as initially open. The valve then closes over a period of one second (a Cv 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 on the Workspace, indicating that transient data is entered for the junction.

Valve Properties

Number: 4  
 Name:   
 Library Jct:   
 Copy Data From Jct:   
 Upstream Pipe: 3  
 Downstream Pipe: 4  
 Elevation:   
 Inlet: 0 meters  
 Outlet: Same as Inlet

Transient Special Condition:   
☒ None  
☐ Ignore Transient State

Initiation of Transient:   
☒ Time  
☐ Dual Event Cycle  
☐ Single Event  
☐ Dual Event Sequential

Transient Data:   
☒ Absolute Values  
☐ Relative To Steady-State Value  
☐ Open Percentage

Data Point	Time (seconds)	Cv
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		

Buttons: OK, Cancel, Jump..., Help

Buttons: Edit Table, Show Graph, Repeat Transient, Transient

**Figure 2.16** Transient input data for the J4 Valve junction

### ***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 2.17).

1. In the Pipe Material dropdown, select Steel - ANSI.
2. In the Size dropdown, select 10 inch.
3. In the Type dropdown, select STD (schedule 40).
4. Specify the Length as 60 meters.
5. Verify the Friction Model is set to Standard.
6. Verify the Pipe Support is set to Thick-Walled Anchored Upstream.

**Figure 2.17 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 junction that states the upstream and downstream pipe in the junction's Properties window. You can jump to that junction's Properties window by double-clicking the connected junction's ID number, or use the Jump button to jump to any other part of the model.

### ***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 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 2.17). Click OK.

### ***Enter data for the 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
3	12	15
4	12	12

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

- **Check for any remaining undefined pipes or junctions.** 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” option 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.

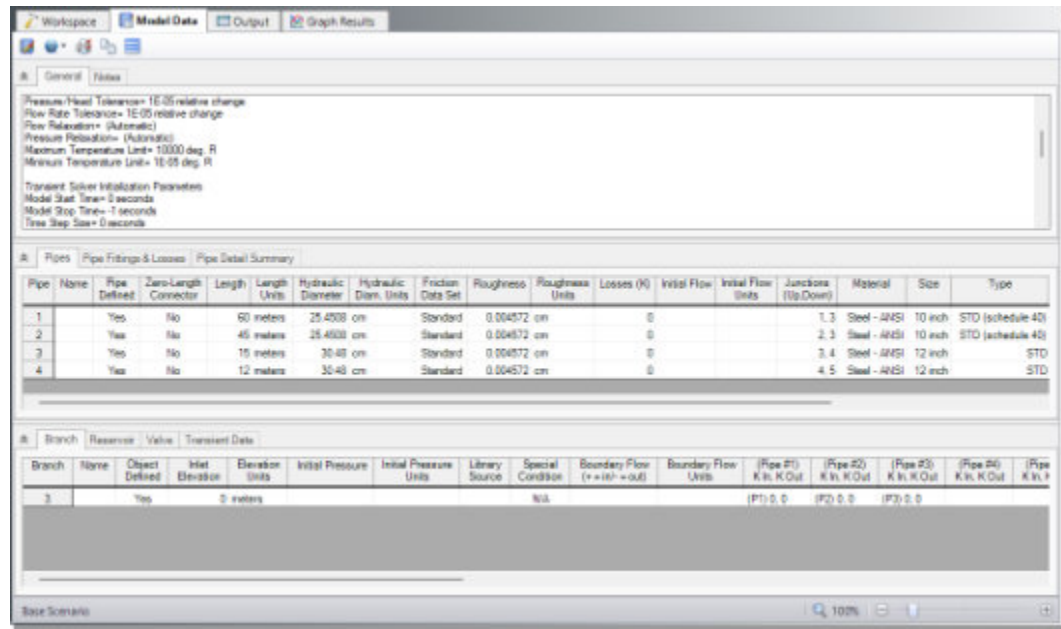
### ***Reviewing input in the Model Data window***

The Model Data window is shown in Figure 2.18. 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 table-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to a PDF, or printed out for review.

Data displays in three sections: the General Data section at the top, the Pipe Data section in the middle, and the Junction Data section at the bottom. Each section collapses using the buttons at the top-left of the section. Further, each section can be resized.

The Model Data window allows access to all Properties windows by double-clicking on any input parameter cell in the row of the pipe or junction you want to access.





The screenshot shows the 'Model Data' window with the following data:

**General / Notes**

Pressure/Head Tolerance= 1E-05 relative change  
Flow Rate Tolerance= 1E-05 relative change  
Flow Relaxation= (Automatic)  
Pressure Relaxation= (Automatic)  
Maximum Temperature Limit= 10000 deg. R  
Minimum Temperature Limit= 1E-05 deg. R

Transient Solver Initialization Parameters  
Model Start Time= 0 seconds  
Model Stop Time= 1 seconds  
Time Step Size= 0 seconds

**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)	Material	Size	Type
1		Yes	No	50 meters		25.4008 cm		Standard	0.004572 cm		0			1.3	Steel - A106	10 inch	STD (schedule 40)
2		Yes	No	45 meters		25.4008 cm		Standard	0.004572 cm		0			2.3	Steel - A106	10 inch	STD (schedule 40)
3		Yes	No	15 meters		30.48 cm		Standard	0.004572 cm		0			3.4	Steel - A106	12 inch	STD
4		Yes	No	12 meters		30.48 cm		Standard	0.004572 cm		0			4.5	Steel - A106	12 inch	STD

**Branch / Reservoir / Valve / Transient Data**

Branch	Name	Object Defined	Initial Elevation	Elevation Units	Initial Pressure	Initial Pressure Units	Library Source	Special Condition	Boundary Flow (x=0 to x=L)	Boundary Flow Units	(Pipe #1) K in, K Out	(Pipe #2) K in, K Out	(Pipe #3) K in, K Out	(Pipe #4) K in, K Out	(Pipe #5) K in, K Out
3		Yes	0 meters					None			(P1) 0.0	(P2) 0.0	(P3) 0.0		

Base Scenario

**Figure 2.18 The Model Data window shows all input in table 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. Pipe Sectioning and Output

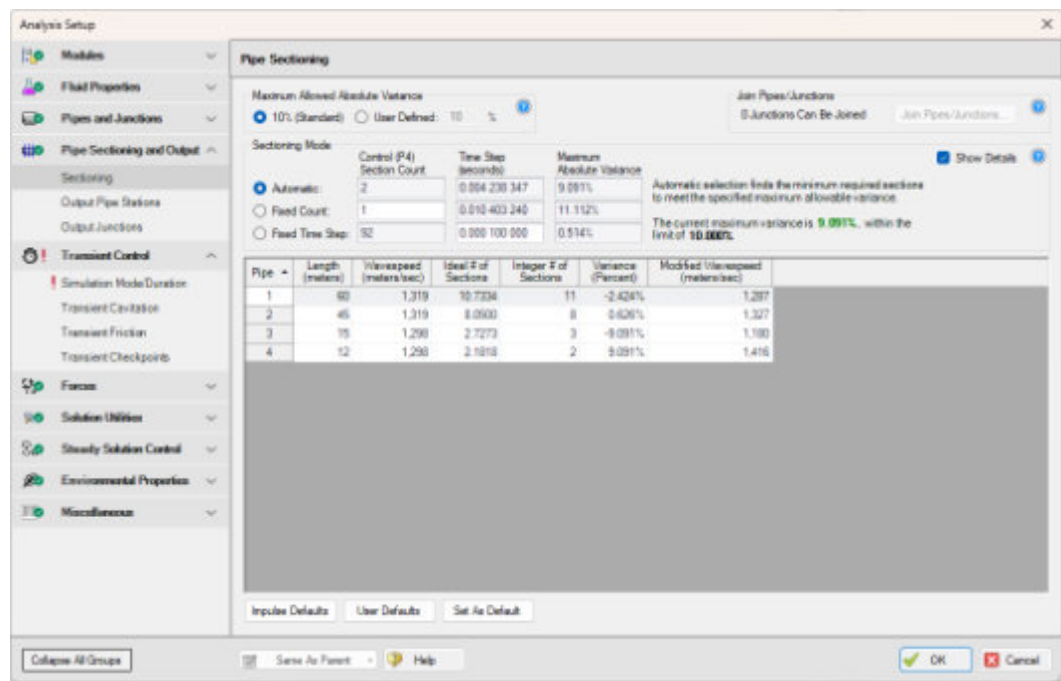
The next group in Analysis Setup is the Pipe Sectioning and Output group. The pipes in the model cannot be sectioned until sufficient data in the Fluid Properties group and Pipe and Junctions group have been specified. The required data includes the fluid specification (this was done in Step 3B when the Fluid panel was defined) and all pipe and junction definitions. After these pieces of 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 2.19). 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, Impulse uses 10% by default.



**Figure 2.19 The Pipe Sectioning panel automates the sectioning process and calculates the time step**

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

The Sectioning panel does not require user input by default. 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. If more sections are needed you can change the Sectioning Mode.

In this example model, the results when the controlling pipe has only one section shows a maximum variance of about 11%. Since this is greater than the default 10% threshold it 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.

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.

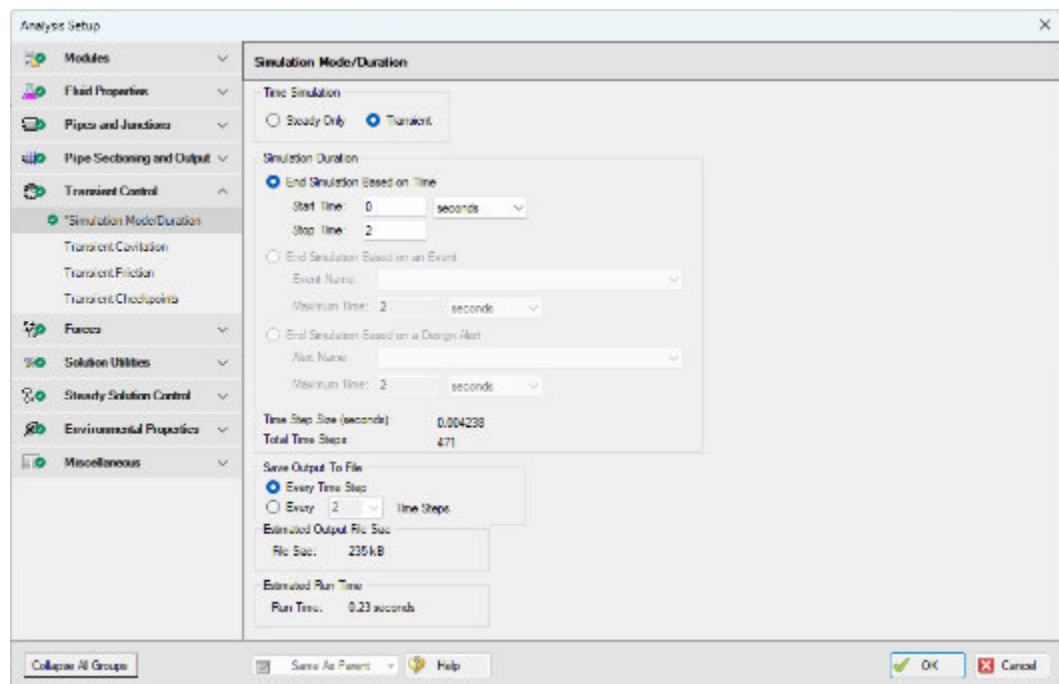
## E. Transient Control

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 2.20.

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

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



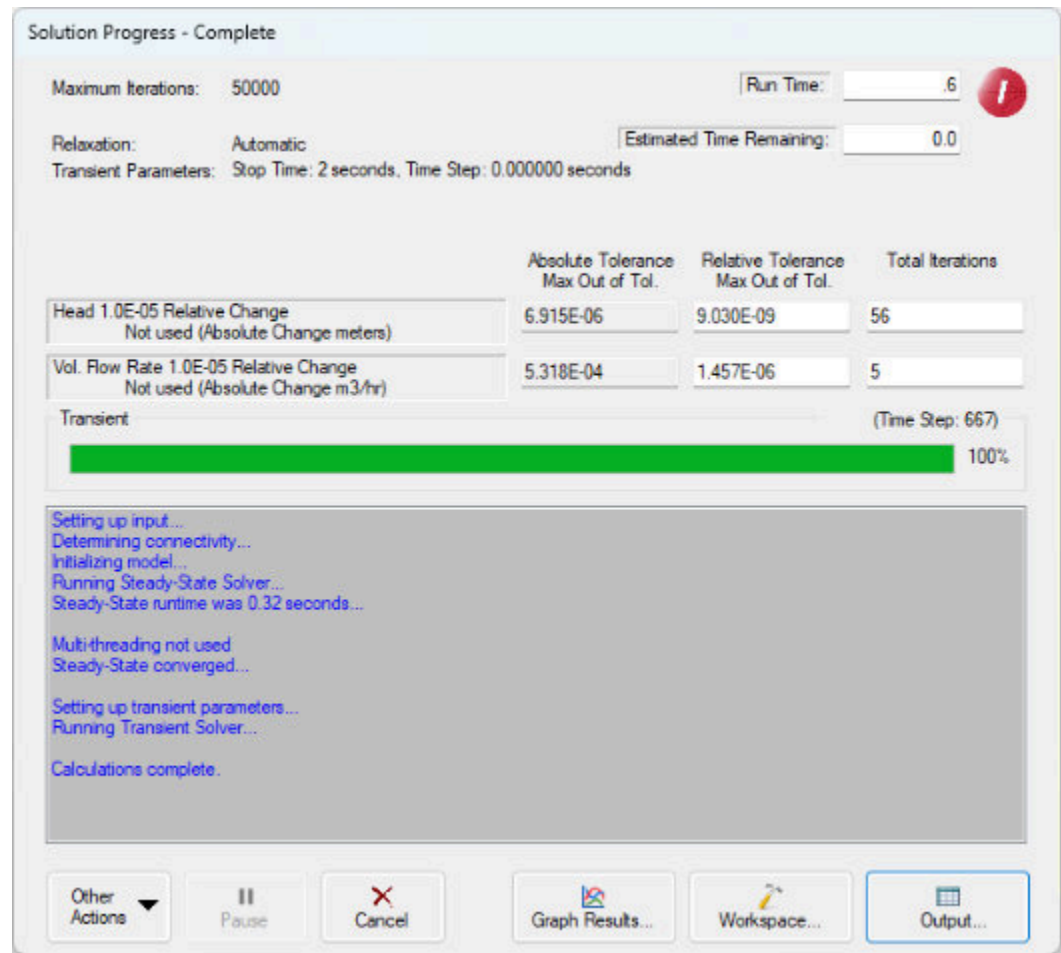
**Figure 2.20** The Simulation Mode/Duration panel offers features to specify the time span for the transient

## 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 model

- Choose “Run Model...” from the Analysis menu or click the Run Model button on the Toolbar. During execution, the Solution Progress window is displayed and can be used to pause or cancel the Solver's activity (Figure 2.21).



**Figure 2.21** The Solution Progress window displays the state of the simulation

## A. The two Solvers

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, 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 table-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 output file

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

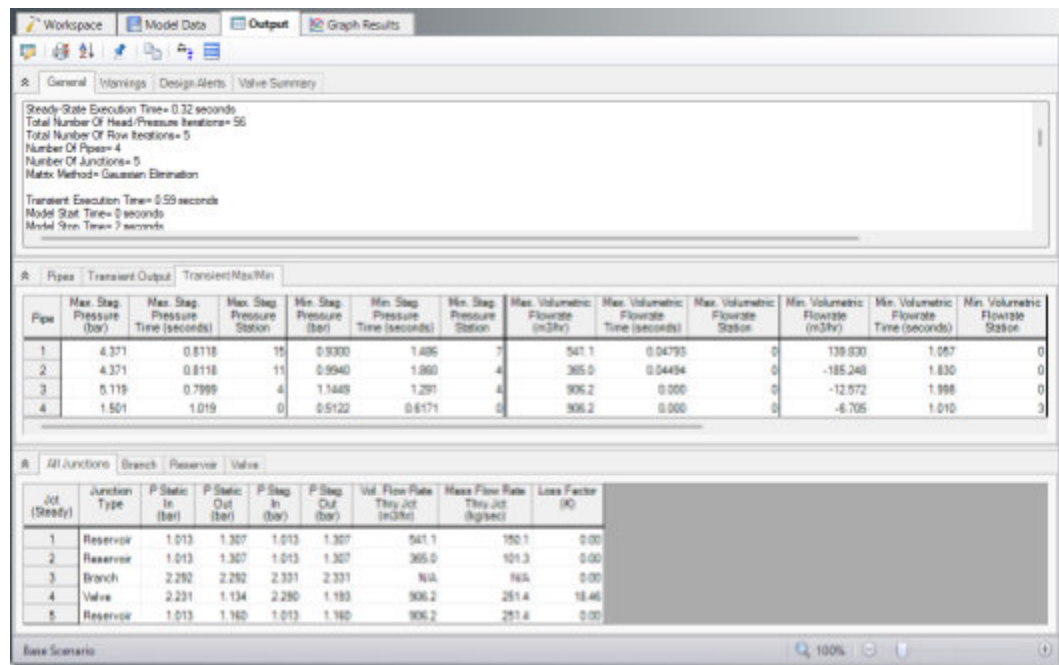
**The output file will only remain on disk if you save the model after it has been run.** Once saved, it will remain until the user erases it or the model input data is modified.

- **Save the model file again** such that the output file is saved to the disk. This means that if you were to close your model right now and then reopen it, you could proceed directly to the Output window for data review without rerunning your model.

## Step 5. Review the Output window

The Output window (Figure 2.22) 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 2.22.



**Figure 2.22 The Output window displays steady-state and transient output in tabular 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 2.23). Moving the slider along the bottom will change the time step shown in the table.

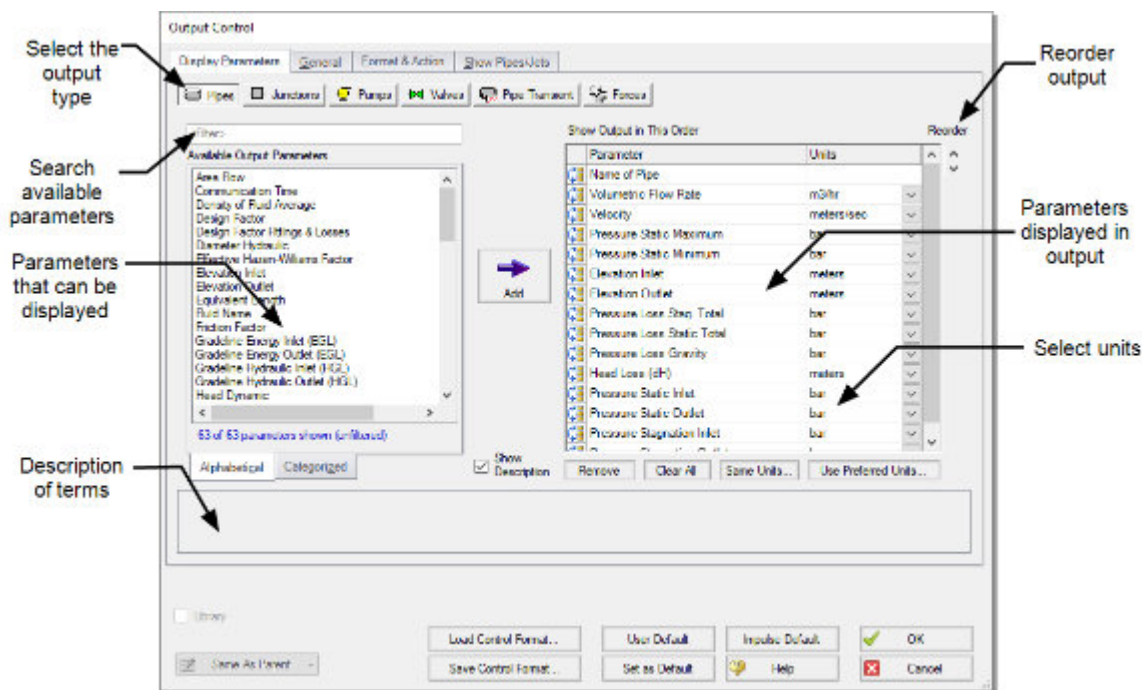
The Output Control window (Figure 2.24) allows you to select the specific output parameters you want to display in your output. You also can choose the units for the output. If you do not change any of the Output Control settings, default Output Control parameters and a default title are assigned.

<div> <span>⌵</span> <span>Pipes</span> <span>Transient Output</span> <span>Transient Max/Min</span> </div>			
<div> Step: 0  Time: 0.00000E+00  Pipe -- Station </div>			
	Stag. Pressure (bar)	Volumetric Flowrate (m3/hr)	Vapor Volume (meters3)
<input type="checkbox"/> 1 -- 0	1.307	541.1	0
1 -- 1	1.375	541.1	0
1 -- 2	1.444	541.1	0
1 -- 3	1.512	541.1	0
1 -- 4	1.580	541.1	0
1 -- 5	1.648	541.1	0
1 -- 6	1.717	541.1	0
1 -- 7	1.785	541.1	0
1 -- 8	1.853	541.1	0
1 -- 9	1.922	541.1	0
1 -- 10	1.990	541.1	0
1 -- 11	2.058	541.1	0
1 -- 12	2.126	541.1	0
1 -- 13	2.195	541.1	0
1 -- 14	2.263	541.1	0
1 -- 15	2.331	541.1	0
<input type="checkbox"/> 2 -- 0	1.307	365.0	0
<input type="checkbox"/> 3 -- 0	2.331	906.2	0
<input type="checkbox"/> 4 -- 0	1.193	906.2	0

⏮
⏪
⏩
⏭
⏴
⏵
⏶
⏷

**Figure 2.23** The Output window displays transient data for each time step in the Transient Output table





**Figure 2.24** Output Control lets you customize the output

### A. Modify the output format

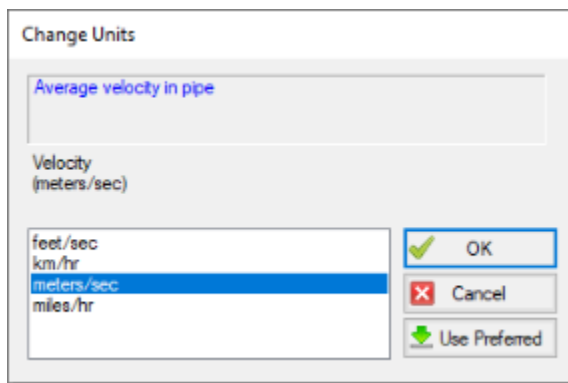
If you selected the default Impulse Output Control, the Pipes table of steady-state results (the tab on the far left in the pipe area) will show volumetric flow rate in the first column with units of m<sup>3</sup>/hr (cubic meters per hour).

- ▶ **Select Output Control from the Tools menu or the main Toolbar.** On the right side of the Pipes section is the list of currently selected output parameters. Click Volumetric Flow Rate and change the units by clicking the arrow beside the units, and then selecting m<sup>3</sup>/sec (cubic meters per second) from the drop-down list.
- ▶ **Click OK to display changes to the current results.** You should see the volumetric flow rate results, still in the first column, in units of m<sup>3</sup>/sec. Notice the Velocity results in the second column.
- ▶ **Select Output Control from the Tools menu** one more time. The Reorder scroll bar on the far right allows you to reorder parameters in the list. You may also reorder parameters by dragging and dropping the icon just to the left of each parameter within the list of currently selected parameters.
- ▶ **Select the Velocity parameter** and use the Reorder scroll bar to move it up to the top of the parameter list.
- ▶ **Click OK to display the changes to the current results.** You will see in the Pipes table that the first column now contains velocity and the second column contains the volumetric flow rate. The Output Control window allows you to control the parameters, units and order you prefer in your output. This flexibility will help you work with Impulse in the way that is most meaningful to you, reducing the possibility of errors.



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

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 2.25). These changes are extended to the Output Control parameter data that is set.



**Figure 2.25** The Change Units window is opened from the Output window tables by double-clicking the column header

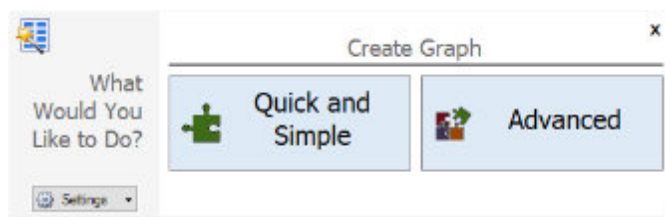
## Step 6. View the graph results

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

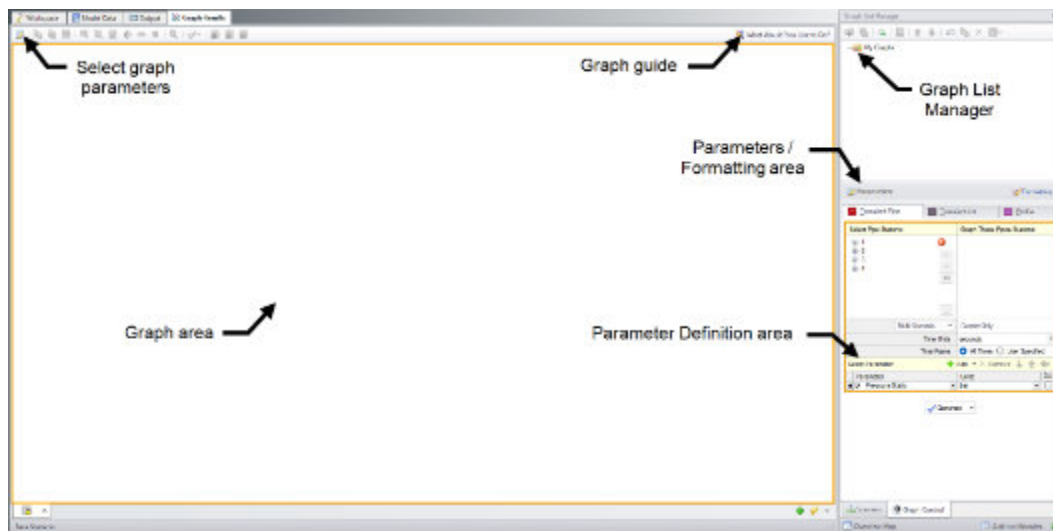
- **Navigate to the Graph Results window** by clicking on the Graph Results tab or pressing CTRL+G. The Graph Results window offers versatile graph generation and preparation.

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 2.26). 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 2.27 shows the Graph Results window with the Graph Control tab enabled on the Quick Access Panel.



**Figure 2.26** 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 2.27** The Graph Results window and Quick Access Panel Graph Control tab is where system parameters (both steady-state and transient) can be graphed

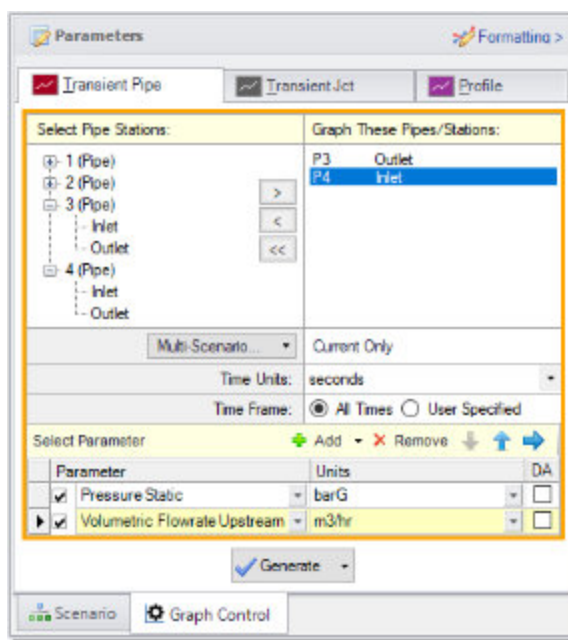
Impulse gives you the ability to create *stacked graphs*. These are graphs that are displayed on top of each other with the same x-axis but with different parameters on the y-axis. This feature is helpful to see how multiple parameters change in the same location of your model at the same time without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

- ▶ **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 Pipe P3 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 drop-down 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 the J4 Valve, 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 2.28).
- ▶ **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 a legend and use the scroll bar to set the font size to 16 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 set the size to 20.

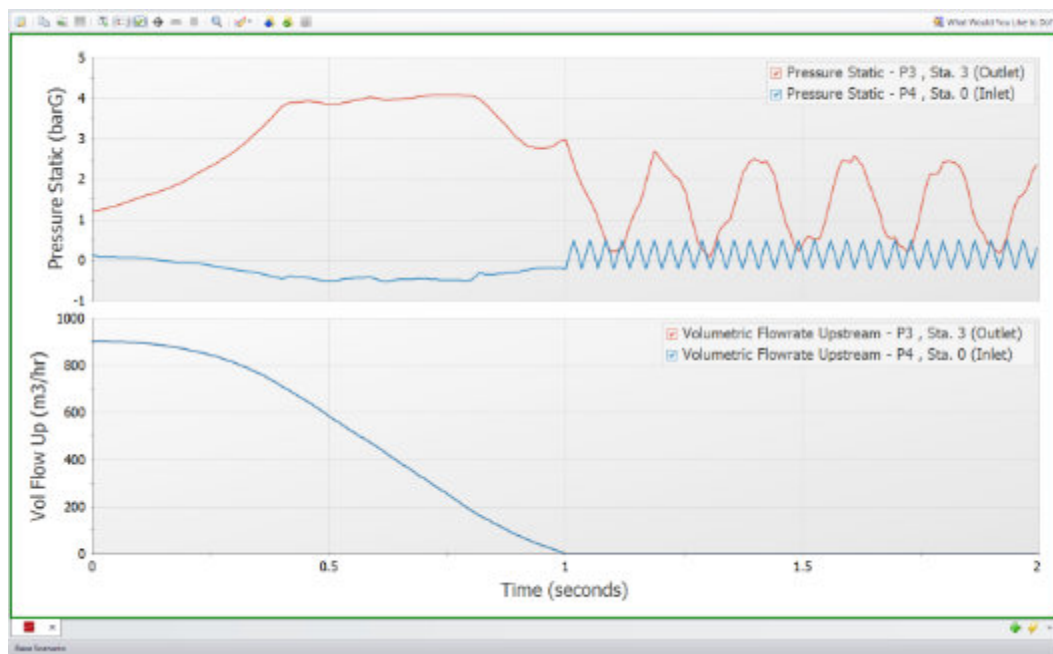
Figure 2.28 shows the input in the Parameters/ Formatting area on the Quick Access Panel.



**Figure 2.28** The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area

The Formatting area on the Graph Control tab can be used to modify the graph colors, fonts, and other elements. Options to copy the graph image or x-y data are available on the Graph Results Toolbar or by right-clicking the graph. The graph image or x-y data can be saved from the File menu.

Figure 2.29 shows the stacked graphs detailing the static pressures and volumetric flowrates at the J4 Valve 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 2.29** Static pressure and volumetric flowrate at the J4 Valve inlet and outlet are shown over 2 seconds

### A. Further review

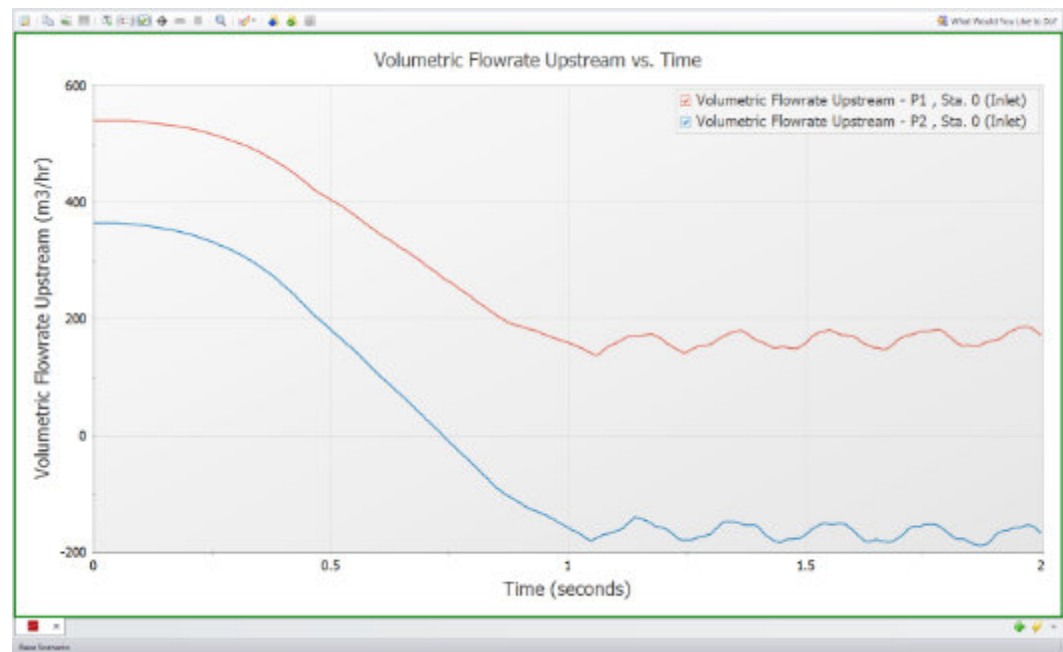
Further review of the valve graph results in Figure 2.29 shows that at time zero, the pressure difference between the top and bottom curves of the pressure graph is about 1.1 barG. This is the steady-state pressure difference across the valve, which can also be found in the Output window to be 1.096 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 flowrate 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 2.30).

- ▶ **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 “m<sup>3</sup>/hr”. Remove the Pressure Static parameter by selecting the row and clicking the Remove button.
- ▶ **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 the J1 Reservoir flows to the J2 Reservoir (Figure 2.30).



**Figure 2.30 Transient flowrates out of the reservoirs**

**Note:** 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 J4 Valve inlet and outlet, you can right-click on the J4 Valve 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 Workspace Layers

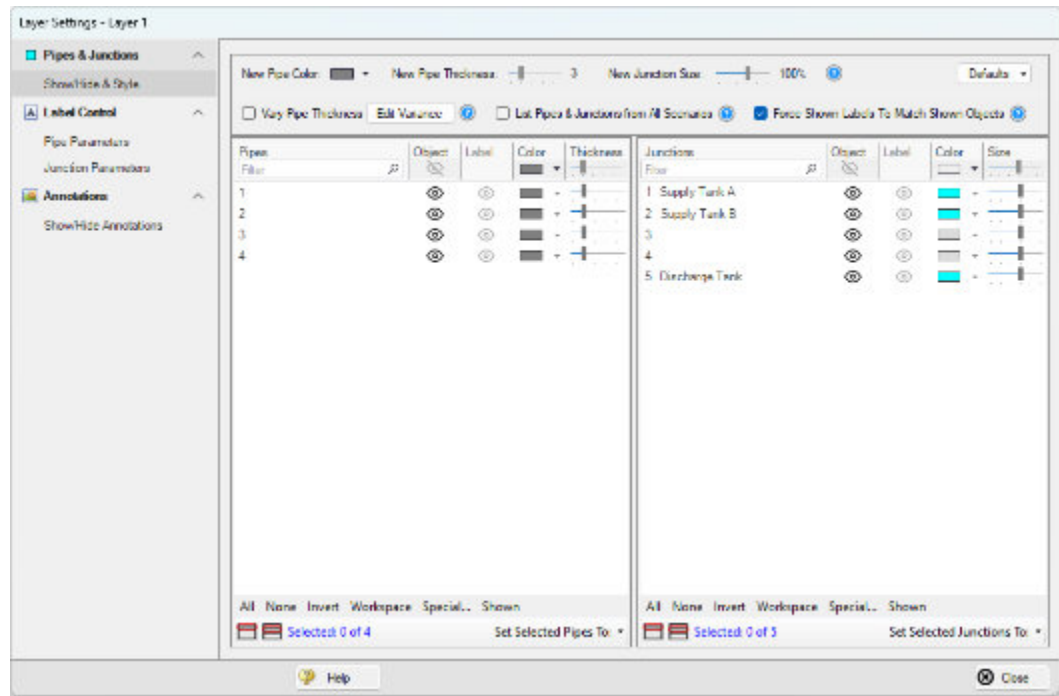
Workspace Layers can display input and output data values superimposed over the graphical model layout. They can also overlay additional graphical modifications, animations, and customize the appearance of objects in the model. The displayed data can be organized on one or more layers which can be toggled on or off, allowing flexibility for viewing results in a graphical manner.

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

Within the Layers section, there are two *layer groups* named Advanced Layers and Standard. We will integrate the model results with the graphical layout of the pipe network by using a Standard layer. All models must have at least one Standard layer, so by default there will be a "Layer 1" automatically created. It will contain all objects in the model (pipes, junctions, annotations, etc); however, objects can be hidden if desired, or the visibility of the entire layer itself can be toggled on/off using the eye icon next to the layer. When additional layers are created they will stack on top of each other, with the layers on top taking precedence over others below. The combination of layers and toggling visibility allows for a wide range of visual customization.

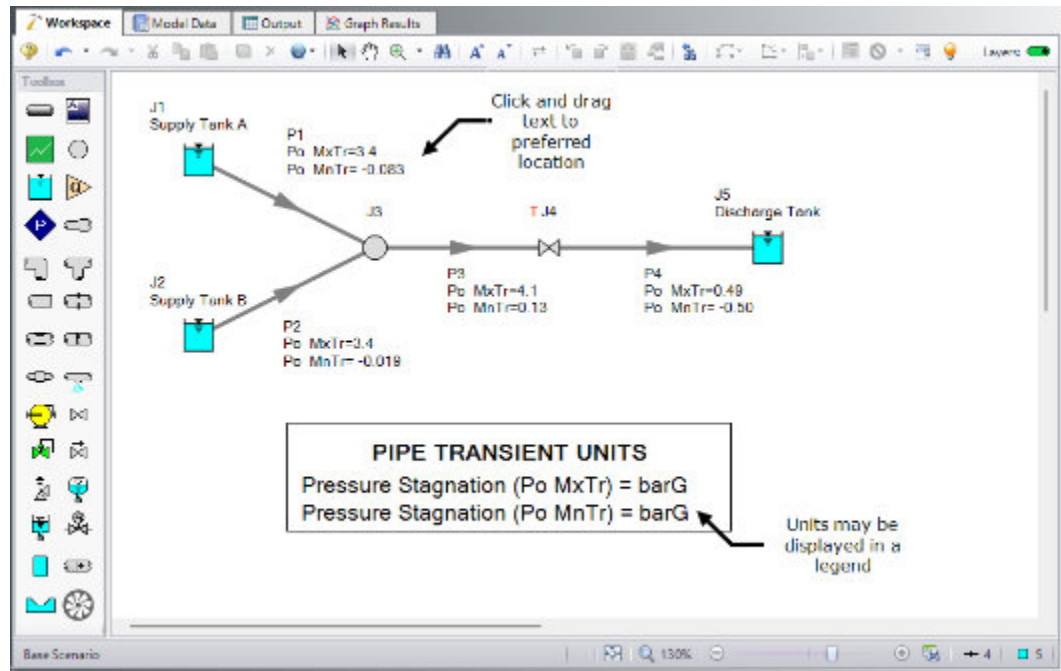
- **Open the Layer Settings window for Layer 1 by double-clicking the layer.** The Layer Settings window is shown in Figure 2.31.

The Layer Settings window provides flexible options for customizing the appearance of the model. The Pipes & Junctions group allows you to show or hide junctions, and change display properties like color, size, and thickness. The Label Control group allows you to display input and output parameters next to objects on the Workspace. The Annotations group allows you to customize annotations.



**Figure 2.31 The Layer Settings window specifies content to be superimposed on the Workspace**

- ▶ **Select the Pipe Parameters item under the Label Control group.** The left pane lists input and output parameters available for display, and the right pane lists the parameters currently being displayed by the layer on the Workspace.
- ▶ **In the left pane, expand the Pipe Max/Min Parameters list.** Then, select Max Pressure Stagnation in the list with units of barG. Add it to the right pane by either double-clicking the parameter, or selecting the parameter and clicking the arrow between the panes. Repeat the process to add the Min Pressure Stagnation parameter.
- ▶ **Add additional clarity by repositioning the labels and adding a legend.** It is common for the text on the Workspace Layers to overlap with objects when first generated. Labels can be dragged to a new area or a smaller font can be selected using the Font Decrease icon on the Workspace Toolbar or from within the Global Layer Settings. Parameter units can be consolidated in a legend which can be added via the Global Layer Settings. In the Layers Toolbar, click the double gear icon to open the Global Layer Settings. Then, enable the checkbox for Show Units in Legend. The result of these adjustments is shown in Figure 2.32.

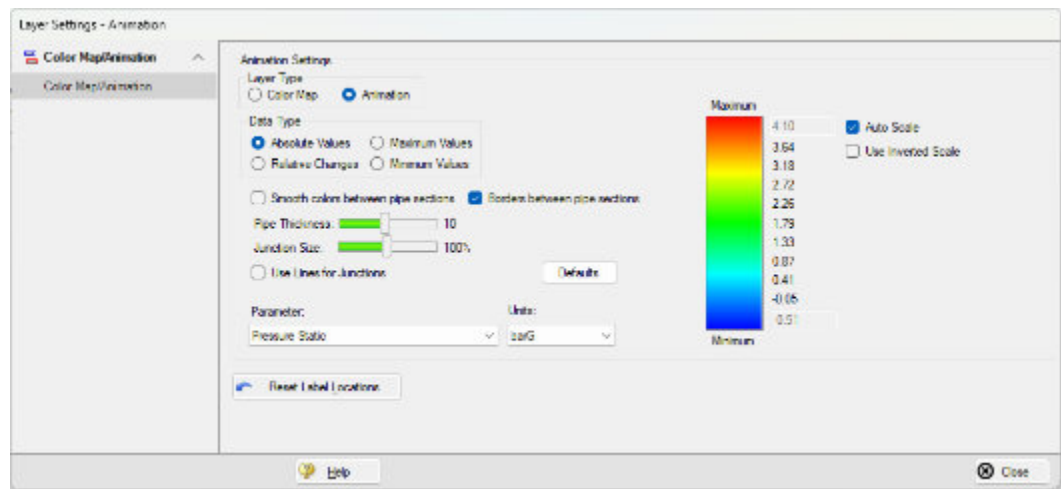


**Figure 2.32 Results are superimposed on the model layout via Workspace Layers**

The Workspace Layers also provides the ability to animate the transient results as a qualitative tool to visualize the behavior in a network. This option can be accessed from the Color Maps/Animations layer type.

- ▶ **Create a new Color Map/Animation layer** using the green plus icon (+) on the Layers Toolbar in the Quick Access Panel.
- ▶ **Open the Layer Settings window and select the Animation option** under the Layer Type. The other Standard Workspace Layers will be visible, but only one Advanced Layer can be shown at a time. Only one parameter can be animated at a time. The 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.
- ▶ **Select Pressure Static as the parameter for the animation using units of barG.** The model setup for animating static pressure can be seen in Figure 2.33.
- ▶ **Close the Layer Settings and add additional clarity by repositioning the legend and time counter.** By default the legend will be drawn in the top-left corner of the Workspace. You can click and drag the legend and time counter to new positions on the Workspace.
- ▶ **Watch the animation by clicking Play** on the animation playback controls at the top of the Workspace. The sliders and plus/minus icons can be used to skip to a different time step or increase the playback speed. The animation can be saved to a file using the Record button.





**Figure 2.33** Workspace Layers provides an animation feature as an alternate way to view the transient results

## Conclusion

You have now used Impulse's Primary Windows to build a simple waterhammer model. The maximum pressure at the valve's inlet during this valve closure was approximately 4.1 barG.

## CHAPTER 3

# 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 the Scenario Manager
- Adding Graphs to the Graph List Manager
- Using animation features

### Required knowledge

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

### Model file

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

C:\Users\[username]\Documents\Datacor\Impulse 11\Examples

- *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 Impulse to understand how the system will respond in each case.

### Step 1. Define the Fluid Properties group

1. Open Analysis Setup using the button on the Common Toolbar or from the Analysis menu.
2. Open the Fluid panel, select the Impulse 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 Impulse is first opened.

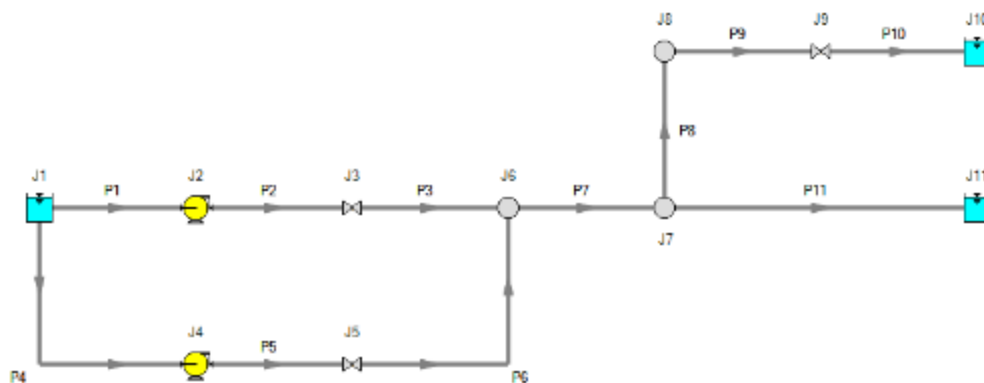
---

### Step 2. 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, or you can select the pipe and use the keyboard shortcut CTRL+SHIFT+Y. 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 needs 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, 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

<b>Time (seconds)</b>	<b>Pump Speed (Percent)</b>
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 on 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. Set the "Loss Source" to "User Specified"
5. Set the "Loss Model" to "Cv"
6. Cv = 1000
7. Click the Transient tab
8. 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. Set the "Loss Source" to "User Specified"
4. Set the "Loss Model" to "Cv"

## 47 Impulse 11 Quick Start Guide

5.  $C_v = 500$
6. Click the Transient tab (Figure 3.2)
7. Initiation of Transient = Single Event
8. Event Type = Pressure Stagnation at Pipe
9. Condition = Greater Than
10. Value = 3.1 barG
11. Pipe = 7 (Pipe) Outlet
12. 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 Jet:  
Copy Data From Jet...

Upstream Pipe: 9  
Downstream Pipe: 10  
Elevation:  
Inlet: 0 meters  
Outlet: Same as Inlet

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.

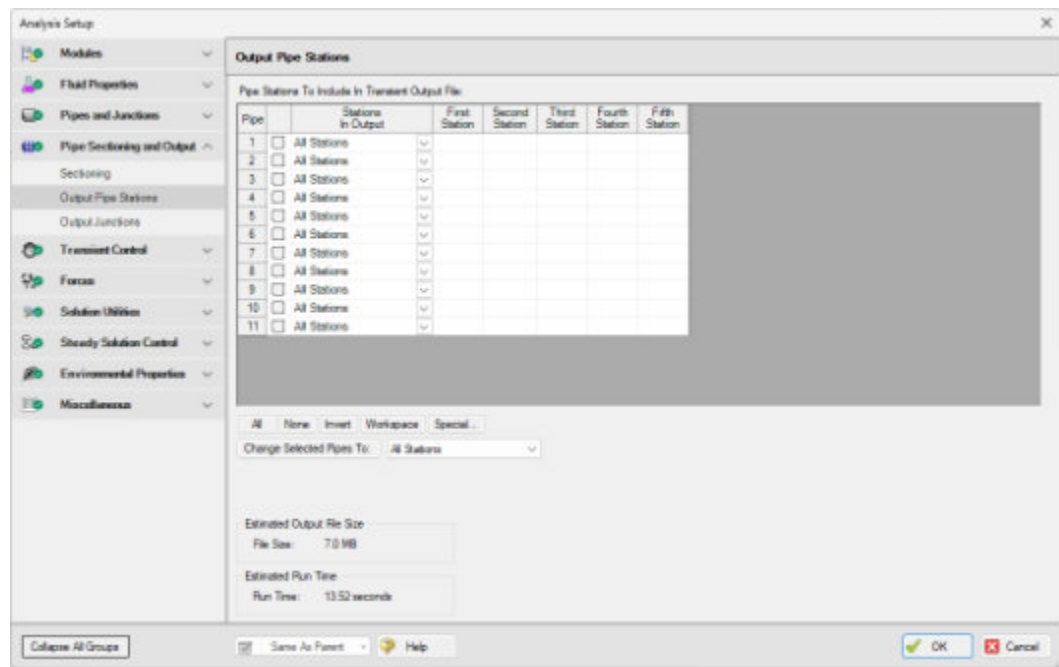
---

## Step 3. Review the Pipe Sectioning and Output group

Open Analysis Setup and go to the Sectioning panel. By default, the best sectioning scheme will be automatically calculated and selected. This is determined by searching for the minimum number of sections in a controlling pipe that will satisfy a maximum wavespeed variance of 10%. The sectioning parameters will be displayed in the table. Keep the default selection of the Automatic sectioning mode.

Navigate to the **Output Pipe Stations** panel. Ensure that all pipes are set to save all stations to the output file. This is the default behavior, but it could differ due to a setting in your User Options. If the pipes are not set to All Stations, then this can be quickly done in bulk by clicking the All button below the list, setting the Change Selected Pipes To drop-down to All Stations, and clicking the Change Selected Pipes To button. This will be useful later for animation purposes.





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

## Step 4. Specify the Transient Control group

Click 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 select the No Cavitation option.

## Step 5. 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

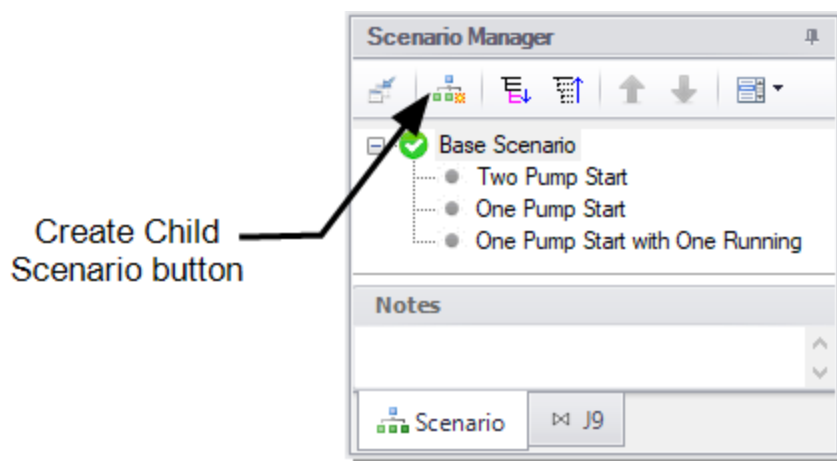
The Scenario Manager is a powerful tool for managing variations of a model, referred to as scenarios. The Scenario Manager allows you to:

- 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 differences between scenarios using the Scenario Comparison Tool
- Propagate changes from a parent scenario to its children

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

- **Click the “Create Child” button on the 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, select the Base Scenario, and 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 Create model variants and keep them organized using the Scenario Manager on the Quick Access Panel**

## 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 set up with two pumps starting, the “Two Pump Start” child does not need to be modified.

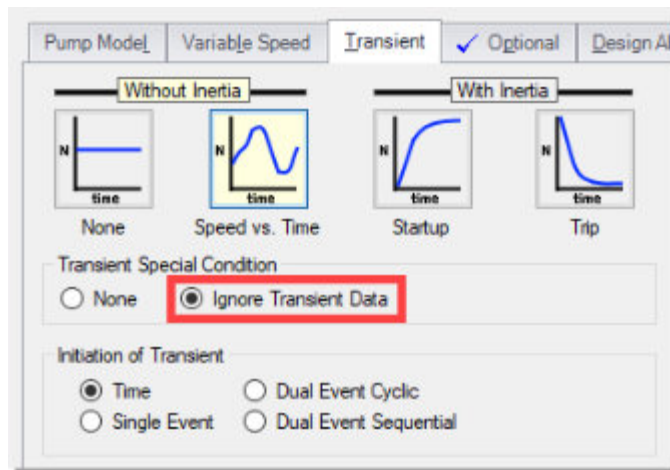
### *One Pump Start*

Double-click the “One Pump Start” scenario in the scenario tree. 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 Pump J4 indicating that the transient data is ignored. Do the same with Valve J5. 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. 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 6. Run the model

Load the "Two Pump Start" scenario. All groups in Analysis Setup should now be complete and the model is ready to run. Run the model by pressing the "Run Model" icon on the Common 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 "Graph Results" button.

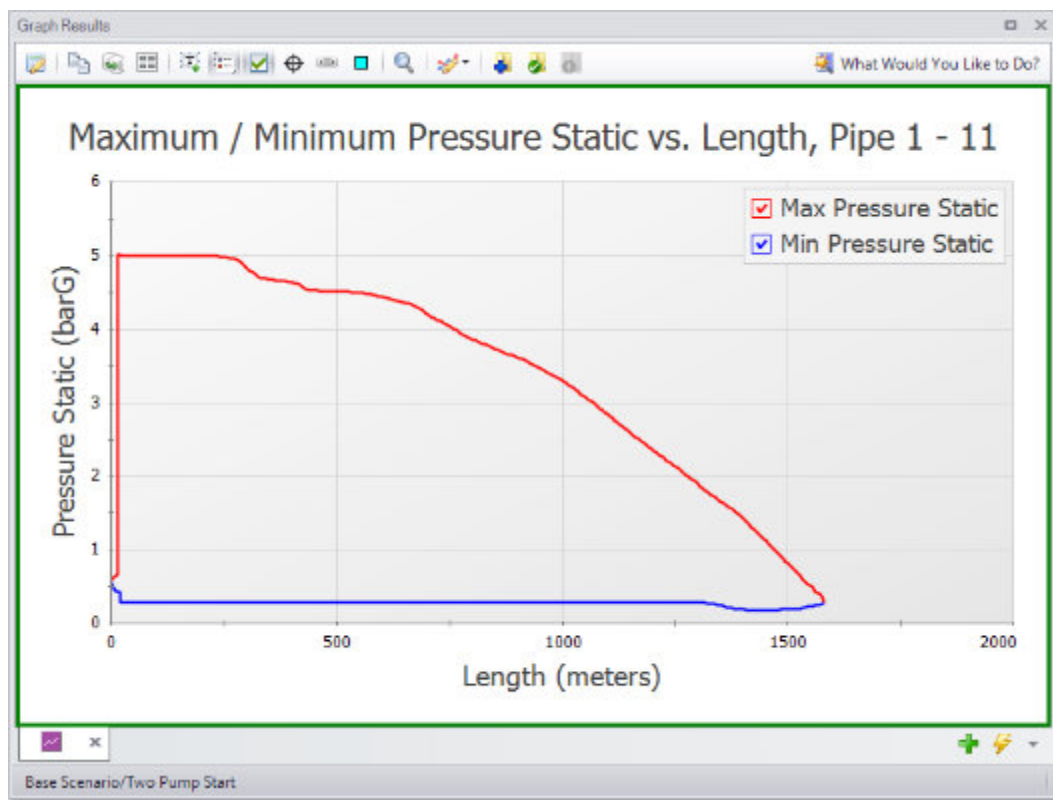
## Step 7. Graph the results

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.



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

To easily recreate this graph in other scenarios or in the future, it can be added to the Graph List Manager. 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.

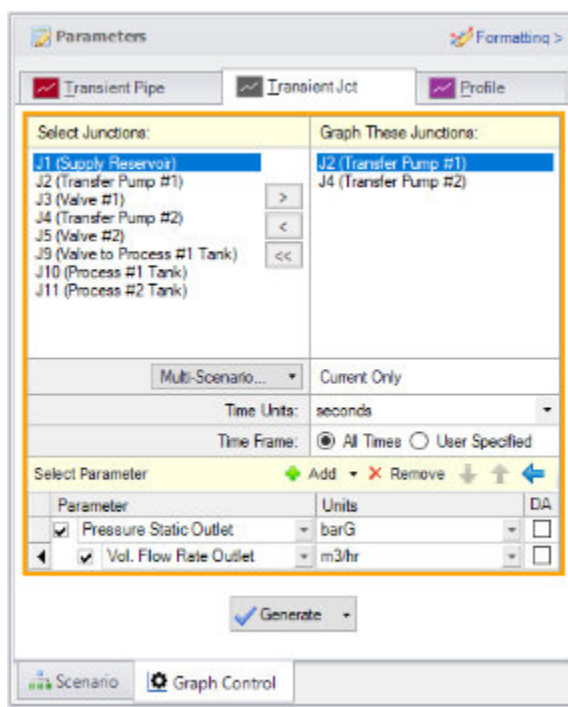
## B. Create a Transient Junction graph

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 Impulse’s dual y-axis graphing abilities to plot both the pressure and volumetric flowrate 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.

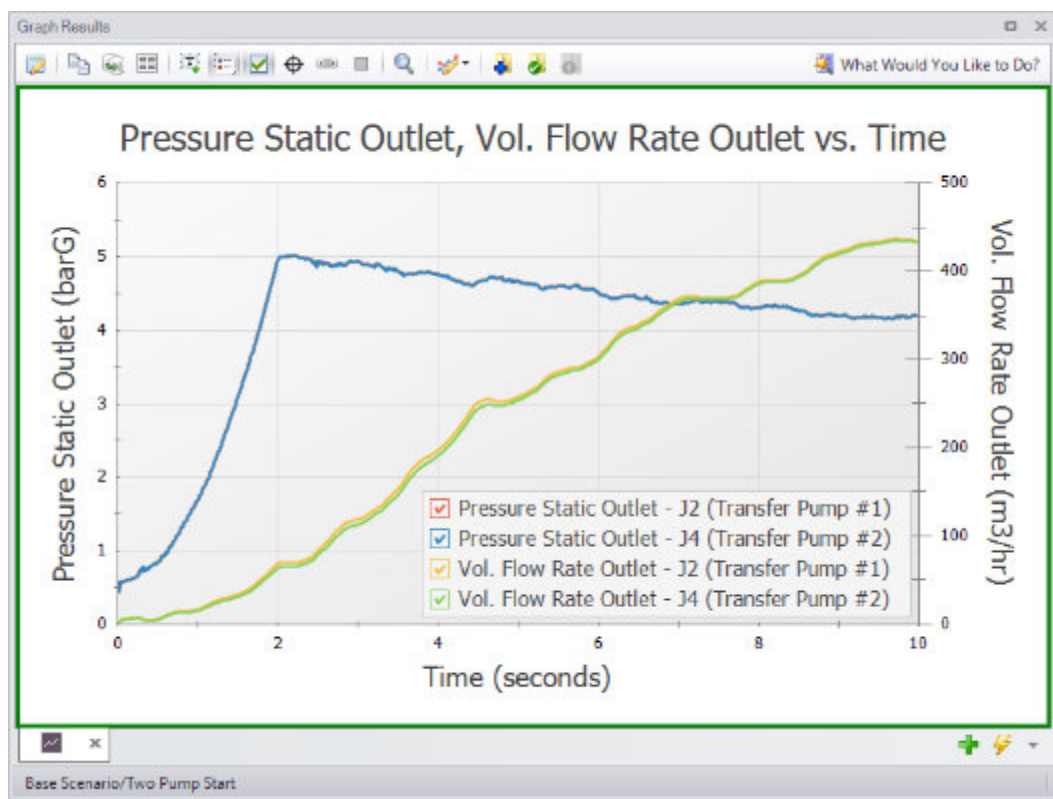
3. To do this, select “Pressure Static Outlet” in the Parameters definition area and choose “barG” as the units.
4. Add a parameter by clicking on the “Add” button next to “Select Parameter” in the Parameter definition area.
5. Change this added parameter to “Vol. Flow Rate Outlet” and choose “m3/hr”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



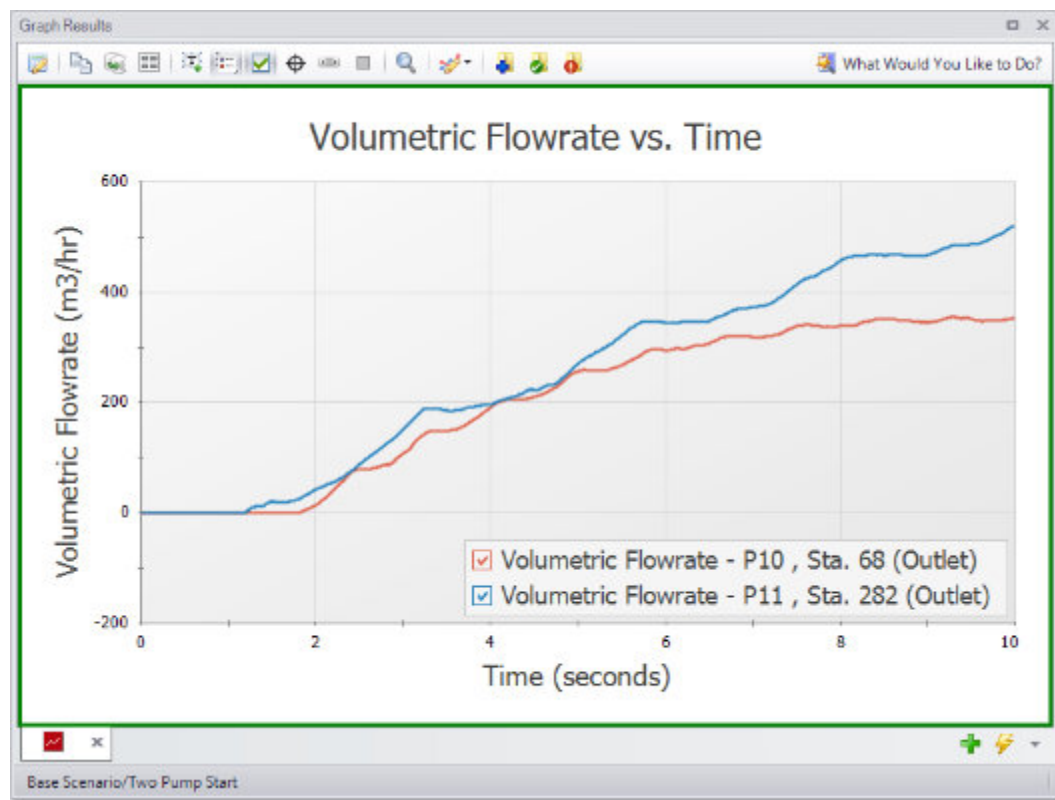
**Figure 3.8** Pressure and volumetric flowrate transients at both pump discharge locations for the Two Pump Start scenario

### 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 graph to the Graph List Manager with the name “Process Tank Flow Rates”.



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

## Step 8. Animate the results

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.

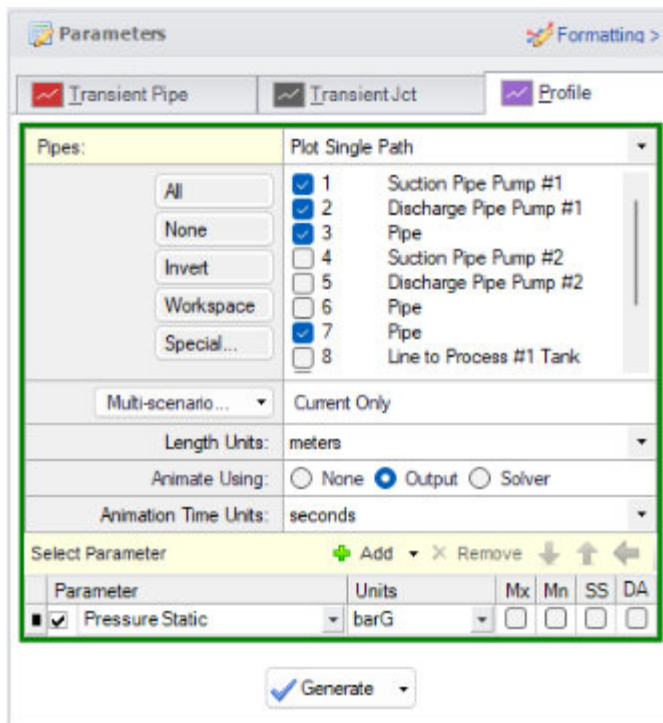
Here, you will animate the pressure and volumetric flowrate responses along the path you previously graphed to gain an understanding of how these hydraulic parameters respond as the pumps start up. To animate these parameters:

1. Load the Graph List item you previously created for the pressure profile by double-clicking the graph name in the Graph List Manager (Pressure Profile Pump #1 to Process Tank #2).
2. Make sure pipes 1, 2, 3, 7, and 11 are selected.
3. Make sure the Length Units are meters.
4. Next to “Animate Using”, specify “Output”, and specify “seconds” next to Animation Time Units (see the following paragraph for an explanation about this option).
5. Uncheck the boxes for Mx and Mn so that the maximum and minimum values will NOT be displayed (See Figure 3.10).



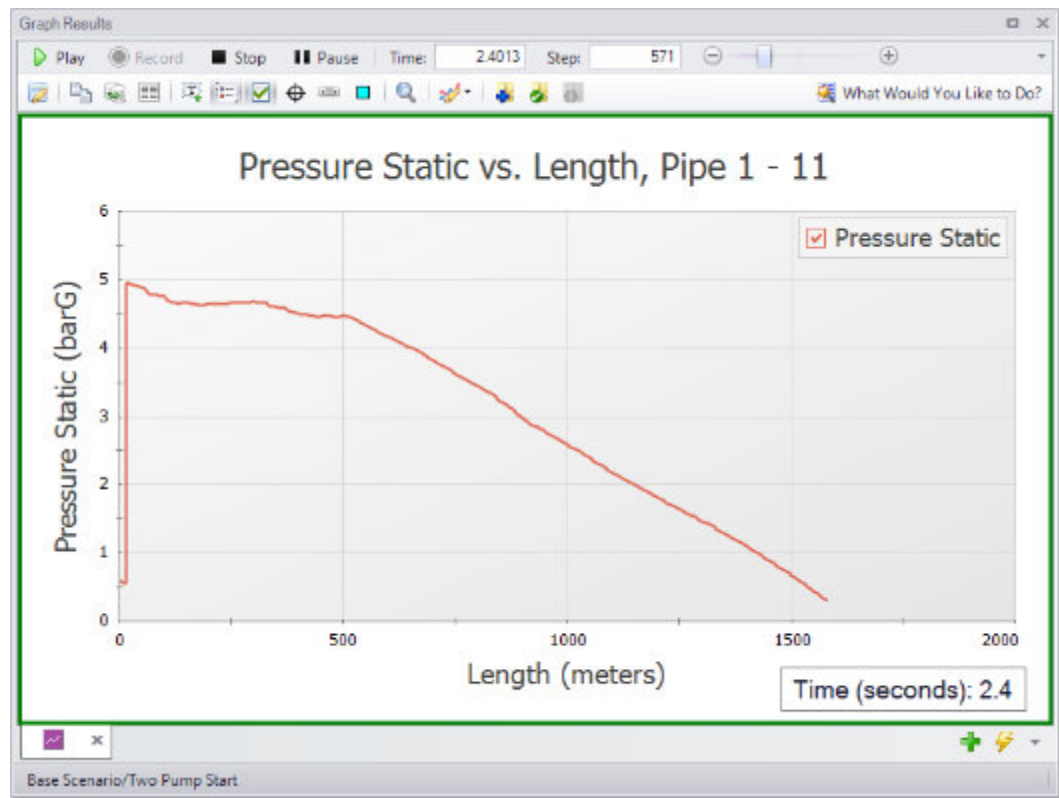
To use the “Animate Using: Output” option for animation, all pipe stations need to be specified to be saved in the Output Pipe Stations panel in the Pipe Sectioning and Output group (this was done previously in Step 4). This animation option has the most flexibility to start, pause, speed up, skip forward/backward,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

Another animation option in Figure 3.10 is “Animate Using: Solver”. This option re-runs the Transient Solver to generate the data for animation, and thus does not need to read it from the output file. Therefore, it does not require all of the pipe station data to be saved in Transient Control. However, it cannot be sped up or skipped forward due to the calculations happening in real time.



**Figure 3.10 Selecting animation in Graph Results for the 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 move. Add this graph to the Graph List Manager with the name “Animated Pressure Profile”.



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

There is also animation available via the Workspace Layers, which can be useful to see how the flow paths interact with each other.

- ▶ **Go to the Workspace window** and select Workspace Layers in the Quick Access Panel. Select New Layer, indicated by the green plus sign in the Layers section. Select Color Map/Animation and give the New Layer a name such as Static Pressure Animation. Open up the Layer Settings with the gear icon.
- ▶ Set the Layer Type to Animation and specify the following settings (Figure 3.12):
  1. Data Type = Absolute Values
  2. Smooth colors between pipe sections = Checked
  3. Parameter = Pressure Static
  4. Units = barG

Close the Layer Settings. The Animation should now be visible on the Workspace. Note that the animation toolbar again appears, similar to when using Graph Animation.

- ▶ **Press the Play button** to watch the pressure transients in the system.

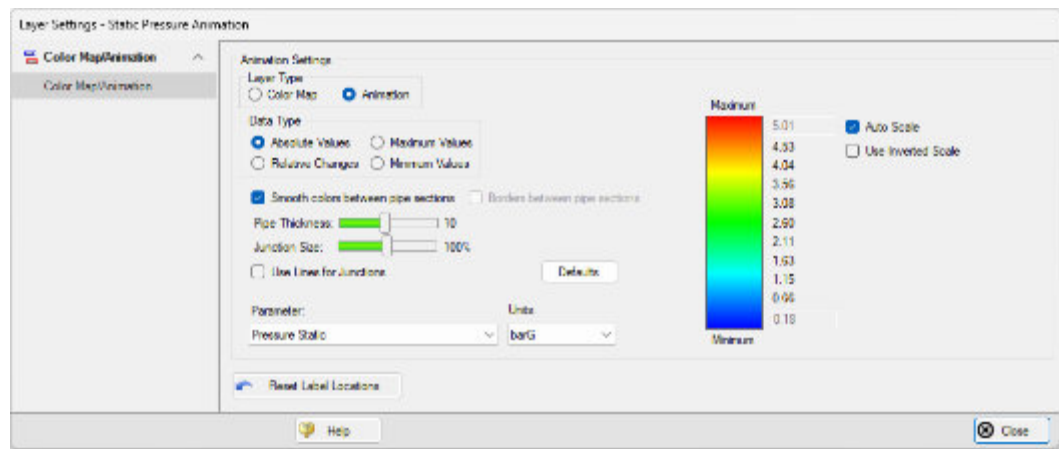


Figure 3.12 Workspace Animation Settings

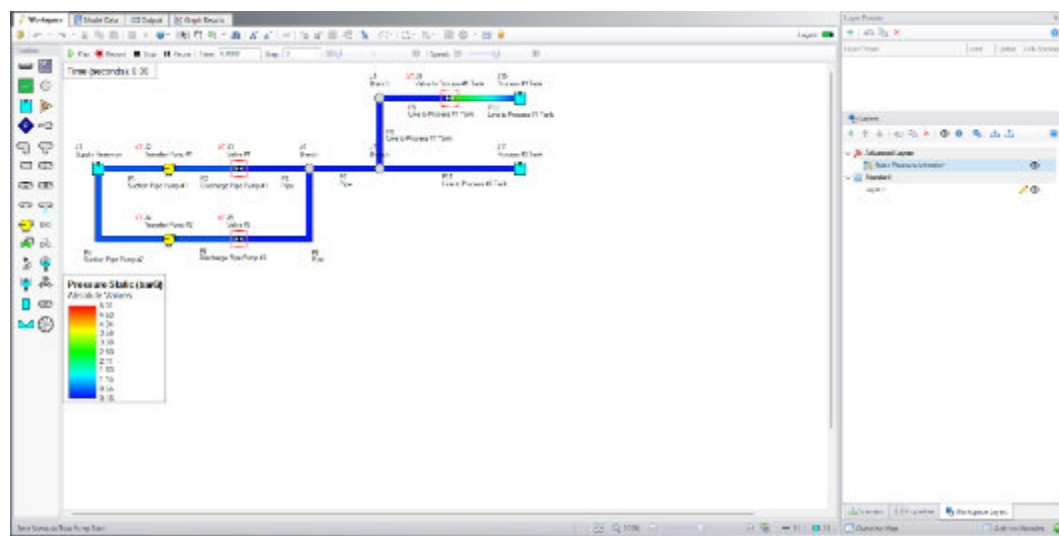


Figure 3.13 Workspace Animation

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

Using Scenario Manager, load the other two scenarios and run them. Reload the Graph List items that were previously saved 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 (between J7 and J11 on the graph). This can also be seen from the Transient Max/Min tab of the Pipe Output section on the Output Primary Window. Also, the pressure drops below atmospheric pressure, but does not reach the vapor pressure of water (-0.995 barG).

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 that reads “Warnings Exist”. The warning can be viewed in the top third section of the Output window on the Warnings tab, and states that there

was reverse flow at Pump J4. 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 static pressure during the transient 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 transient static pressure for the three scenarios**

<b>Case</b>	<b>Max. Static Pressure (bar)</b>
Two Pumps Start	6.028
One Pump Start	5.944
One Pump Start With One On	6.079

## CHAPTER 4

# 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 curve 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

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

### Model file

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

C:\Users\[username]\Documents\Datacor\Impulse 11\Examples

- *Metric - Pump Trip with 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. Define the Fluid Properties group

1. Open Analysis Setup using the button on the Common Toolbar or from the Analysis menu.
2. Open the Fluid panel, select the Impulse 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 2. 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.

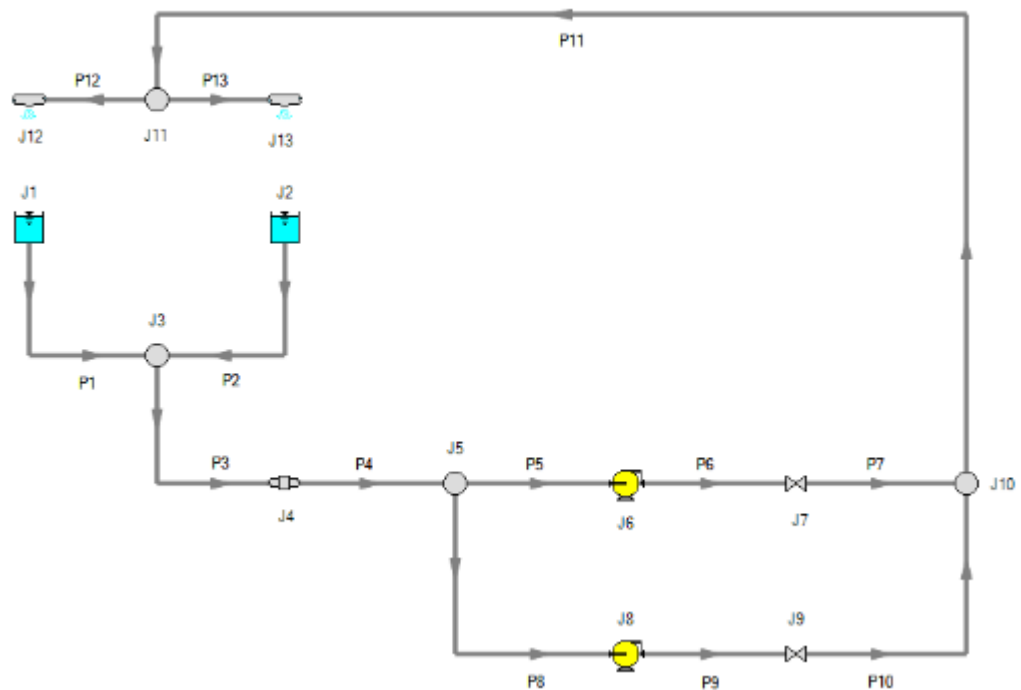


Figure 4.1 Layout of the cooling tower system

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

The system is in place, but now the input data for the pipes and junctions needs 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 =

<b>Volumetric</b>	<b>Head</b>
<b>m3/hr</b>	<b>meters</b>
0	0
4542	6.7
9085	26.8

5. Curve Fit Order = 2
6. Click Generate Curve Fit Now, then click OK

#### ***J5 & J10 - Branches***

Elevation = 0 meters

#### ***J11 - Branch***

Elevation = 15 meters

#### ***J6 & J8 - Pumps***

1. J6 Name = CWP-1
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



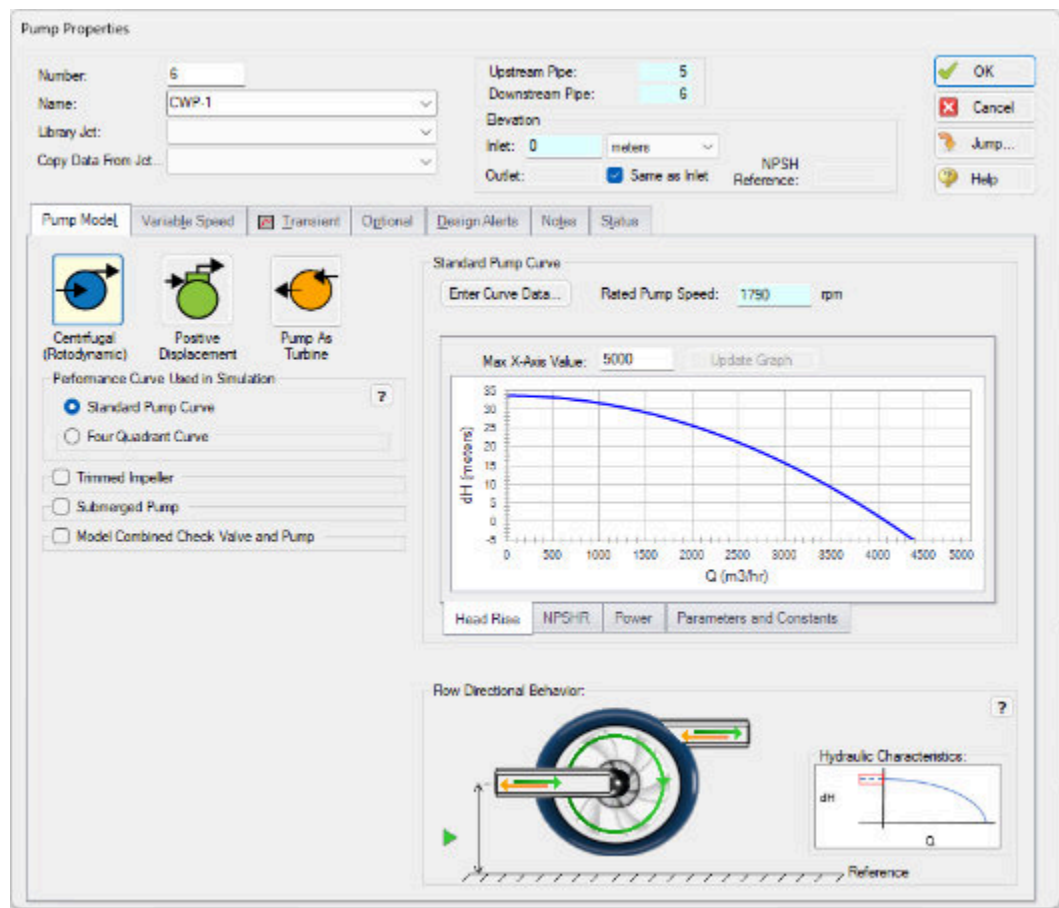
<b>Volumetric</b>	<b>Head</b>	<b>Power</b>
<b>m3/hr</b>	<b>meters</b>	<b>kW</b>
2825	18.3	156.5

7. Curve Fit Order = 2
8. Click Generate Curve Fit Now, then click OK
9. Rated Pump Speed = 1790 rpm
10. Click the Transient tab
11. Transient = Trip (With Inertia)
12. Initiation of Transient = Single Event
13. Event Type = Time Absolute
14. Condition = Greater Than or Equal To
15. Value = 0 seconds
16. Total Rotating Inertia = User Specified
17. Value = 5.9 kg-m2

---

**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 where 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. Loss Source = User Specified
3. Loss Model = Cv
4. Cv = 10,000
5. Click the Transient tab
6. Transient Data =

Time (seconds)	Cv
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. Cd = 0.6
8. Discharge Flow Area = 0.18 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 3. 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 4. Run the model**

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

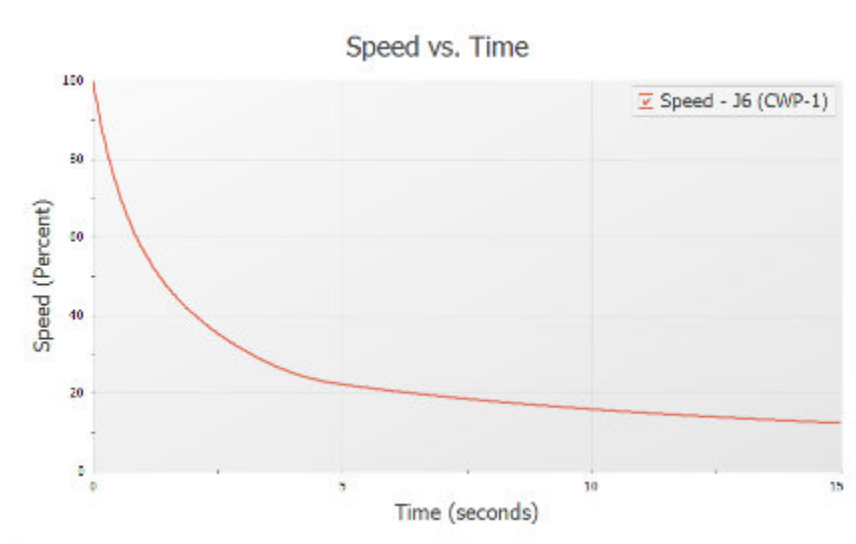
## **Step 5. Review results**

A warning is given in the output 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. In general, if the reverse flow is low in magnitude and duration, then using the zero-flow head may be a reasonable approximation and the warning could be dismissed. We will examine the impact of using a four quadrant curve as suggested by this warning later in this example.

## A. Graph the pump speed decay

1. Change to the Graph Results window.
2. On the Transient Junction tab, add Pump J6.
3. In the Parameter definition area, choose Speed as the parameter.
4. Choose Percent in the Units column.
5. Click Generate.
6. 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".

The speed of the two pumps should decay similarly. Figure 4.3 shows the speed decay for CWP-1. Here the pump speed decays quickly at first, then begins to slow after the discharge valve closes at about 5 seconds.



**Figure 4.3** The pump speed decay is shown, where the speed decays quickly at first, and then slows down after the valve closes and the flow goes to zero

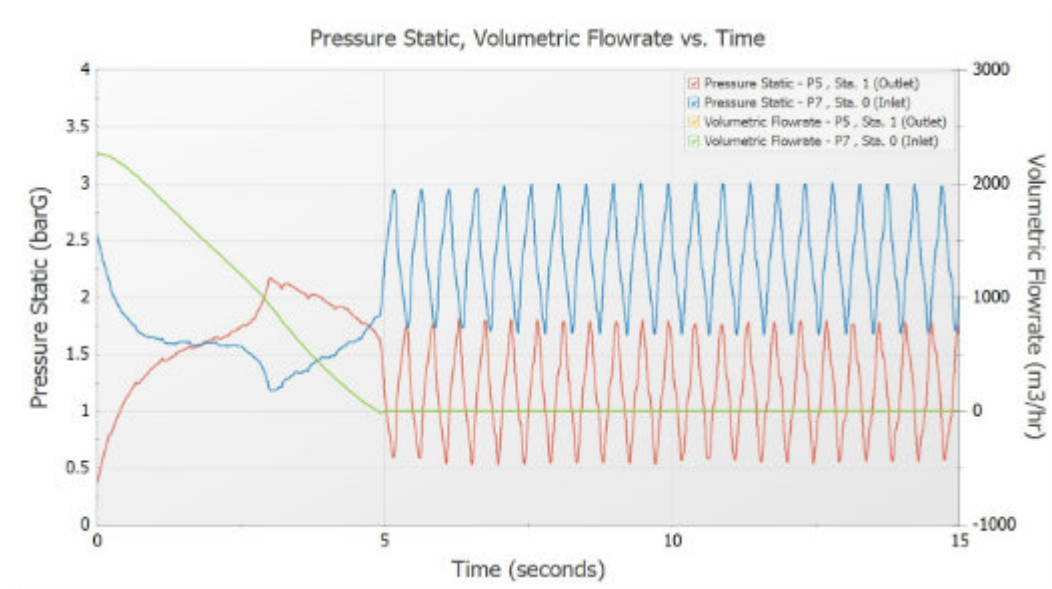
## B. Graph the transient pressures and flows at the pump station

1. Return to the Graph Control tab on the Quick Access Panel.
2. Go to the Transient Pipe tab on the Graph Control tab and add Pipe P5 Outlet and Pipe P7 Inlet under the Graph These Pipes/Stations list. 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 as the second parameter to be

graphed.

5. Move the Volumetric Flowrate to the secondary y-axis by clicking the Make Secondary Y-axis button, which is the blue, right arrow icon, or by double-clicking in the column to the left of the parameter. This parameter should now be indented.
6. Set the units to barG for the static pressure and m3/hr for the volumetric flowrate.
7. Click Generate.
8. Add the graph to the Graph List Manager with the name “Pressure & Flowrate vs. Time”.

In Figure 4.4 you can see that initially the pressure increases at the pump inlet and decreases at the valve outlet, but later on the pressures begin to oscillate once the discharge valve becomes fully closed.



**Figure 4.4** Pressure and volumetric flow rate transients using the standard pump curve data

## Step 6. 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.

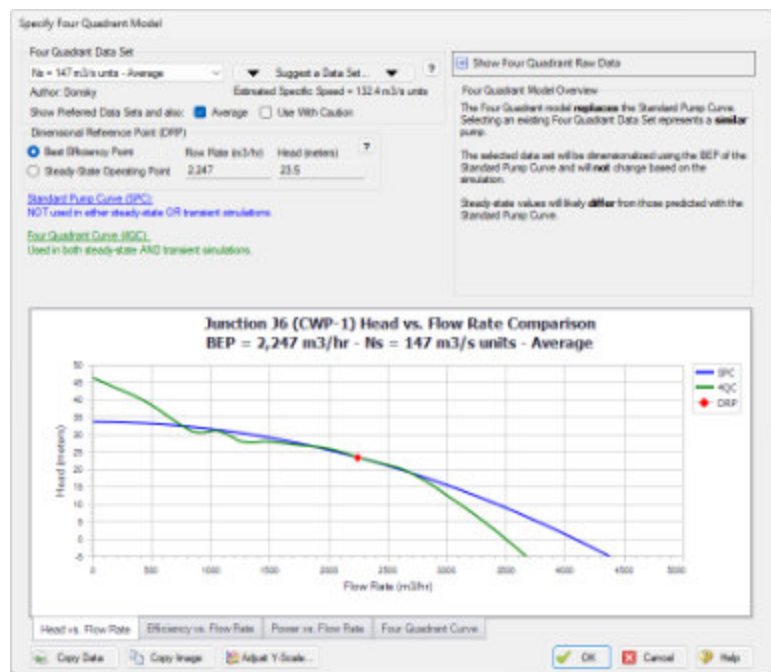
To do this, click the Create Child button on the Scenario Manager on the Quick Access Panel. Name the child “Standard Pump Curve”. A new scenario will appear below the Base Scenario in the scenario tree. Select the Base Scenario, create another child, and call it “Four Quadrant BEP”. We will later clone 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 4.6).
2. Under the Four Quadrant Curve option click User Selected, and click Specify Model.
3. 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 Impulse will use to recommend a data set. To do this, click Suggest a Data Set under Four Quadrant Data Set.
4. By default, 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.

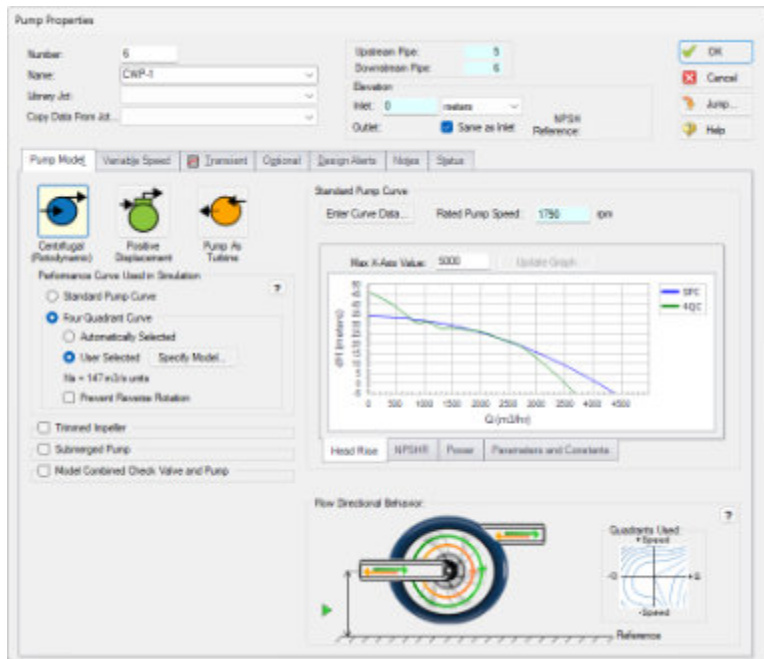


**Figure 4.5 Four Quadrant Model settings**

5. The specific speed is estimated to be 132.4 m³/s units, and the suggested data set is Ns = 147 m³/s units. Click Select Suggested Data Set and Close. The Ns = 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 4.5).

**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 the above steps for junction J8. The required input for this scenario will then be complete.



**Figure 4.6** The Pump Model tab defined using the Four Quadrant Curve, where the Flow Directional Behavior section shows the pump behavior for both forwards and reverse flow and speed possibilities

### 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 both pumps, 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 7. 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.

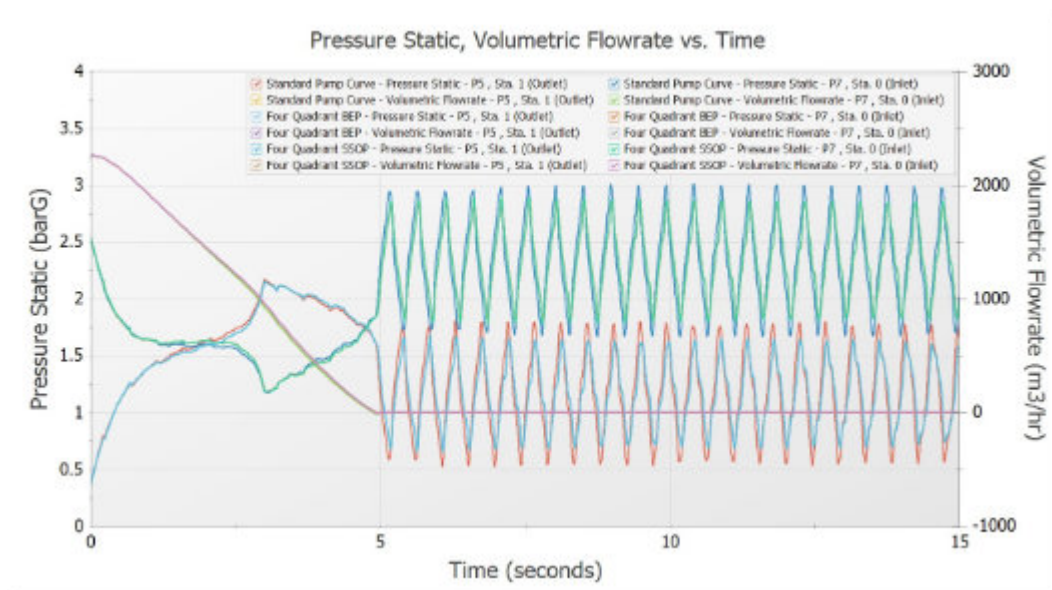
Multi-scenario graphing can be used to compare the results for all of these scenarios. To create the multi-scenario graph, do the following.

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.

As shown in Figure 4.7, the colored curves of the Standard Pump Curve scenario and the two four quadrant scenarios show no major differences.

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 are minor in the standard pump curve scenario.

On the Output window Warnings tab for each four quadrant scenario, information will be given on the four quadrant options that are defined in the pump, as well as listing topics which provide more background information on four quadrant data.



**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



## Step 8. Adjust the valve closure speed

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 9. Specify the Transient Control group

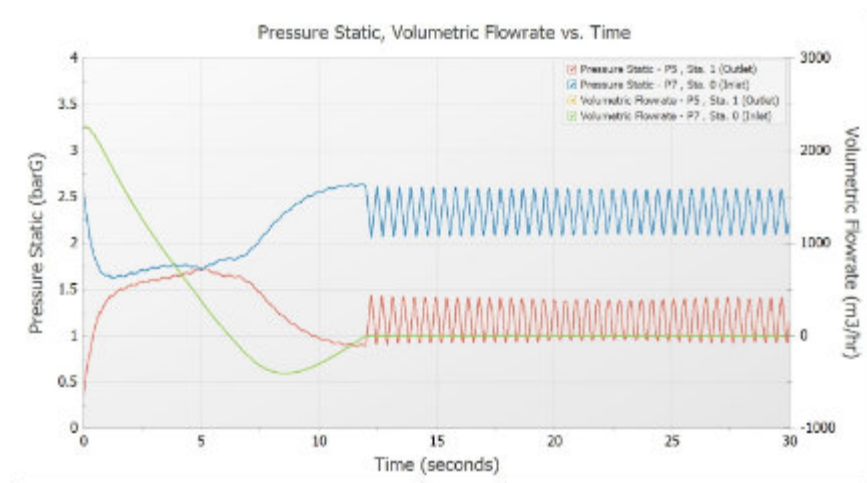
Since the valve closure takes a longer time, it will be useful to extend the simulation Stop Time to see more oscillations in pressure and flowrate when graphing. Open the Simulation Mode/Duration panel from the Analysis Setup and change the Stop Time to 30 seconds, then click OK.

## Step 10. Run the model

The model should now be fully defined and ready to run. Select Run Model from the Common Toolbar or the Analysis menu. View the results by clicking Graph Results at the bottom of the Solution Progress window.

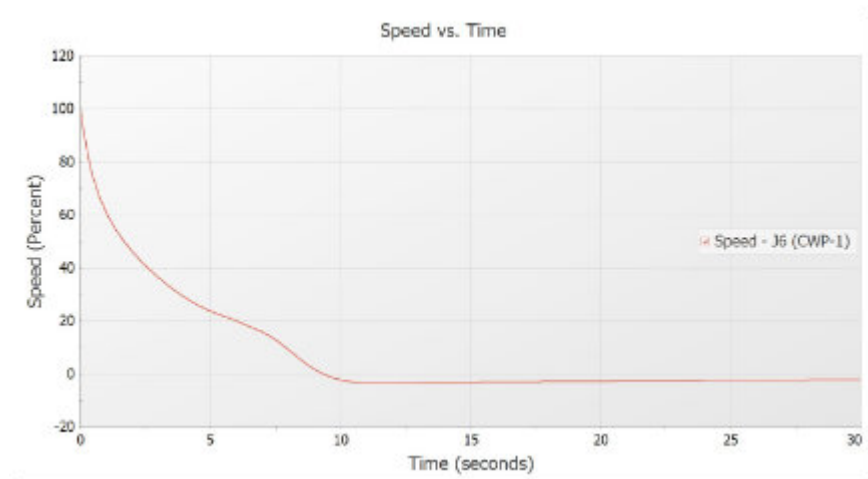
## Step 11. Graph results

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

Load the “Speed vs. Time” graph from the Graph List Manager, as seen in Figure 4.9. Note how at about 9 seconds the pump begins to experience reverse rotation. This means that from 6 to 9 seconds, the pump will have forward rotation with reverse flow, then from 9 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 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.

## CHAPTER 5

# Pipe Forces from Valve Closure 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

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

### Model file

A partially completed model file will be utilized as the starting point for this example. A completed version of the model file can be referenced if desired. Both the initial and final model files can be found in the Examples folder as part of the Impulse installation:

C:\Users\[username]\Documents\Datacor\Impulse 11\Examples

- *Metric - Pipe Forces from Valve Closure - Initial.imp*
- *Metric - Pipe Forces from Valve Closure - Final.imp*

### Step 1. Open the 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 Pipe Drawing Mode

The previous example models were drawn using the default Pipe Drawing Mode, 2D Freeform. However, 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. It is not recommended to switch the Pipe Drawing Mode once a model has been created and saved. 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 either Auto-rotate Icon or Customize Icon.

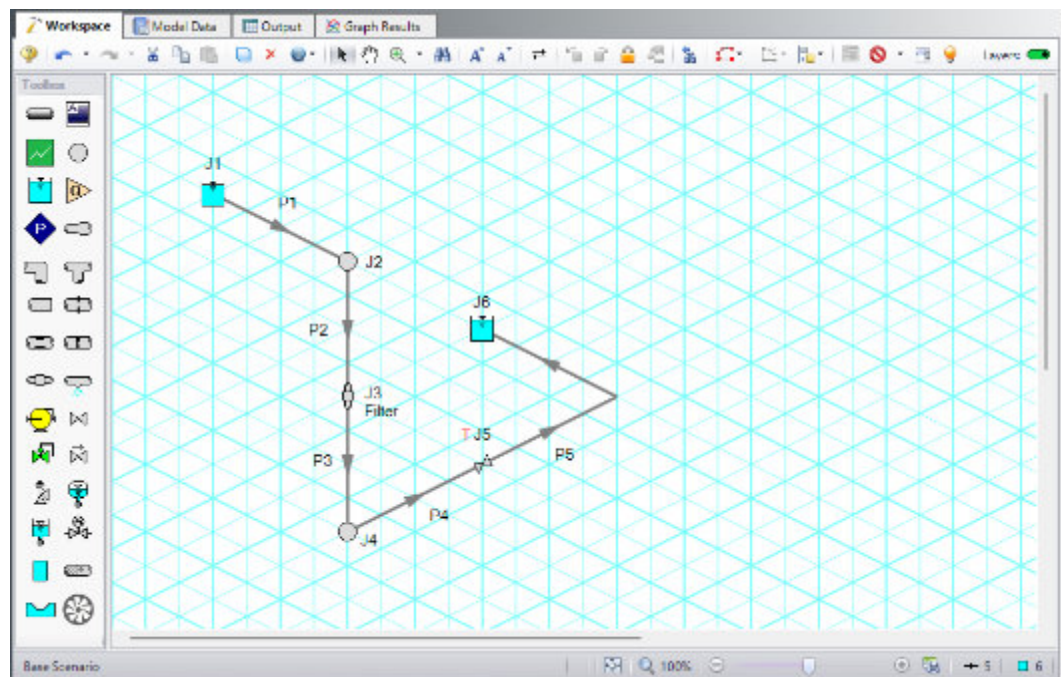


Figure 5.1 Layout of the gravity drain system with isometric grid on

## Step 2. Specify the Transient Control group

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

## Step 3. Define the Forces group

Click the Force Definitions panel in the Forces group.

### A. Force Types

The *Difference* force type is intended to model the specific case where all of the following criteria is true:

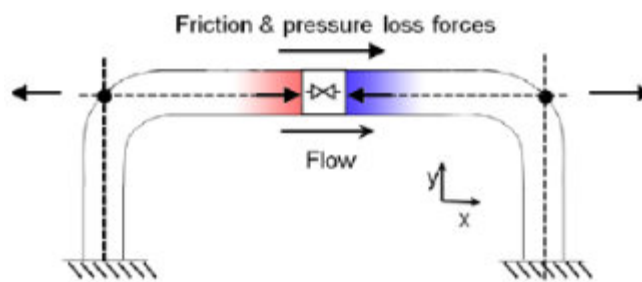
1. The force is between a pair of 90 degree elbows
2. The force is an axial pipe force
3. All pipes, fittings, and components between the elbow pair are in-line
4. There are no flow splits or branching paths between the elbow pair

The force equations in the Solver are specific to this geometry. The pipes and components between the elbow pair can be horizontal, vertical, or sloped, but the axial component of the fluid weight is not accounted for in the forces. However, elevation and hydrostatic pressure are accounted for in the pressure and flow solution. Furthermore, Impulse is a hydraulic solver; therefore, only hydraulic forces are calculated. These are forces which act on the pipe from the fluid.

---

**Note:** Pipes are assumed to be rigid bodies and Impulse does not account for the effect of pipe deflection on forces.

---



**Figure 5.2 Schematic of the Difference force type**

The *Point* force type would be used for calculating the force at a Dead End junction. It is calculated using the difference in fluid pressure and ambient pressure, multiplied by the pipe area.

The *Difference Exit* force type would be used between a single 90 degree elbow and a location where the fluid is leaving the system, such as an exit valve, exit orifice, or nozzle.

The force sets in this example will use the Difference force type to calculate the axial forces between 90 degree elbow pairs.

## **B. Defining Force Sets**

Generally, it is best to represent each elbow with a unique junction on the Workspace (either an Elbow or Branch). This example assumes that the static pressure loss in the elbows is negligible, and therefore, they are modeled as Branch junctions which are lossless, with exception of the last elbow in Pipe P5 which is visually depicted with a pipe segment.

Using explicit junctions on the Workspace will make the Force Definitions simple and straightforward, and will create computing stations at the exact locations where the forces are being calculated. However, this can increase the runtime if the pipes between the elbows have a short length compared to the rest of the pipes in the model. For this reason, Force Sets can also be defined using an intermediate computation station inside a pipe.

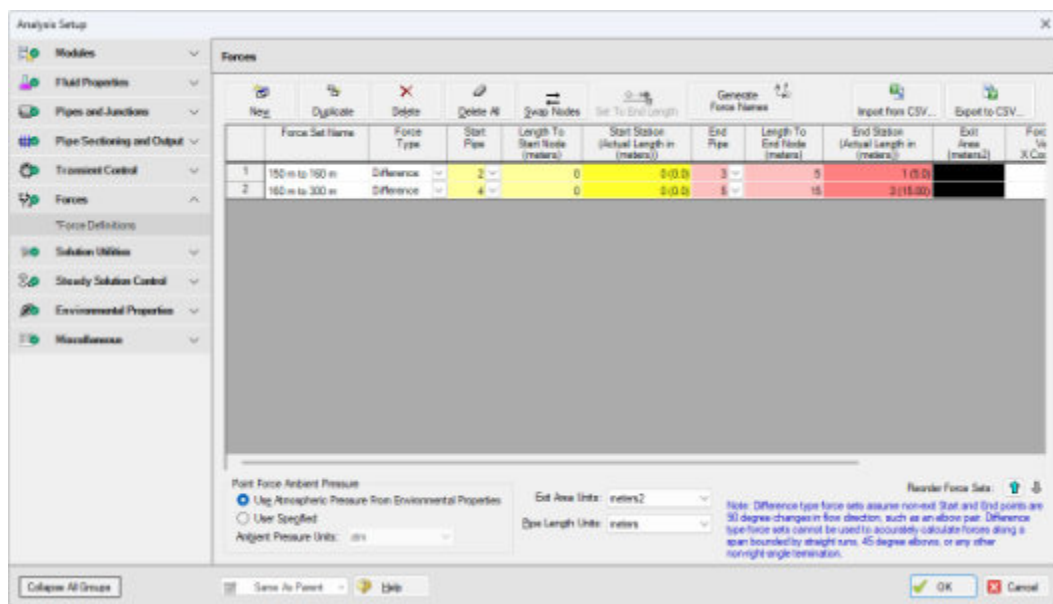
The first elbow pair is represented by J2 and J4 which will demonstrate the setup when using explicit junctions. The second elbow pair is represented by J4 and the segment in Pipe P5 which will demonstrate the setup when using an intermediate station.

---

**Reminder:** Segments are simply a visual representation of a bend that can be added to a pipe. They do not represent a pressure loss or elevation change.

---

- ▶ **Create a force set by clicking New.** Then enter the data as shown in Figure 5.3. Repeat this process for the second force set.
  - Force Set 1:
    - Name = 150 m to 160 m
    - Force Type = Difference
    - Start Pipe = 2
    - Length To Start Node = 0 meters
    - End Pipe = 3
    - Length To End Node = 5 meters
  - Force Set 2:
    - Name = 160 m to 300 m
    - Force Type = Difference
    - Start Pipe = 4
    - Length To Start Node = 0 meters
    - End Pipe = 5
    - Length To End Node = 15 meters



**Figure 5.3 Completed Force Definitions panel**

The first force set spans between the first and last computation stations that are between the J2-J4 elbow pair. The start corresponds to the Pipe P2 inlet, therefore, the Start Pipe is P2 and the Length to Start Node is 0 meters. The end corresponds to the Pipe P3 outlet, therefore, the End Pipe is P3 and the Length to End Node is 5 meters.

The second force set starts at J4 but ends in the middle of Pipe P5. The start corresponds to the Pipe P4 inlet, therefore, the Start Pipe is P4 and the Length to Start Node is 0 meters. The end corresponds to the middle of Pipe P5 (which is 30 meters in total length), therefore, the End Pipe is P5 and the Length to End Node is 15 meters.

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 this example, the 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.

- Click OK to close Analysis Setup.

## Step 4. Run the model

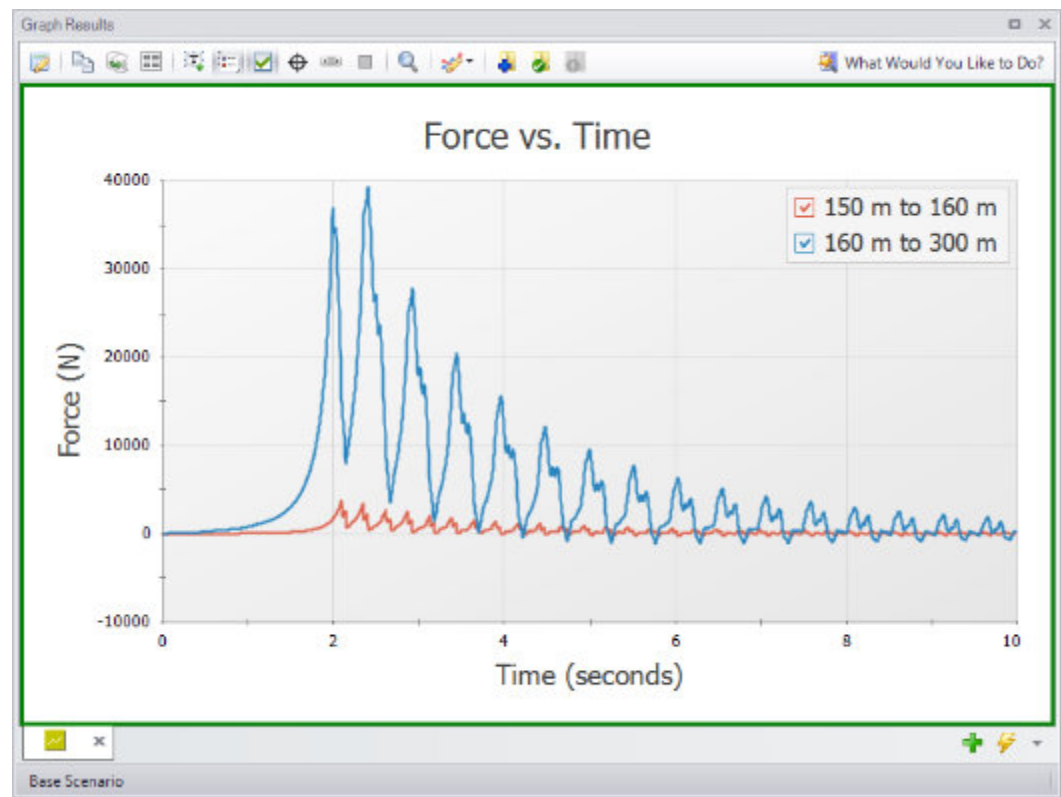
All groups in Analysis Setup should now be complete and the model is ready to run. Run the model by either pressing the Run Model icon on the Common Toolbar, selecting Run Model in the Analysis menu, or pressing the CTRL+R



keyboard shortcut. This will open the Solution Progress window. When complete, press the Output button.

## Step 5. 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 boxes next to both Force Sets.**
- ▶ **Click Generate**, and the following force vs time graph is displayed (Figure 5.4).



**Figure 5.4 Force vs. Time for both force sets**

Note that at time zero, which represents the initial, steady-state results, there is no force. This is the expected result 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 5.4 was created with friction and momentum ignored, then the steady-state values would be calculated as approximately -5,100 N and -11,940 N. Due to this effect, Impulse will always include both friction and momentum when graphing force sets.

**Note:** An 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.

---

## Conclusion

In summary, there are several important points to be observed:

1. 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. The Difference force type calculates the axial pipe force between a pair of 90 degree elbows.
3. In some cases, ignoring friction and momentum force balance components will result in forces 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.

Data that is obtained from the force sets in Impulse can be exported to CAESAR II, ROHR2, TRIFLEX or other pipe stress analysis software for further analysis on the impact of forces on the structural supports.

## CHAPTER 6

# 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

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

### Required knowledge

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

### Model files

The following examples have completed versions of the model files that can be referenced from the Examples folder as part of the Impulse installation:

C:\Users\username\Documents\Datacor\Impulse 11\Examples

- *Metric - Pipe Sizing for Sand Transfer - SSL.imp*
- *Metric - Positive Displacement 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. Activate the SSL module

Open the Analysis Setup window and go the Modules panel under the Modules group. Then, select the option to Activate SSL.

## 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, the Solids Definition panel, and the Slurry Definition panel.

### A. Carrier Fluid panel

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

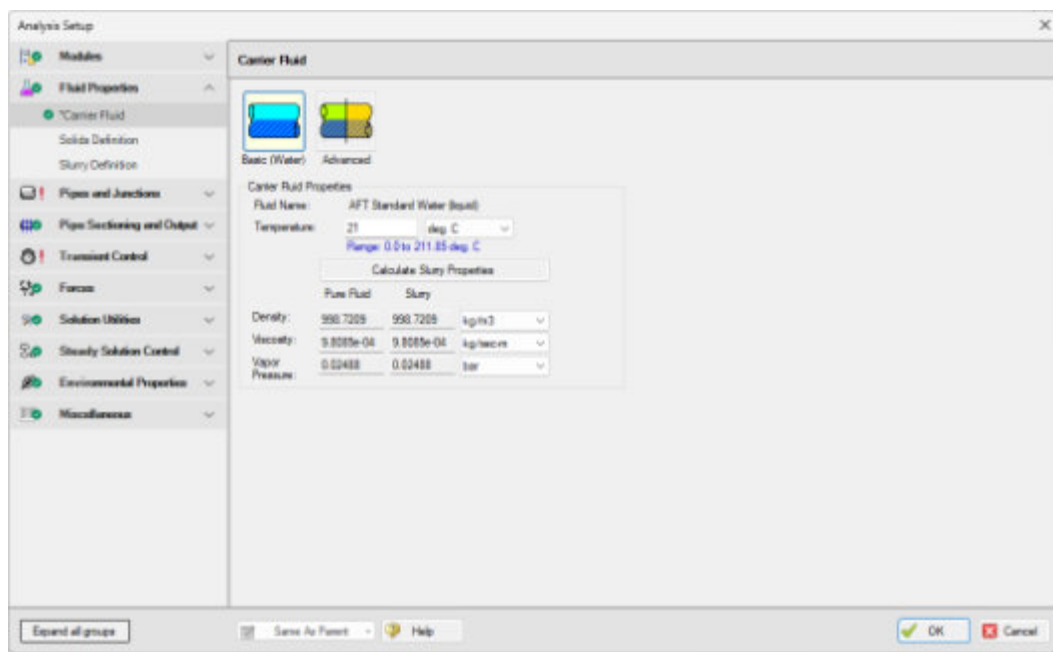


Figure 6.1 Carrier Fluid panel

## B. Slurry Definition panel

1. Go to the Slurry Definition panel (Figure 6.2)
2. Slurry Model = Settling Slurry - Wilson, Addie, Clift
3. Slurry Calculation Method = Detailed
4. Concentration Type = Volume Fraction
5. Amount Solids Added = 20 Percent

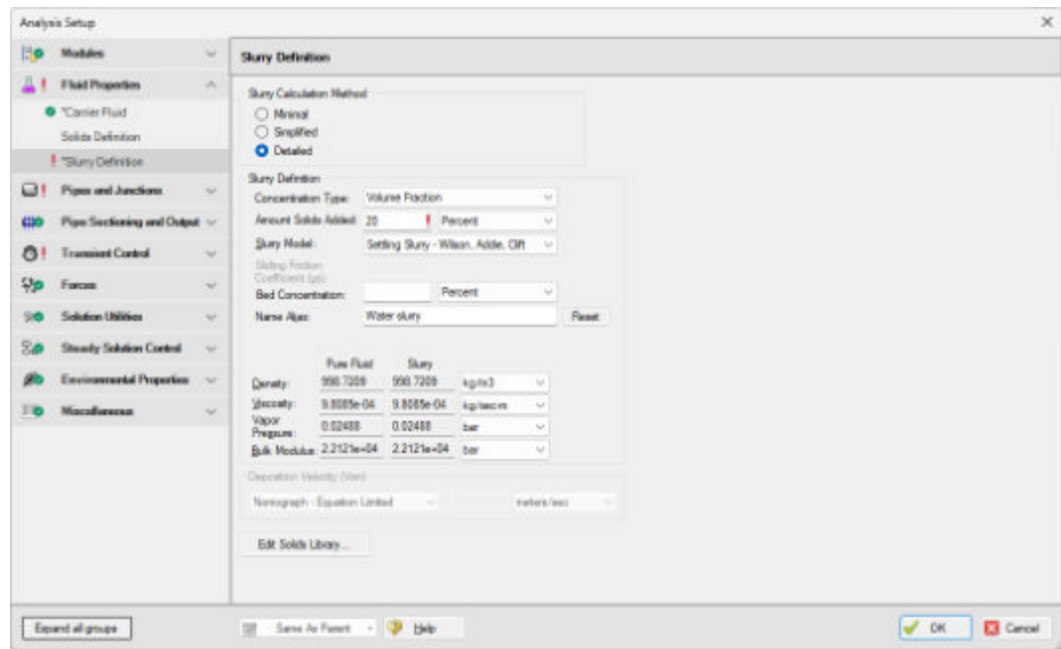


Figure 6.2 Completed Slurry Definition panel

## C. Solids Definition panel

1. Go to the Solids Definition panel (Figure 6.3)
2. Solids Specifications = User Specified Solids Added
3. M (Stratification Ratio Exponent) = Calculated
4. Terminal Velocity Parameter,  $V_t/V_{ts} (X_i) = 0.55$
5. Density = 2.65 S.G. water
6.  $d_{50} = 0.07$  cm
7.  $d_{85} = 0.17$  cm
8. Bulk Modulus = 16000 MPa
9. Click OK to save and close Analysis Setup

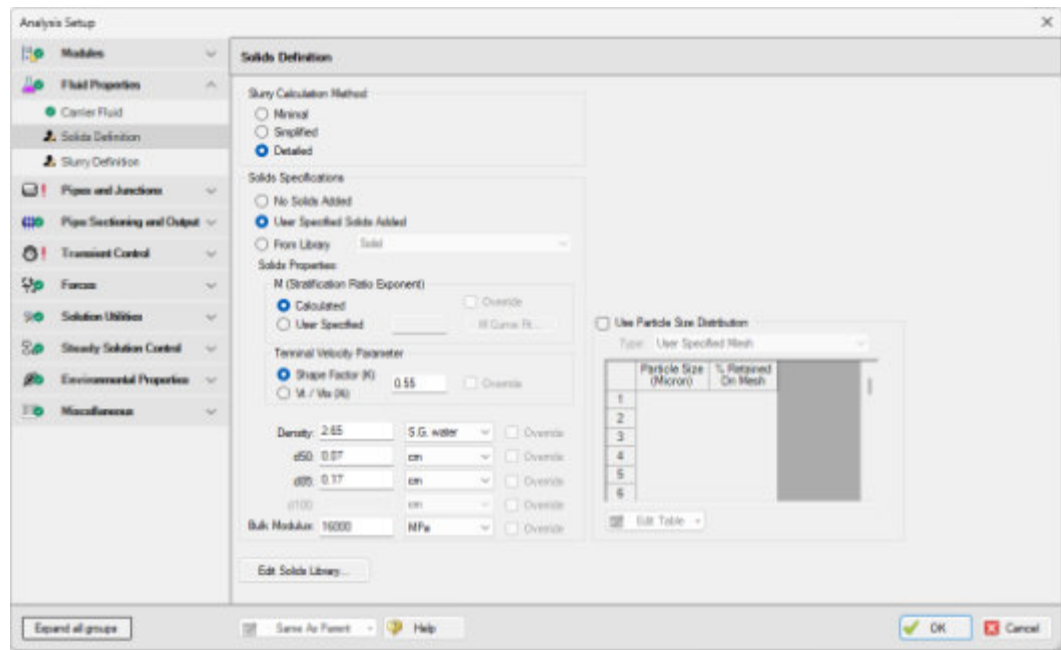


Figure 6.3 Completed Solids Definition panel

## 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.

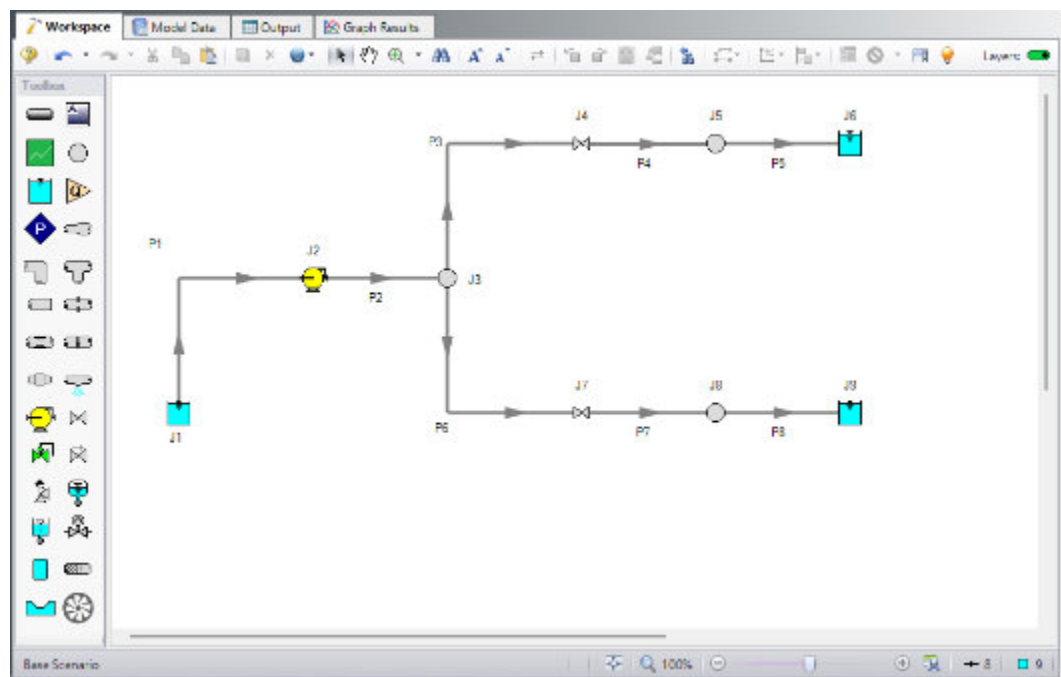


Figure 6.4 Layout of the pipe system for the Sand Transfer to Two Locations example

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

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

***Pipes***

All pipes will use the following properties:

1. Pipe Material = Steel - ANSI
2. Size = 10 inch
3. Type = STD (schedule 40)
4. Friction Model = Standard
5. Wavespeed, Pipe Support = Thick-Walled Anchored Upstream
6. Length = see the table below

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

***J2 - Pump***

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

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

5. Curve Fit Order = 2
6. Click Generate Curve Fit Now, then click OK

**J3 - Branch**

Elevation = 1.2 meters

**J4 - Valve**

1. Name = Valve to Deposit #1
2. Inlet Elevation = 1.2 meters
3. Loss Source = User Specified
4. Loss Model = Cv
5. Cv = 500
6. Click the Transient tab
7. Transient Data =

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

**J5 - Branch**

Elevation = 6 meters

**J6 - Reservoir**

1. Name = Deposit #1
2. Liquid Surface Elevation = 6 meters
3. Liquid Surface Pressure = 0 barG
4. Pipe Depth = 0 meters



### **J7 - Valve**

1. Name = Valve to Deposit #2
2. Inlet Elevation = 1.2 meters
3. Loss Source = User Specified
4. Loss Model = Cv
5. Cv = 400
6. Click the Transient tab
7. Transient Data =

Time (seconds)	Cv
0	400
5	500
10	500

### **J8 - Branch**

Elevation = 9 meters

### **J9 - Reservoir**

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

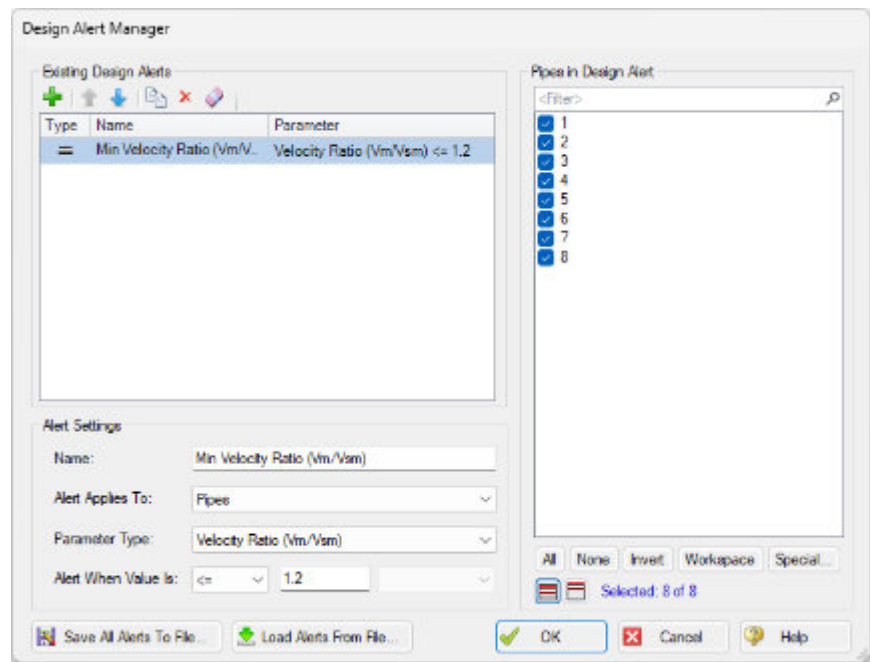
## **C. Create a Design Alert**

In this system the Velocity Ratio (mixture to settling velocity ratio or  $V_m/V_{sm}$ ) must remain greater than 1.2 in each of the pipes to avoid the potential of the sand falling out of the slurry and plugging the pipe. Use the Design Alert Manager to add a Design Alert to each pipe. This will give a warning message in the output if the velocity ratio is below the minimum value.

- **From the Tools menu, select Design Alerts to open the Design Alert Manager.** Click the green plus icon to create a new Design Alert.

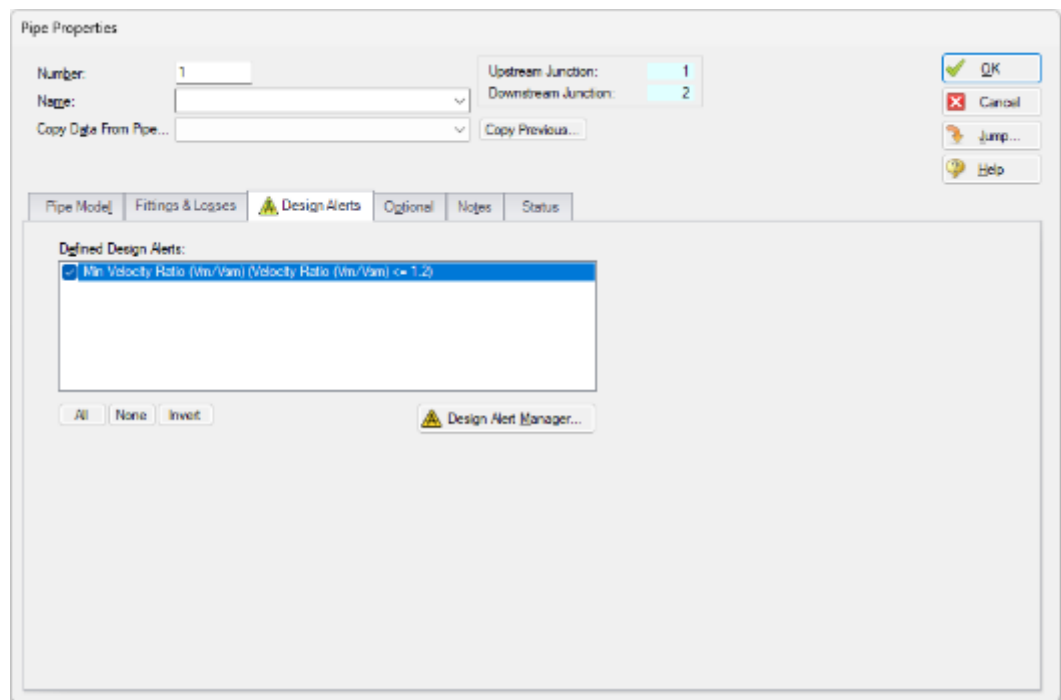
1. Name = Min Velocity Ratio ( $V_m/V_{sm}$ )
2. Alert Applies To = Pipes
3. Parameter Type = Velocity Ratio ( $V_m/V_{sm}$ )
4. Alert When Values Is = Less Than or Equal To ( $\leq$ )
5. Value = 1.2
6. Click the All button under the Pipes in Design Alert list

7. The Design Alert Manager window should now appear as shown in Figure 6.5
8. Click OK to return to the Workspace



**Figure 6.5 Creating a minimum velocity ratio Design Alert**

Open the Pipe Properties window of any pipe and click the Design Alerts tab. Check to confirm that the Design Alert you created has been applied (Figure 6.6).



**Figure 6.6 Design Alerts tab of the Pipe Properties window**

### **D. 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 is ready to run. Run the model by pressing the “Run Model” icon on the Common 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**

Clicking “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 6.7, it can be seen that the minimum  $V_m/V_{sm}$  drops below this value in Pipe P4 (scroll to the left to find the column). The Design Alert is violated which results in the cell being highlighted.

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	2.657	1.346	0	2.482	6.993	0
2	3.356	1.344	0	3.136	6.996	0
3	1.924	0.000	0	1.276	9.996	0
4	1.712	0.000	0	1.135	9.995	0
5	1.924	0.02094	0	1.276	9.619	0
6	1.874	9.997	0	1.431	2.463E-03	0
7	1.598	10.000	0	1.220	0.000	0
8	1.874	10.000	0	1.431	0.02463	0

**Figure 6.7 Design Alert violation in Pipe P4**

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 could 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.

### B. Increase the Pipe Diameter and Re-run the Model

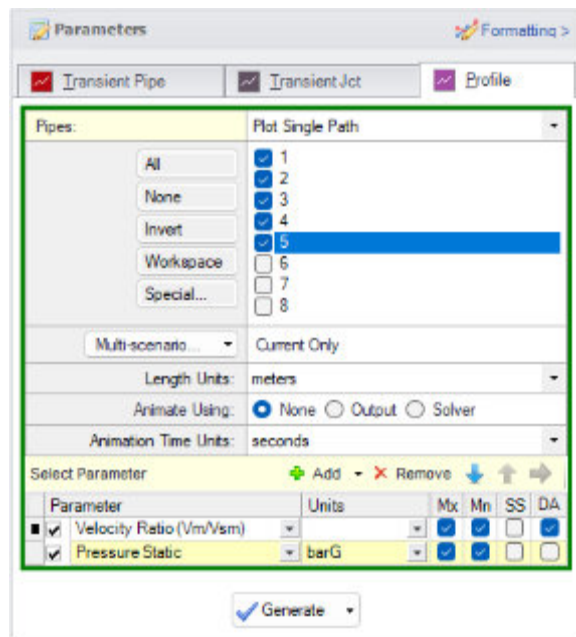
Return to the Workspace and change Pipes P1 and P2 to have a 14 inch diameter. Since the size of the pipes was changed, the pipes need to be re-sectioned by opening the Sectioning panel in the Analysis Setup window. Click OK to accept the results.

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.

### C. Graph the velocity ratio and pressures to Deposit #1

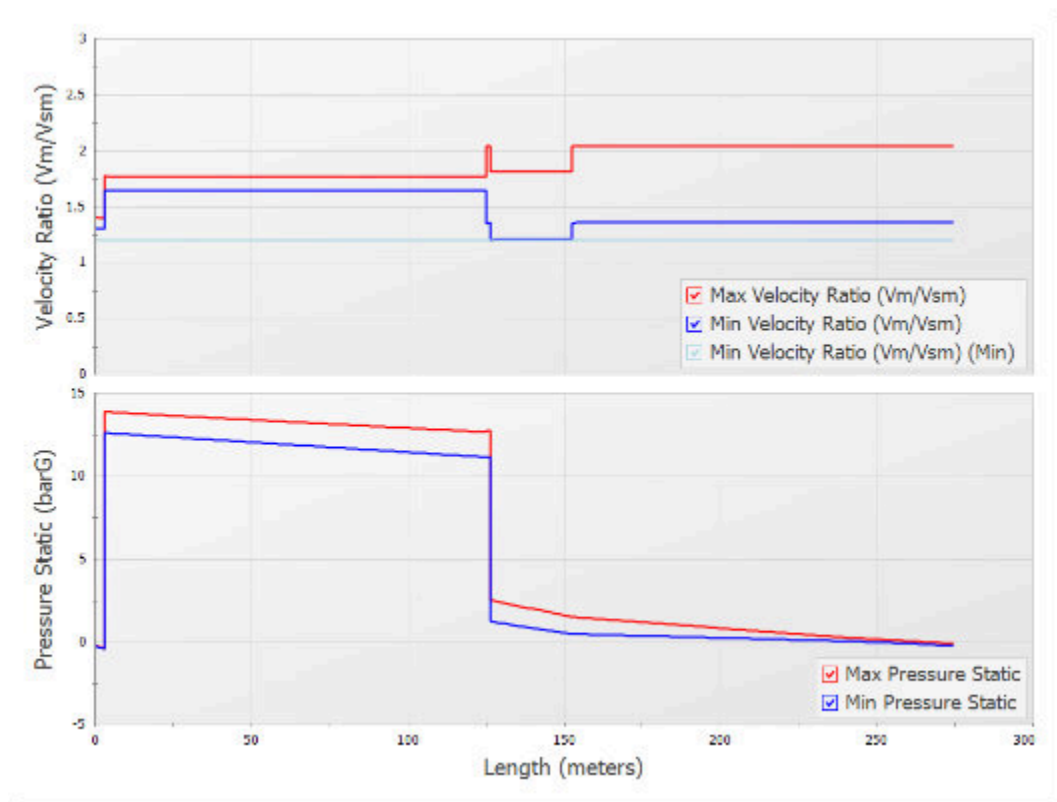
Stacked graphs will be used to simultaneously view the maximum and minimum velocity ratio and pressures in the system during the transient.

1. Change to the Graph Results window by either clicking on the Graph Results tab, from the Window menu, or by pressing CTRL+G.
2. Click the Profile tab in the Parameters definition area.
3. Select Pipes P1 through P5 (Figure 6.8). These represent the flow path to Deposit #1.
4. Add a second Parameter by clicking the green “+” icon. Choose “Velocity Ratio (Vm/Vsm)” for one parameter and “Pressure Static” in units of barG for the other.
5. Check the boxes for Mx, Mn, and DA in order to display the maximum, minimum, and Design Alert.
6. Click Generate. The resulting graphs are shown in Figure 6.9.



**Figure 6.8** Data selection to view the maximum and minimum values of the velocity ratio and static pressure along the flow path to Deposit #1

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 6.9** Maximum and minimum velocity ratio and static pressure values in the flow path to Deposit #1, where the velocity ratio is above the minimum Design Alert of 1.2 at all locations

#### D. 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.

1. On the Transient Pipe tab, select Pipes P4 Inlet and P7 Inlet and add them to the Graph These Pipes/Stations list (Figure 6.10). These are the two sloped pipes which have the lowest velocity ratio.
2. Choose Velocity Ratio ( $V_m/V_{sm}$ ) as the Parameter to graph.
3. Check the DA box to display Design Alerts on the graph.
4. Click Generate.

The resulting graph is shown in Figure 6.11. This shows that, as Valve J4 closes and Valve J7 opens, the velocity ratios in the pipe immediately downstream change but stay in the acceptable range.

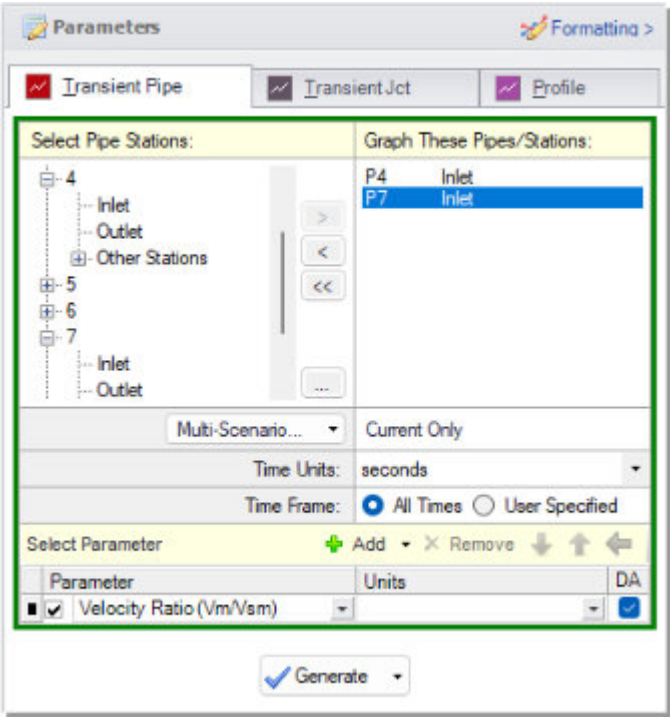


Figure 6.10 Data selection to view the velocity ratio over time

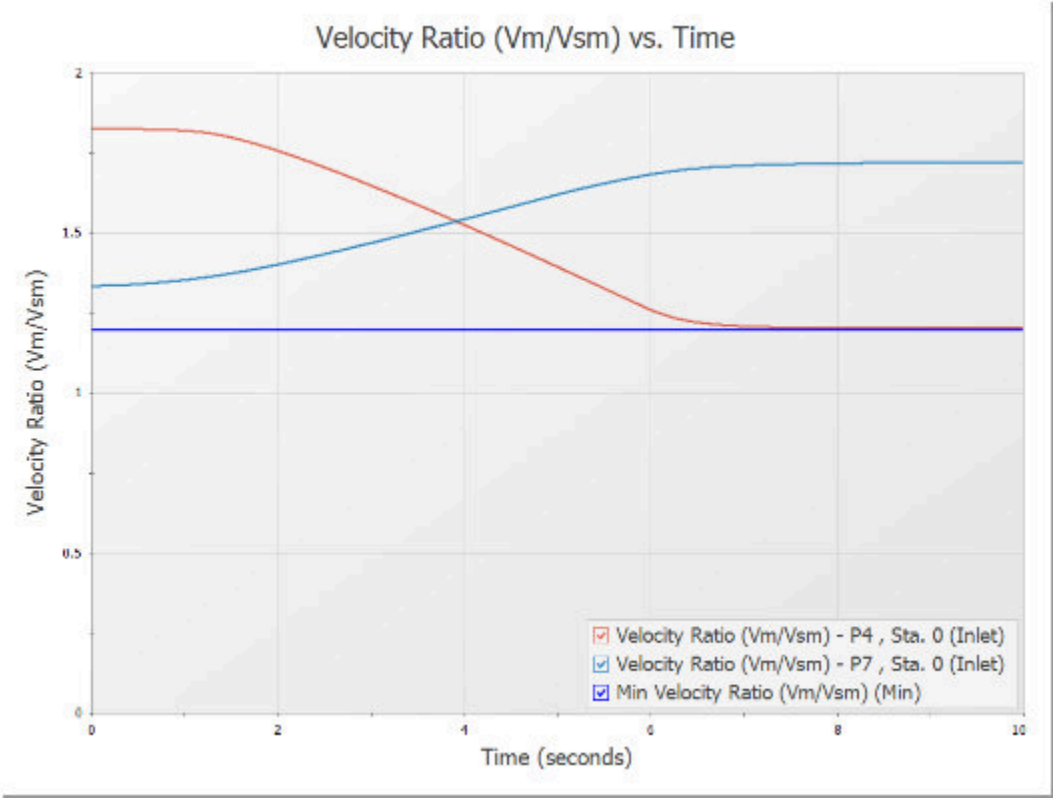


Figure 6.11 The velocity ratio in the sloped pipes as the valves change position

## SSL Conclusion

A slurry of sand and water flowing to two deposits was modeled. The velocity ratio ( $V_m/V_{sm}$ ) 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 and the created Design Alert was violated. 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. Activate the PFA module

Open the Analysis Setup window and go the Modules panel under the Modules group. Then, select the option to Activate PFA.

## PFA Step 2. Define the Fluid Properties group

1. Open Analysis Setup using the button on the Common Toolbar or from the Analysis menu.
2. Open the Fluid panel and make sure User Specified 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

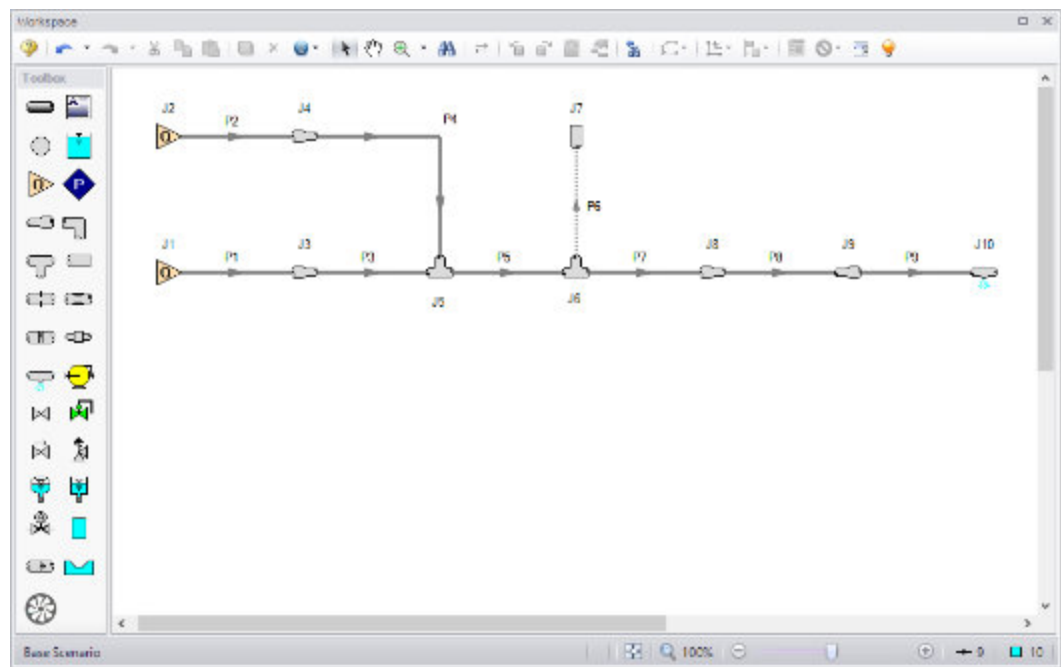
## PFA Step 3. Define the Pipes and Junctions group

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 6.12.





**Figure 6.12** Layout of the 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 needs 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 a user-specified material (scroll to the top of the Pipe Material list) 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**

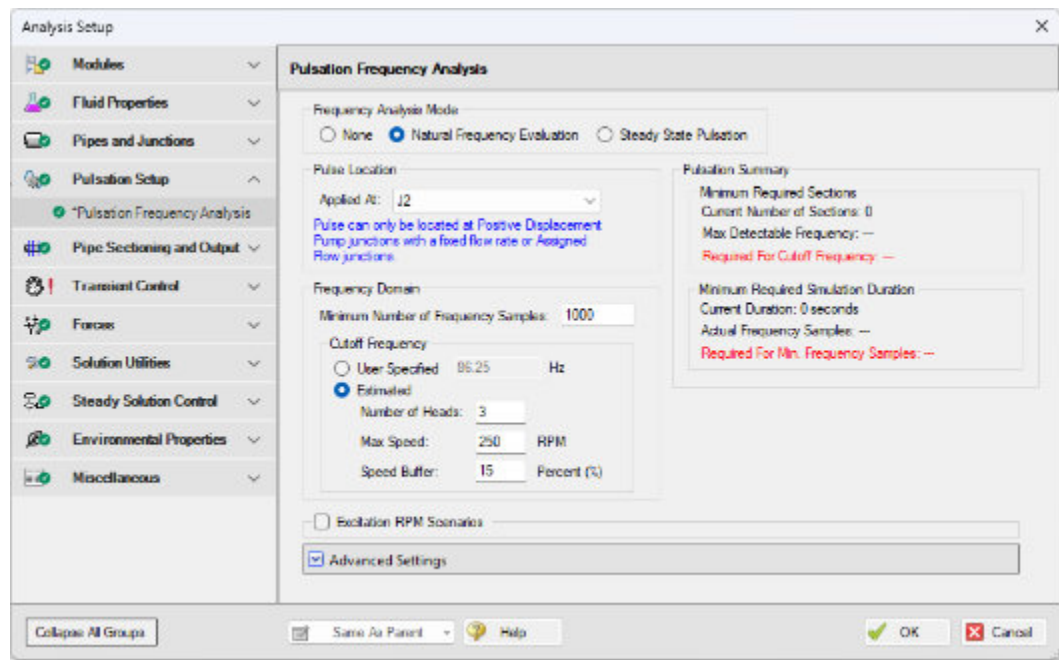
Defining the initial pulse or “ring” is an important step in determining the natural frequencies that different stations in the model will respond to. This is the numerical equivalent of hitting the system with a hammer to see how it responds at different frequencies. This first step does not represent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

Open Analysis Setup and go to the Pulsation Frequency Analysis panel in the Pulsation Setup group. The Pulsation Frequency Analysis panel is the main interface for specifying input regarding the initial pulse and other information about the pulsation source. There are two types of analysis modes available in the Pulsation Frequency Analysis panel. The Natural Frequency Evaluation will produce a single transient pulse at a specified assigned flow or fixed flow rate positive displacement pump junction so as to determine the frequencies that would excite the system. The Steady State Pulsation option will determine how a pulsation source, such as a reciprocating pump, would interface with a system in steady operation. The Natural Frequency Evaluation mode can be used to generate Steady State Pulsation analyses at problematic pump speeds based on the system's resonant frequency.

### **A. Define Pulse Location and Frequency Domain**

Define the Pulsation Frequency Analysis panel (Figure 6.13) using the following values:

1. Frequency Analysis Mode = Natural Frequency Evaluation
2. Applied At = J2
3. Minimum Number of Frequency Samples = 1000
4. Cutoff Frequency = Estimated
5. Number of Heads = 3
6. Pump Speed = 250 RPM
7. Speed Buffer = 15 Percent (%)
8. Value = 86.25 Hz (populates after completing the steps above)

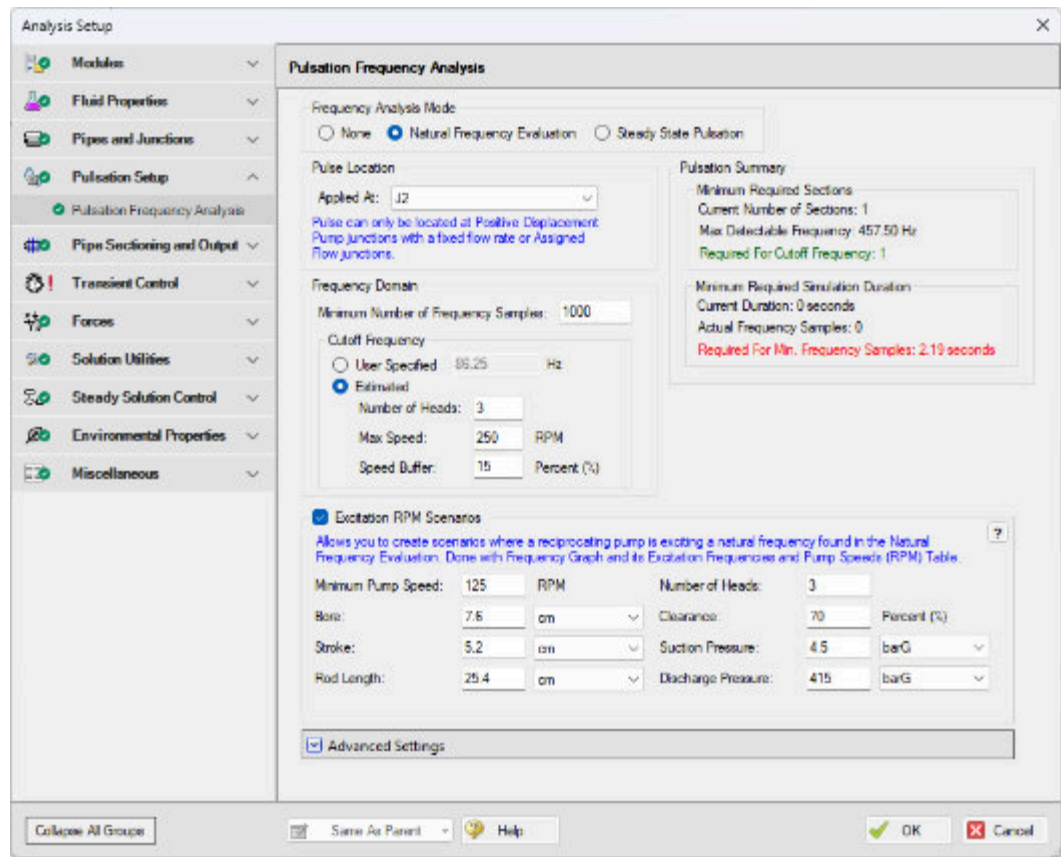


**Figure 6.13 Completed Pulse Location and Frequency Domain sections of Pulsation Frequency Analysis**

### B. Define Excitation RPM Scenarios

Define the Excitation RPM Scenarios section (Figure 6.14) using the following values:

1. Excitation RPM Scenarios checked
2. Minimum Pump Speed = 125 RPM
3. Bore = 7.6 cm
4. Stroke = 5.2 cm
5. Rod Length = 25.4 cm
6. Number of Heads = 3
7. Clearance = 70 Percent (%)
8. Suction Pressure = 4.5 barG
9. Discharge Pressure = 415 barG

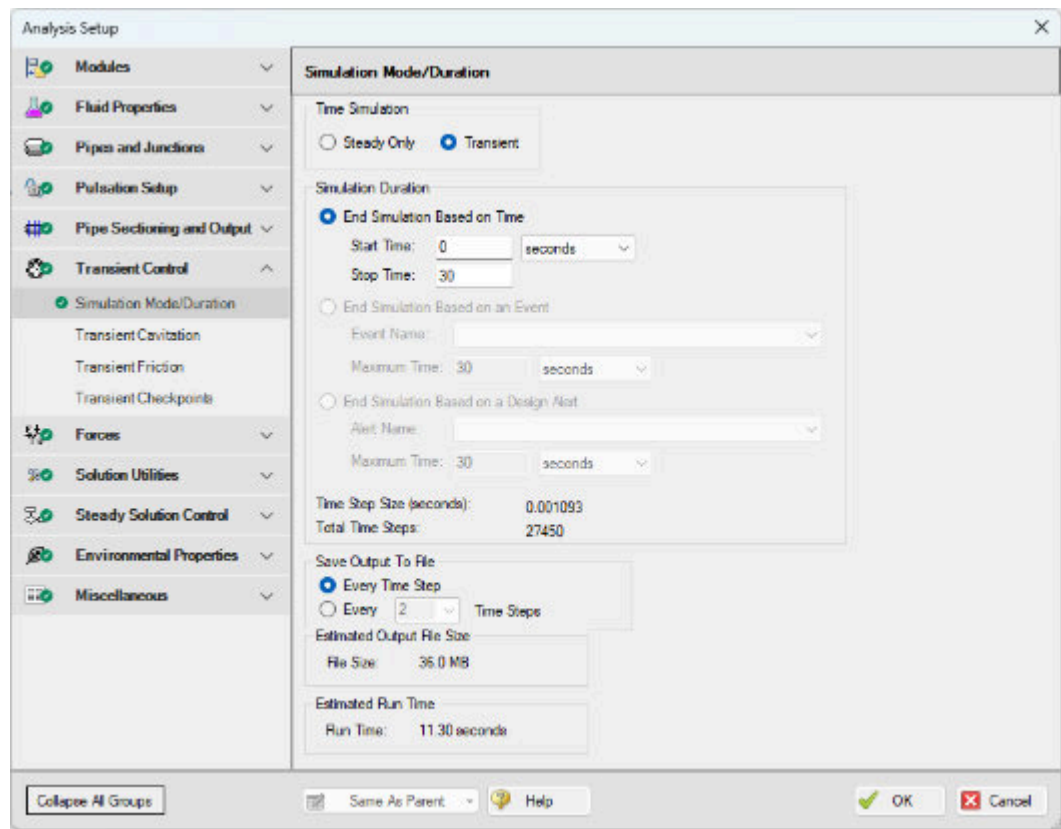


**Figure 6.14** Excitation RPM Scenarios are used to generate read only Steady State Pulsation child scenarios

## PFA Step 5. Define the Transient Control Group

According to the Pulsation Summary in the Pulsation Frequency Analysis panel, the Minimum Required Simulation Duration is 2.19 seconds. Several factors go into simulation duration selection for pulsation analysis. To have a sufficient number of samples for frequency analysis, the simulation duration should be at least an order of magnitude greater than the Minimum Required Simulation Duration and one to two orders of magnitude greater than the total communication time for the system. For this system, the total wave communication time from J2 to J10 and back is 0.116 seconds, which implies that the simulation duration should be at least 12 seconds. However, this is less than the length of time needed to ensure that the simulation duration is an order of magnitude greater than the Minimum Required Simulation Duration, so a duration of 30 seconds will be used to comfortably satisfy both of these recommendations.

Go to the Simulation Mode/Duration panel in the Transient Control group of Analysis Setup. In the Stop Time field, enter 30 with units of seconds (Figure 6.15). Hit OK. A message will appear confirming that the PFA Module has applied a forcing function to J2. Pipe sectioning has been automatically applied and the model is ready to run.



**Figure 6.15 Simulation Mode/Duration panel**

Open J2 to verify that the numerical “ring” has been applied to the transient (see Figure 6.16 for the transient input). This transient is a brief jump in flow rate at junction J2, which can be displayed with Show Graph.

Assigned Flow Properties

Number: 2  
 Name:   
 Library Jct:   
 Copy Data From Jct:

Upstream Pipe: None  
 Downstream Pipe: 2  
 Elevation: 0 meters

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: 0 m3/hr

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	0.00218578	16
4	0.00327868	16
5	0.00437157	8
6	29.99887	8
7		
8		
9		
10		
11		
12		

Edit Table Show Graph...  
☐ Repeat Transient  
 Transient

Figure 6.16 Assigned Flow Properties with the forcing function applied to J2

## PFA Step 6. Run the model

All groups in Analysis Setup should now be complete and the model is ready to run. Run the model by pressing the “Run Model” icon on the Common Toolbar, selecting “Run Model” in the Analysis menu, or pressing CTRL+R on the keyboard. The Waterhammer Assistant window may open when the model is run. This window is a tool which provides advice to help simplify and improve the model before it is run. In this case we are intentionally using the Assigned Flow junctions to represent the PD Pumps and the forcing function, so click OK to dismiss the Waterhammer Assistant. This will open the Solution Progress window. When complete, press the “Graph Results” button.

## PFA Step 7. Review results

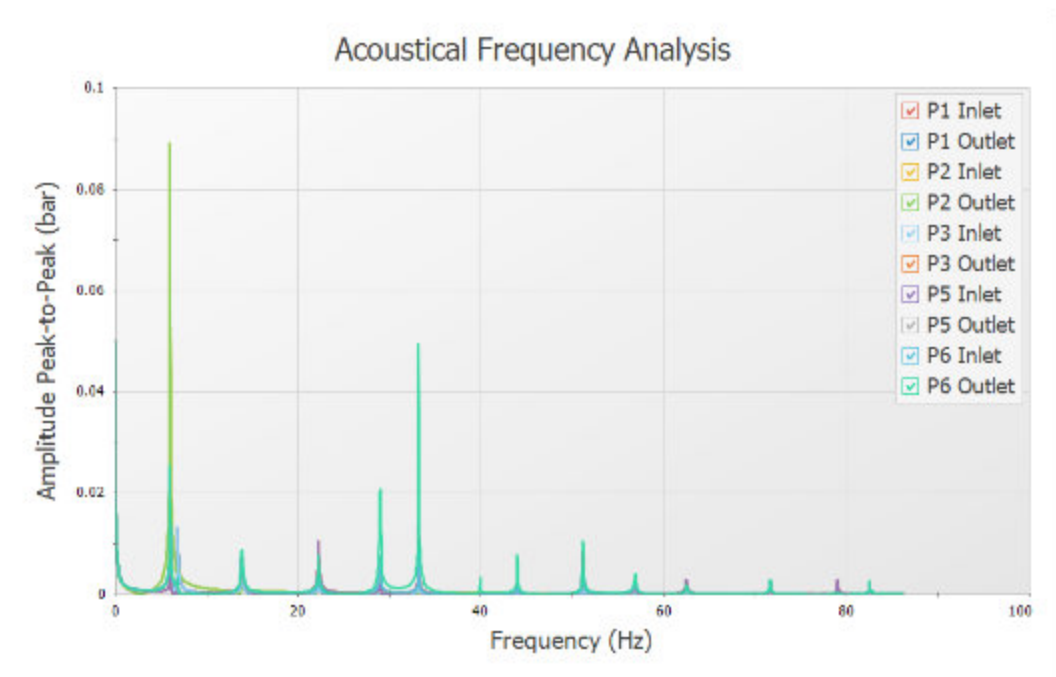
### A. View the Frequency Response Function

1. On the Graph Control tab on the Quick Access Panel, click the Frequency tab to begin generating an Acoustical Frequency Analysis graph.
2. We need to determine which pump speeds within our specified range excite the resonant frequencies of the system. For the purposes of this example, we will graph the Frequency Response Function 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 6.17) represents the frequencies that are excited by the pulse defined in the Pulsation Setup group.

From the graph in Figure 6.17, 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 6.17 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 Acoustical Frequency Analysis graph with the magnitude displayed on a logarithmic scale, generate the frequency graph as discussed above. Right-click on the y-axis and select the box next to Logarithmic.



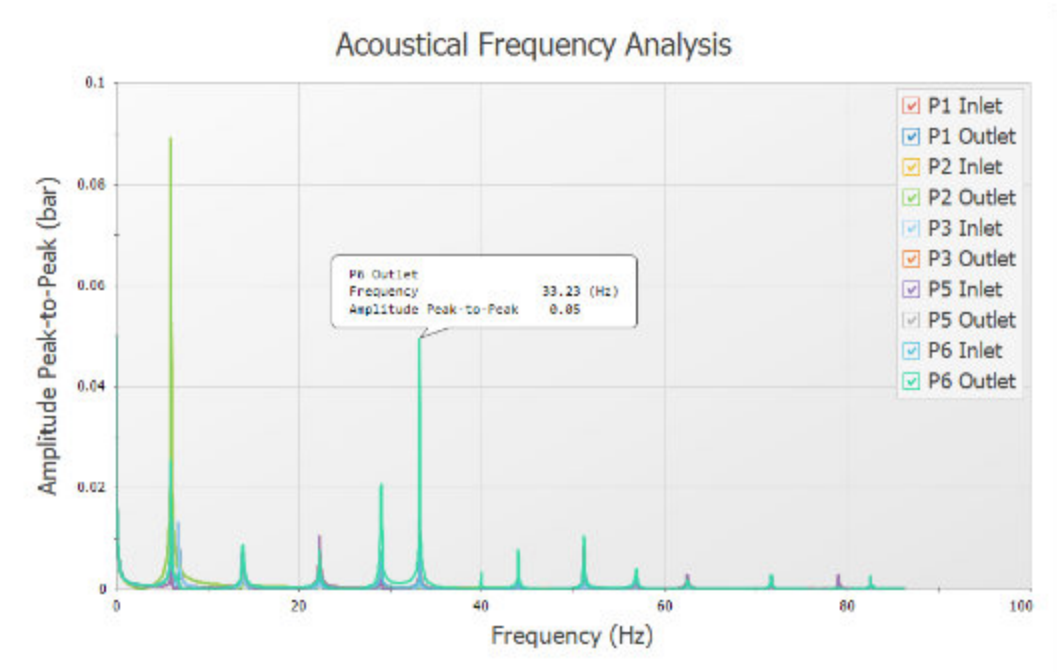


**Figure 6.17 Acoustical Frequency Analysis graph**

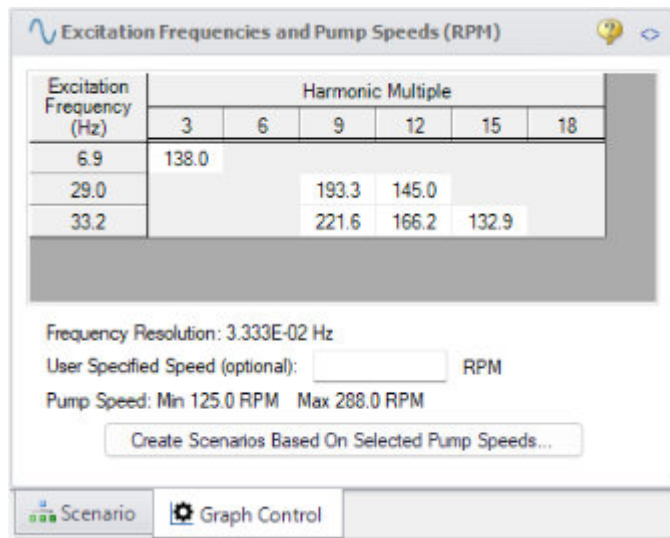
### **B. Choose excitation frequencies to evaluate**

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 6.18.
3. Select "Evaluate Excitation Frequency" to add this frequency to the Excitation Frequencies and Pump Speeds (RPM) 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 6.19.



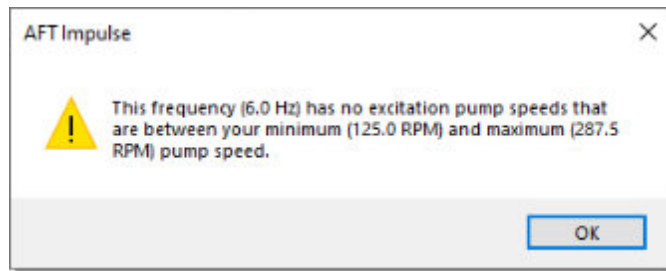
**Figure 6.18** Acoustical Frequency Analysis with the local maxima selected, where right-clicking on the Dialog Flag provides the option to Evaluate Excitation Frequency



**Figure 6.19** Excitation Frequencies and Pump Speeds (RPM) 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 the Pulsation Frequency Analysis panel. For example, try selecting

the largest maxima, which occurs at about 6.0 Hz. The message in Figure 6.20 appears.



**Figure 6.20** Message indicating no excitation pump speeds between the min and max pump speeds exist

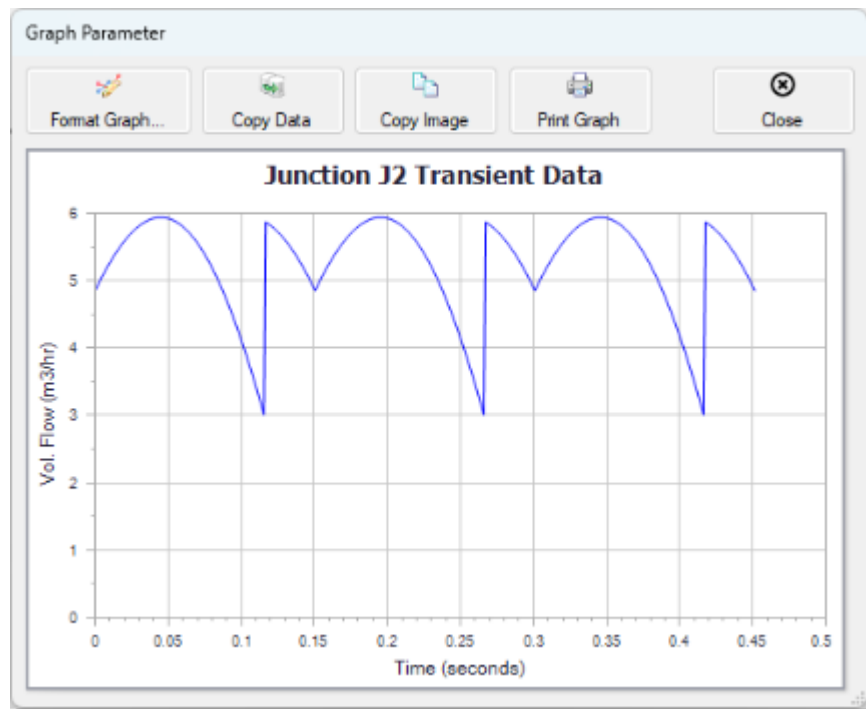
### **C. Create excitation RPM child scenarios and determine the pressure response of the system at excited frequencies**

In the Excitation Frequencies and Pump Speeds (RPM) 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 Excitation Frequencies and Pump Speeds (RPM) panel are pre-selected. Clicking a speed harmonic multiple will toggle it on or off. Speeds that are toggled off will appear with the number in italic font. 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 selected pump speeds in the Excitation Frequencies and Pump Speeds (RPM) 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 Excitation Frequencies and Pump Speeds (RPM) panel. Click Yes to continue creating the child scenarios.
3. Double-click on the “132.9 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 6.21.

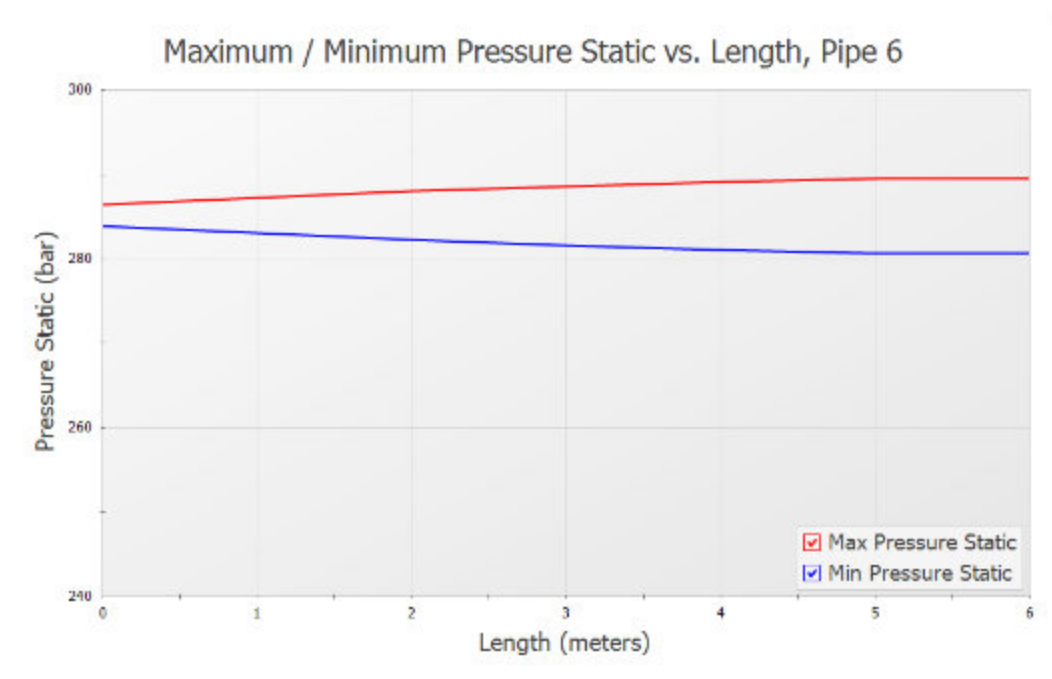


**Figure 6.21 New Assigned Flow transient representing the PD Pump operating at the specified RPM**

Run the “132.9 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 select the Profile tab
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 and adjust the scale by unchecking the Auto Scale box, then changing the Minimum to 240 and the Major Val to 20.

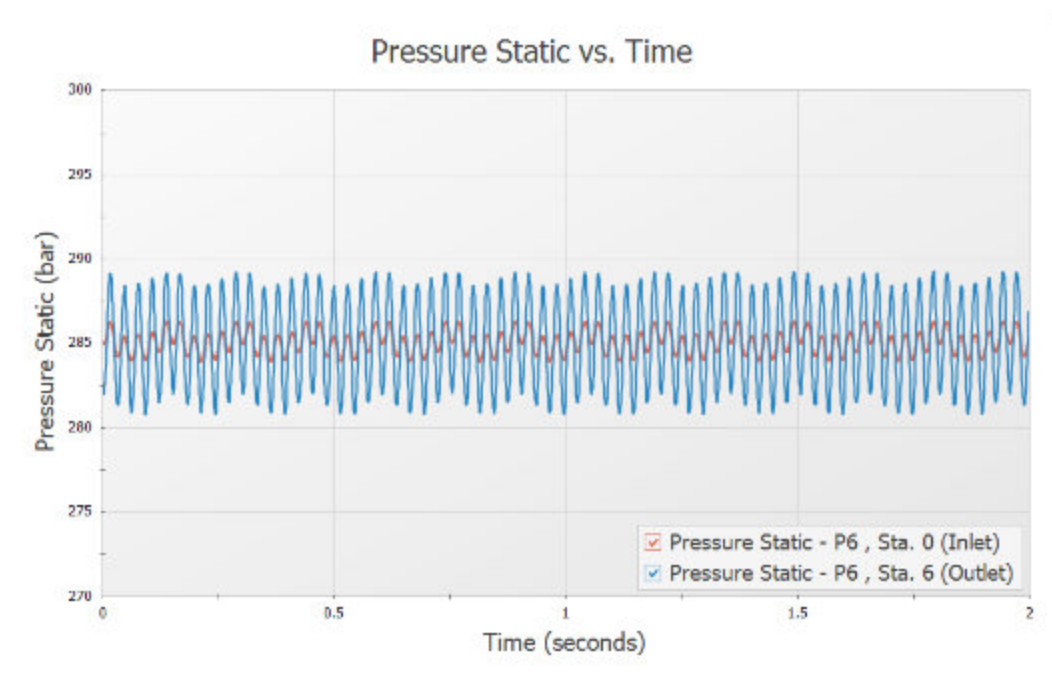
Figure 6.22 shows the profile plot for Pipe P6 at a pump speed of 132.9 RPM.



**Figure 6.22 Max/Min Pressure Profile of Pipe P6 for 132.9 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.
7. Right-click on the y-axis and uncheck the Auto Scale option. Set the Minimum to 270, the Maximum to 300, and the Major Val to 5. The resulting graph is shown in Figure 6.23.



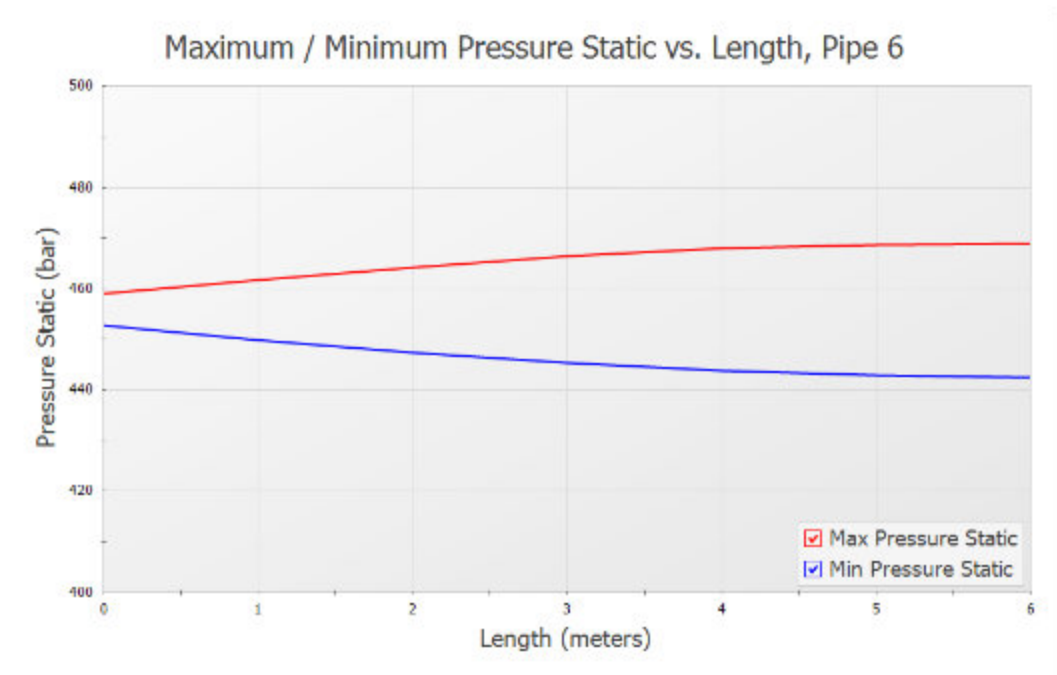
**Figure 6.23 Pressure Oscillations at the Inlet and Outlet of Pipe P6 for 132.9 RPM**

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.

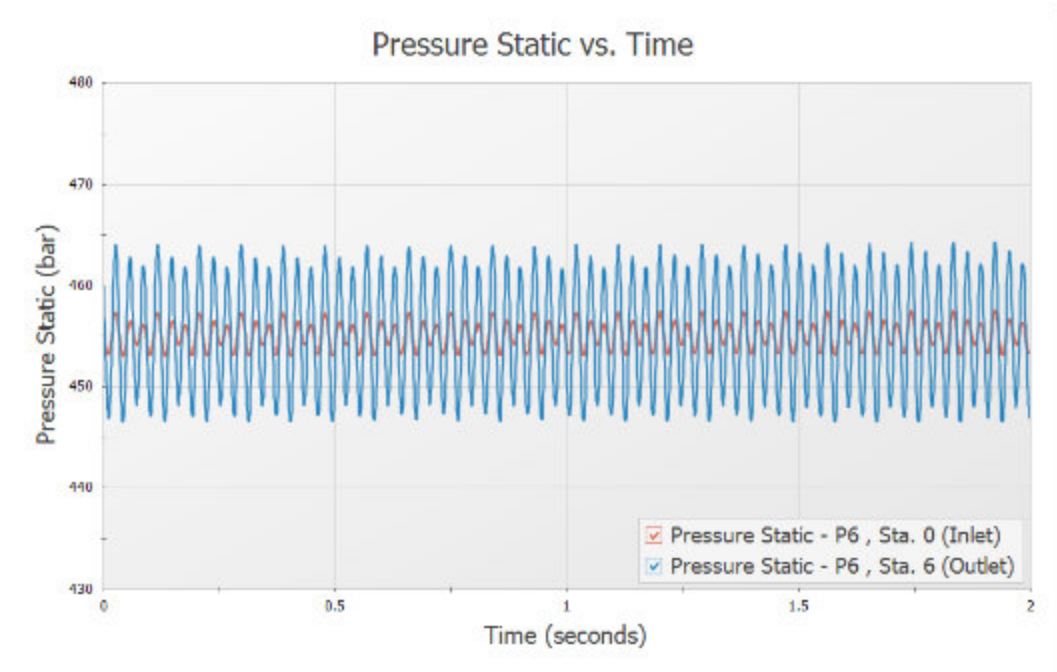
Figure 6.24 shows the Max/Min Pressure Profile for Pipe P6 for the pump speed of 221.6 RPM. The y-axis has been adjusted to scale from 400 to 500, with a Major Val of 10.



**Figure 6.24 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 6.25 (the y-axis has been adjusted to scale from 430 to 480, with a Major Val of 10). 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 resulting in failure. It should be noted in the Output that this scenario also violates the API 674 standard.



**Figure 6.25 Pressure Oscillations at the Inlet and Outlet of Pipe P6 for 221.6 RPM**

## PFA Conclusion

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. It was determined that the pressure response violated the API 674 standard, and a warning was given in the software.



## CHAPTER 7

# Other Impulse Capabilities

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

You can find a number of other example model discussions on the Datacor Pipe Flow Modeling documentation help website. To view additional examples, open the Help menu and choose Show Examples or go to [docs.aft.com](https://docs.aft.com), select the Impulse Help Site, expand the Examples section in the sidebar, then select Walk-Through Examples.

### Microsoft Excel™ data integration

Impulse includes Excel importing and 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

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 Impulse and EPANET Files (\*.INP) can be both imported and exported. Additionally, force files can be exported for use in CAESAR II, TRIFLEX, and ROHR2.

Piping Component Files (\*.PCF) from AutoCAD Plant 3D, SmartPlant, PDS, CADWorx, and other software can be imported into Impulse. These options are all accessed from the File Menu. PCF importing supports multiple files at once and uses zero-length connectors to handle welded junctions automatically. Pipe length units can be adjusted during import, and a dedicated window allows pipe materials to be defined before importing.

### 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**

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**

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.

### **Reciprocating pumps**

Impulse's Reciprocating Pump modeling supports both single-acting and double-acting configurations, along with customizable cylinder arrangements. Multiple cylinders can be defined to simulate a wide range of positive displacement pump setups. This allows accurate representation of real-world pump behavior in the model.

### **Pumps with viscosity corrections**

When the liquid is sufficiently viscous, centrifugal pump performance will be degraded. 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**

Impulse can model flow and pressure control valves. 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. Impulse can model air valves, including different inflow and outflow geometries. Gas volume is assumed to remain fixed at a single computation point and does not travel down the pipeline.

### **Surge tanks**

Surge tanks are a surge suppression device used in low pressure systems with non-volatile fluids. 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**

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**

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, Impulse can model varying elevations along a single pipe.

### Fitting library

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

### Design alerts

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

### Design Alert Layer

The Design Alert Layer displays color-coded indicators directly on the Workspace to show where Design Alerts have been triggered. Each highlighted object is labeled with the violated condition and its evaluated value. This allows quick identification of areas that exceed system requirements across multiple conditions.

### Libraries

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

### Workspace Layer animations

The Workspace Layers provides the ability to animate the transient results as a qualitative tool to visualize the behavior in a network. 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 also be recorded to a file.

### Compare input from different scenarios

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

### Scenario Comparison Layer

The Scenario Comparison Layer highlights differences between the current scenario and a selected comparison scenario directly on the Workspace. Color-coded objects indicate changes, and each one displays specific parameter and value differences through labels and the Inspection Window. This provides a visual way to analyze scenario variations within complex models.

### 4-Component Slurry Model

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

### Pipe filling and draining

Impulse can simulate pipe filling and draining by tracking liquid-gas interfaces, such as gas pockets, moving through the system. It uses the Method of Characteristics for the liquid and a compressible model for the gas to capture flow splits, surge effects, and changes in system forces. This capability helps analyze how filling, draining, or batch movement affects system behavior.

### End transient simulation based on event, Design Alert, or set time

Impulse can end a transient simulation based on specific events, Design Alerts, or a set time. A run can stop automatically when conditions like pressure limits are met, allowing for more precise control of the simulation. When this occurs, output is truncated, a checkpoint file is created, and a message is logged for review.

### Parametric Study tool

The Parametric Study tool in Impulse allows systematic variation of input parameters to analyze how changes affect system performance. It supports sensitivity analysis, optimization, and visual trend exploration to identify key trade-offs and operating conditions. This assists in evaluating scenarios and supports data-driven decision-making.

### Graph Annotation tool

The Graph Annotation tool places auto-updating graphs directly in the Workspace, displaying results that refresh with each simulation run. These graphs can be resized, moved, hidden, and layered like Color Maps, providing a clear visual comparison of output data. This allows for real-time analysis of model behavior without navigating away from the main workspace.

### **Global Junction Morph**

Impulse can convert multiple junctions from one type to another in a single step. This tool allows system structures to be updated by selecting and changing groups of junctions at once.

# Index

## #

4-Component Slurry Model 117

## A

Air Valve junction 2

Animation

    Graph Animation 56

    Workspace Layers Animation 40, 116

AutoCAD Plant 3D 113

## B

Bingham Plastic 115

Brecht & Heller method 115

## C

CADWorx 113

CAESAR II 113

Cavitation 2

Change Units window 33

Controlling pipe 25

## D

Defining objects

    Undefined Objects window 17

Design Alert Layer 116

Design Alerts 89, 116

Duffy method 115

## E

EPANET 113

Event transient specification 46

Excel 113

Exporting 113

## F

Fathom 3

Fitting Library 116

Fluid Libraries

    ASME Steam/Water 3

    Chempak 3

    Impulse Standard 3

    NIST REFPROP 3

Fluid panel 16, 62

Force sets 76

## G

Global Junction Morph 118

Graph Annotation 117

Graph Guide 33

Graph List 53

Graph Results window 4, 33, 52, 68, 92

    Design Alerts 116

    Forces tab 81

    Secondary Y-axis 54

## H

Highlight feature 19

## I

ID numbers 10, 17

Importing 113

Impulse

    Engineering assumptions 2

    Overview 4

    Summary of capabilities 2

Inspection feature 19

Intermediate elevations 116

Isometric 11, 77

### J

#### Junctions 9

- Air Valve 2, 115
- Area Change 98
- Assigned Flow 98
- Gas Accumulator 2
- Pump 45, 87
- Relief Valve 115
- Reservoir 17-18, 44, 88
- Spray Discharge 67, 98
- Surge Tank 2, 115
- Valve 20, 88

### L

- Libraries 116
- Library Manager 116
- Liquid column separation 2
- Lock Object 14

### M

- Method of Characteristics 3, 25
- Model Data window 4, 24
- Modulus of elasticity 23

### N

- Newton-Raphson method 3
- Non-Newtonian fluid 115

### O

- Open Pipe/Jct Window 17
- Output File 29
- Output window 4, 29, 67
  - SSL output 91
- Overview of AFT Impulse 4

### P

- Parametric Study 117
- PCF 113

#### PDS 113

- Periodic transient behavior 115
- PFA Module 83, 96
  - Excitation Frequency Analysis 104
  - Pulsation Setup group 99

#### Pipe Drawing tool 12

#### Pipe forces 76

#### Pipe Object 10

#### Pipe Properties window 22-23

##### Design Alerts 116

##### Intermediate elevations 116

#### Pipes

##### flow direction 14

#### Poisson Ratio 23

#### Power Law 115

#### Preferred Units 18

#### Primary Windows 4

#### Properties windows

##### Using tabs 19

#### Pulp and paper modeling 115

#### Pump junction 45

##### Entering pump curves 45

##### Four quadrant modeling 70

##### Positive displacement pump modeling 114

##### Pump as turbine (PAT) 114

##### Variable speed controllers 114

##### Viscosity corrections 114

### Q

#### Quick Access Panel 9

##### Pinning 9

#### Quick Graph Formatting 36

### R

#### Reciprocating pumps 114

#### Reference positive flow direction 14

#### Relief Valve junction 115



Reservoir junction 44

Reverse Direction 14

Reverse flow 14

Run Model 28

## S

Scenario Comparison Layer 117

Scenario Manager 9, 49

Sectioning panel 25, 48, 91

Show Object Status 17, 19, 24

SmartPlant 113

Solution Progress window 28, 52, 67, 80, 91, 103

SSL Module 83

Stacked Graphs 34

Status Bar 15

Steady Solution Control 15

Steady-state solver 3, 28

Surge Tank junction 2

## T

Toolbars 15

Toolbox 9, 12

- Annotation tool 9

- Pipe Drawing Tool 9

Tooltips 9

Transient cavitation 2, 114

Transient Control group 27, 49, 67, 73, 78, 91

Transient solver 3

## U

Undefined Objects window 17

User Options 17

## V

Valve junction 20

- Event transient specification 46

VFD 114

Viscosity model 16

## W

Wavespeed 23, 25

- Assumption that it remains constant 2

Workspace window 4, 9

- Workspace Layers 38