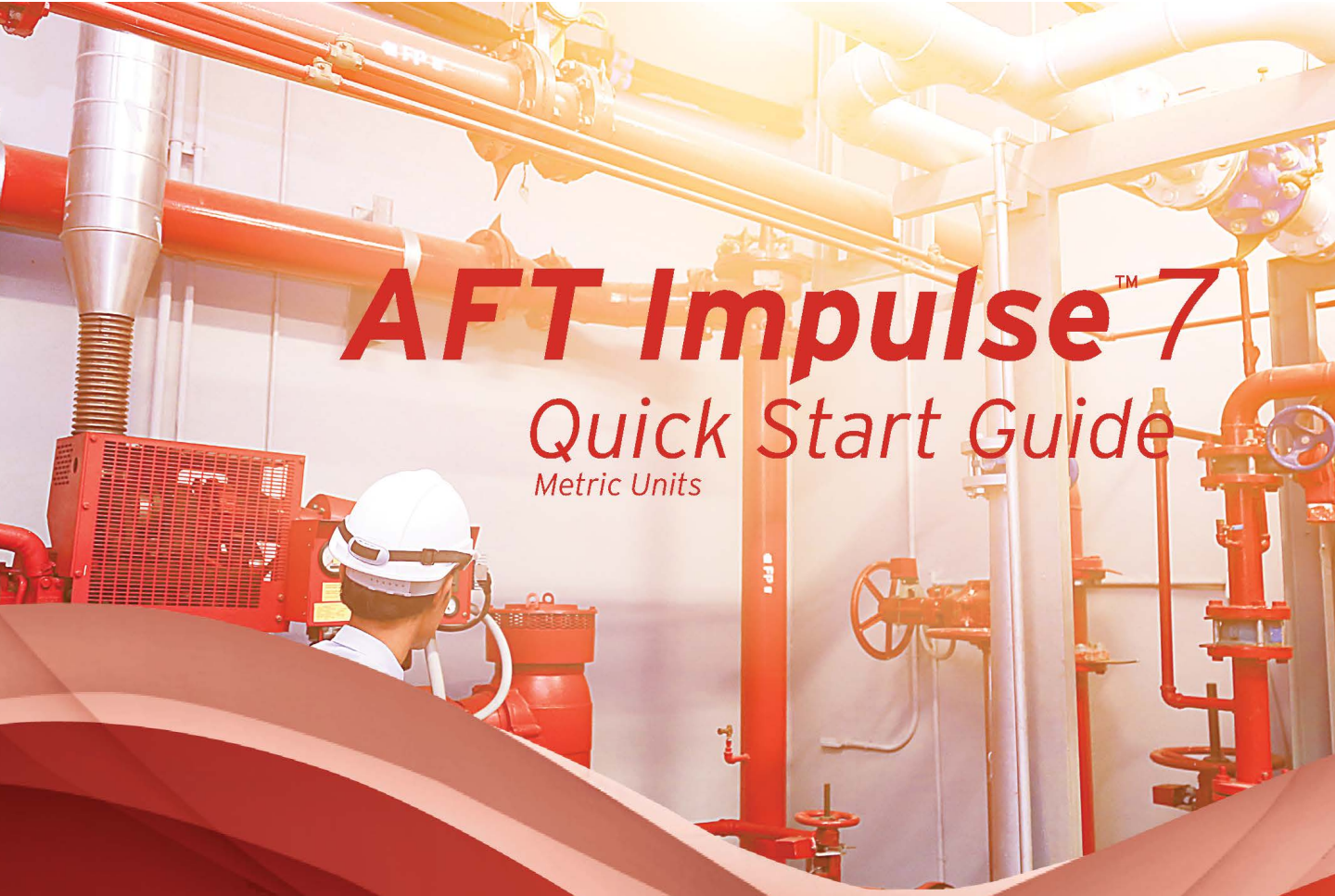


AFT Impulse™ 7

Quick Start Guide

Metric Units

Dynamic solutions for a fluid world ™

Applied Flow Technology

AFT Impulse™

Quick Start Guide

Metric Units

AFT Impulse Version 7
Waterhammer Modeling in Piping Systems

Dynamic solutions for a fluid world ™

CAUTION!

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

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

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

Printed in the United States of America.

First printing.

Contents

CHAPTER 1

Introducing AFT Impulse

Welcome to AFT Impulse™ 7, Applied Flow Technology's powerful waterhammer modeling tool. With AFT Impulse you can model transients caused by a wide range of pipe system behavior. This will allow you to understand transient pressure extremes and, when necessary, size and locate surge suppression equipment.

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

AFT Impulse 7 has two available add-on modules which extend AFT Impulse's extensive 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 PFA Pulsation Frequency Analysis (PFA) module allows the engineer to determine resonant frequencies that can result in excessive vibration and damage to systems.

Designing for waterhammer

Waterhammer can cause catastrophic failure of pipe systems and damage expensive equipment. Properly addressing such issues at the design stage is essential. An AFT Impulse model allows the engineer to better understand and predict the dynamic behavior of the pipe system. When undesirable waterhammer transients are identified at the design stage different strategies to reduce surge pressures can be evaluated. These

may include surge suppression equipment, modifying the design, or modifying the system operation.

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

Modeling capabilities

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

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

Add-on module capabilities (optional)

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

The steady-state solver

Before a waterhammer model can be run, the initial steady-state conditions are required. AFT Impulse obtains the steady-state solution using a Newton-Raphson matrix solution algorithm to obtain a mass and

momentum balance. The algorithm is similar to that used in the acclaimed AFT Fathom™.

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

The transient solver

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

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

Thermophysical property data

AFT Impulse derives physical properties from one of five sources. The first is the standard AFT Impulse set of incompressible fluids which contains data for about 10 common fluids (called AFT Standard).

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

The fourth is the NIST REFPROP database, first introduced in AFT Impulse 7. REFPROP is licensed from the National Institute of Standards and Technology and is included in AFT Impulse. REFPROP has a database of approximately 150 fluids and supports user specified fluid mixtures. AFT Impulse is restricted to non-reacting mixture calculations.

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

Engineering assumptions in AFT Impulse

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

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

AFT Impulse Primary Windows

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

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

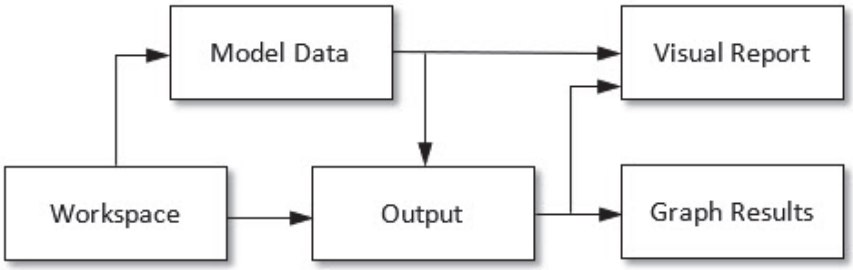


Figure 1.1 Primary Window workflow in AFT Impulse

Input windows

The two windows that function exclusively as input windows are the Workspace window and the Model Data window. These two windows,

one graphical and the other text-based, work together to process model input data with immense flexibility. The tools provided in these two windows allow you to model a large variety of pipe networks.

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

Output windows

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

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

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

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

CHAPTER 2

Valve Closure Example

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

A number of other example model discussions are included in a Help file distributed with AFT Impulse called ImpulseExamplesMetric.chm. It can be opened from the Help menu by choosing "Show Examples" and selecting “Metric Units…”.

Topics covered

- Model building basics
- Entering pipe and junction data
- Specifying System Properties
- Entering transient data
- Sectioning pipes
- Graphing output results

Required knowledge

No prior knowledge is required for this example.

Model file

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

- *Valve Closure Metric.imp*

This example is also provided in US customary units under the English Units file name, Valve Closure.imp.

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

Problem Statement

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

This model will simulate the closure of the valve over a period of 1 second. The valve will stay closed for the remaining simulation duration.

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

Step 1. Start AFT Impulse

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

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

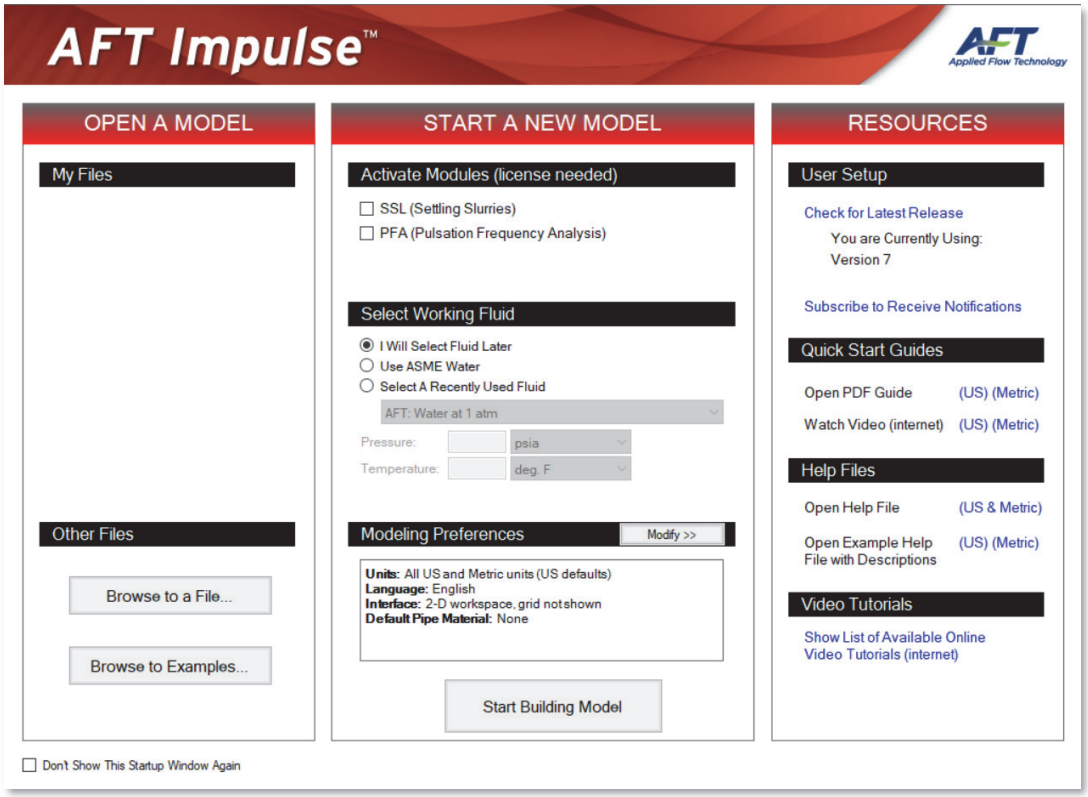


Figure 2.1 AFT Impulse startup window

Some of the actions available from the AFT Impulse Startup window are:

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

- Access other Resources, such as Quick Start Guides, Help Files, and Video Tutorials

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

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

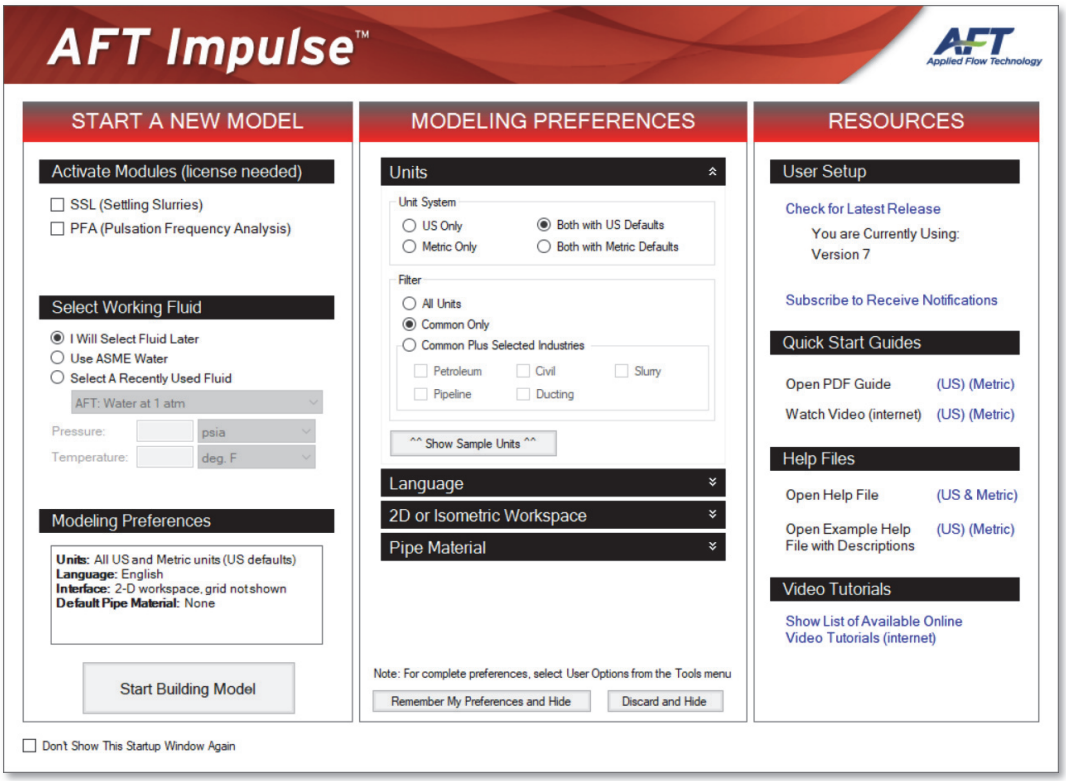


Figure 2.2 Startup Window with Modeling Preferences Expanded

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

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

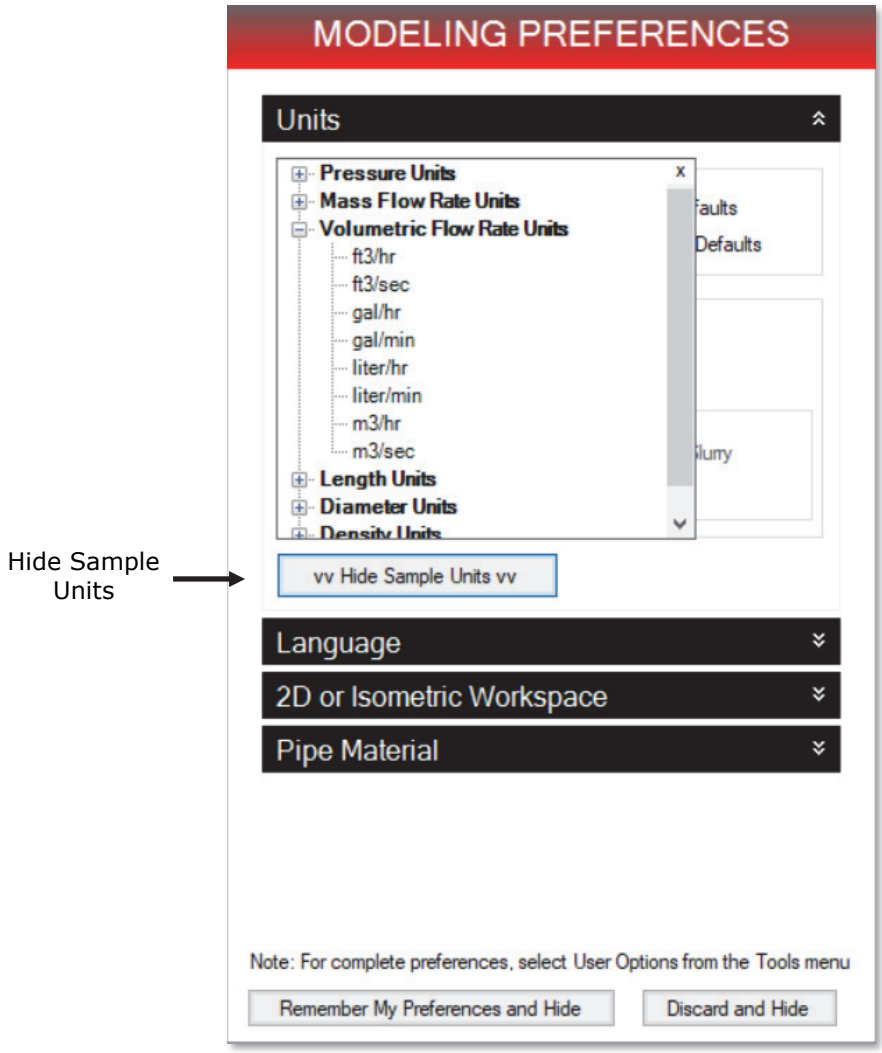


Figure 2.3 Show Sample Units with “Common Only” Selected

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

The Workspace window is the initial active (large) window, as seen in Figure 2.2. The five tabs in the AFT Impulse window represent the five Primary Windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

The Workspace window

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

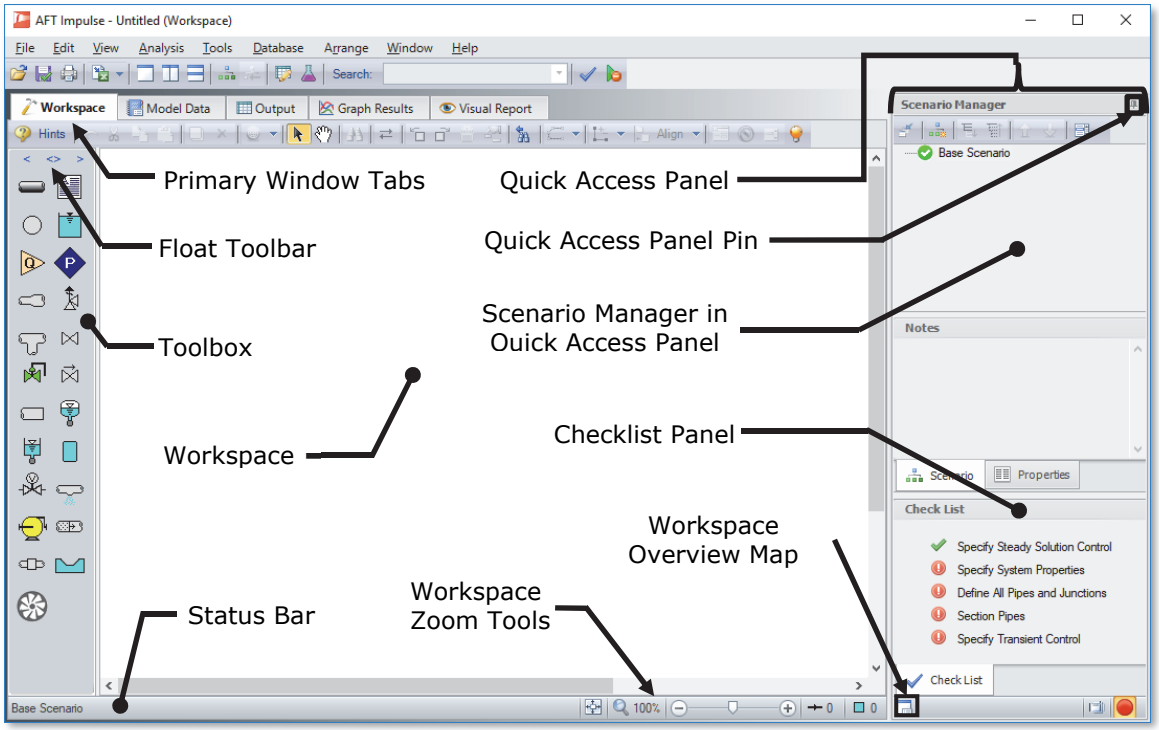


Figure 2.4 **The Workspace window is where the model is built. The other four Primary Windows are found on the tabs along the top of the Workspace. The Status Bar shows the model status**

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

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

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

Unpinning the Quick Access Panel

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

Step 2. Lay out the model

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

A. Place a reservoir

➢ **To start, drag a reservoir junction from the Toolbox** and drop it on the Workspace. Figure 2.3a shows the Workspace with one reservoir.

Objects and ID numbers

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

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

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

Editing on the Workspace

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

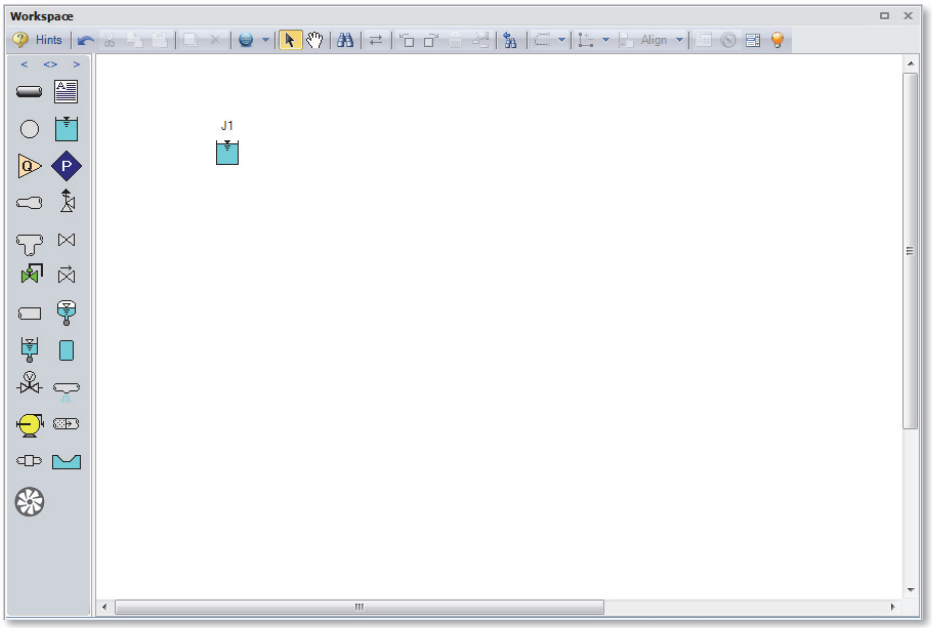


Figure 2.3a Valve closure model with first junction placed

B. Place the other junctions

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

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

Place a branch, valve and third reservoir junctions from the Toolbox and drop them on the Workspace so that your model appears similar to that shown in Figure 2.5b.

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

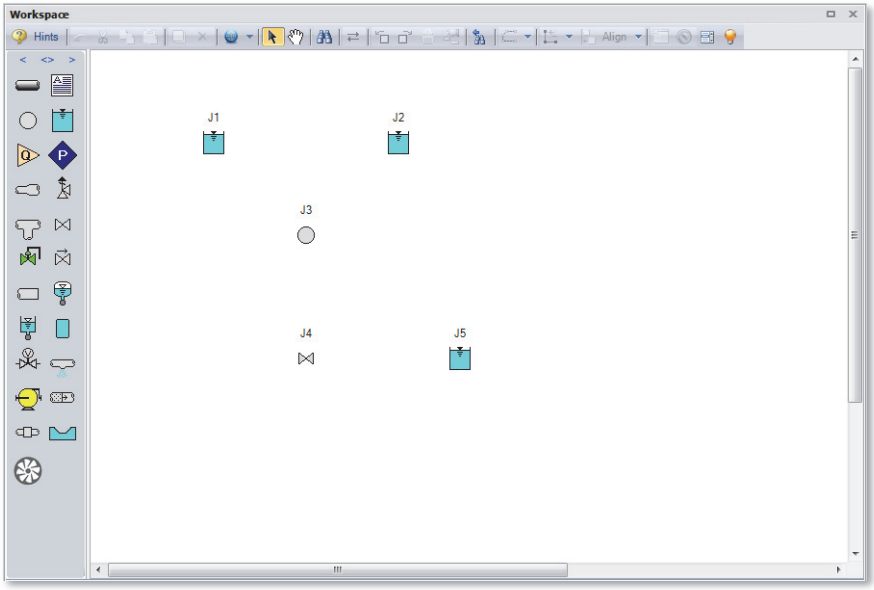


Figure 2.5b Valve closure model with all junctions placed

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

C. Draw a pipe between J1 and J3

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

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

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

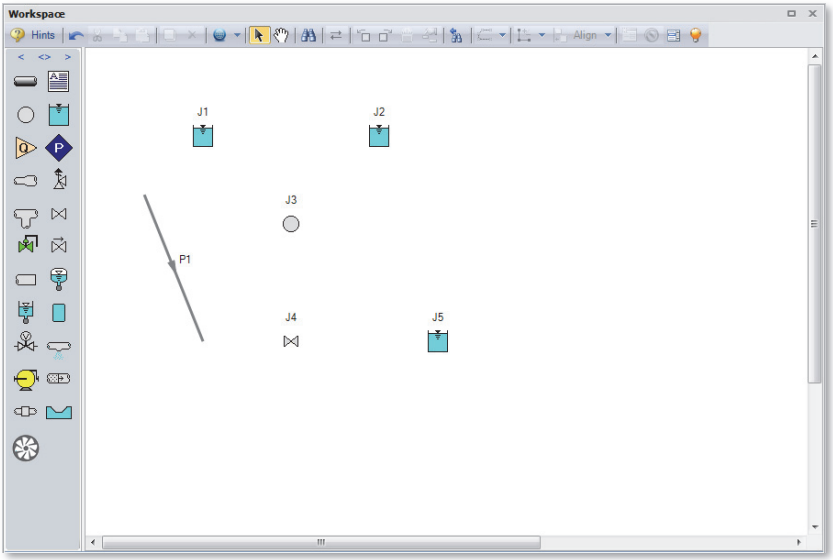


Figure 2.5c Valve closure model with first pipe drawn

➢ **To place the pipe between J1 and J3,** use the mouse to grab the pipe in the center, drag it so that its left endpoint falls within the J1 Reservoir icon, then drop it there (see Figure 2.5d). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J3 Branch icon (see Figure 2.5e).

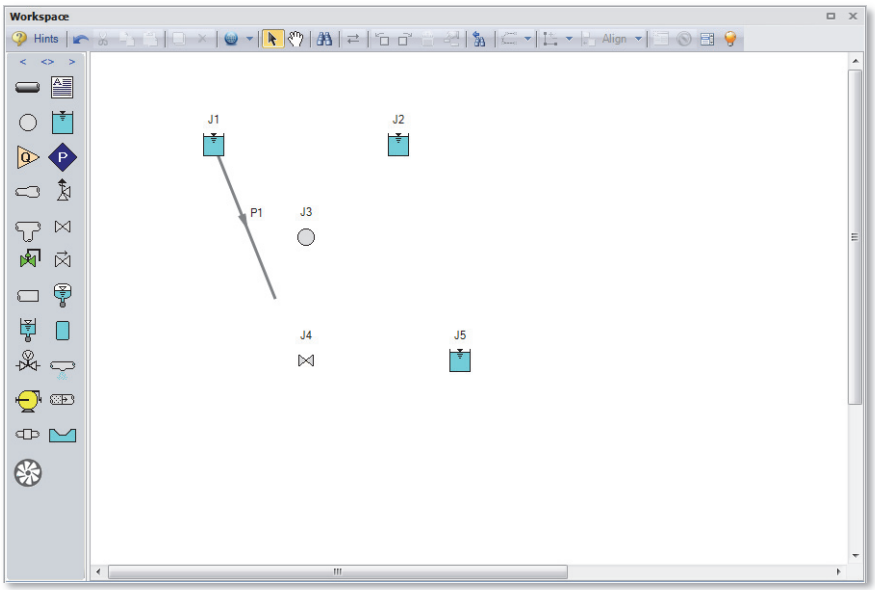


Figure 2.5d Valve closure model with first pipe partially connected

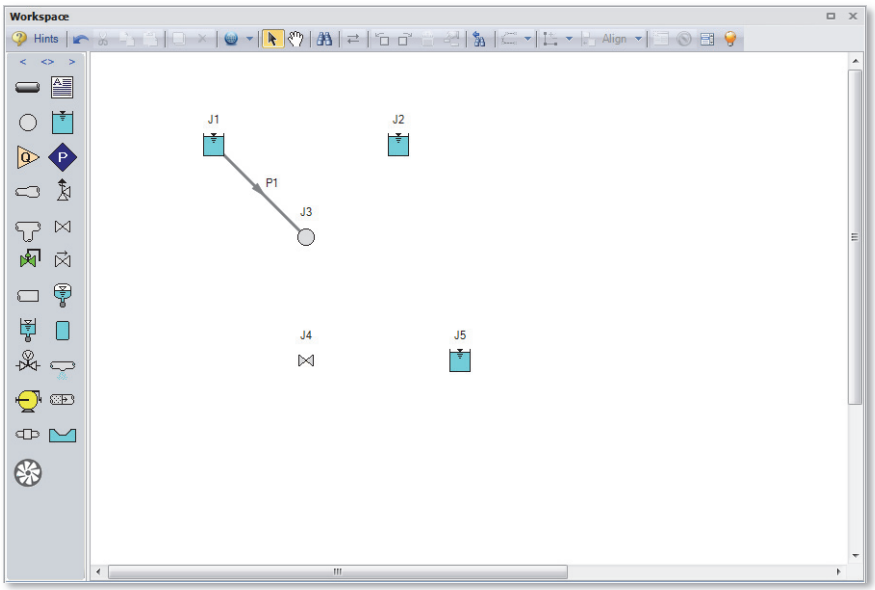


Figure 2.5e Valve closure model with first pipe connected

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 2.5f. After all the pipes are drawn, all the objects in the model are graphically connected. Save the model by selecting Save in the File menu or by clicking on the diskette button on the Main Toolbar.

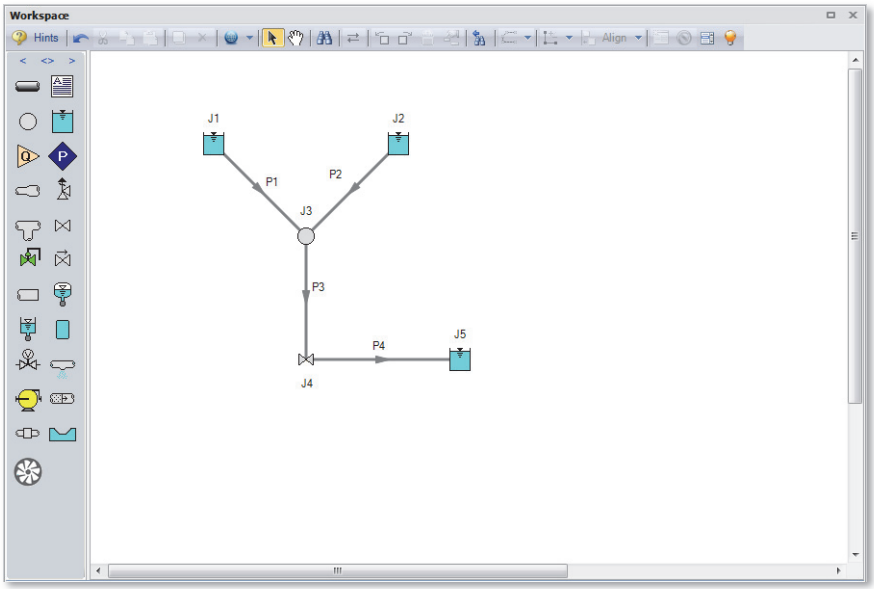


Figure 2.5f Valve closure model with all pipes and junctions placed

Reference positive flow direction

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

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

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

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

Step 3. Complete the first three checklist requirements

- **Next, click the checkmark on the Main Toolbar** at the top of the AFT Impulse window. This opens the Checklist window (Figure 2.6). The Checklist window (also simply called the Checklist) contains five items. Each item needs to be completed before AFT Impulse allows you to run the Solver.

Note: The Checklist contains five items, but if you are running your model in steady state mode only, the two items at the bottom of the list, Section Pipes and Specify Transient Control, will appear faded and you will be unable to open them because they pertain to the transient analysis only. To change back and forth between the steady state and transient analyses, choose either “Steady Only” or “Transient” from the Time Simulation selection on the Analysis menu.

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

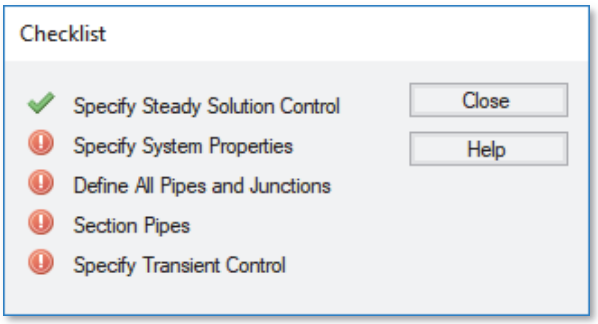


Figure 2.6 The Checklist window tracks the defined and undefined model input

A. Specify steady solution control

The first item, "Specify Steady Solution Control", is always checked when you start AFT Impulse because AFT Impulse assigns default solution control parameters for the steady-state part of the analysis. In general, you do not need to adjust these values. If necessary, you can make adjustments by opening the Steady Solution Control window from the Analysis menu.

B. Specify system properties

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

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

➢ **Select System Properties from the Analysis menu** to open the System Properties window (Figure 2.7). For this example, use the AFT Standard database and select "Water at 1 atm" in the upper list, then click the "Add to Model" button. Enter a temperature of 70 deg. F and click the Calculate Properties button to obtain the Density, Dynamic Viscosity, Bulk Modulus and Vapor Pressure.

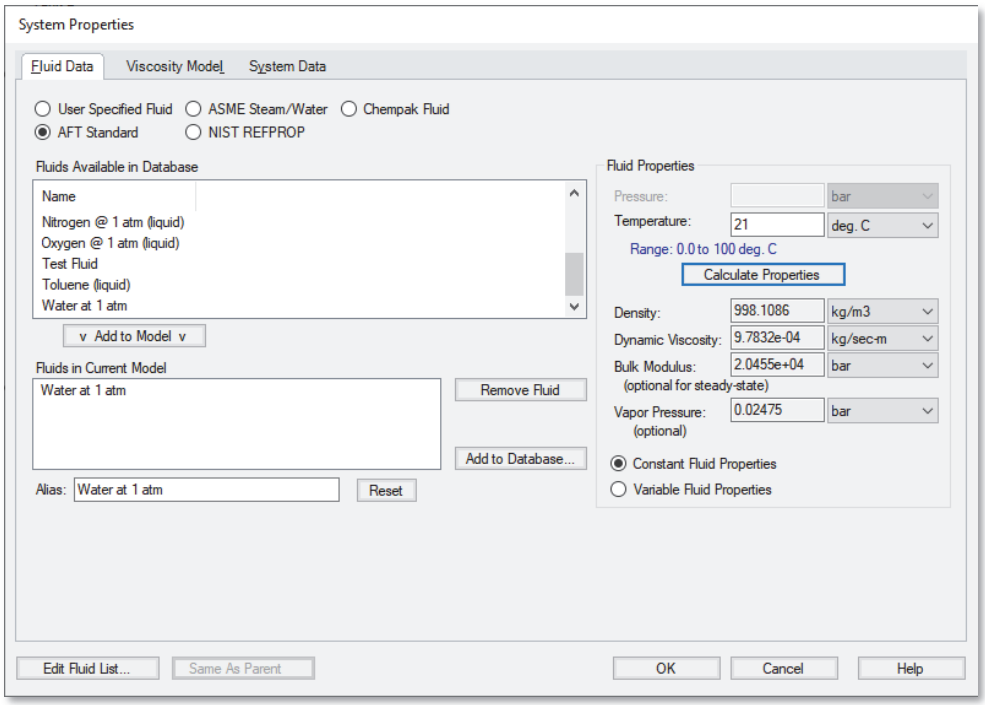


Figure 2.7 System Properties window is where you specify the fluid

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

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

The third item on the Checklist, “Define All Pipes and Junctions”, is not as straightforward to satisfy as the first two. This item encompasses the proper input data and connectivity for all pipes and junctions.

Object status

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

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

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

Showing undefined objects

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

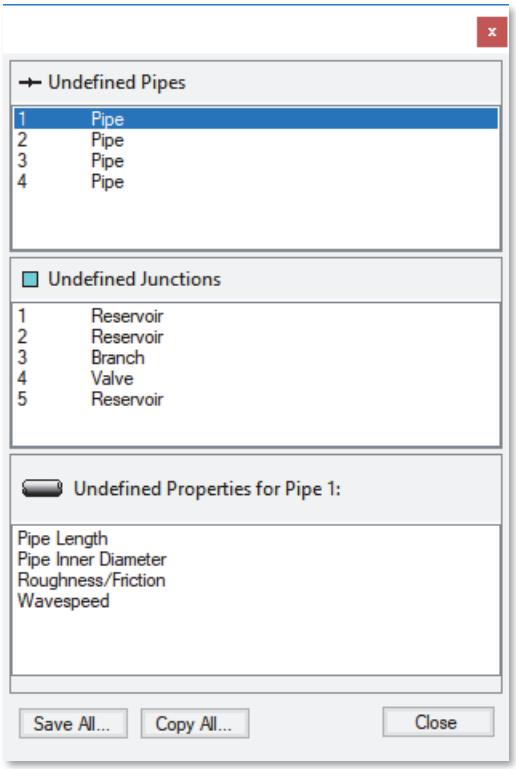


Figure 2.8 The Undefined Objects list lets you see the undefined properties for each undefined object

A. Enter data for reservoirs

As shown in Figure 2.5f, the J1 junction is a Reservoir junction.

➢ **To define the first reservoir,** open the J1 Reservoir Properties window (Figure 2.9) by double-clicking on the J1 icon. The pipe depth should now be enabled for entry, showing that Pipe 1 is connected to the reservoir.

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

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

1. Set the Tank Model to “Infinite Reservoir”.
2. Enter 15 meters for the Liquid Surface Elevation. You can assign any unit of pressure or elevation found in the adjacent drop-down menu of units
3. Enter 0 barG (0 kPa(g)) 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: You can choose default units for many parameters (such as meters for length) in the User Options window in the Tools menu under Preferred Units.

You can give the object a name, if desired, by entering it in the Name field at the top of the window. By default, the junction’s name indicates the junction type. In Figure 2.9, the name of this reservoir has been changed to Supply Tank A. The name can be displayed on the Workspace, Visual Report or in the Output.

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

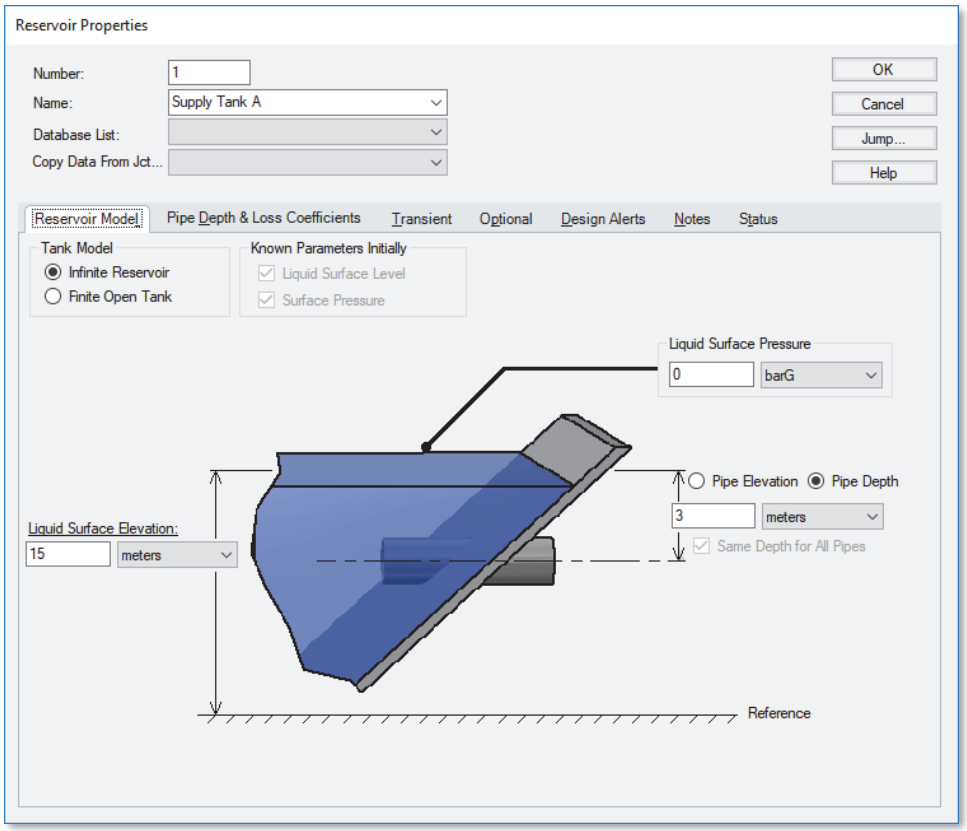


Figure 2.9 Properties window for Reservoir J1

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

Using the folder tabs in the Properties windows

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

If there is only one pipe connected to the reservoir, the depth can be entered directly on the diagram on the Reservoir Model tab for convenience. The depth can also be entered no matter the number of pipes on the Pipe Depth & Loss Coefficients tab. The pipe table allows

you to specify entrance and exit loss factors for each pipe connected to the tank (in this case there is only one). The default selection is for the loss factors to be specified as zero. To change the loss factors later, click within the pipe table and enter the loss.

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

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

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

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

➢ **Next, repeat this process for junctions J2 (Supply Tank B) and J5 (Discharge Tank) with the following data for the Infinite Reservoir tank model**:

- For J2, use a Liquid Surface Elevation of 14 meters, a Pipe Depth of 3 meters, and a Liquid Surface Pressure of 0 barG (0 kPa(g))
- For J5 use a Liquid Surface Level of 1.5 meters, a Pipe Depth of 1.5 meters, and a Liquid Surface Pressure of 0 barG (0 kPa(g))

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 2.10.

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

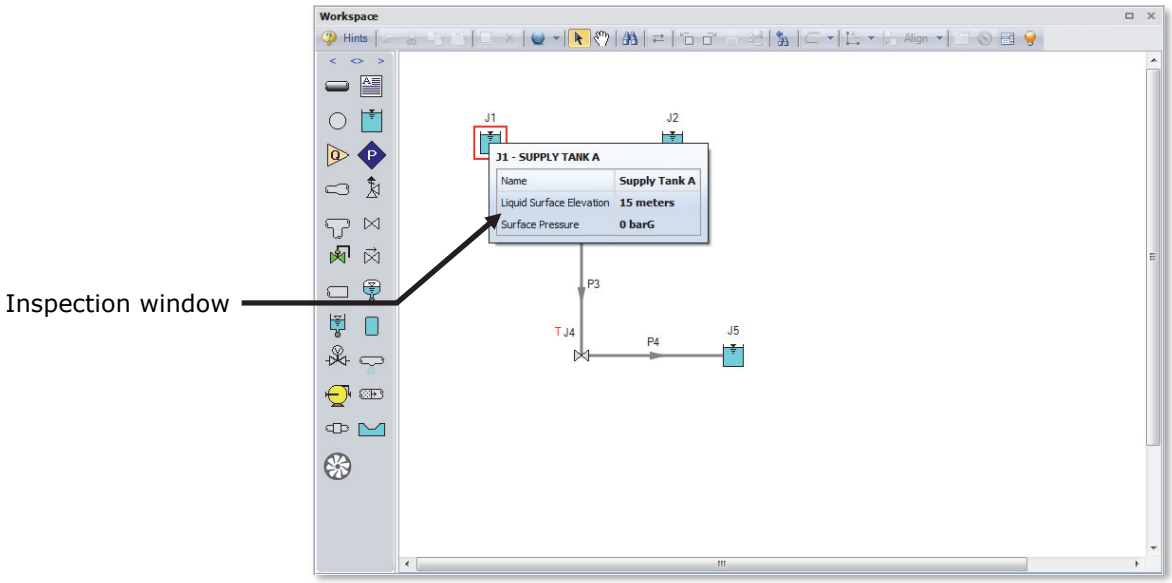


Figure 2.10 Inspecting junction properties from the Workspace by clicking and holding the right mouse button

B. Enter branch data

➢ **Open the J3 Branch Properties window** and enter an elevation of 0 meters.

C. Enter valve data

The J4 junction is a Valve junction. This junction will be the initiator of the transient.

➢ **Double-click the J4 junction icon** to open the Valve Properties window (Figure 2.11)

1. Enter an elevation of 0 meters
2. Choose the Loss Model as Cv and enter a Cv of 1000 (this is the steady-state Cv)

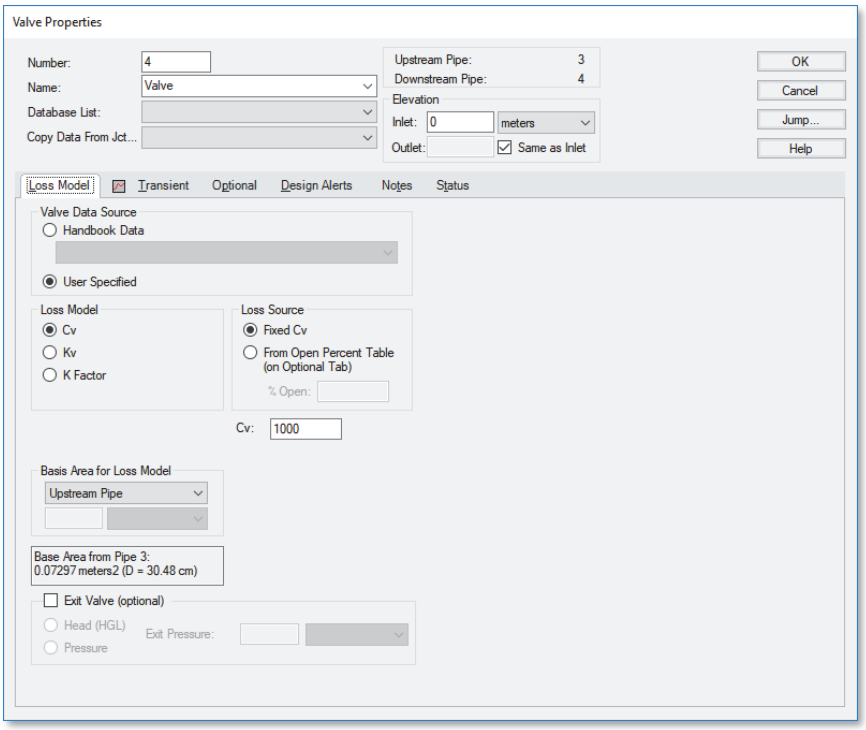


Figure 2.11 Input data for Valve junction J4

3. Click the Transient tab and select Time under "Initiation of Transient" to indicate that the valve closure will be initiated at the time you specify, then enter the following Time vs. Cv data for the valve's Cv at various times during the simulation (Figure 2.12).

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

The first data point (Cv = 1000 at time zero) must match the steady-state value (entered in step 2). 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.

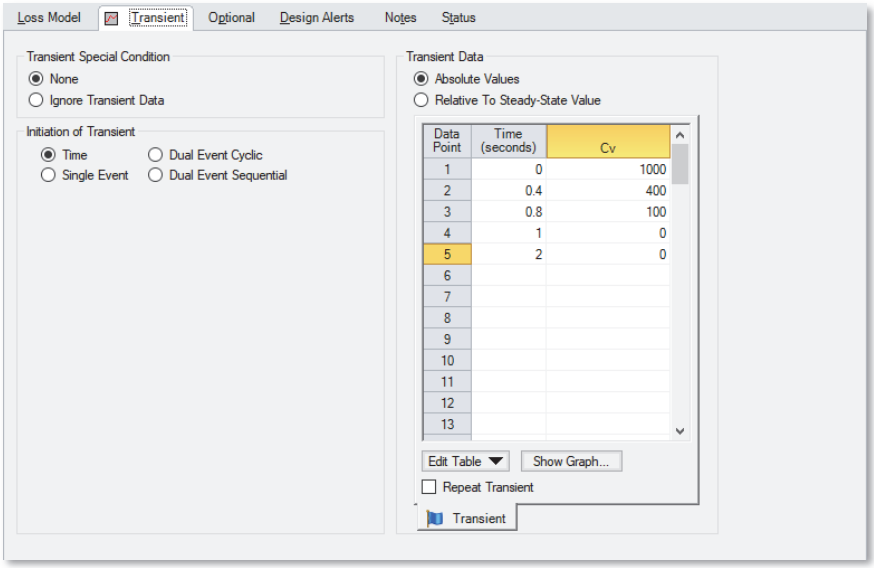


Figure 2.12 Transient input data for Valve Junction J4

When transient data is entered for a junction, a “T” symbol is shown next to the junction number on the Workspace.

4. Click the OK button

D. Enter pipe data for 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 folder tab opens by default (Figure 2.13).

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

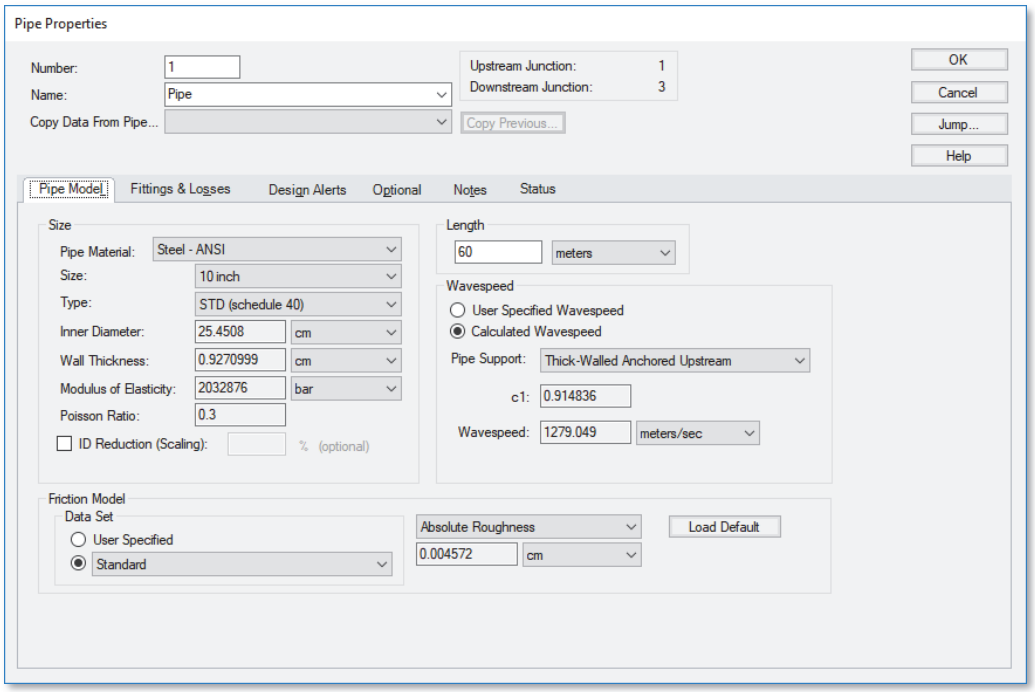


Figure 2.13 Properties window for Pipe P1

The Pipe Properties window

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

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

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

By double-clicking the connected junction number, you can jump directly to the junction's Properties window. Or you can click the Jump button to jump to any other part of you 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 databases supplied with AFT Impulse, and was automatically obtained when the Steel - ANSI, 10-inch, STD (schedule 40) option was chosen. The calculated wavespeed is 1279 meters/sec (Figure 2.13).

E. Enter data for other pipes

Similar to Pipe P1, enter the following data for the other pipes:

Pipe	Length (meters)	Size (inches)	Pipe Support	Friction Model
1	60	10	Thick-Walled Anchored Upstream	Standard
2	45	10	Thick-Walled Anchored Upstream	Standard
3	15	12	Thick-Walled Anchored Upstream	Standard
4	12	12	Thick-Walled Anchored Upstream	Standard

F. Check pipes and junctions

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

G. 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 2.14. To change to this window, you can click on the Model Data tab, select it from the Window menu, or press Ctrl+ M. The Model Data window gives you a text-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, exported into excel, saved to a formatted file, printed to an Adobe™ PDF, or printed out for review.

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

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

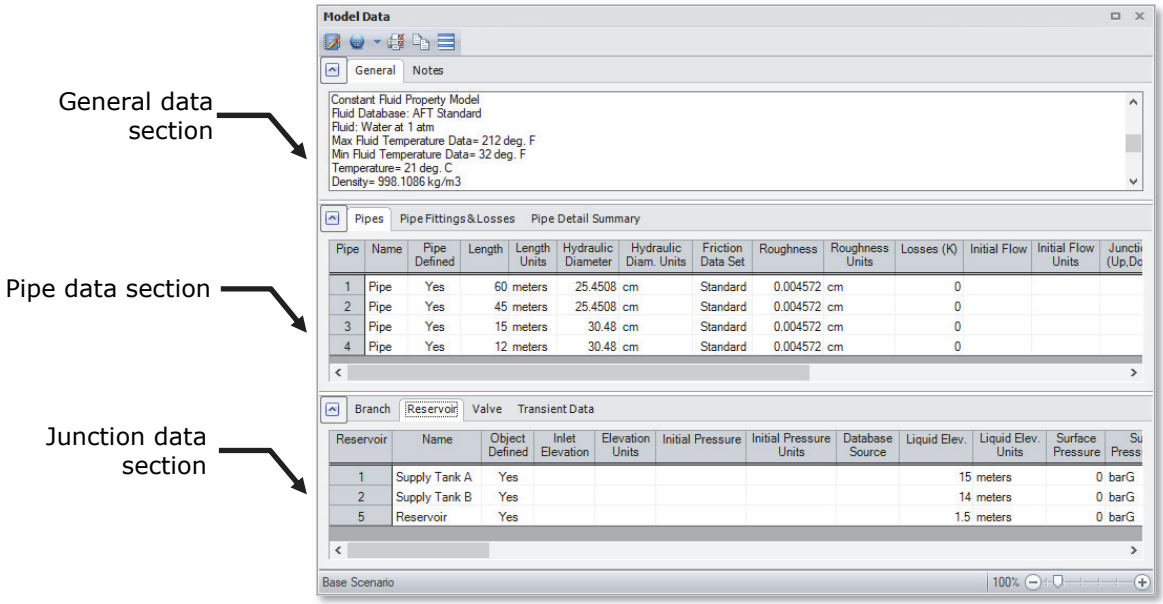


Figure 2.14 The Model Data window shows all input in text form

Step 5. Complete the last two checklist requirements

After completing the first three Checklist items, sufficient information exists to run the model in steady-state. The final two Checklist items are for transient modeling. To run the model in steady-state, select “Time

Simulation" from the Analysis menu and then choose "Steady Only". If one does this, the final two Checklist items are grayed out and the model can be run.

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.

A. Specify pipe sectioning

The fourth Checklist item is Section Pipes. This window cannot be opened until sufficient data in the System Properties and Pipe Properties windows has been specified. This required data includes that the fluid must be selected (this was done in Step 3B when the System Properties were entered). Additionally, all pipes must have a length and wavespeed entered. After this data has been entered, the pipes can be sectioned. The Section Pipes window helps divide the pipes into computation sections in a manner which is consistent with the Method of Characteristics (MOC).

Select Section Pipes on the Analysis menu to display the Section Pipes window (Figure 2.15). 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 Δt 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 Section Pipes window automates this process by searching for sectioning which satisfies the required tolerance. While a 15% variance in wavespeed is often acceptable, AFT Impulse uses 10% by default.

Results for one to five pipe sections

Details for all pipes when 2 sections in the controlling pipe is selected above

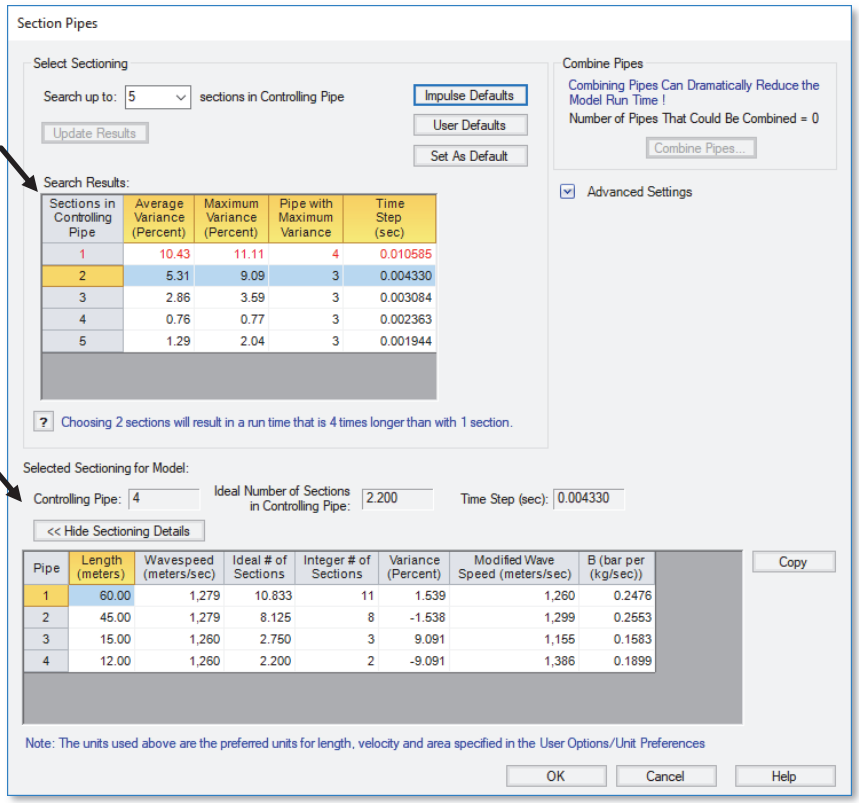


Figure 2.15 The Section Pipes window automates the sectioning process and calculates the time step

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

In this example model, the results when the controlling pipe has only one section show a maximum variance of 11%. Since this is greater than the default 10% threshold (shown when the “Show Advanced Settings” button is selected), the first row is colored red and was not automatically selected. The option to have two sections in the controlling pipe is selected since that has the fewest sections with an acceptable maximum variance. The time step is shown to be 0.00433 seconds.

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

B. Specify transient control

The fifth and final Checklist item is Specify Transient Control. This window allows you to specify the time at which the transient starts and ends, as well as how much data to include in the output file.

- **Select Transient Control on the Analysis menu** to display the Transient Control window (Figure 2.16). Enter 0 seconds in the Start Time field and 2 seconds in the Stop Time field.

  The Transient Control window allows you to enable or disable transient cavitation modeling and select which model to use. It also offers control over how AFT Impulse should respond to artificial transients. Artificial transients are a problem that can sometimes occur when steady-state and transient conditions are inconsistent.

  At the bottom of the window, the projected output file size is shown. You should pay attention to this number, as the output file size can grow very large. In this case, the output file will be 213 kB. If the output file does become excessively large, you will want to limit the number of time steps and pipe output written to disk.

- **Click OK to accept the current settings**. The last Checklist item should be completed. The model is ready to be solved.

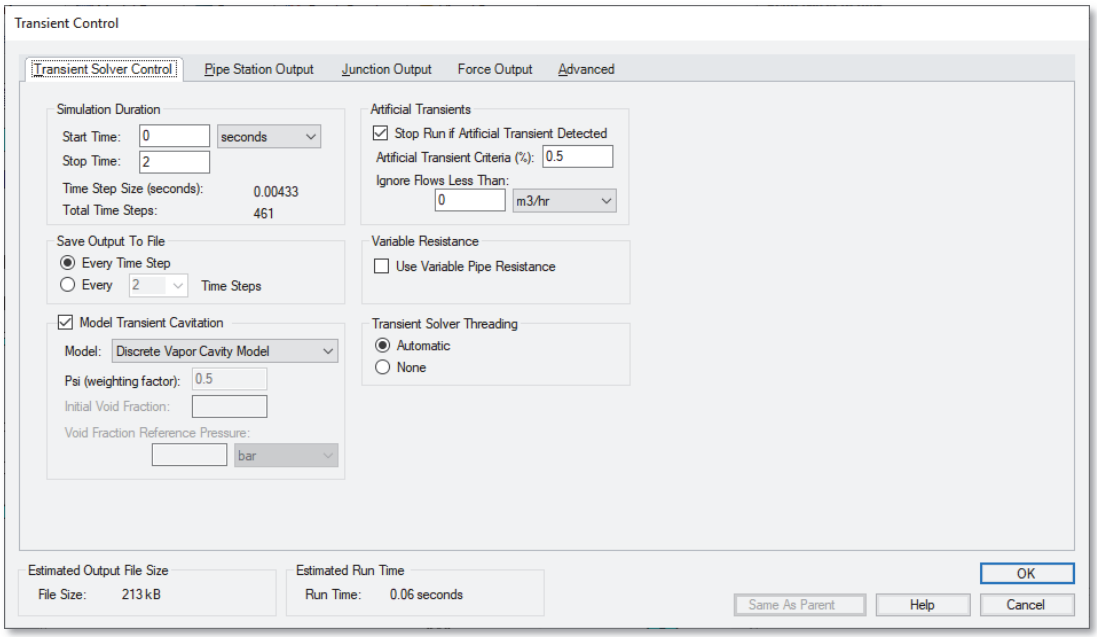


Figure 2.16 The Transient Control window offers features to specify the time span for the transient and what output data is written to disk

C. Save the model

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

Step 6. Run the solver

➢ **Choose "Run Model…" from the Analysis menu** or click the arrow icon on the Toolbar. During execution, the Solution Progress window is displayed (Figure 2.17). You can use this window to pause or cancel the Solver's activity.

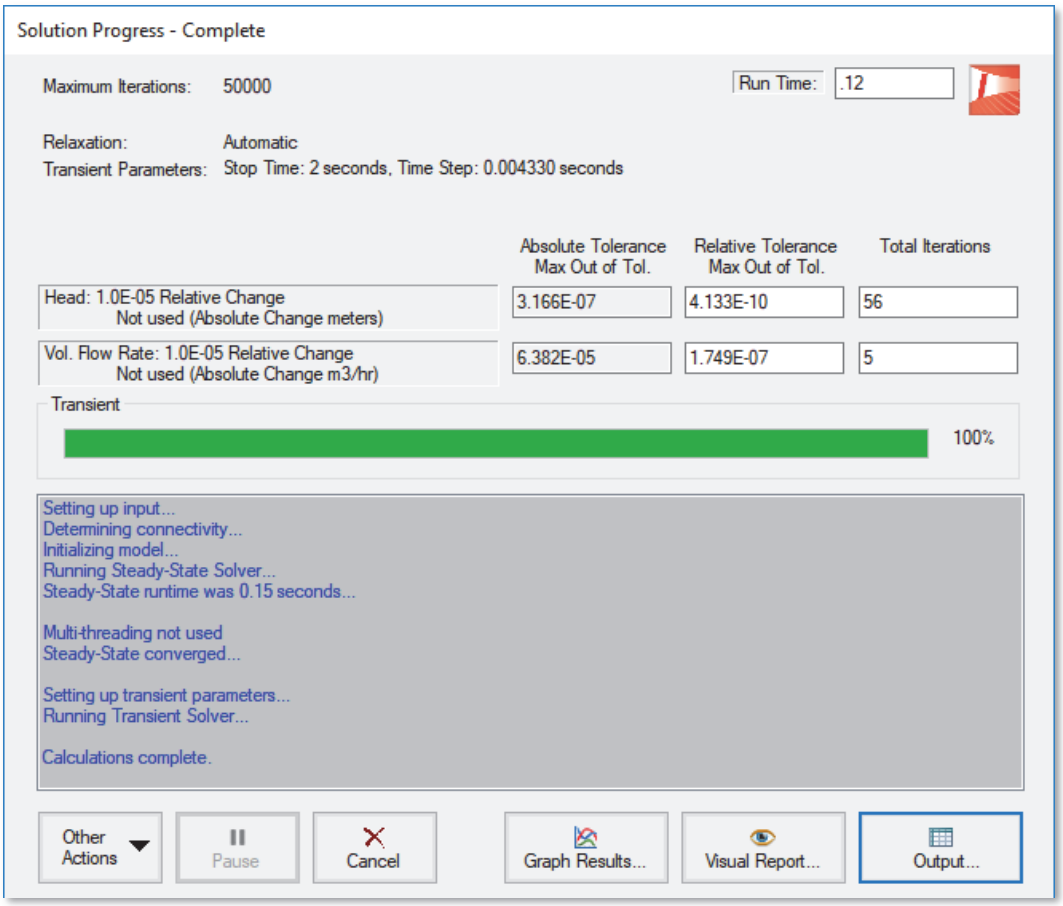


Figure 2.17 The Solution Progress window displays the state of the simulation

The two solvers

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

Before a transient simulation can be initiated, the initial conditions are required. These initial conditions are the steady-state solution to the system. After the steady-state solution is obtained by the Steady-State Solver, AFT Impulse uses the results to automatically initialize the Transient Solver and then run it.

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

The transient output file

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

The output file will remain on disk until the user erases it or the input model is modified. This means that if you were to close your model right now and then reopen it, you could proceed directly to the Output window for data review without rerunning your model.

Step 7. Review the Output window

The Output window (Figure 2.18) is similar in structure to the Model Data window. Three areas are shown (including the general output section, the pipe output section, and the junction output section), and you can expand or collapse each area by dragging the boundary between the areas up or down, or by clicking the arrow beside the area label. The items displayed in the tables are those items you 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 output 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 output section and is the active tab displayed in Figure 2.19.

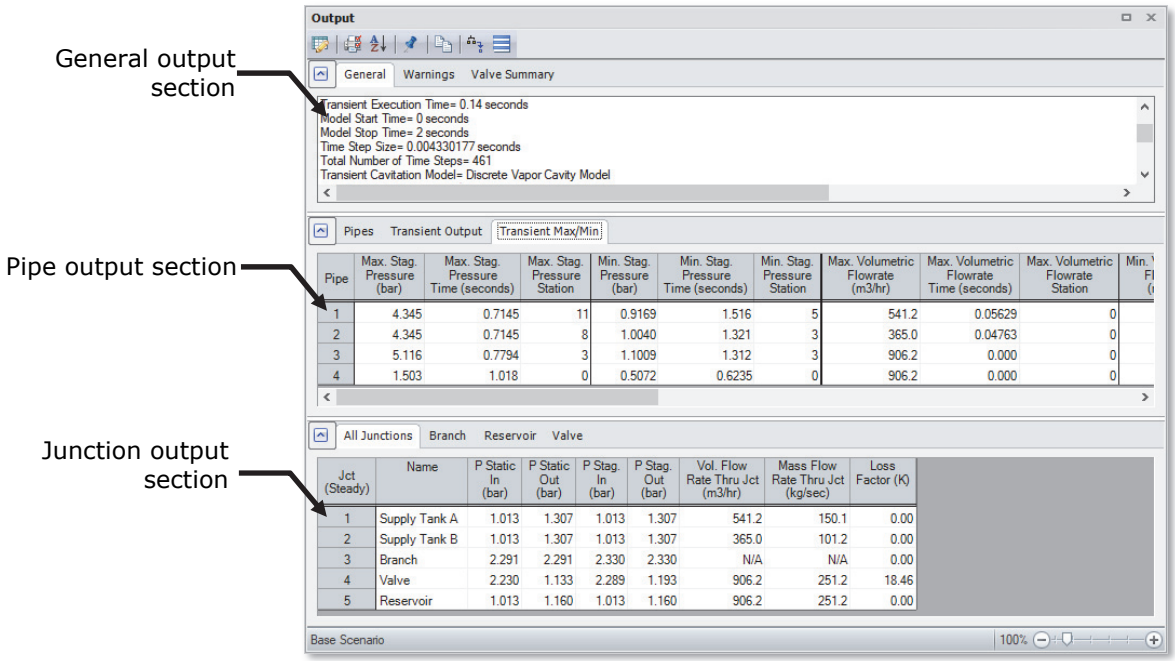


Figure 2.18 The Output window displays steady and transient output in text form

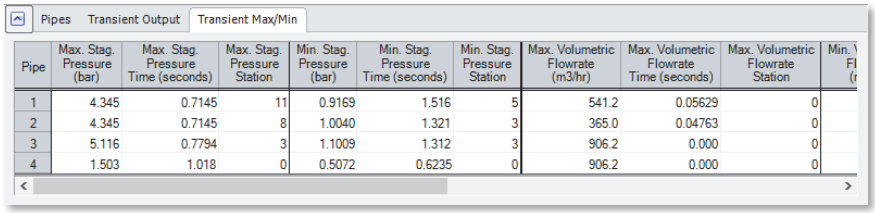


Figure 2.19 The Output window displays maximum and minimum transient data in the Transient Max/Min table

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 output section (Figure 2.20). Moving the slider along the bottom will change the time step shown in the table.

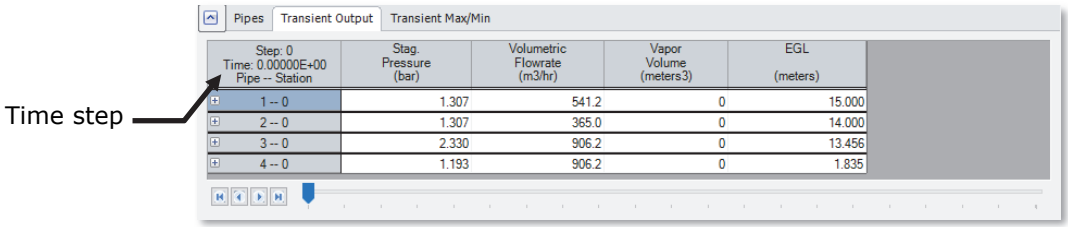


Figure 2.20 The Output window displays transient data for each time step in the Transient Output table

The Output Control window (Figure 2.21) 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.

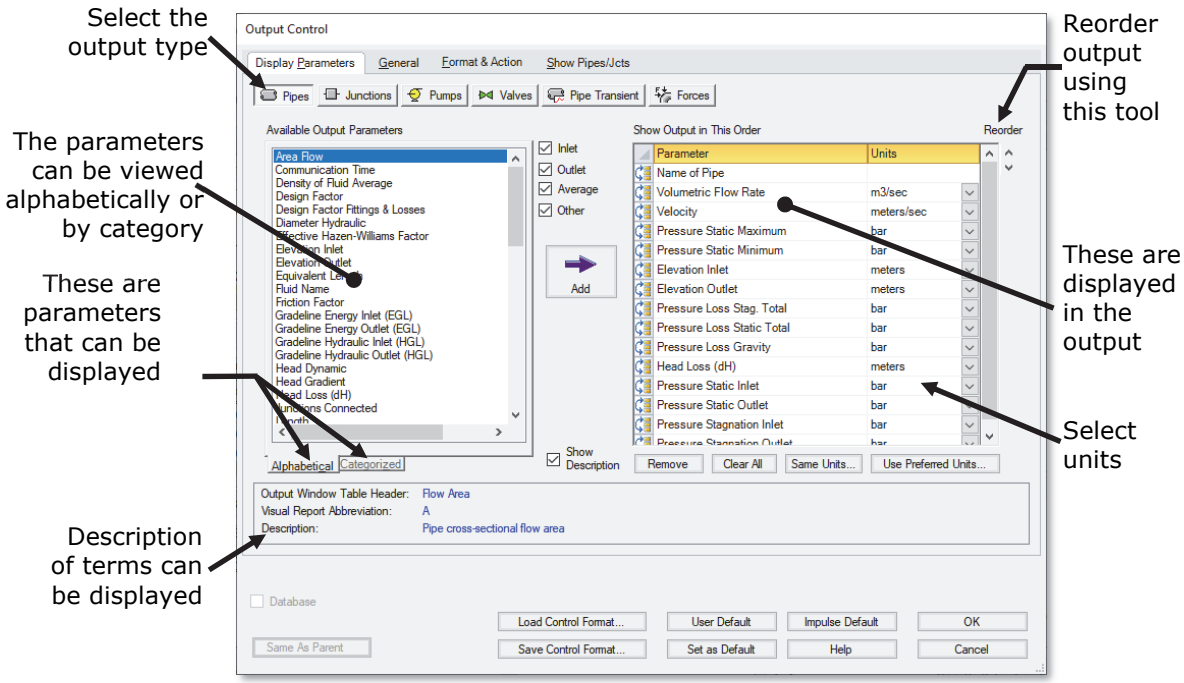


Figure 2.21 The Output Control window lets you customize the output

Units for each column in the Output section can also be changed by double-clicking the column header. This will open a window in which you can select the units again if you prefer (Figure 2.22). These changes are extended to the Output Control parameter data that is set.

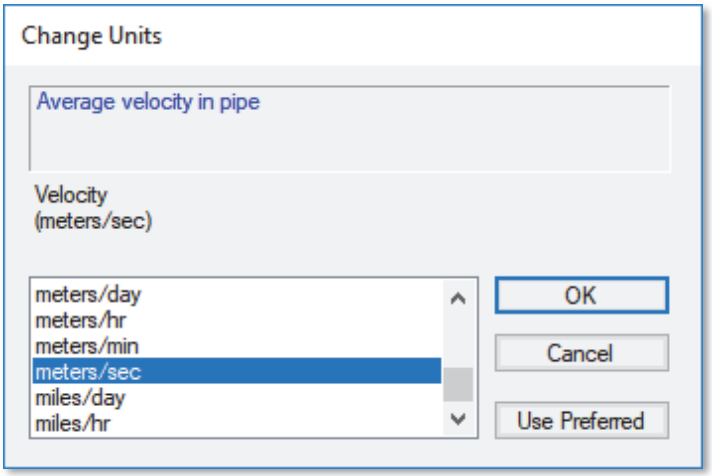


Figure 2.22 The Change Units window is opened from the Output window tables by double-clicking the column header

Step 8. View the graph results

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

AFT Impulse's Graph Guide feature, accessed by clicking on the "What Would You Like to Do?" button located at the top right of the Graph Area, provides assistance by guiding you through the creation of a "Quick and Simple" graph, or an "Advanced" graph (Figure 2.23). 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 remember that you can use it whenever you would like assistance in creating a graph.

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

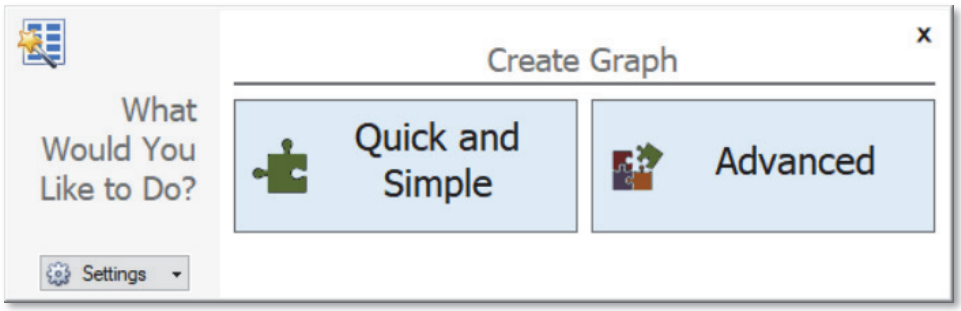


Figure 2.23 The Graph Guide can be toggled on and off by clicking the "What Would You Like to Do?" button on the Graph Results window

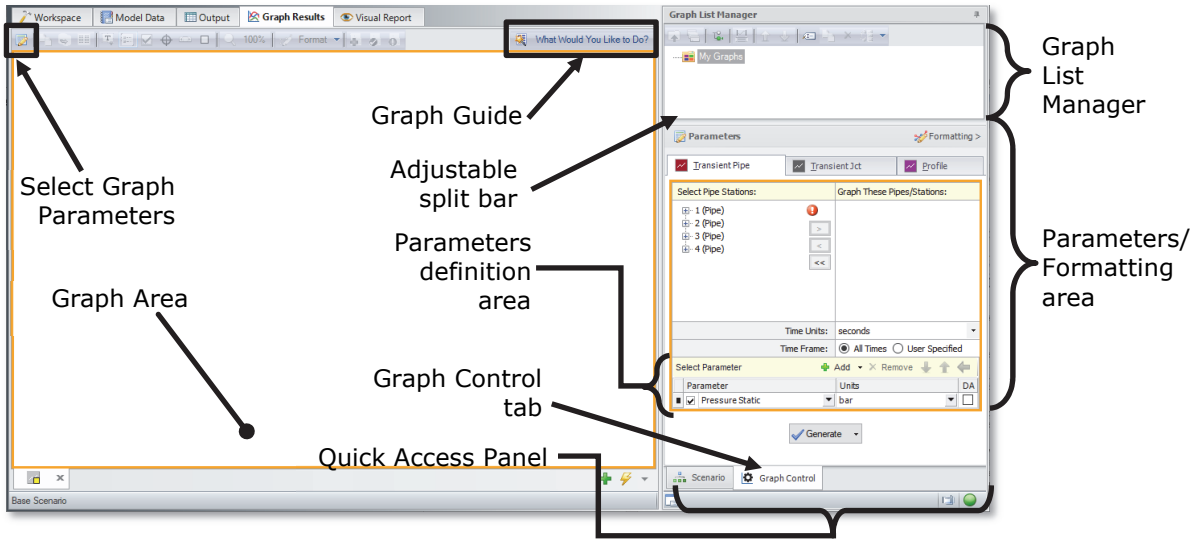


Figure 2.24 The Graph Results window and Quick Access Panel Graph Control tab is where various system parameters (both steady-state and transient) can be graphed

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

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

- **First, change to the Graph Results window** by clicking on the Graph Results tab, by clicking "Graph Results" from the Solution Progress window after running the model, or by pressing Ctrl+G on the keyboard. The Graph Results window offers full-featured Windows plot preparation.
- **Access the graphing parameters by opening the Graph Control tab on the Quick Access Panel (this tab is automatically opened 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 the "Select Pipe Stations" on the Transient Pipe tab, expand the P3 pipe stations and double-click on Outlet**, which is the pipe computing station at the valve inlet. Alternatively, you can click on the right arrow button after selecting the pipe station you want to graph to add it to the "Graph These Pipes/Stations" list. Also add the inlet of Pipe P4 which is the valve exit.
- **Select seconds for the Time Units**, and select "All Times" next to "Time Frame".
- **In the Parameters definition area, select "Pressure Static" and choose "barG" in the dropdown box under "Units"** to graph the static pressure in barG at the valve inlet and outlet over time.
- **In order to add the stacked graph over the static pressure graph showing the volumetric flowrate at the inlet and outlet of Valve J4, click the "Add" button**, which has the green "+" icon next to "Select Parameter". A new row under the Parameter definition area will appear (See Figure 2.25).
- **Choose "Volumetric Flowrate Upstream" in this new parameter row,** and select units of "m^3/hr".
- **Click Generate at the bottom of the Quick Access Panel** to create the static pressure and volumetric flowrate graphs at the inlet and outlet of the valve over the duration of the simulation.

- **To format the legend font size, right-click on each legend and use the scroll bar next** to decrease the font size to 9 for both graphs. Drag the legend to the upper right corner of the graph.
- **To format each axis font size, right click on each axis title and use the scroll bar** to decrease the size until the font size on each axis appears as you want it.

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

Figure 2.26 shows the stacked graphs detailing the static pressures and volumetric flowrates at the Valve J4 inlet and outlet over 2 seconds. Here you can see that the maximum pressure at the valve inlet is about 1.1 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.

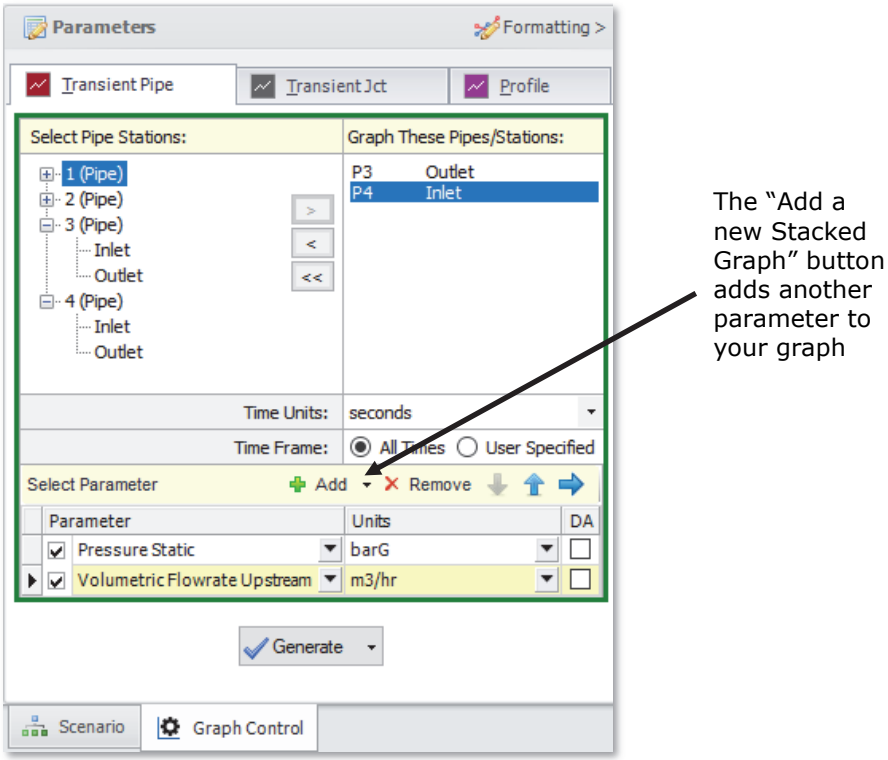


Figure 2.25 The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area

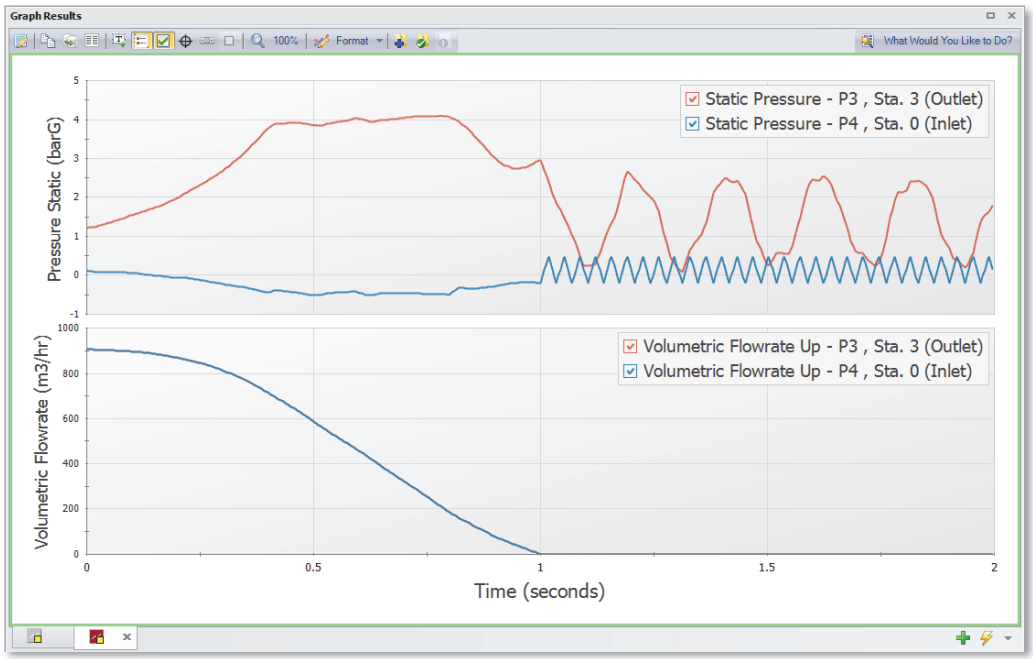


Figure 2.26 The Graph Results window offers full-featured plot generation. Here the static pressure and volumetric flow at the Valve J4 inlet and outlet are shown over 2 seconds.

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

Further review

Further review of the valve graph results in Figure 2.26 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 1 second, the valve closes entirely and the pipes upstream and downstream

of the valve are isolated from each other and will decay to the steady-state conditions which exist for a closed valve.

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

- **Create a new graph tab** by clicking the "New Tab" button which is the green plus icon located on the bottom right, immediately below the graph area (Figure 2.27)
- **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.
- **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^3/hr".
- **Click the Generate** button to display the graph. A graph of these stations shows the flowrates out of the two supply reservoirs. Note that after the valve closes, the fluid in Reservoir J1 flows to Reservoir J2 (Figure 2.27).

The "New Tab" button creates a new graph tab

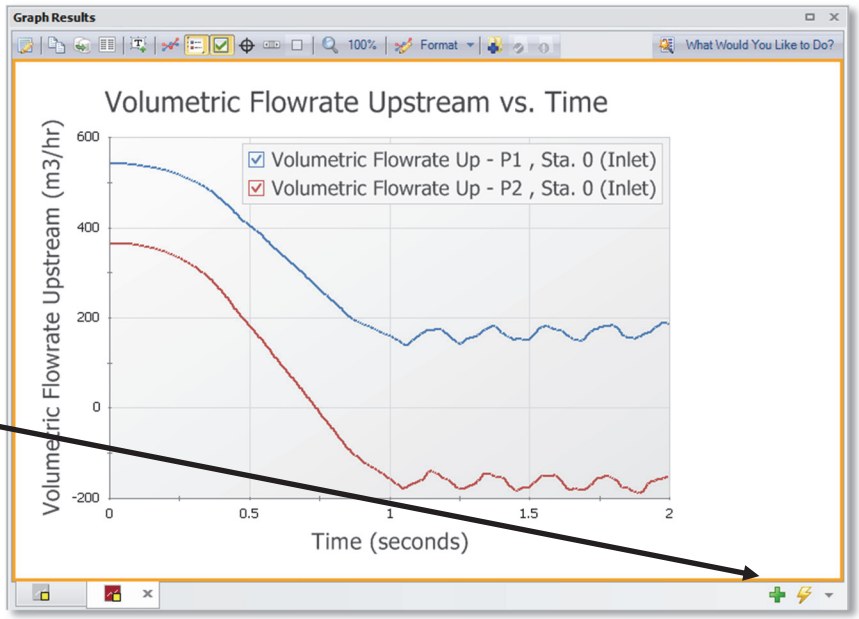


Figure 2.27 Transient flowrates out of the reservoirs

Note: AFT Impulse allows you to quickly generate several types of graphs by right-clicking on pipes or junctions on the Workspace after your model has output. To quickly generate the static pressure plot at the Valve J4 inlet and outlet, you can right-click on Valve J4 on the Workspace, then select "Create Transient Graph for connected pipes (Inlet/Outlet)", 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 9. View the Visual Report

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

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

- **Click the Visual Report Control button on the Visual Report Toolbar** (or Tools menu) and open the Visual Report Control window, shown in Figure 2.28. Default parameters are already selected, but you can modify these as desired. For now, select "Max Pressure Stagnation" and "Min Pressure Stagnation" in the Pipe Results area. Click the Show button. The Visual Report window graphic is generated (Figure 2.29).

  It is common for the text in the Visual Report window to overlap when first generated. You can change this by selecting smaller fonts or by dragging the text to a new area to increase clarity. You can also use the Visual Report Control window to display units in a legend to increase the clarity of the display. These adjustments have already been done in Figure 2.29. This window can be printed or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.

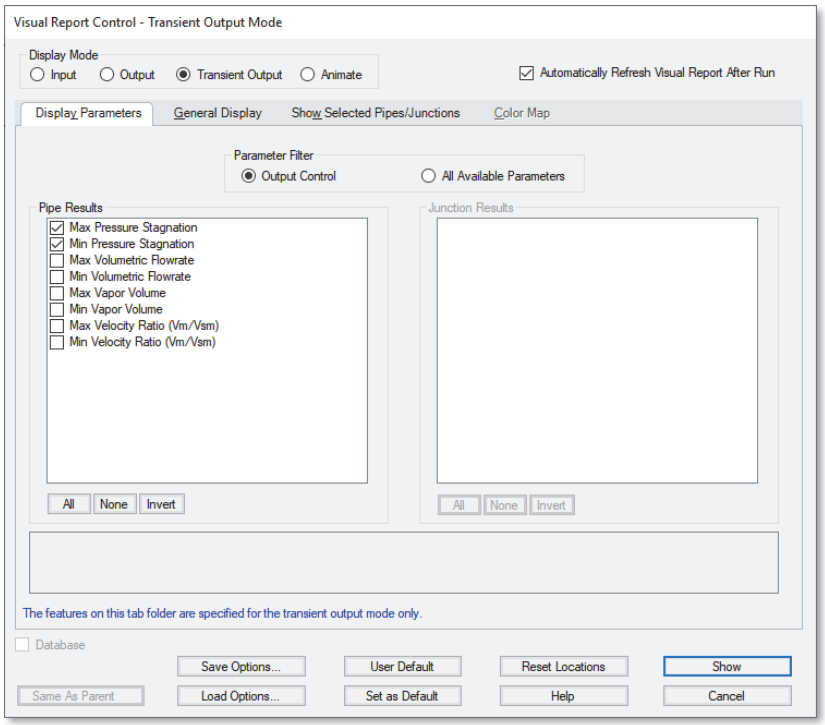


Figure 2.28 The Visual Report Control window selects content for the Visual Report window

The Visual Report window also provides the ability to animate the transient results as a qualitative tool to visualize the behavior in a network. This option can also be accessed from the Visual Report Control by selecting the Animate Display Mode at the top of the window (see Figure 2.28). When using the animation mode, text results cannot be displayed for the report. Instead, one parameter can be animated using absolute values or values relative to the steady state, or a static map of all maximum/minimum transient values can be generated. The animation can be recorded to a file. The model setup for animating static pressure can be seen in Figure 2.30.

Conclusion

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

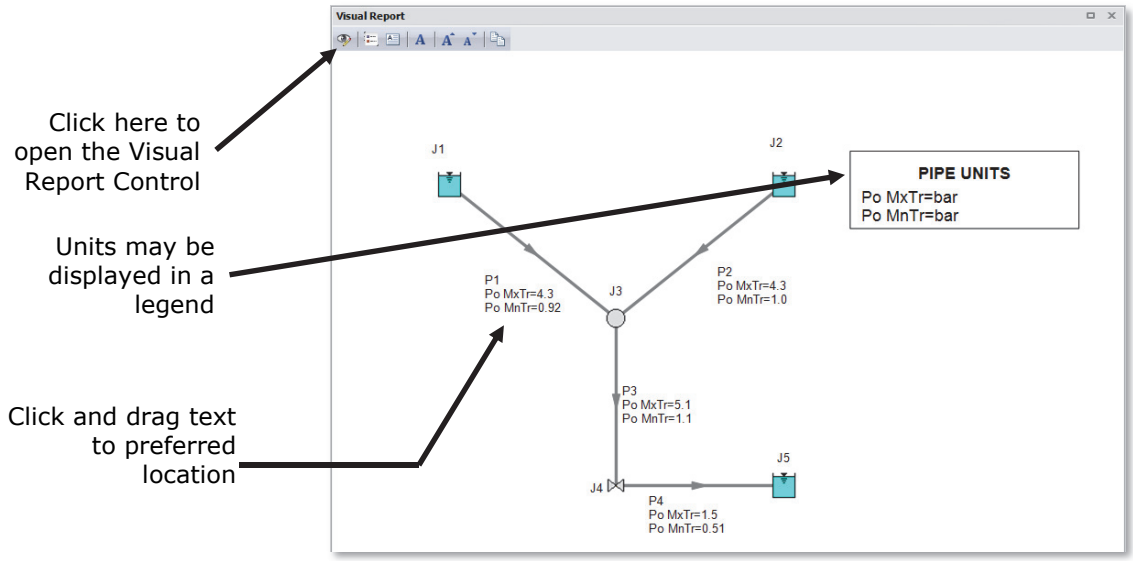


Figure 2.29 The Visual Report integrates results with model layout

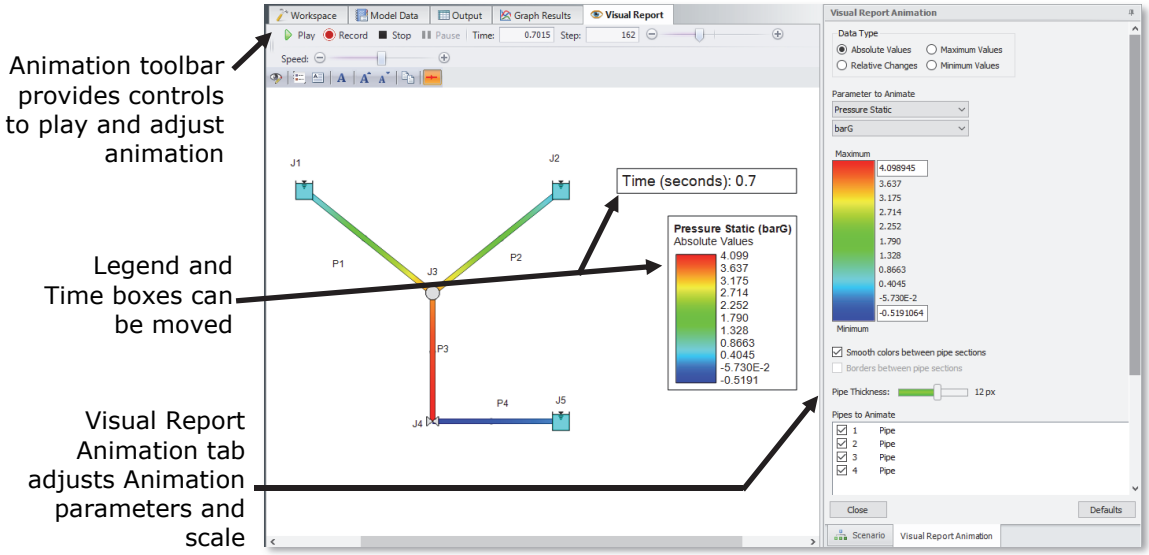


Figure 2.30 The Visual Report provides an animation feature as an alternate way to view the transient results

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

This example will cover the following topics:

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

Required knowledge

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

Model file

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

- *Pump Startup With Event Transient_Metric.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. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

Step 2. Specify system properties

1. Open the System Properties window by selecting “System Properties” in the Analysis menu.
2. On the Fluid Data Tab, select the AFT Standard database and then select "Water at 1 atm" in the “Fluids Available in Database” window.
3. Click "Add to Model" to select water for use in this model.
4. Specify 16 deg. C in the Temperature field and click "Calculate Properties". This calculates the fluid properties to use in the model.
5. Click OK

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

Step 3. Build the model

A. Place the pipes and junctions

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

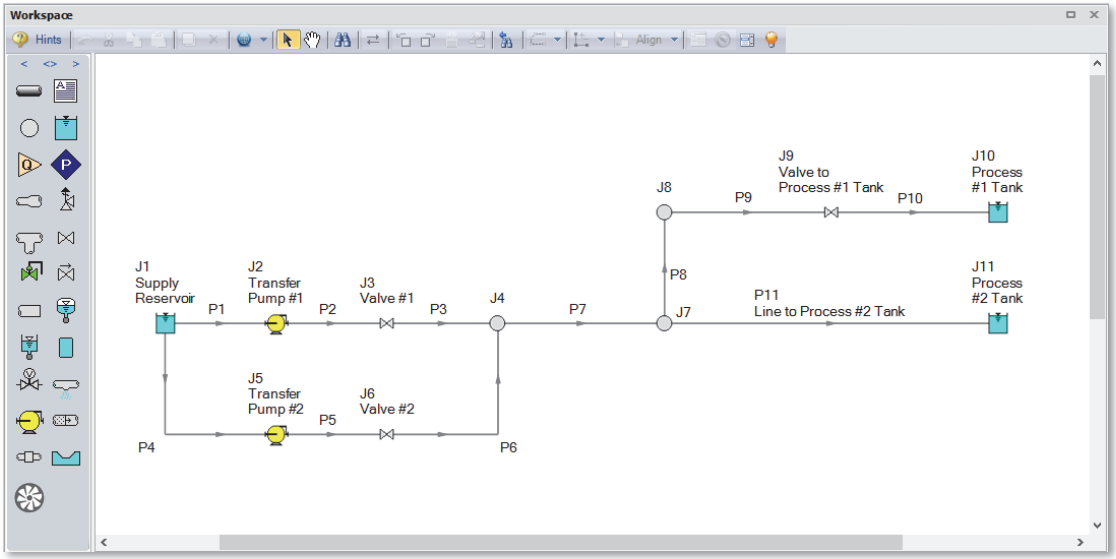


Figure 3.1 Layout of pipe system for pump startup example

B. Enter the pipe and junction data

The system is in place, but now the input data for the pipes and junctions need to be entered. Double-click each pipe and junction and enter the following data in each Pipe and Junction Properties window. Pipes P1-P11 are Steel – ANSI, STD, Thick-Walled Anchored Upstream, and the following data:

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

J1 - Reservoir

1. Name = Supply Reservoir
2. Liquid Surface Elevation = 6 meters
3. Liquid Surface Pressure = 0 barG (0 kPa (g))
4. Pipe Depth = 6 meters

J10 - Reservoir

1. Name = Process #1 Tank
2. Liquid Surface Elevation = 30 meters
3. Liquid Surface Pressure = 0 barG (0 kPa (g))
4. Pipe Depth = 3 meters

J11 - Reservoir

1. Name = Process #2 Tank
2. Liquid Surface Elevation = 3 meters
3. Surface pressure = 0 barG (0 kPa (g))
4. Pipe Depth = 3 meters

J4, J7 and J8 - Branch

Elevation = 0 meters

J2 and J5 - Pumps

1. J2 Name = Transfer Pump #1
2. J5 Name = Transfer Pump #2
3. Elevation = 0 meters
4. Pump Model = Centrifugal (Rotodynamic)
5. In the "Reverse Flow" area choose "Never", as the pump discharge valves are designed to prevent reverse flow through the pumps.
6. Click the "Enter Curve Da" tab and enter the following data:

Q (m^3/hr)	dH (m)
0	45.7
227	42.7
454	36.6

7. In the Curve Fitting area, click the "Generate Curve Fit Now" button. Then click the OK button
8. On the Transient tab, select "Speed vs. Time" under "Without Inertia" for the transient type
9. Enter the following data for pump speed to specify that the pumps will come up to speed in two seconds

Time (sec)	Speed (%)
0	0
2	100
10	100

10. On the Optional tab, select the Special Condition as "Pump Off With Flow Through". This 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

J3 and J6 - Valves

1. J3 Name = Valve #1
2. J6 Name = Valve #2
3. Elevation = 0 meters
4. Cv = 1000

5. On the Transient tab, enter the following data:

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

6. On the Optional tab, select the Special Condition as Closed. (When you close the valves, the adjacent pipes are displayed with dashed lines to denote that the pipes are closed)

J9 - Valve

This valve will use what is called an event transient. This means that the time zero in the Transient Data table is with respect to some condition occurring in the system. If the condition is never reached, the valve transient is never initiated. Here the valve should open when the pressure at Branch J7 is sufficient to cause the 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. Elevation = 0 meters
3. Cv = 500

On the Transient tab, enter the following data (Figure 3.2)

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

In the “Initiation of Transient” area select “Single Event”

4. Event Type = Pressure Stagnation at Pipe
5. Condition = Greater Than
6. Value = 3.10 barG
7. Pipe = 7, Outlet
8. On the Optional tab, select the Special Condition as Closed

Note: Multiple condition events for the transient can be defined by selecting Multi-Condition for the Event Type, then using the Multi-Condition Event Manager to group the events.

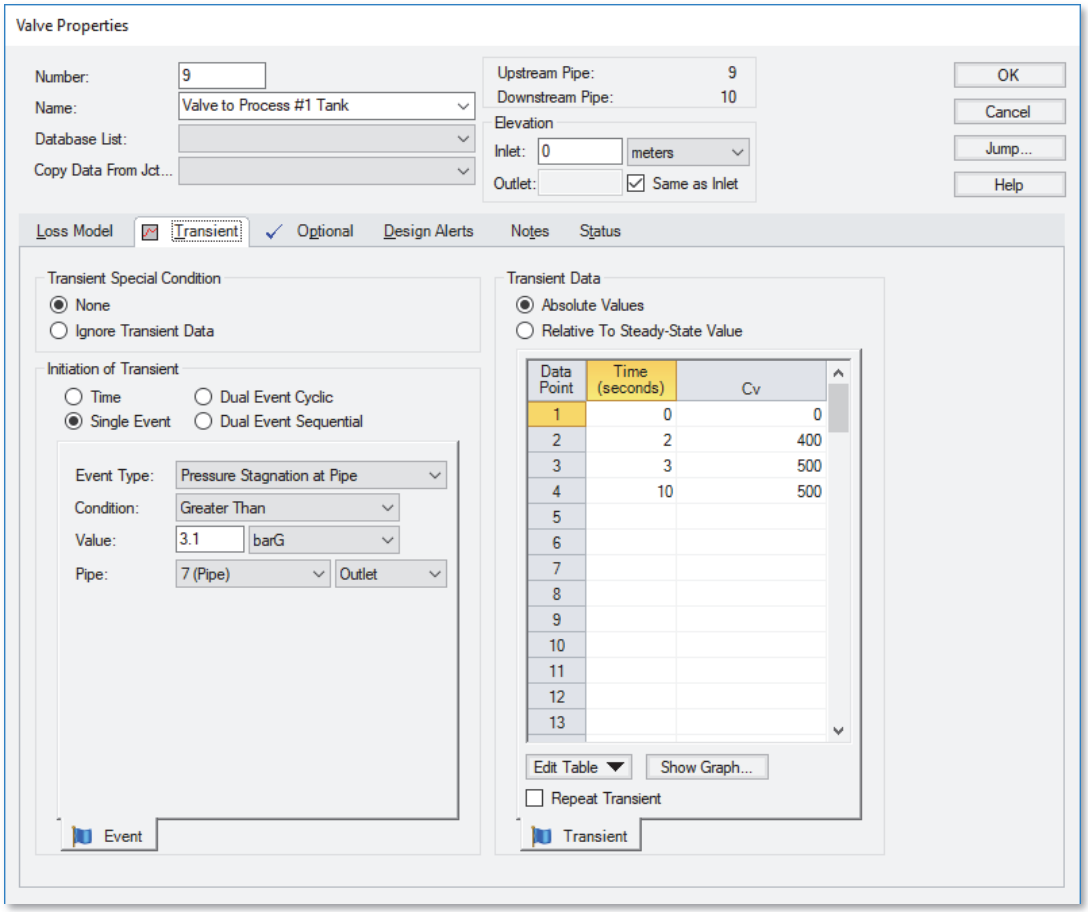


Figure 3.2 Specifying an event transient for Valve J9

C. Check if the pipe and junction data is complete

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

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

Step 4. Section the pipes

Open the Section Pipes window from the Analysis menu. When the Section Pipes window is first opened, it will automatically search for the best (least average variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Click OK to accept the displayed sectioning.

Step 5. Specify transient control

1. Open the Transient Control window from the Analysis menu
2. Enter the Start Time as 0 seconds and the Stop Time as 10 seconds on the Transient Solver Control tab
3. Clear the checkbox for Model Transient Cavitation on the Transient Solver Control tab (cavitation will be ignored until it becomes clear that it should be modeled)
4. To minimize run time and output file size, the table on the Pipe Station Output tab uses “Inlet and Outlet” for all pipes by default. This will cause the output for only the inlet and outlet station of each pipe to be saved to file. Data can be saved for all pipe stations or only selected stations. Click the All button to select all the pipes in the table. Drop down the list next to the “Change Selected Pipes To” button and select “All Stations”. Then click the “Change Selected Pipes To” button (Figure 3.3). This will save all pipe station data for all pipes, which will be useful later for animation purposes
5. Click the OK button

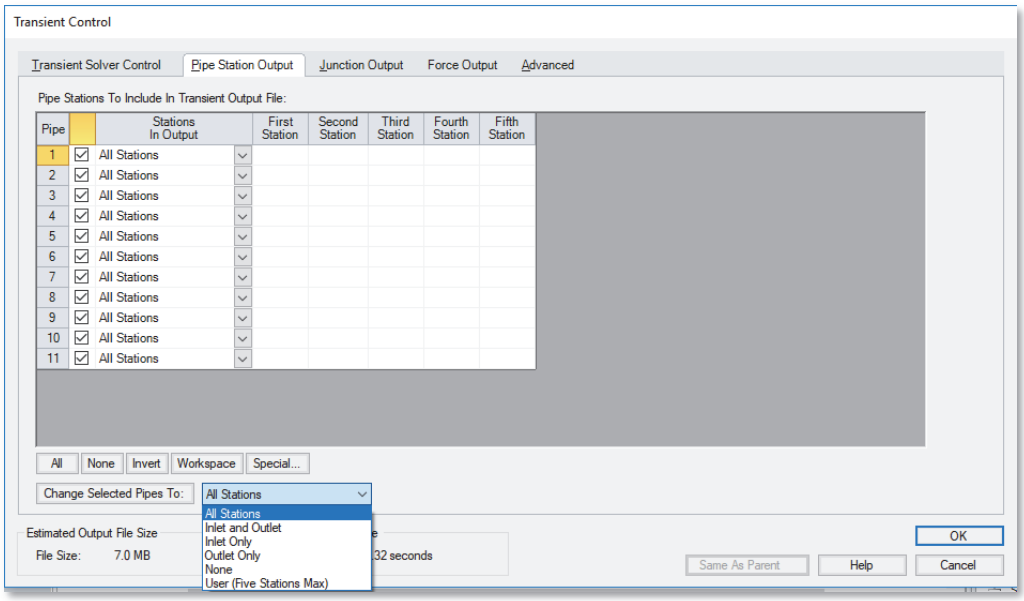


Figure 3.3 Transient Control window pipe station selection

Step 6. Create scenarios to model the three startup cases

A. Create scenarios

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.

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

- Create, name and organize scenarios
- Select the scenario to appear in the Workspace (the ‘current’ scenario)
- Delete, copy and rename scenarios
- Duplicate scenarios and save them as separate models

- Review the source of a scenario's properties
- Pass changes from a scenario to its variants

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

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

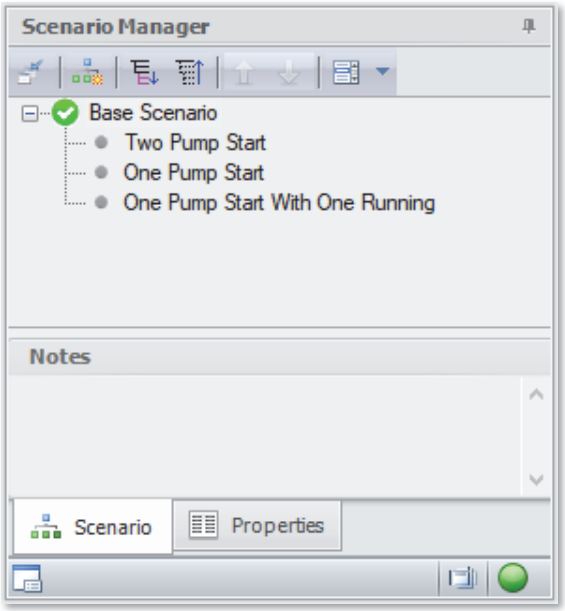


Figure 3.4 The Scenario Manager on the Quick Access Panel displays the scenario tree that allows you to create model variants (scenarios) and keep them organized within the same model file

B. Set up scenarios

Child scenarios "inherit" data from their ancestors. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent. Since the Base Scenario already has been setup with two pumps starting, the "Two Pump Start" child does not need to be modified.

One Pump Start

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

Here, only Pump J2 will be started while keeping Pump J5 off. Deleting the transient input data for Pump J5 and Valve J6 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 J5 Pump 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. Do the same with the J6 Valve. The required input for this scenario is now complete.

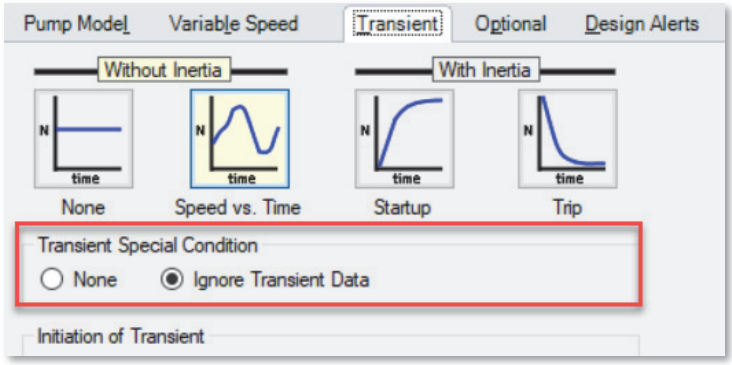


Figure 3.5 Transient Special Condition section on the Transient tab of the Pump Properties window.

One Pump Start With One Running

Double-click the "One Pump Start With One Running" scenario from the scenario tree in the Scenario Manager on the Quick Access Panel. This

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

Here, Pump J2 will be running at 100% speed during the steady-state and transient, and Pump J5 will be started.

- **Open the J2 Pump window** and select the Optional tab. Set the Special Condition to “None”. This specifies that during the steady-state the pump will be on and operate on its curve.
- **Select the Transient tab** and set the Transient Special Condition to “Ignore Transient Data”, since this pump will run at 100% speed during the transient. Alternatively, the speed data could be deleted.

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

The required input for this scenario is now complete.

Step 7. Run the first scenario

Double-click the “Two Pump Start” scenario from the scenario tree in the Scenario Manager on the Quick Access Panel. Select “Run Model” in the Analysis menu. After the model has finished running, view the results by pressing “Output” at the bottom of the Solution Progress window.

Note that several warnings in red appear in the Output stating that several pipes listed use constant friction factors, and that Variable Pipe Resistance is recommended. The variable pipe resistance can be turned on from the Transient Solver Control tab in the Transient Control window.

As background to this warning, traditional waterhammer modeling has ignored the effect of varying pipe resistance (i.e., friction) with flowrate. This is for several reasons. First, for highly turbulent flow (i.e., large Reynolds numbers) the pipe resistance does not depend on flowrate at all. On Moody charts this is sometimes referred to as the "fully rough zone". Second, for turbulent flows that are not in the fully rough zone, the dependence of pipe resistance is weak. Third, the waterhammer

phenomenon is not affected very much by relatively small changes in pipe resistance. And finally, introducing varying pipe resistance requires more steps in the calculation process.

Therefore, accounting for varying pipe resistance is not important in many waterhammer applications. One application where it is important is when the initial steady conditions have no flow in some pipes. For instance, when the transient being modeled is a pipe with a closed valve which is being opened. In such cases, it is difficult to obtain the steady-state pipe resistance because the flow is zero and thus no Reynolds number exists. Thus it is probably best to use variable resistance.

If you choose to not use variable resistance, the pipe frictional resistance obtained in the steady-state part of the simulation will be used as a constant in the transient part of the simulation. On the other hand, if you allow the resistance to vary, the pipe friction will differ for each pipe section because the flowrate (and hence Reynolds number) is varying along the pipe due to the waterhammer transient. Further, the resistance will vary with time as the flowrate changes.

If variable resistance is modeled, the required run time increases. Experience suggests the run time increases by about a factor of three.

For the three scenarios in this chapter, using variable pipe resistance does not significantly affect the results.

Step 8. Graph the results

Change to the Graph Results window by clicking on the Graph Results Primary Window tab, by clicking “Graph Results…” from the Solution Progress window after running the model, or by pressing Ctrl+G on the keyboard.

1. From the Graph Control tab on the Quick Access Panel, select the Profile tab
2. Next to Pipes, select “Plot Single Path”
3. 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 returning to the Graph Results tab and selecting “Workspace” to automatically choose all pipes currently selected on the Workspace

4. Choose “meters” for the Length Units
5. In the Parameters definition area, select “Pressure Static” and specify “barG” in the dropdown box under “Units”
6. 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)
7. Click the Generate button

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.

You can show the location of junctions within the pipeline path that you have plotted. Click the “Show Junction Locations” icon on the Graph Results Toolbar (see Figure 3.6).

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

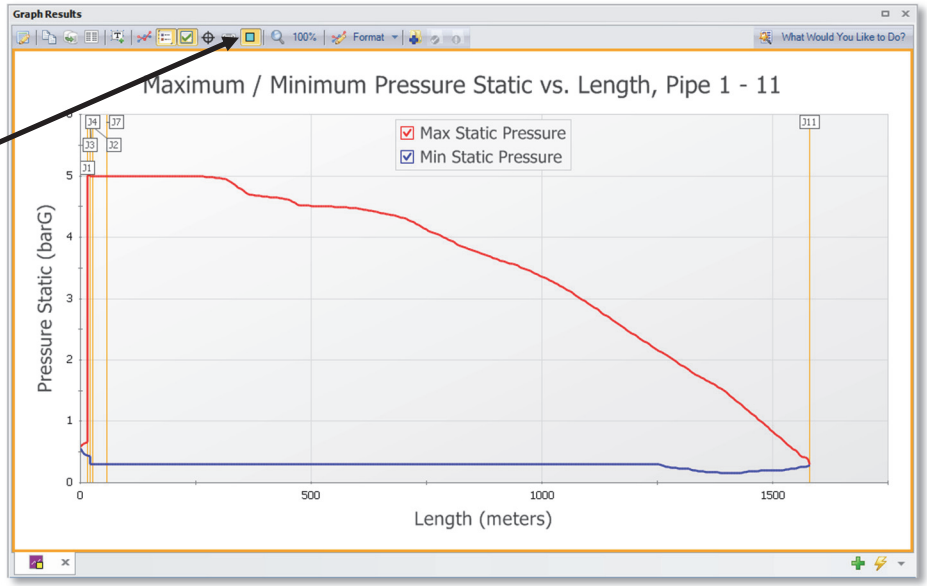


Figure 3.6 Profile of the maximum and minimum pressures through Pump #1 to Process #2 Tank for Two Pump Start scenario

To easily recreate this graph in other scenarios or in the future, it can be added to the Graph List. This can be done using the Graph List Manager on the Quick Access Panel. After generating the graph, click the “Add Graph To List” button on the Graph Results Toolbar (this icon is a blue

"+" sign on top of a file folder image). Give the graph a name like "Pressure Profile Pump #1 to Process #2 Tank". The graph will be added to the currently selected folder in the Graph List Manager. Note that graphs added to the Graph List Manager are not permanently saved until the model itself is saved again.

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

Since the maximum pressure is at the pump discharge, it is of interest here to plot the pressure vs. time. The volumetric flowrate vs. time at the pump discharge is also of interest. You will use AFT Impulse's dual Y-axis graphing abilities to plot both the pressure and volumetric 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 J5 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 "m^3/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.

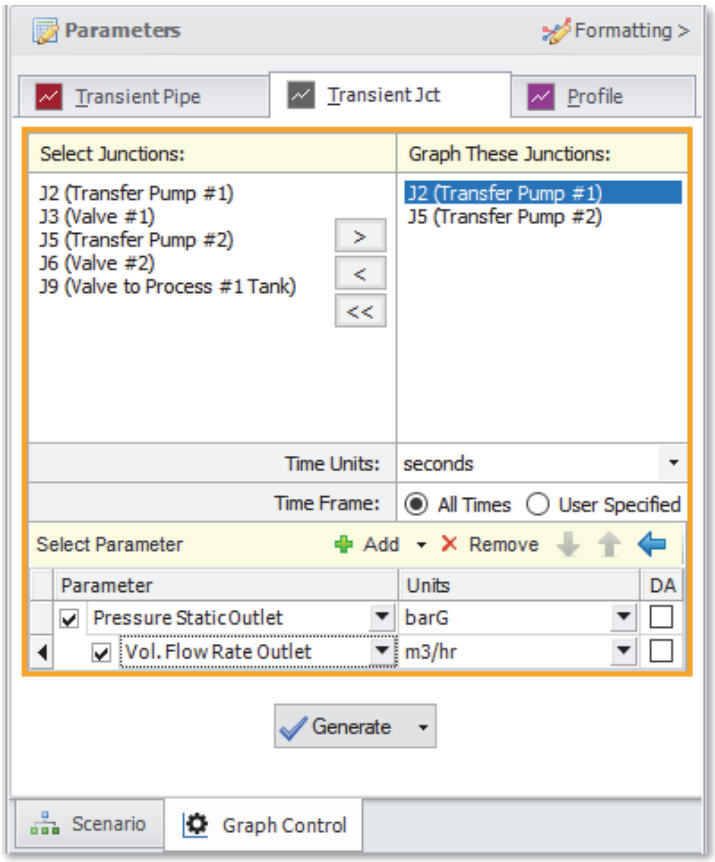


Figure 3.7 Parameter definitions for dual Y-axes graph of static pressure (primary Y-axis) and volumetric flowrate (secondary Y-axis) on the Graph Control tab on the Quick Access Panel

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

As would be expected, the pressure and volumetric flowrate transients are very similar for both pumps. The resulting graph is shown in Figure 3.8. Add this graph to the Graph List.

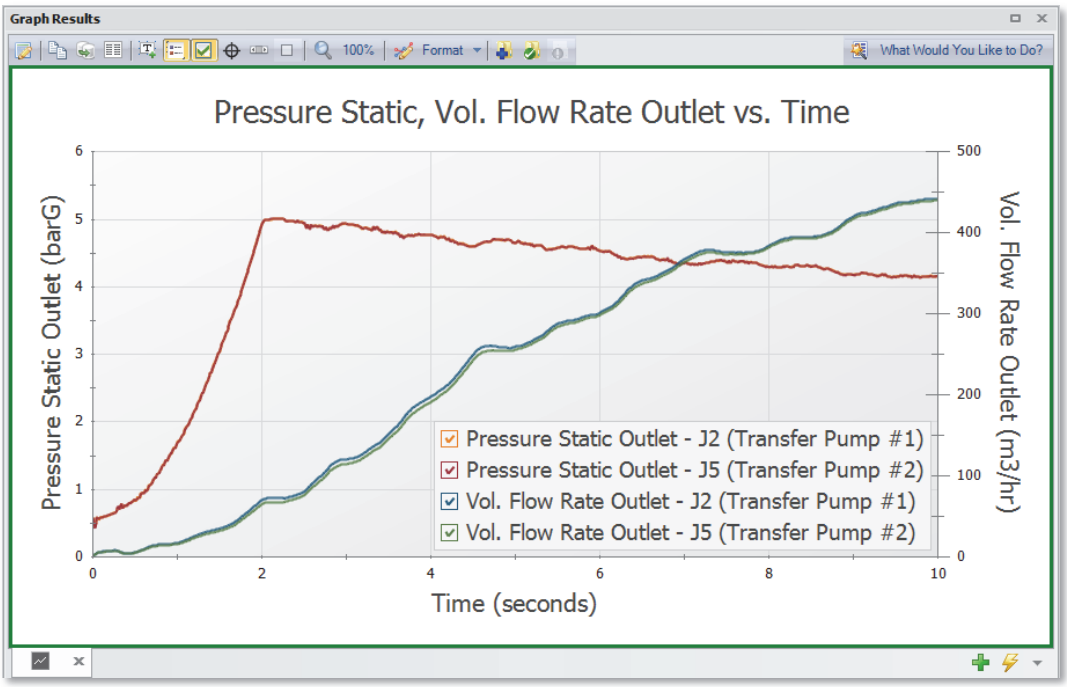


Figure 3.8 Pressure and volumetric flowrate transients at both pump discharge locations for Two Pump Start scenario

The flowrates into the two process tanks are of interest.

1. From the Transient Pipe tab on the Graph Control tab, click the double left arrow button between the “Select Pipe Stations” list and “Graph These Pipes/Stations” list to clear all selections from the “Graph These Pipes/Stations” list
2. Then, 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. Because you are only graphing the volumetric flowrate, ensure that any other parameters are removed from the Parameter definition area. If there are any other parameters here, be sure to select the parameter and then click the red “X” remove button
5. Choose units of m^3/hr
6. Click the Generate button

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

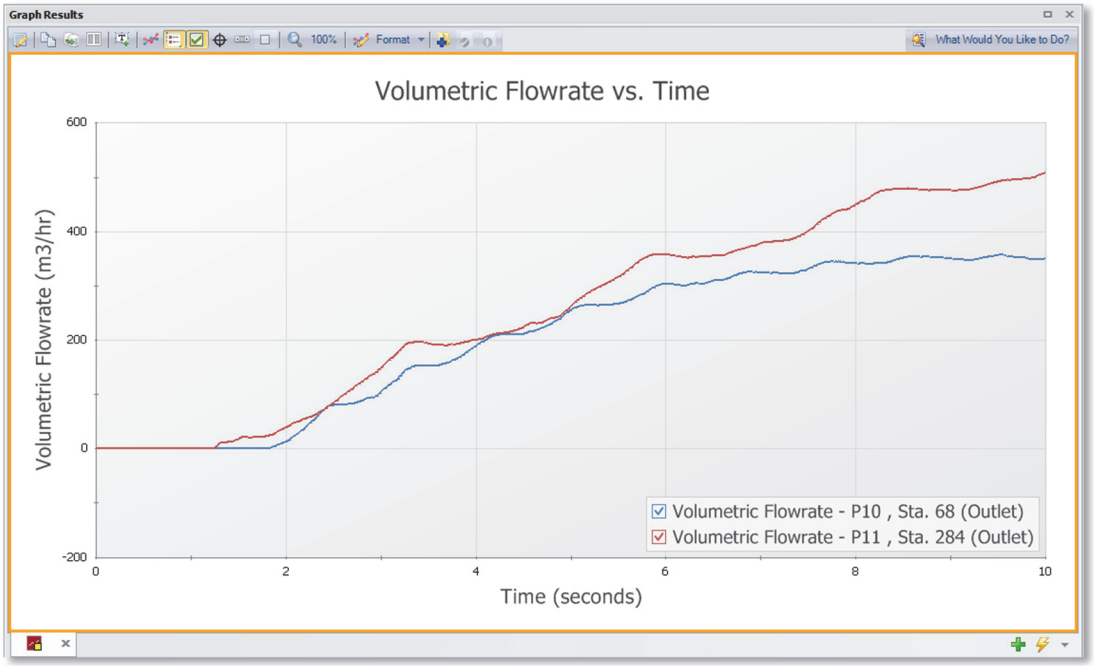


Figure 3.9 Flowrate transient at process tanks for Two Pump Start scenario

Step 9. Animate the results

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

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

1. Select the Profile tab under the Graph Control tab on the Quick Access Panel. Alternatively, you can access the Select Graph Parameters window by clicking on the “Select Graph Parameters” icon located at the top left of the Graph Results window.
2. Select pipes 1, 2, 3, 7 and 11

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

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

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

To use the “Animate Using: Output” option for animation, which is done here, all pipe stations need to be saved in Transient Control (Step 5: Specify transient control). In addition, it frequently is best to save all time step data as well. Both of these were selected previously. This animation option has the most flexibility to start, pause, and restart animation.

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

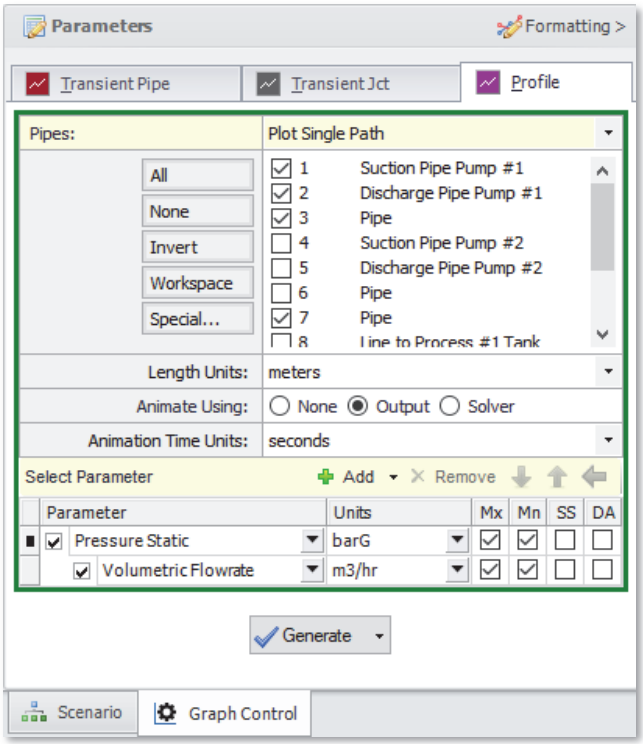


Figure 3.10 Selecting animation in Graph Results for Two Pump Start scenario

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

Step 10. Run the other scenarios and graph the results

Using Scenario Manager, load the other two scenarios and run them. Then create graphs similar to Figures 3.6-3.9. This will reveal that the maximum pressure for the “One Pump Start” scenario occurs at the pump discharge location, similar to the first scenario, but in the “One Pump Start With One Running” scenario the maximum pressure occurs in Pipe P11. Also, the pressure drops below atmospheric pressure, but does not reach the vapor pressure of water.

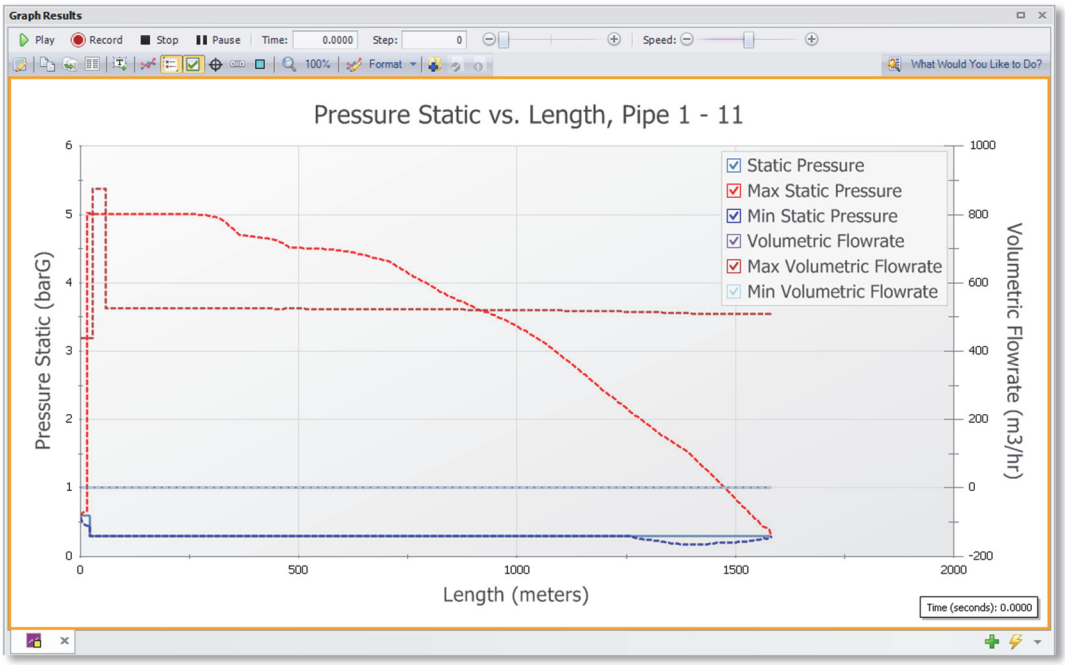


Figure 3.11 Animating output in Graph Results for Two Pump Start scenario

In the “One Pump Start With One Running” scenario, it should also be noted that there is a warning given for the pump. This is indicated by the red message in the status bar. The warning can be viewed in the top third section of the Output window on the Warnings tab, and states that there was reverse flow at pump junction J5. This warning message indicates that though reverse flow was not expected at pump J5, the calculations for the model found that reverse flow would occur at this point. It is recommended to adjust the model to better account for this reverse 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 closure profile to prevent reverse flow to the pump. This is only acceptable if the changes are physically accurate for the system. Otherwise, it will be necessary to choose an appropriate four quadrant data set for the pump to accurately predict the head at reverse flow conditions. This will be discussed further in the next chapter.

Conclusion

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

Table 3.1 Summary of maximum static pressure for the three cases

Case	Max. Static Pressure (barG)
Two Pumps Start	5.01
One Pump Start	4.92
One Pump Start With One On	5.01

CHAPTER 4

Pump Trip Example

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

Topics covered

This example will cover the following topics:

- How to specify a pump curve and power curve in the Pump Properties window
- How to use one of the transient pump models which accounts for pump inertia
- Use of 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 AFT Impulse such as placing junctions, connecting pipes, entering pipe and junction properties, and entering transient data. Refer to the Valve Closure Example in Chapter 2 for more information on these topics.

Model files

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

Pump Trip With Backflow_Metric.imp

Problem Statement

In this problem, two parallel pumps in a cooling tower 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 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 determined to be 12 seconds, in which case reverse rotation will occur at the pump.

Step 1. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu
2. On the Fluid Data Tab, select the AFT Standard database and then select "Water at 1 atm" in the “Fluids Available in Database” window
3. Click "Add to Model" to select water for use in this model
4. Specify 38 deg. C in the Temperature field
5. Click "Calculate Properties"

This calculates the fluid properties to use in the model.

Note: The Fluid Data can also be specified in the Startup window when Impulse is first opened.

Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to "Define All Pipes and Junctions". In the Workspace window, use the Toolbox on the left to drag and drop two reservoirs and a pump onto the Workspace. Then connect these with a pipe using the pipe drawing tool. The default flow direction will be from the first object you select to the second object. An arrow indicates the default flow direction. The Workspace should look like Figure 4.1.

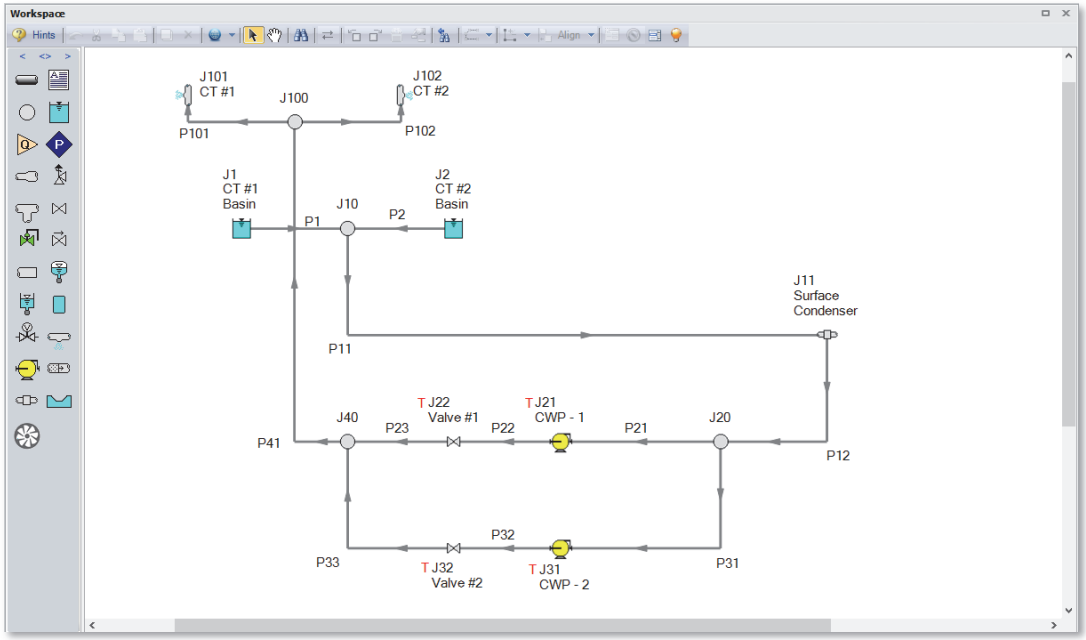


Figure 4.1 Workspace for pump trip example

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 or junction and enter the following data in the Pipe or Junction Properties window.

All pipes in the model will have Material = Steel – ANSI, standard roughness, and the pipe support as Thick-walled Anchored Upstream.

Pipe	Name	Length (meters)	Size (inches)	Type
1	Pipe	12	18	STD
2	Pipe	12	18	STD
11	Main Return	99	30	STD
12	Main Return	11	30	STD
21	CWP-1 Suction	3	20	STD(schedule 20)
22	CWP-1 Discharge	3	20	STD(schedule 20)
23	Pipe	3	20	STD(schedule 20)
31	CWP-2 Suction	3	20	STD(schedule 20)
32	CWP-2 Discharge	3	20	STD(schedule 20)
33	Pipe	3	20	STD(schedule 20)
41	Main Supply to CT	122	30	STD
101	Pipe	12	20	STD(schedule 20)
102	Pipe	12	20	STD(schedule 20)

Reservoir J1 and J2

1. Name = CT #1 Basin and CT #2 Basin
2. Tank Model = Infinite Reservoir
3. Liquid Surface Elevation = 12 meters
4. Liquid Surface Pressure = 0 barG (0 kPa(g))
5. Pipe depth = 1 meter

Branch J10

Elevation = 11 meters

Branch J20 and J40

Elevation = 0 meters

Branch J100

Elevation = 15 meters

General Component J11

1. Name = Surface Condenser
2. Elevation = 0 meters
3. Loss Model = Resistance Curve
4. Click Enter Curve Data button and create a curve fit for the following data:

Q (m^3/hr)	H (m)
0	0
4542	6.7
9085	26.8

Pumps J21 and J31

1. Name = CWP-1 and CWP-2
2. Elevation = 0 meters
3. Select Centrifugal (Rotodynamic) as the pump model, then click the Enter Curve Data Button and enter the following data:

Q (m^3/hr)	H (m)	Power (kW)
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
2725	18.3	156.5

4. In the Curve Fitting area select the All button then click the Generate Curve Fit now button. Then click the OK button.
6. In the Rated Pump Speed enter 1790 rpm.
7. In the "Reverse Flow" area choose "Possible".
8. Select Standard Pump Curve.
9. Check the box to Model Combined Check Valve and Pump, then set the Forward Velocity and Delta Head both as zero. (Figure 4.2)
10. On the Transient tab, select “Trip” under “With Inertia”
11. Under Total Rotating Inertia choose User Specified = 5.9 kg-m^2

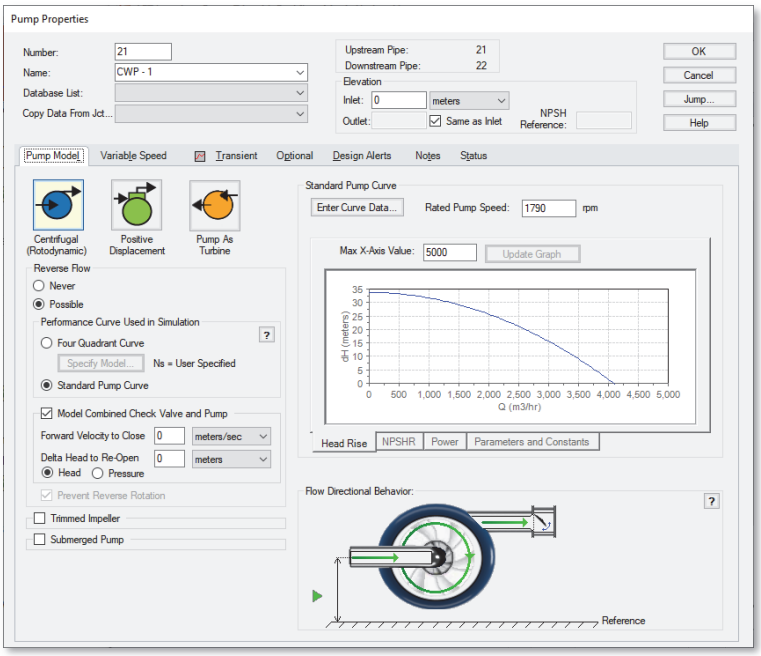


Figure 4.2 Pump Model tab for Pump J21. The internal check valve and flow behavior can be seen in the Flow Directional Behavior section.

12. Set the Initiation of Transient to be "Single Event", for a Time Absolute greater than or equal to zero.

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. In this case an internal check valve is required to prevent reverse flow, since there is not enough data in the standard curve to calculate the inertial pump trip if reverse flow did occur. We will come back later and compare these results to the Four Quadrant option.

Valves J22 and J32

1. J22 is named Valve #1
2. J32 is named Valve #2
3. Elevation = 0 meters
4. Cv = 10,000

5. On the Transient tab enter the following data.

Time (sec)	Cv
0	10000
3	1000
5	0
15	0

Spray Discharge J101 and J102

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

1. Name = CT #1 and CT #2
2. Elevation = 24 meters
3. Loss Model = Cd Spray (Discharge Coefficient)
4. Geometry = Spray Nozzle
5. Exit Pressure = 0 barG (0 kPa(g))
6. Cd = 0.6
7. Discharge Flow Area = 0.93 m^2

C. Check if the pipe and junction data is complete

Turn on the “Show Object Status” button from the View menu to verify if all data is entered. If so, the third Checklist item will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Section the pipes

Open the Section Pipes window from the Analysis menu. When the Section Pipes window is first opened, it will automatically search for the best (least average variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Click OK to accept the displayed sectioning.

Step 5. Specify transient control

Open the Transient Control window from the Analysis menu.

1. Enter the Stop Time as 15 seconds.
2. To minimize run time and output file size, select to Save Output to File Every 10 Time Steps.
3. On the Junction Output tab ensure the pump speed for J21 (CWP-1) is being saved by checking the box next to it.
4. Click the OK button.

Step 6. Run the model

The Checklist should now be complete and the model ready to run. Run the model by pressing the “Run Model” green arrow icon on the Workspace Toolbar, selecting “Run Model” in the Analysis menu, or pressing Ctrl+R on the keyboard. This will open the Solution Progress window. This window allows you to watch the progress of the Steady-State and Transient Solvers. When complete, press the “Output” button at the bottom of the window.

Step 7. Review results

Clicking “Output” from the Solution Progress window, selecting the Output tab, or pressing Ctrl+U on the keyboard will take you to the Output window, which will display any warnings (if they exist). There should not be any warnings here after running this model. You will also see several tabs in the top third of the window displaying the Event Messages in the model. This tab only appears in the case where a conditional transient has been activated, such as a check valve or relief valve.

Click on the Event Messages by Time tab. You should see a message stating that the internal check valve that we defined in pump J21 has closed. This tells us that reverse flow will occur in the pump in the actual system, but we prevented it with a check valve so that we could use the Standard Pump Curve rather than defining a Four Quadrant Data Set.

The Graph Results window will be useful in evaluating the impact of this check valve. We will graph the static pressure and volumetric flowrate at

the pump suction and valve discharge as stacked graphs to see how these parameters vary with time.

A. Graph the transient pressures at the pump

1. Change to the Graph Results window from the Window menu, by clicking on the Graph Results tab, or by pressing Ctrl+G.
2. Go to the Transient Pipe tab on the Graph Control tab and add Pipe P21 Outlet and Pipe P23 Inlet under “Graph These Pipes/Stations”. These locations represent the pump suction and valve discharge piping, respectively.
3. Select the “Time Units” as seconds and ensure that the “Time Frame” is showing “All Times”.
4. Add a second parameter to be graphed by clicking on the “Add” button, represented with the green “+” icon. Select “Pressure Static” as one parameter to be graphed and “Volumetric Flowrate Upstream” as the second parameter to be graphed.
5. Move the Volumetric Flowrate Upstream to the secondary Y-axis by clicking the “Make Secondary Y-axis” button, which is the blue, right arrow icon. This parameter should now be indented.
6. Set the units to “barG” for the static pressure and “m^3/hr” as the volumetric flowrate.
7. Click the Generate button. The resulting pressure and flow transients are shown in Figure 4.3.
8. Click the Add Graph to List button from the Toolbar to save this graph to the Graph List Manager so that it can be regenerated later, and name it “Pump Suction & Valve Outlet Pressures and Flows”.

In Figure 4.3 you can see that the pressure increases at the pump inlet and decreases at the valve outlet until a maximum pressure of 2.23 barG at the pump inlet and a minimum pressure of 1.08 barG at the valve outlet is reached. The pressures then begin to oscillate once the discharge valve becomes fully closed.

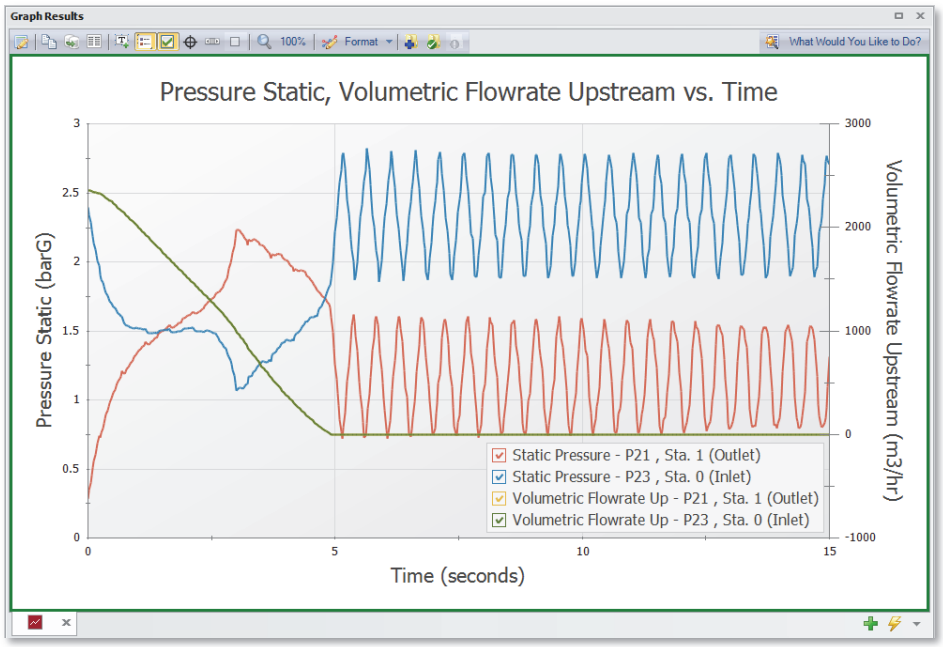


Figure 4.3 Pressure and volumetric flow rate transients using the standard pump curve data.

B. Graph the pump speed decay

You can graph certain junction properties for junctions in your model. To generate these graphs, the junction output data will need to be saved to file. You can specify which junctions you wish to save output data for by checking the box next to the junctions of interest in the Junction Output tab on the Transient Control window (Step 5: Specify transient control).

To graph the pump speed decay,

1. Return to the Graph Control tab on the Quick Access Panel.
2. On the Transient Junction tab, add Pump J21.
3. In the Parameter definition area, choose “Pump Speed” as the parameter to be graphed.
4. Choose “Percent” in the Units column.
5. Click the Generate button.

The predicted pump speed is then shown in Figure 4.4 Click the Add Graph to List button from the Toolbar to save this graph to the Graph

List Manager so that it can be regenerated later, and name it “CWP-1 Pump Speed Decay”.

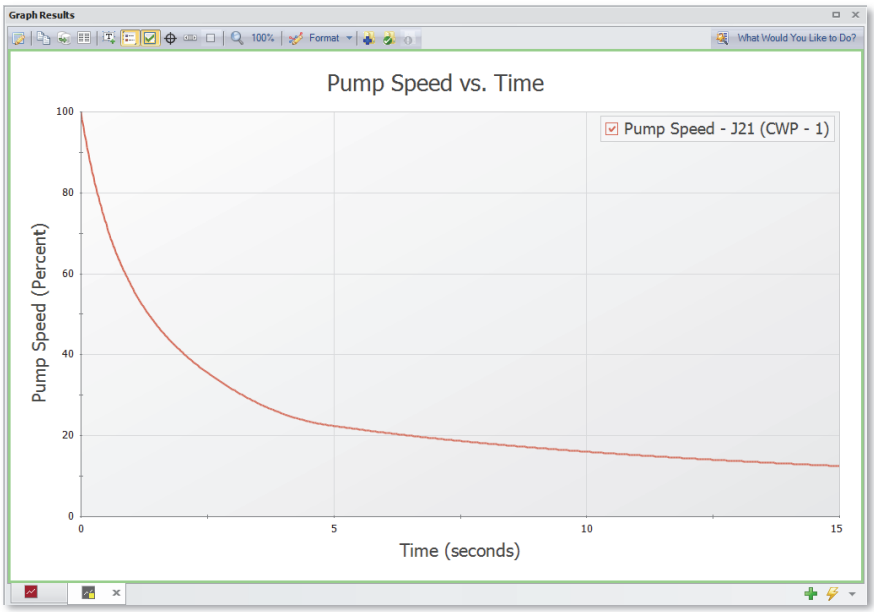


Figure 4.9 The pump speed decay is shown. The speed decays quickly at first, and then slows down after the check valve closes and the flow goes to zero

Step 8. Create additional pump scenarios

To check the validity of our results when using the standard pump curve with an internal check valve 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 children 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.

Four Quadrant BEP

Load the “Four Quadrant BEP” scenario by double-clicking the item in the Scenario Manager. Open the Pump Properties window for pump J21.

1. Select the option for “Four Quadrant Curve”. (Figure 4.5)
2. Uncheck the box to “Model Combined Check Valve and Pump” to disable the internal check valve.
3. Uncheck the box to “Prevent Reverse Rotation”.
4. Under the “Four Quadrant Curve” option click the “Specify Model…” button.
5. AFT Impulse provides multiple pre-defined Four Quadrant Data sets which are characterized by specific speed. To choose a data set to use, we will first estimate the specific speed for the pump, which AFT Impulse will use to recommend a data set. To do this, click “Suggest a Data Set” under “Four Quadrant Data Set”.

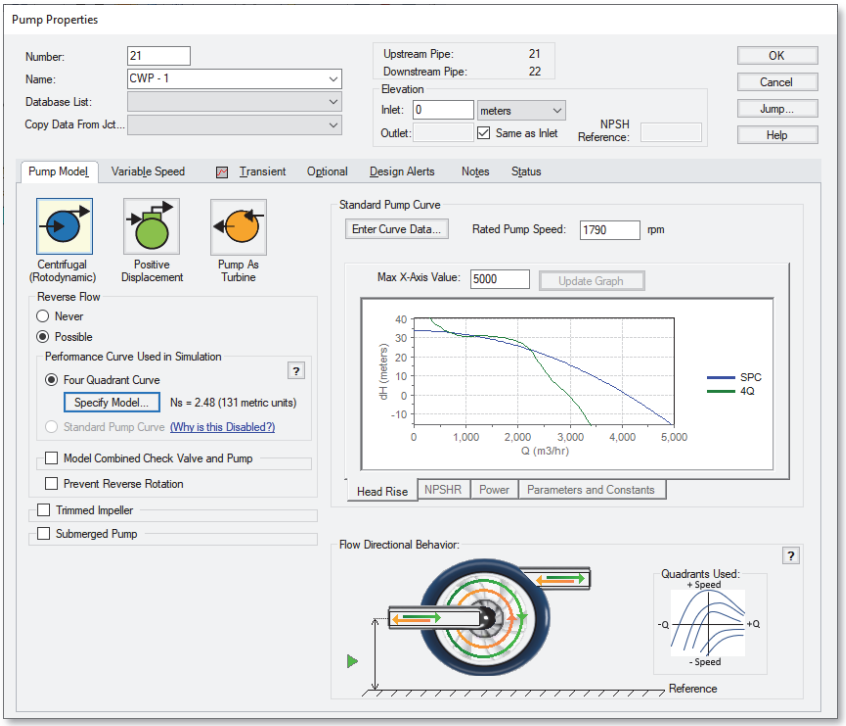


Figure 4.5 The Pump Model tab defined using the Four Quadrant Curve. Note that the Flow Directional Behavior section shows both forwards and reverse flow possibilities.

6. By Default AFT Impulse will select “Calculated from Standard Pump Curve”, which allows most of the data to be automatically populated from the Pump Properties window. In this case the pre-defined data is accurate for the pump, so click Calculate.

7. The specific speed in this case is estimated to be 2.50, and the suggested data set is therefore Ns = 2.48. Click “Select Suggested Data Set and Close”. The Ns = 2.48 data set will now be selected, and the dimensionalized curve will be displayed against the previously entered Standard Pump Curve. (Figure 4.6)

8. Make sure that the Four Quadrant Curve Dimensional Reference Point is set to use the Best Efficiency Point for this scenario. We will select the Steady State Operating Point option later for the third scenario. Click OK on this window, then OK on the Pump Properties window to accept the changes.

9. Repeat steps 1-8 for junction J31. The required input for this scenario is now complete.

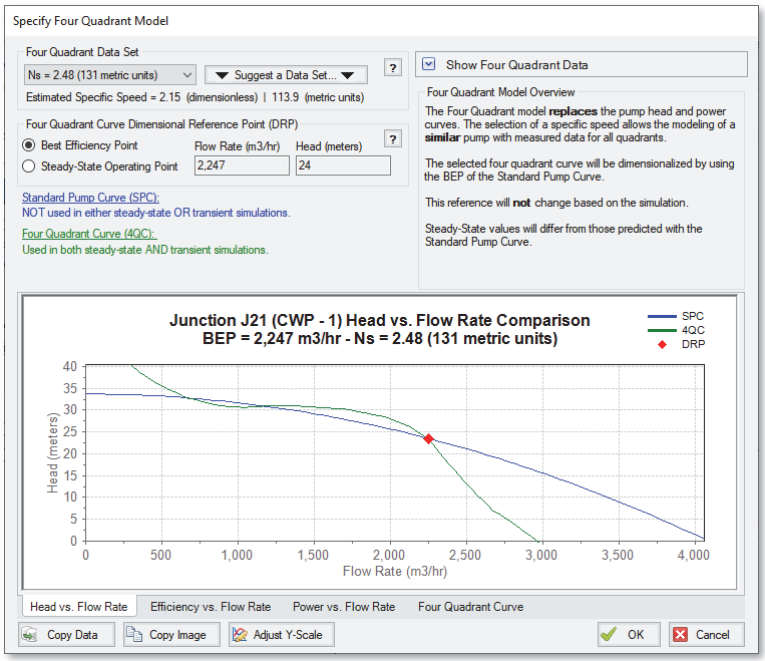


Figure 4.6 The Specify Four Quadrant Model window can be used to select and view options for the dimensionalized four quadrant data set.

Four Quadrant SSOP

Clone the “Four Quadrant BEP” scenario by right-clicking the scenario name in the Scenario Manager and choosing “Clone Without Children”. Name this cloned scenario “Four Quadrant SSOP”. For pump J21 and J31, open the Pump Properties window and click the Specify Model… button under Four Quadrant Curve. Change the Four Quadrant Curve Dimensional Reference Point to Steady State Operating point, then click Ok on all open windows to accept the changes. The required input for this scenario is now complete.

Step 9. Run the four quadrant scenarios

Using the Scenario Manager load each of the four quadrant scenarios and run them, then go to the Graph Results tab. For each scenario generate the “Pump Suction & Valve Outlet Pressures and Flows” graph by double clicking the graph name in the Graph List Manager on the Quick Access Panel. The results for each of the scenarios can be seen in Figures 4.7 and 4.8. In comparing the transient results for the two four quadrant scenarios, there are no major visible differences.

Now navigate to the Output window for each of the scenarios, then select the Pump Summary tab in the top third section. Notice that for the Four Quadrant SSOP and standard pump curve scenarios, the steady state flow rate is about 2362 m^3/hr, while the steady state flow rate for the BEP scenario is 2287 m^3/hr. This difference in the Four Quadrant BEP scenario results from the fact that the pump is not operating exactly at BEP, as is the case for most pumps.

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

Comparison between the four quadrant graphs (Figure 4.7 & 4.8) and standard pump curve graph (Figure 4.3) reveals that while the results are similar, 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. These differences can be attributed to the small flow oscillations that are neglected in the standard pump curve scenario due to the internal check valve.

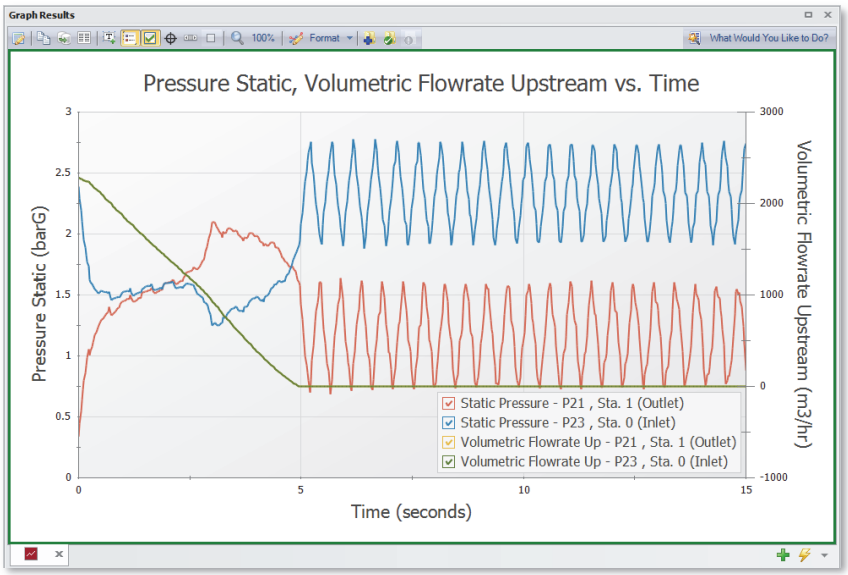


Figure 4.7 **Pressure and volumetric flow rate transients using a Four Quadrant Curve dimensionalized at the BEP.**

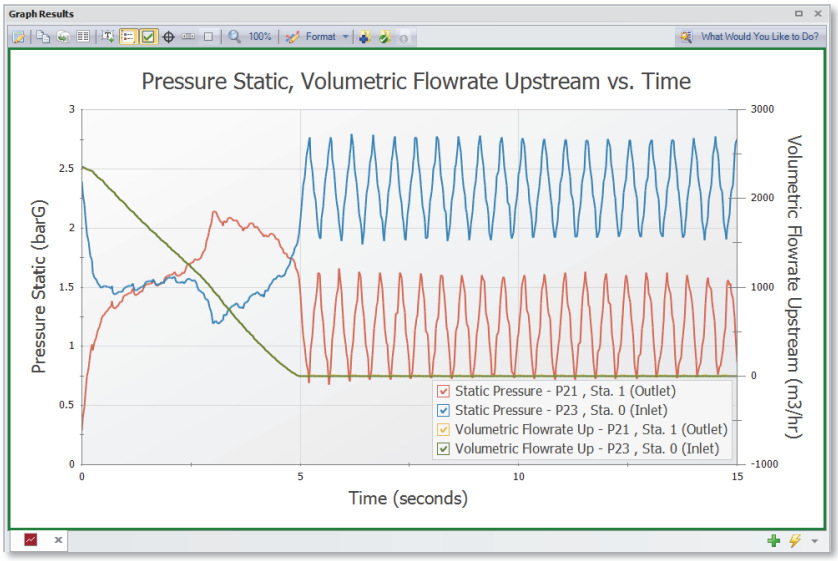


Figure 4.8 **Pressure and volumetric flow rate transients using a Four Quadrant Curve dimensionalized at the steady state operating point.**

Step 10. Adjust the valve closure

Now let's consider the case where the valves at the discharge of the pumps cannot be closed in 5 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 J21 and J31 will already inherit the four quadrant model from the Four Quadrant BEP scenario, but we will need to adjust the valves. For Valves J22 and J32 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 (sec)	Cv
0	10000
5	1000
12	0
30	0

Step 11. Specify transient control

Since the valve closure takes a longer time, it will be useful to extend the run time. Open the Transient Control window from the Analysis menu and change the stop time to 30 seconds, then click OK.

Step 12. Run the model

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

Step 13. Graph results

Load the “Pump Suction & Valve Outlet Pressures and Flows” graph from the Graph List Manager, as seen in Figure 4.9. Note that as expected, a significant amount of reverse flow occurs at the pump from about 6-12 seconds, requiring data beyond the standard data which is typically used for forward flow with forward rotation at the pump. The resulting transient results show that the maximum pressures at the valve outlet and the pump suction have been reduced.

The pump speed decay is shown in Figure 4.10. Note how the pump speed decreases at a faster rate than Figure 4.4. Additionally, at about 10 seconds the pump begins to experience reverse rotation. This means that from 6 seconds to 10 seconds, the pump will have forward rotation with reverse flow, then from 10-12 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.

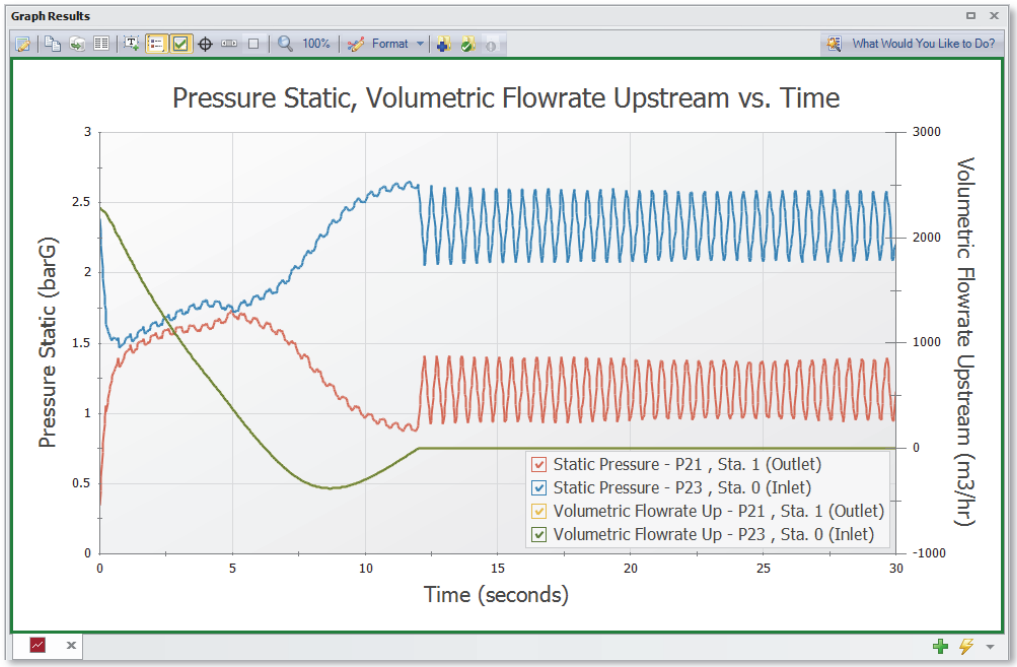


Figure 4.9 Predicted transient pump suction and discharge pressures and volumetric flowrates with a 12 second valve closure rate

Conclusion

For the valve closure of 5 seconds a minimal amount of reverse flow was experienced at the pump. As a result, the standard pump curve with an internal check valve 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, but to have a 80 m^3/hr difference in the steady state operating point. 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.

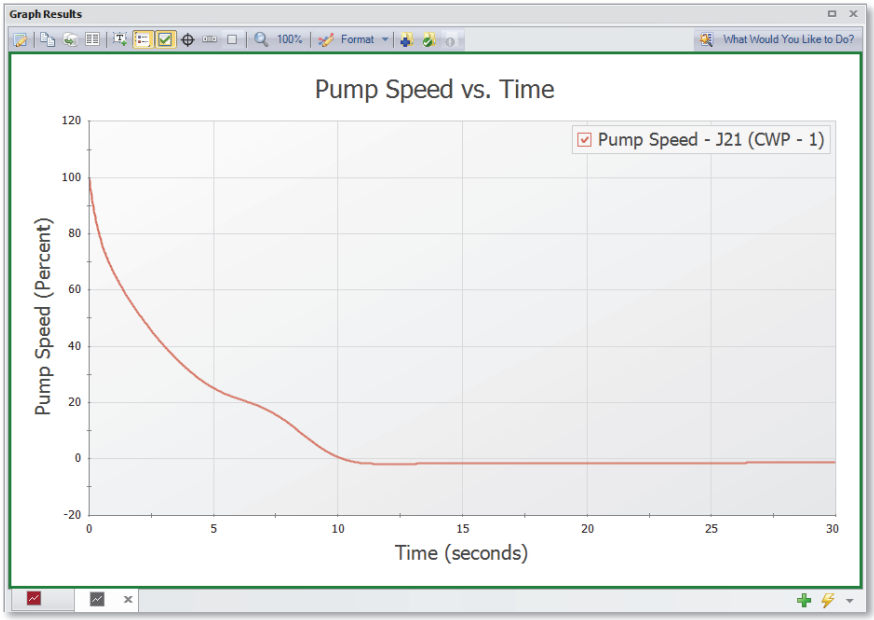


Figure 4.10 Pump Speed decay when the valves are modified to close fully at 12 seconds

CHAPTER 5

Valve Closure With Force Sets Example

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

Topics covered

This example will cover the following topics:

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

Required knowledge

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

Model Files

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

Valve Closure With Pipe Forces_Metric – Quick Start.imp

Step 1. Start AFT Impulse

From the Start Menu choose AFT Products and AFT Impulse.

Step 2. Open model file

For this example, we will be starting from a pre-built model file which has the first three checklist items 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.

Isometric Drawing Mode

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

For future reference when using this feature, the Isometric drawing mode can be turned on in the Arrange menu under Pipe Drawing Mode. The isometric grid can also be turned on or off from the Arrange menu. When drawing segmented pipes such as P5, a red-dashed preview line will show how the pipe will be drawn on the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse. You can hold the "Alt" key while adjusting a pipe by the endpoint to add an additional segment. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

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

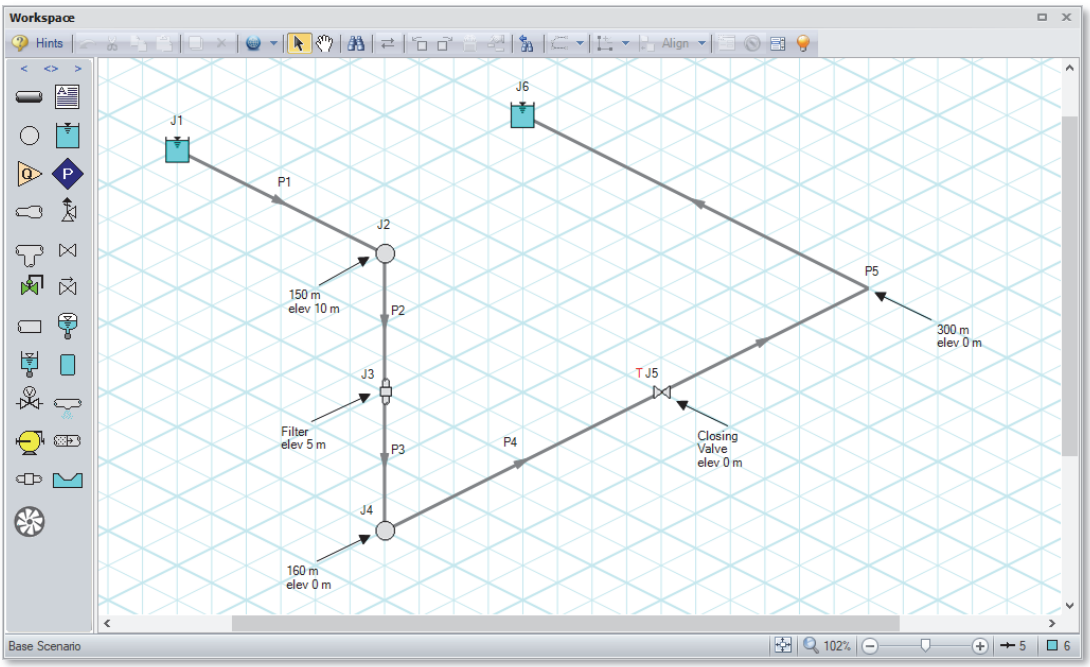


Figure 5.1 Layout of gravity drain system with isometric grid on.

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

Step 3. Section the pipes

- **Open the Section Pipes window** from the Analysis menu. When the Section Pipes window is first opened it will automatically search for the best (least average variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Note that the default option under "Search Results" shows the number of sections in the controlling pipe is 1.

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

➢ **Click OK** to accept the results of using one section in the controlling pipe.

Step 4. Specify transient control

➢ **Open the Transient Control window** from the Analysis menu and enter a Stop Time of 10 seconds.

➢ **Select the Force Output tab** and define a force set by clicking New, then entering the data as shown in Figure 5.2. Repeat this process for the second force set.

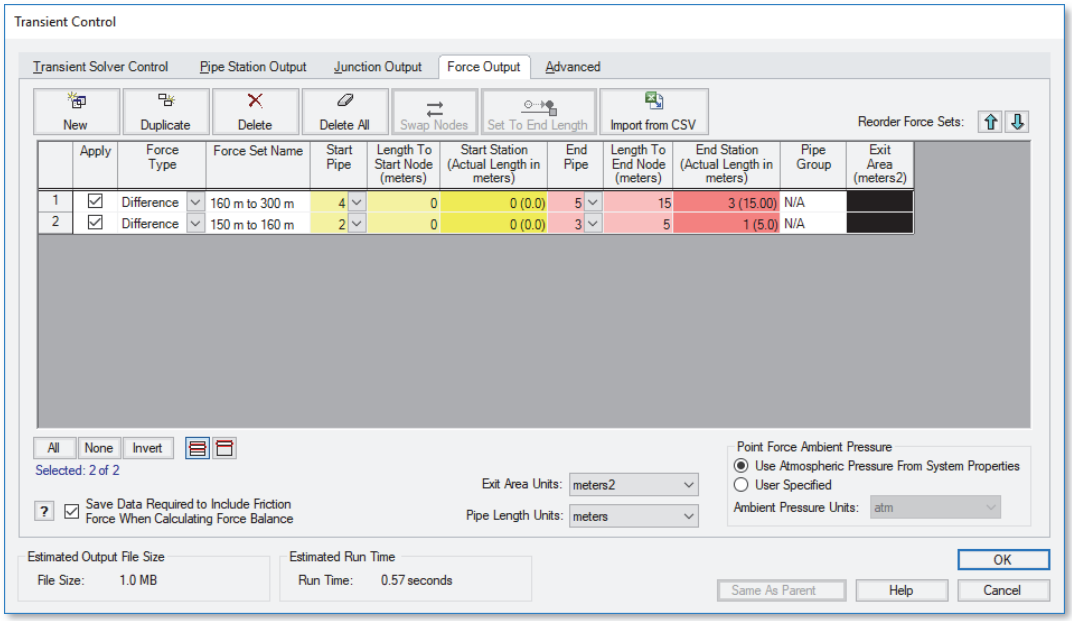


Figure 5.2 Force sets defined in Transient Control.

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

across more than two pipes, or for cases where a user defined exit area would be useful, such as for forces across a nozzle.

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

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

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

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

- **Click the OK button** to close Transient Control.

Step 5. Run the model

- **Select Run Model** in the Analysis menu. When completed the Solution Progress window will indicate that the run has completed. View the results in the Output window by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Graph pipe forces

- **Select the Forces tab** from the Graph Parameters section on the Quick Access Panel. The force sets defined in Transient Control will appear as available forces to graph.

  Check the box next to the "150 m to 160 m" force set. This corresponds to the elevation change between the branch junctions of 10 meters and 0 meters specified from pipes P2 and P3. Also check the box next to the

"160 m to 300 m" force set, which corresponds to the force over the closing valve. Click Generate and the following force vs time graph is displayed (Figure 5.3).

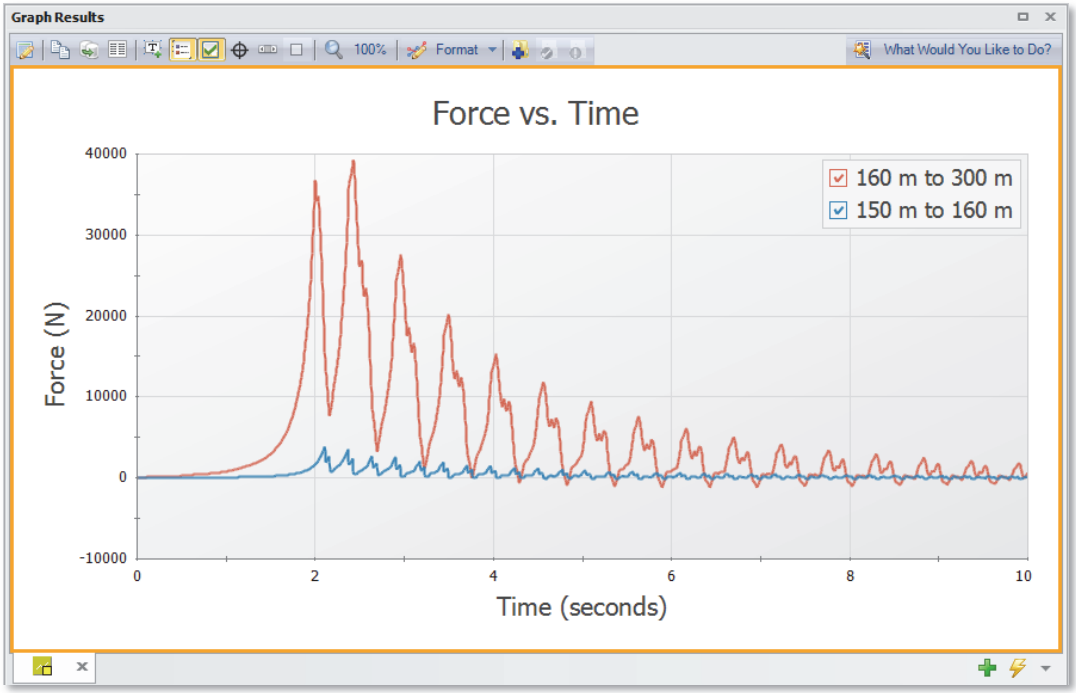


Figure 5.3 Unbalanced Force vs. Time for force set "150 m to 160 m" with friction.

Note that at time 0, which represents the initial, steady state results, there is no force imbalance. This is the expected results for a system at steady state.

Some traditional methods of analyzing force sets will not have this same result, since they do not include the effects of friction or momentum in their force balances. If the graph from Figure 5.3 is created with friction and momentum ignored, the steady state values are actually calculated as approximately -4790 N and -11940 N. Due to this effect, AFT Impulse will always include both friction and momentum by default when graphing force sets.

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

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

Final notes

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

Analysis summary

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

CHAPTER 6

AFT Impulse Add-on Modules Examples

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

Topics covered

These examples will cover the following topics:

- Defining settling slurry properties
- Reviewing slurry output
- "Ringing" systems to find natural acoustic frequencies
- Finding system excitation frequencies
- Calculating pressure response of system at ‘worst case’ PD speeds

Required knowledge

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

Model files

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

- *Sand Transfer to Two Locations – SSL_Metric.imp*
- *PD Pulsation Study – PFA_Metric.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 (Vsm) must be kept above 1.2 to avoid potentially plugging the pipe. Determine if the pipe size and velocity ratio is acceptable during the transient where one valve starts to close while the other opens further.

SSL Step 1. Start AFT Impulse

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

SSL Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu.
2. Select Basic Water Slurry Input at the upper right (this should be the default selection). See Figure 6.1
3. Select “Detailed” under “Slurry Calculation Method”.
4. In the Solids Specifications area select the “User Specified Solids Added” button.
5. Select the calculated *M* and *Vt/Vts* factor of 0.55.

6. Enter a solids density of 2.65 S.G. water, a *d50* of 0.07 cm, a *d85* of 0.1 cm, and a Bulk Modulus of 16,000 MPa.
7. Enter a temperature for water of 21 deg. C.
8. Set the Concentration Type to “Volume Fraction”, and for the Amount Solids Added field, specify 20%.
9. Select the Settling Slurry – Wilson, Addie, Clift model.
10. Select OK.

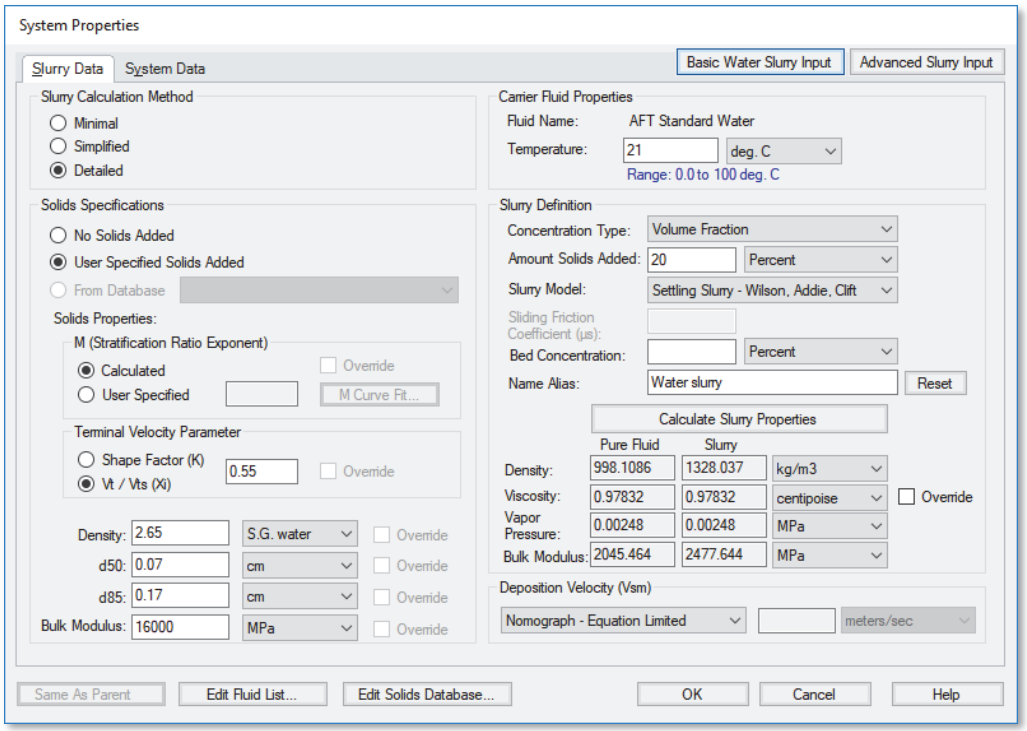


Figure 6.1 Data entry for sand slurry properties

SSL Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to “Define All Pipes and Junctions”. In the Workspace window, assemble the model as shown in Figure 6.2.

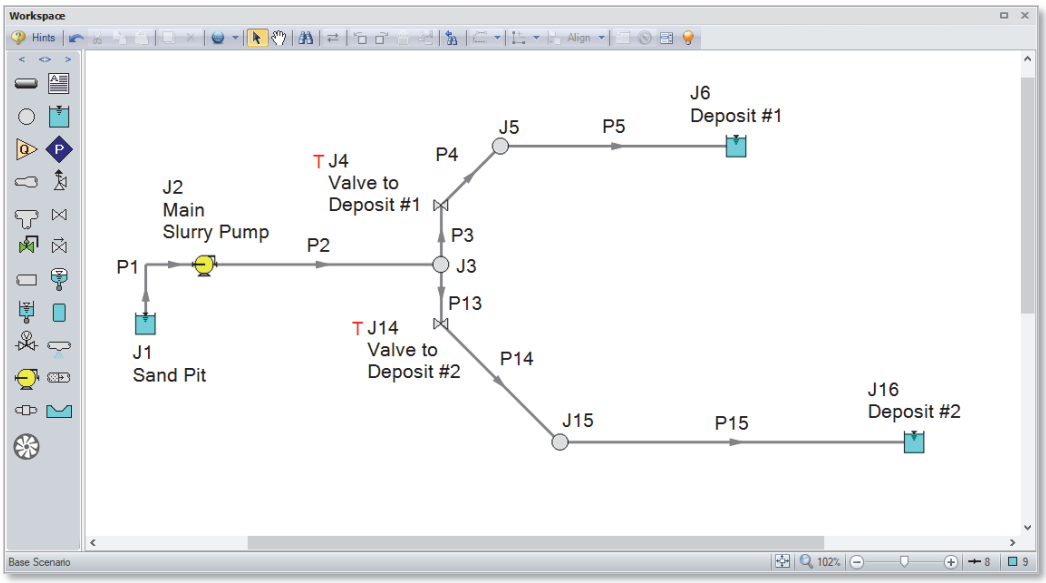


Figure 6.2 Layout of pipe system for Sand Transfer to Two Locations example

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions.

All pipes are Steel - ANSI, with standard roughness, thick-walled anchored upstream and the following data:

Pipe	Length (meters)	Size	Type
P1	3	10 inch	Schedule 40
P2	122	10 inch	Schedule 40
P3	1.5	10 inch	Schedule 40
P4	26	10 inch	Schedule 40
P5	122	10 inch	Schedule 40
P13	1.5	10 inch	Schedule 40
P14	30.5	10 inch	Schedule 40
P15	213	10 inch	Schedule 40

C. Enter the junction data

J1 - Reservoir

1. Name = Sand Pit
2. Liquid Surface Elevation = 0 meters
3. Liquid Surface Pressure = 0 barG (0 kPa (g))

J3 - Branch

Elevation = 1.2 meters

J5 - Branch

Elevation = 6 meters

J15 - Branch

Elevation = 9 meters

J6 - Reservoir

1. Name = Deposit #1
2. Liquid Surface Elevation = 6 meters
3. Liquid Surface Pressure = 0 barG (0 kPa (g))

J16 - Reservoir

1. Name = Deposit #2
2. Liquid Surface Elevation = 9 meters
3. Liquid Surface Pressure = 0 barG (0 kPa (g))

J2 - Pump

1. Name = Main Slurry Pump
2. Elevation = 1.2 meters
3. Choose the Centrifugal (Rotodynamic) model button
4. In the “Reverse Flow” area choose “Never”, since the discharge valves are designed to prevent reverse flow through the pump.

5. Click the “Enter Curve Data” button and enter the following data (Figure 6.3). Ensure that Power is selected in the Efficiency/Power area):

Q (m^3/hr)	H (meters)	Power (kW)
0	152	764
908	145	820
1590	122	894
2044	91	1006

6. In the Curve Fitting area, select the All button then click the Generate Curve Fit now button. Then click OK

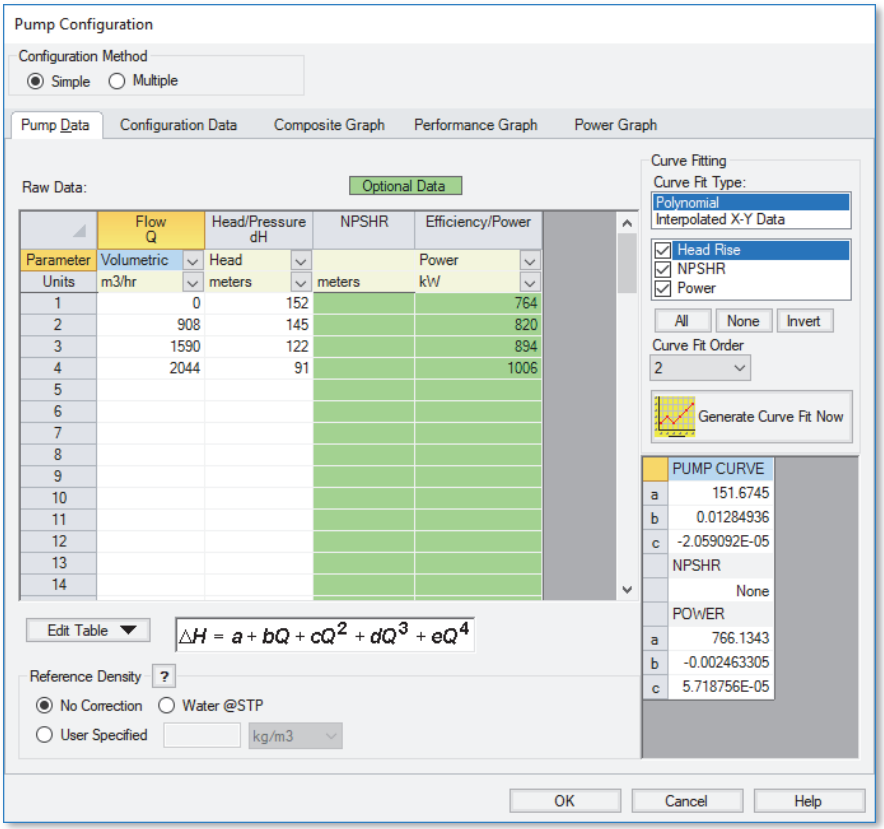


Figure 6.3 Pump J2 performance data

J4 - Valve

1. Name = Valve to Deposit #1
2. Elevation = 1.2 meters
3. Cv = 500
4. Click the Transient tab, select “Time” under “Initiation of Transient”, then enter the transient data as follows:

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

J14 - Valve

5. Name = Valve to Deposit #2
6. Elevation = 1.2 meters
7. Cv = 400
8. Click the Transient tab, select “Time” under “Initiation of Transient”, then enter the transient data as follows:

Time (sec)	Cv
0	400
5	500
10	500

D. Check if the pipe and junction data is complete

Turn on “Show Object Status” from the View menu to verify that all the necessary data is entered. If so, the “Define All Pipes and Junctions” Checklist item on the Quick Access Panel will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data. You can also open the List Undefined Objects window from the View menu to see what data is missing.

SSL Step 4. Section the pipes

Open the Section Pipes window from the Analysis menu. When the Section Pipes window is first opened, it will automatically search for the best (least average variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Click OK to accept the displayed sectioning.

SSL Step 5. Specify transient control

Open the Transient Control window from the Analysis menu. Enter the Stop Time as 10 seconds, then click the OK button.

SSL Step 6. Run the model

The Checklist should now be complete and the model ready to run. Select “Run Model” in the Analysis menu. This will open the Solution Progress window. This window allows you to watch the progress of the Steady-State and Transient Solvers. When complete, press the “Output” button at the bottom of the window to view the results.

SSL Step 7. Review results

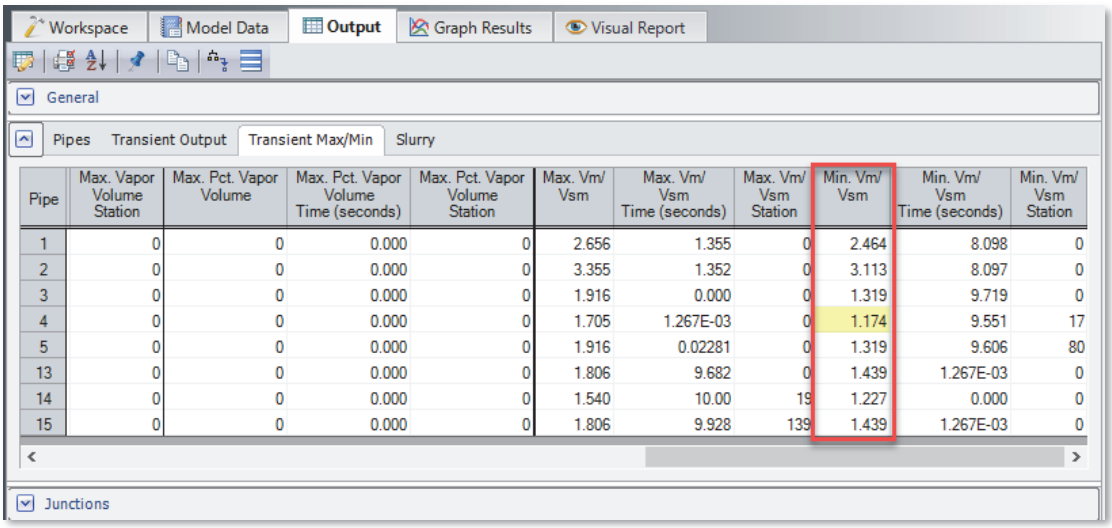
A. Check the Transient Max/Min in the Output

Clicking “Output” on the Solution Progress window will take you to the Output window (Figure 6.4). As a design objective, the system is required to have at minimum a Vsm of 1.2 to avoid the potential of the sand falling out of the slurry and plugging the pipe.

By inspecting the Min Vsm column in the Transient Max/Min tab, it can be seen that the minimum Vsm drops below this value in pipe P4.

Note: A Design Alert could be defined in the Tools menu to automatically evaluate if this design objective is met, and issue a warning if the condition is not satisfied. A predefined Design Alert for this model can be seen by opening the *Sand Transfer to Two Locations – SSL_Metric.imp* model file which is installed in the Examples folder in the AFT Impulse 7 folder.

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



Pipe	Max. Vapor Volume Station	Max. Pct. Vapor Volume	Max. Pct. Vapor Volume Time (seconds)	Max. Pct. Vapor Volume Station	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	0	0	0.000	0	2.656	1.355	0	2.464	8.098	0
2	0	0	0.000	0	3.355	1.352	0	3.113	8.097	0
3	0	0	0.000	0	1.916	0.000	0	1.319	9.719	0
4	0	0	0.000	0	1.705	1.267E-03	0	1.174	9.551	17
5	0	0	0.000	0	1.916	0.02281	0	1.319	9.606	80
13	0	0	0.000	0	1.806	9.682	0	1.439	1.267E-03	0
14	0	0	0.000	0	1.540	10.00	19	1.227	0.000	0
15	0	0	0.000	0	1.806	9.928	139	1.439	1.267E-03	0

Figure 6.4 Design Objective violation in pipe P4 station 17

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 Section Pipes window. Click OK to accept the results. Then open Transient Control and click OK.

Rerun the model and view the Output. There should not be any warnings now that the pipes were changed. The Graph Results window will be more useful in understanding the results.

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

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

1. Change to the Graph Results window from the Window menu, by clicking on the Graph Results tab, or by pressing Ctrl+G on the keyboard.

2. Open the Profile tab on the Graph Control tab.
3. For the Pipes section, select “Plot Single Path”.
4. Select pipes P1, P2, P3, P4 and P5 (Figure 6.5). These represent the flow path to Deposit #1.

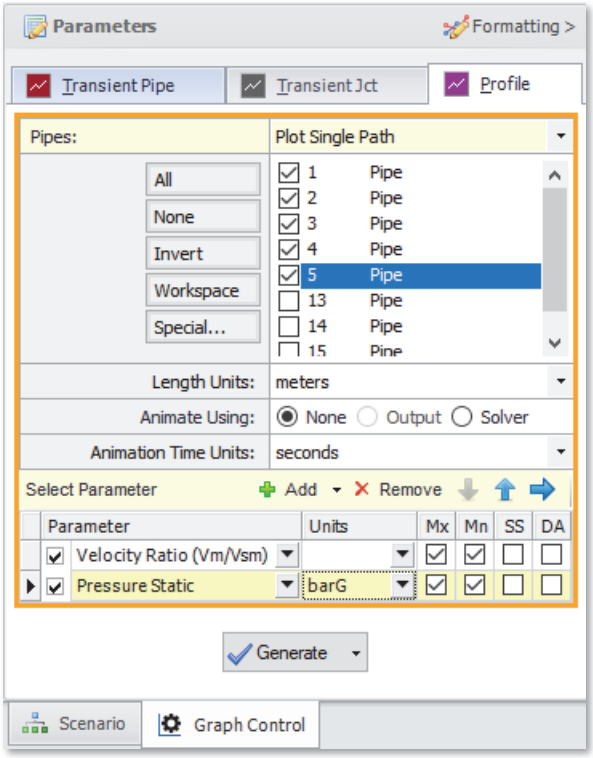


Figure 6.5 Data selection to view the maximum and minimum values of the velocity ratio and static pressure along the flow path to Deposit #1.

5. Choose “meters” as the Length Units.
6. Because you are not animating these values in this example, ensure that “None” is selected next to Animate Using.
7. In the Parameters definition area, ensure that there are two parameters by selecting the green “+” icon to add a parameter to graph. Choose “Velocity Ratio (Vm/Vsm)” for one parameter and “Pressure Static” for the other.
8. Specify “barG” as the Units for the Pressure Static parameter.

9. Check the boxes under the “Mx” and “Mn” columns for both parameters to ensure that the maximum and minimum values are being displayed for both graphs.

10. Click the Generate button. The resulting graphs are shown in Figure 6.6.

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.

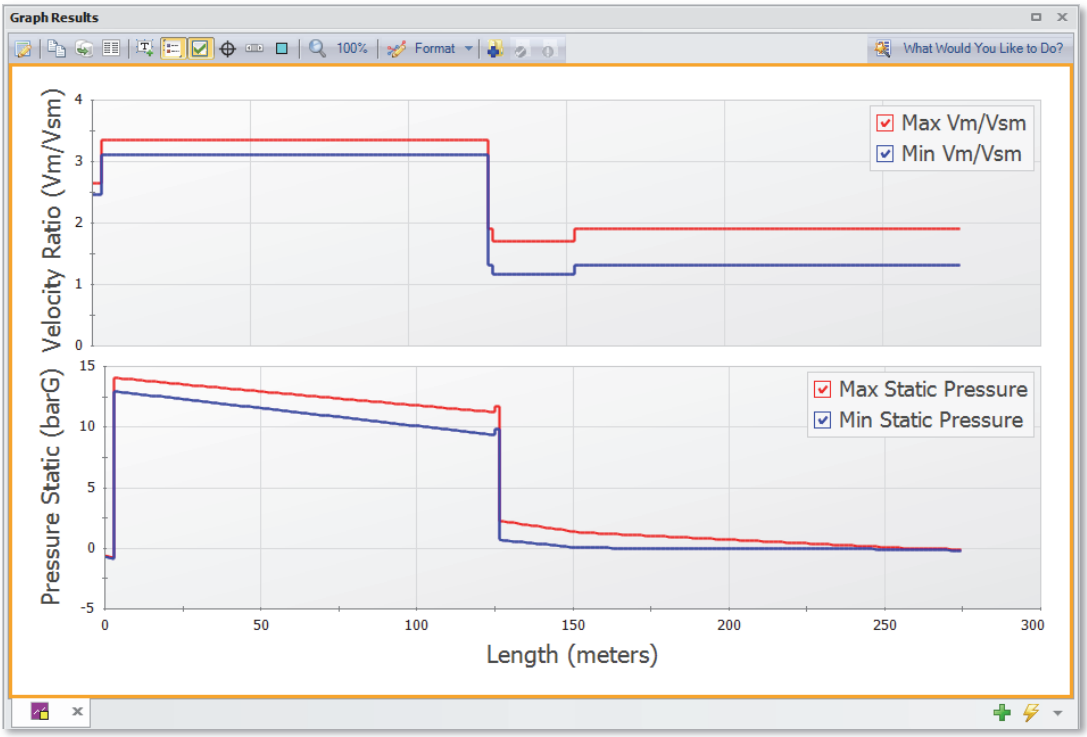


Figure 6.6 Maximum and minimum velocity ratio and static pressure values in the flow path to Deposit #1. Note that the velocity ratio is above the minimum Design Alert of 1.2 at all locations.

C. Graph the velocity ratio over time

A time comparison of the velocity ratio in the critical pipes along the two flow paths is important to consider. To graph this profile, complete the following steps:

1. Return to the Graph Control tab on the Quick Access Panel.
2. On the Transient Pipe tab, select Pipes P4 Inlet and P14 Inlet and add them to the pipe stations to graph (Figure 6.7). These are the two sloped pipes which have the lowest velocity ratio
3. Specify the Time Units as seconds
4. Specify “All Times” as the Time Frame
5. In the Parameters definition area, choose Velocity Ratio (Vm/Vsm)
6. Click the Generate button

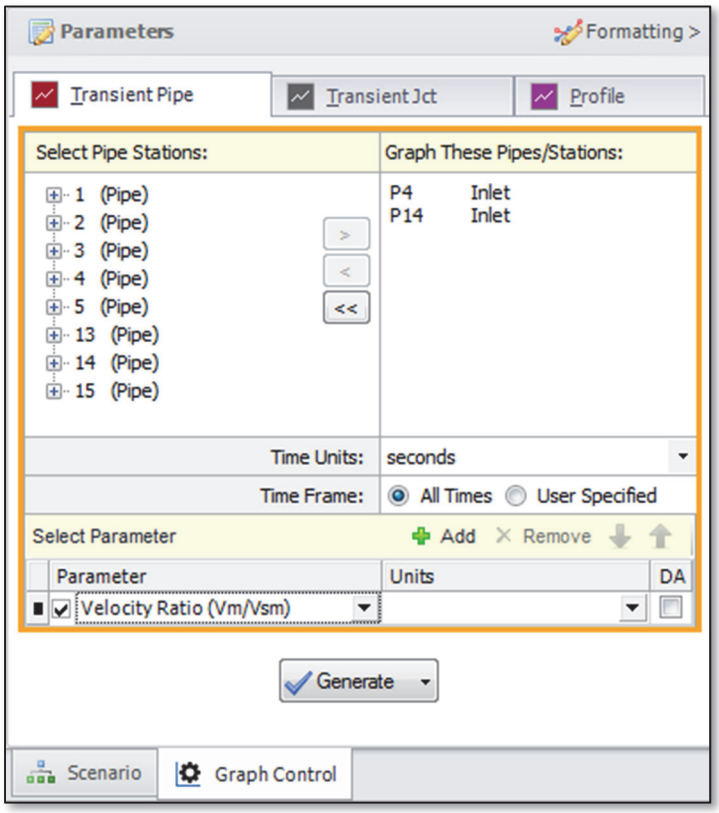


Figure 6.7 Data selection to view the velocity ratio over time

The resulting graph is shown in Figure 6.8. This shows that, as Valve J4 closes and Valve J14 opens, the velocity ratios in the pipe immediately downstream change but stay in the acceptable range.

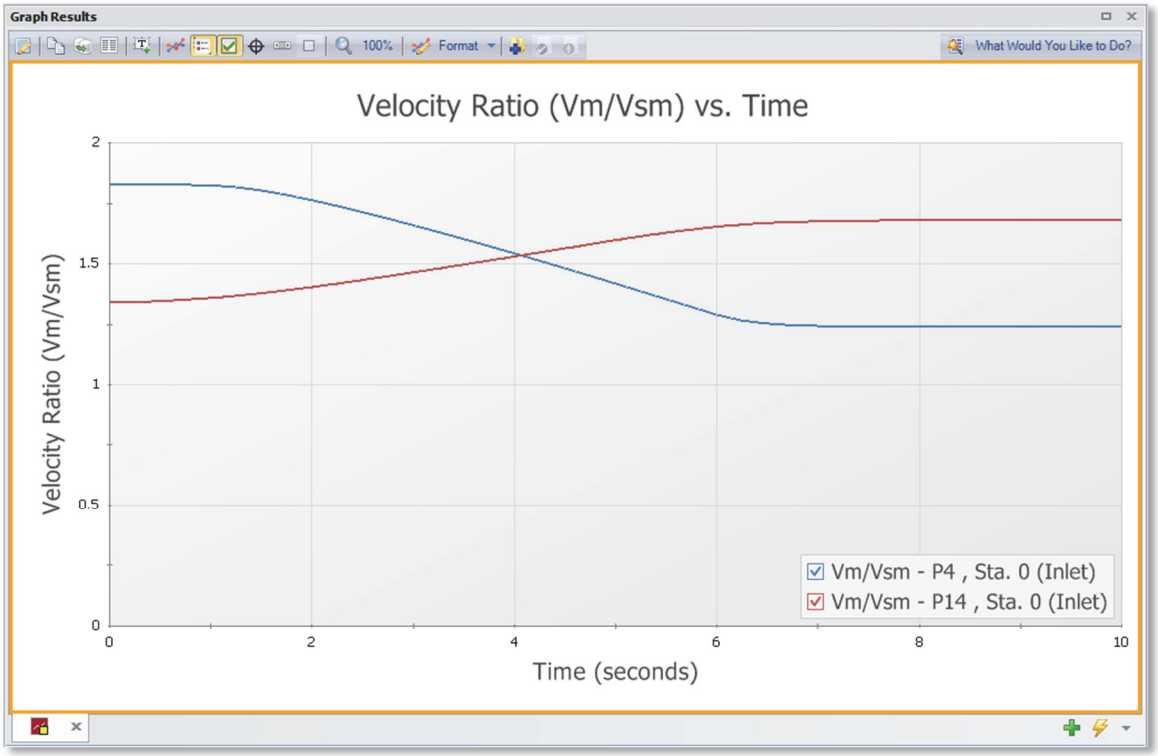


Figure 6.8 The velocity ratio in the sloped pipes as the valves change position

SSL analysis summary

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

PFA problem statement

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

PFA Step 1. Start AFT Impulse

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

PFA Step 2. Specify system properties

1. Open the System Properties window by selecting System Properties in the Analysis menu.
2. On the Fluid Data tab, select User Specified Fluid
3. Enter 0.92 g/cm^3 in the fluid density box
4. Enter 0.95 cP in the fluid dynamic viscosity box
5. Enter 9653 bar in the fluid bulk modulus box
6. Click OK

PFA Step 3. Build the model

A. Place the pipes and junctions

At this point, the first two items are completed on the Checklist. The next Checklist item is to "Define Pipes and Junctions". In the Workspace window, assemble the model as shown in Figure 6.9.

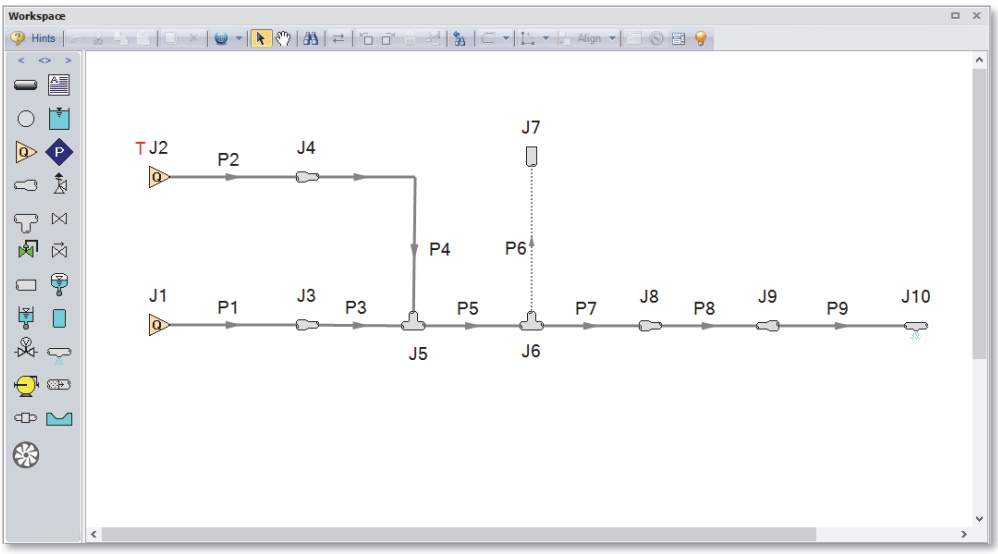


Figure 6.9 Layout of PD Plunger Pulsation Study example

B. Enter the pipe data

The input data for the pipes and junctions now needs to be entered. Double-click on each pipe to open the properties window and enter the following data (or use the Global Pipe Editing window).

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

Pipe No.	Inner Diameter (cm)	Length (m)
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

C. Enter the junction data

Double-click each junction and enter the following data in the Properties window (or use the Global Junction Editing window).

J1 and J2 – Assigned Flow

For the sake of simplicity, assigned flow junctions are being used to represent the PD pumps with the information listed below. The Pulsation Setup window 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. Volumetric Flow Rate = 8 m^3/hr

J3 and J4 - Area Change

1. Elevation = 0 meters
2. Type = Abrupt Transition (K factor is automatically calculated)

J8 and J9 - Area Change

1. Elevation = 0 meters
2. Type = Conical Transition (K factor automatically calculated)
3. Angle = 90 degrees

J5 and J6 - Tees

1. Elevation = 0 meters
2. Loss Model = Simple (no loss)

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 = 1.01 bar
6. Cd (Discharge Coefficient) = 1
7. Discharge Flow Area = 0.148 cm^2

J7 - Dead End

Elevation = 0 meters

D. Check if the pipe and junction data is complete

Turn on Show Object Status from the View menu to verify that all the necessary data is entered. If so, the "Define Pipes and Junctions" Checklist item on the Quick Access Panel will have a check mark. If not, the incomplete pipes or junctions will have their number shown in red. If this happens, go back to the incomplete pipes or junctions and enter the missing data. You can also open the List Undefined Objects window from the View menu to see what data is missing.

PFA Step 4. Define pulsation setup properties

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 the Pulsation Setup window from the Analysis menu. The Pulsation Setup window is the main interface for specifying input regarding the initial pulse and other information about the pulsation source. There are two tabs on the Pulsation Setup window directly associated with the PFA module: the Pulsation Setup tab and the PD Pump Settings tab. The PD Pump Settings tab only requires input if the source of pulsation is a positive displacement pump. Because the source of pulsation in our model is a positive displacement pump, we will need to complete the input on both of these tabs as shown below:

1. Define the Pulse Properties on the Pulsation Setup tab (see Figure 6.10 for all input on the Pulsation Setup tab in the Pulsation Setup window)

    a. Applied At = J2 (Assigned Flow)

    b. Start Time = 0 seconds

2. Check the box next to Use Variable Pipe Resistance

3. Verify that the Magnitude is being calculated automatically. This is equal to twice the steady-state flow. In this case, the Peak is 16 m^3/hr.

4. Define the Pulsation Source
   a. Select Positive Displacement (PD) Pump
   b. Enter 3 for the number of heads
   c. Enter 250 rpm for the pump speed
   d. Select 15% for the speed buffer
5. Verify that the Low-Pass Filtering defaults are applied
   a. Filter Type = Chebyshev
   b. Order = 10
   c. Ripples Allowed = 1 dB
6. Verify that the Cutoff Frequency is calculated automatically using the PD Pump Characteristics. In this example, the Cutoff Frequency is calculated to be 86.25 Hz.

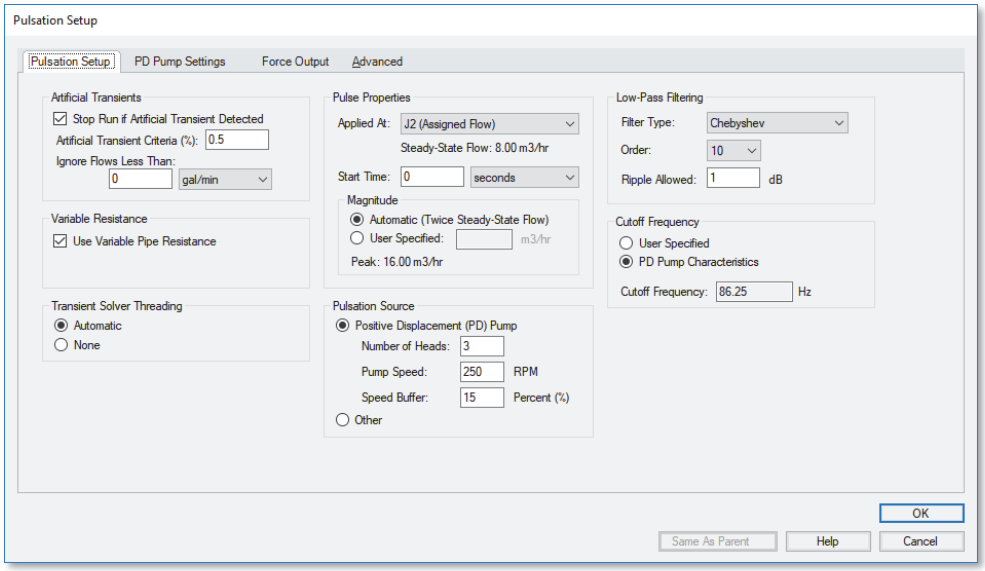


Figure 6.10 Pulsation Setup window with input

7. Now that all required input has been entered in the Pulsation Setup tab, enter the following information on the PD Pump Settings tab (see Figure 6.11).
   a. Minimum Pump Speed = 125 RPM

b. Bore Diameter = 7.6 cm

c. Stroke Length = 5.2 cm

d. Rod Length = 25.4 cm

e. Clearance = 70 Percent (%)

f. Suction Pressure = 4.5 bar

g. Discharge Pressure = 415 bar

h. Phase Lag = 0 degrees

8. Click OK to keep the changes and close the Pulsation Setup window.

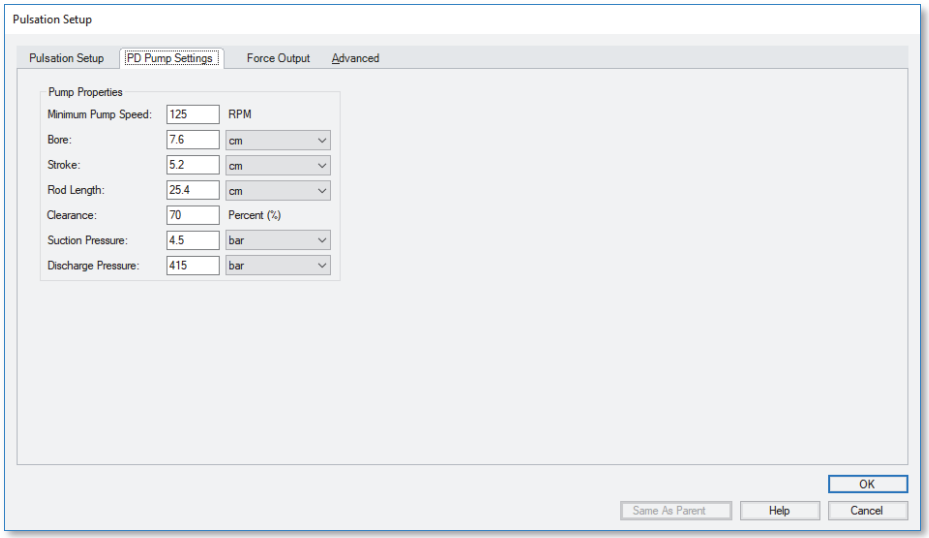


Figure 6.11 PD Pump Settings tab with input

PFA Step 5. Section the pipes

Open the Section Pipes window from the Analysis menu. While using the AFT Impulse PFA module, the pulsation setup needs to be completed prior to sectioning the pipes in the model. To section the model's pipes for the simulation, complete the following:

1. Select the significant pipes that should consider error. In our example, we will consider the error in all modeled pipes, except for

pipe P8. You can see that this pipe has been excluded in Figure 6.12. This is because this pipe represents a meter in the physical system and is not an actual pipe. Ensure that the box next to each pipe is selected under Consider Variances, except for pipe P8.

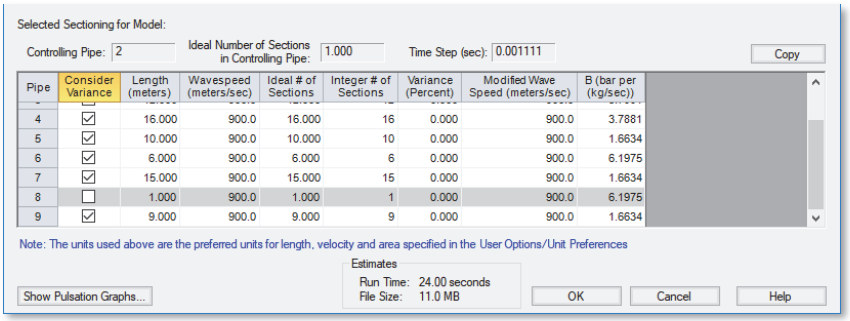
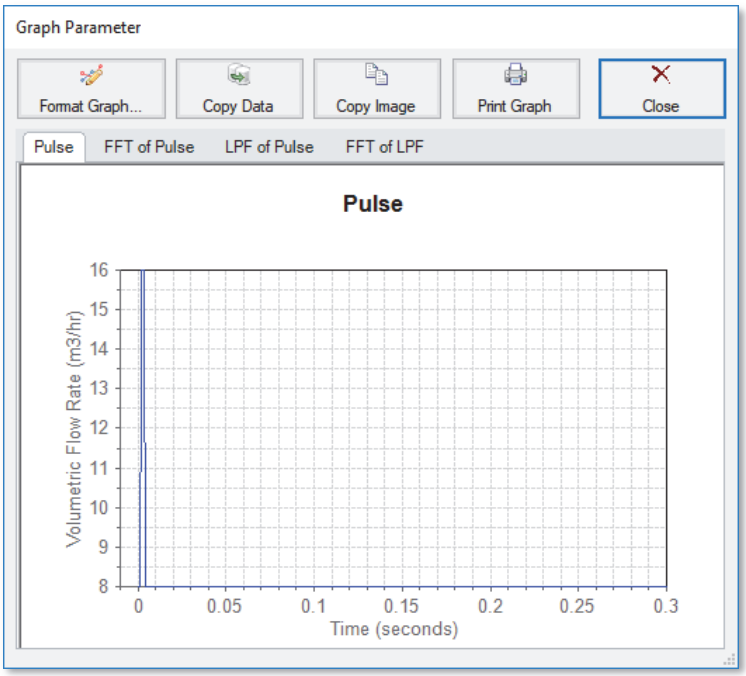


Figure 6.12 Section Pipes window excluding the variance at P8

2. Select 1 section in the controlling pipe by clicking on 1 section in the table under Search Results.
3. Select Show Pulsation Graphs located at the bottom left of the Section Pipes window. A graph of the pulse will appear (see Figure 6.13). You can also view graphs for the FFT of the Pulse, the LPF of the Pulse FFT, and the FFT of the LPF in the Graph Parameter graph that appears. After you are done viewing these graphs, click Close.
4. Click OK to close the Section Pipes window, and click OK for the pop-up window notifying you that the forcing function has been placed on J2 (the junction that was selected in the Pulsation Setup window) as transient data.
5. Open J2 to verify that the numerical “ring” has been applied to the transient (see Figure 6.14 for the transient input). This transient is a brief jump in flow rate at junction J2.

PFA Step 6. Run the model

The Checklist should now be complete and the model ready to run. Select Run Model in the Analysis menu. This will open the Solution Progress window. This window allows you to watch the progress of the Steady-State and Transient Solvers. When complete, click the Graph Results button at the bottom of the Solution Progress window.



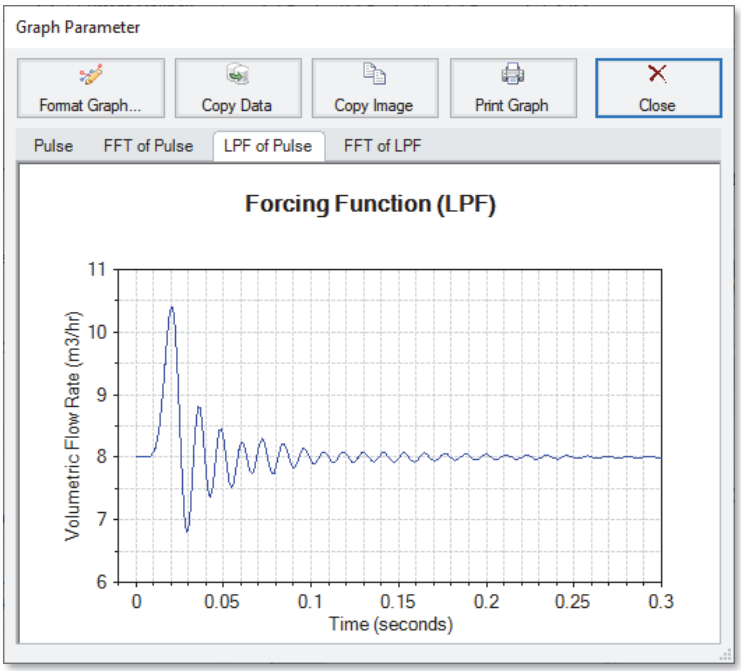


Figure 6.13 Graphs of Pulse and LPF of Pulse in this example

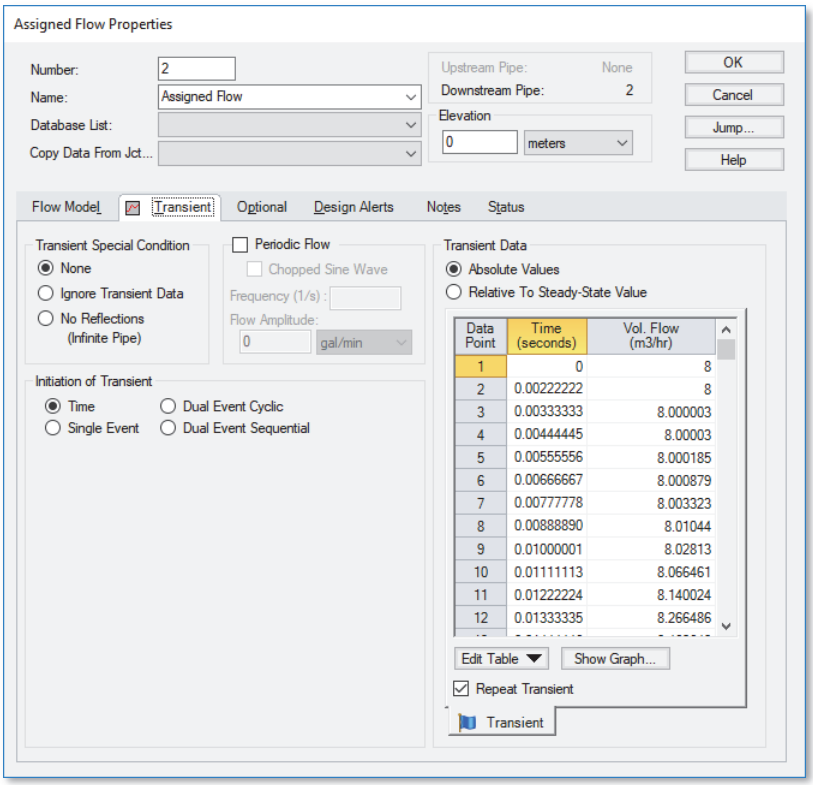


Figure 6.14 Assigned Flow Properties with forcing function applied to J2. Choosing "Show Graph…" under the data shows this in graphical form.

PFA Step 7. Review results

Find excitation frequencies to study

1. On the Graph Control tab on the Quick Access Panel, click the Frequency tab to begin generating an Excitation Frequency Analysis graph.

2. We need to determine which frequencies excite the pump speeds within the pump speed range we have specified. For the purposes of this example, we will graph the Excitation Frequency graph for 10 different pipe stations. Note that it is the engineer's responsibility to evaluate all pipe stations that could be excited by various frequencies and to perform the necessary analysis.

3. On the Graph Results tab, select the Frequency tab and select the inlet and outlet stations for the following pipes: 1, 2, 3, 5, and 6.
4. Click Generate. The resulting graph (shown in Figure 6.15) represents the frequencies that are excited by the pulse defined in the Pulsation Setup window.

From the graph in Figure 6.15, 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 frequencies that will most excite the system at the selected stations.

Note that the graph in Figure 6.15 can quickly be changed to display the magnitude on a logarithmic scale in order to allow for easier viewing of the magnitude behavior changes with a changing frequency. To view the Excitation Frequency Analysis graph with the magnitude displayed on a logarithmic scale, generate the frequency graph as discussed above. Right-click on either axis and select the box next to Logarithmic.

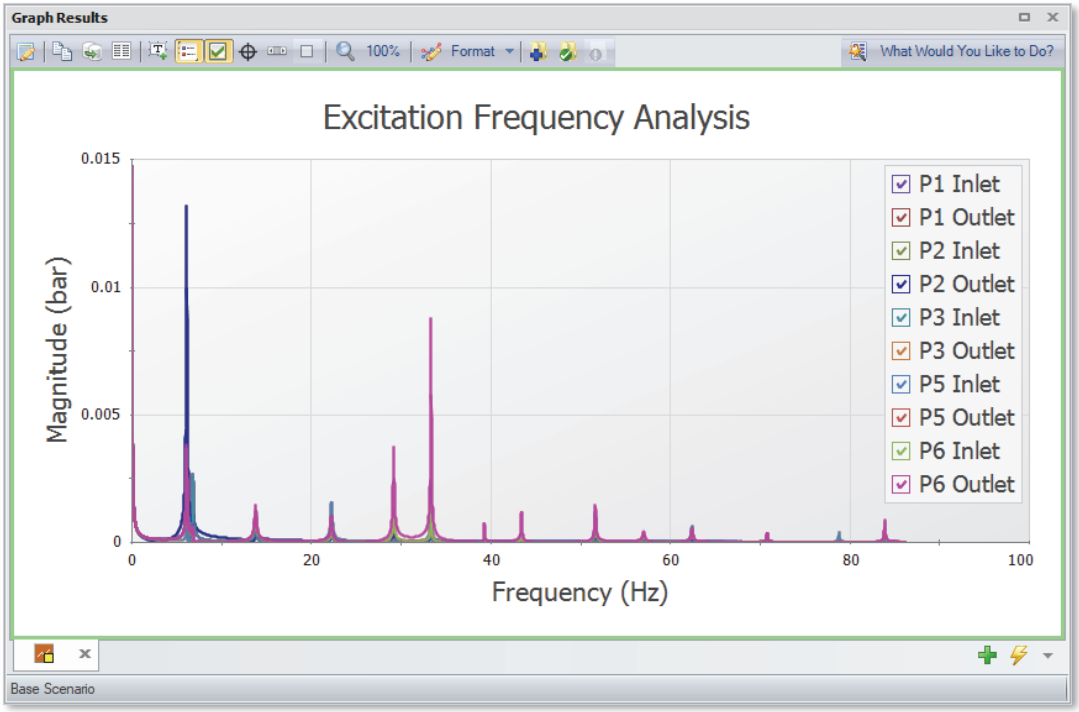


Figure 6.15 Excitation Frequency Analysis graph

Evaluate excitation frequencies

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

1. Left-click near the frequency of around 33 Hz and drag the mouse over the peak on the graph near this frequency. This will highlight and flag the local maximum. Figure 6.16 shows the frequency graph after the dialog flag appears.
2. Right-click on the dialog flag that appears on the peak of this frequency
3. Select "Evaluate Excitation Frequency" to add this frequency to the Pump RPM Evaluation panel, which is labeled with "Excitation Frequencies and Pump Speeds (RPM)". Repeat steps 1 through 3 for the local maxima at the frequencies of 29.17 Hz and 6.79 Hz. The panel should now appear as shown in Figure 6.17.

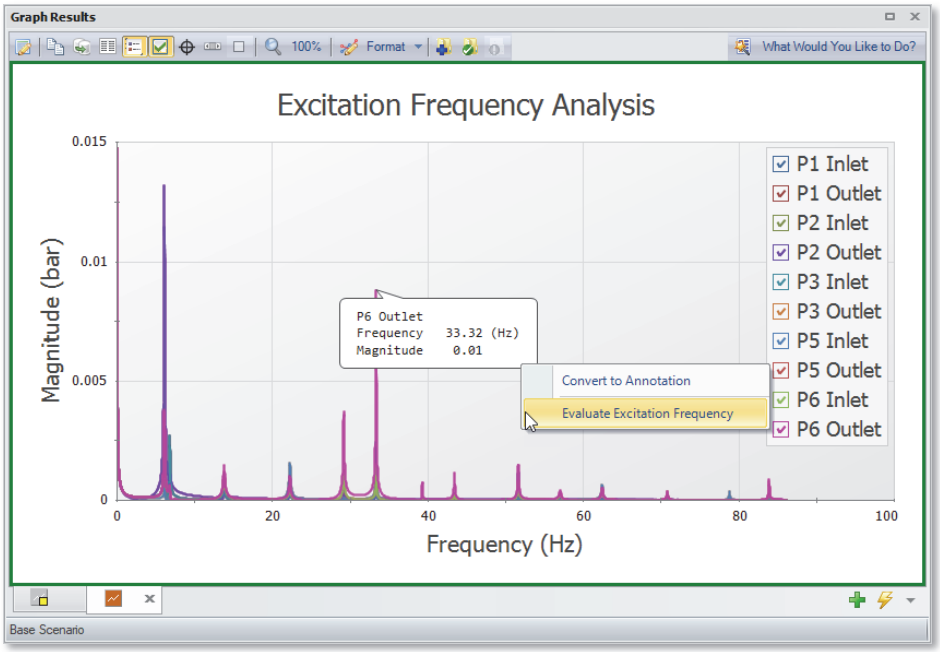


Figure 6.16 Excitation Frequency Analysis with Dialog Flag. Right-clicking on the Dialog Flag provides the option to Evaluate Excitation Frequency.

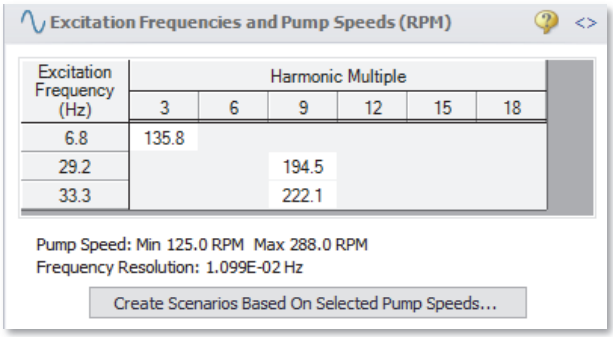


Figure 6.17 Pump RPM Evaluation Panel with Excitation Frequencies and their Harmonic Multiples Displayed in the bottom of the Quick Access Panel.

When choosing low or high frequencies on the graph, it is possible to select a frequency which is excited in the system but does not have a pump speed within the minimum and maximum pump speeds for the analysis. The minimum pump speed is defined directly in the PD Pump Settings in the Pulsation Setup window, while the maximum is calculated using the Pump Speed and Speed Buffer defined under “Pulsation Source” in Pulsation Setup. For example, try selecting the largest maxima, which occurs at 6.1 Hz. The message in Figure 6.18 appears.

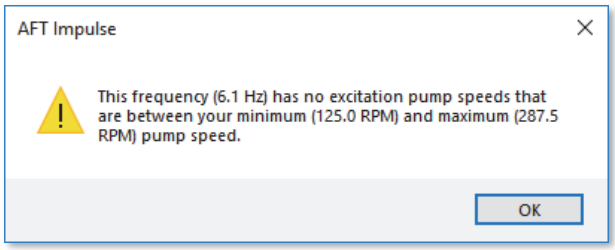


Figure 6.18 Message Indicating No Excitation Pump Speeds Between the Min and Max Pump Speeds Exist

Determine the pressure response of the system at excited frequencies

In the Pump RPM Evaluation panel, corresponding pump speeds can be selected for study. At these speeds, the pressure response is important to characterize. By default, all speeds in the Pump RPM Evaluation panel are pre-selected and must be clicked to be turned off. Child Pump Speed

scenarios can then be created that will have the Flow vs. Time profile at the selected speed (RPM) that will excite the evaluated frequencies.

We will create child scenarios for all three Excitation Frequencies in the Pump RPM Evaluation panel. To do this, ensure that the pump speeds of 135.8, 194.5, and 222.1 RPM are selected (not italicized).

1. Click “Create Scenarios Based On Selected Pump Speeds…”
2. A dialog box will appear stating that three scenarios will be created based on the selected speeds in the Pump RPM Evaluation panel. Click Yes to continue creating the child scenarios.
3. Double-click on the "135.8 RPM (6.8 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 135.8 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 6.19.

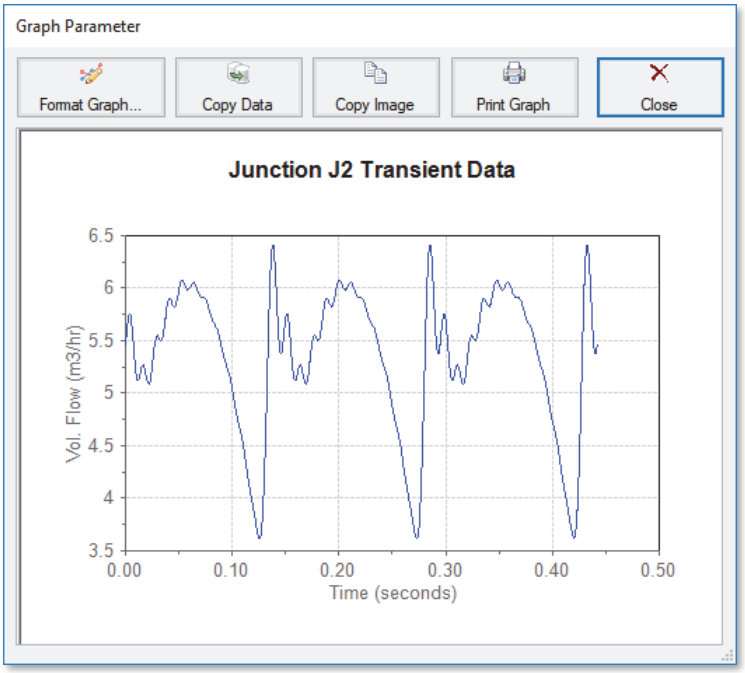


Figure 6.19 New Assigned Flow transient representing the PD Pump operating at 135.8 RPM

Run the "135.8 RPM (6.8 Hz)" scenario to see the pressure response. Often, the most noticeable pressure response will occur in stagnant branched lines. For the purposes of this example, we will evaluate the pressure responses through pipe P6. However, engineers must ensure that all pipes that could experience significant pressure responses are analyzed.

1. To create a pressure profile graph in pipe P6, go to the Graph Results tab and in the Graph List Manager, select Profile for the graph type.
2. Check the box for Pipe P6.
3. Select Pressure Stagnation for the Parameter.
4. Use bar for the units.
5. Click Generate.
6. Right-click on the y-axis to adjust the y-axis scale by unchecking the Auto Scale box, then changing the Minimum to 240 and the Major Val to 20.

Figure 6.20 shows the profile plot for Pipe P6 at a pump speed of 135.8 RPM.

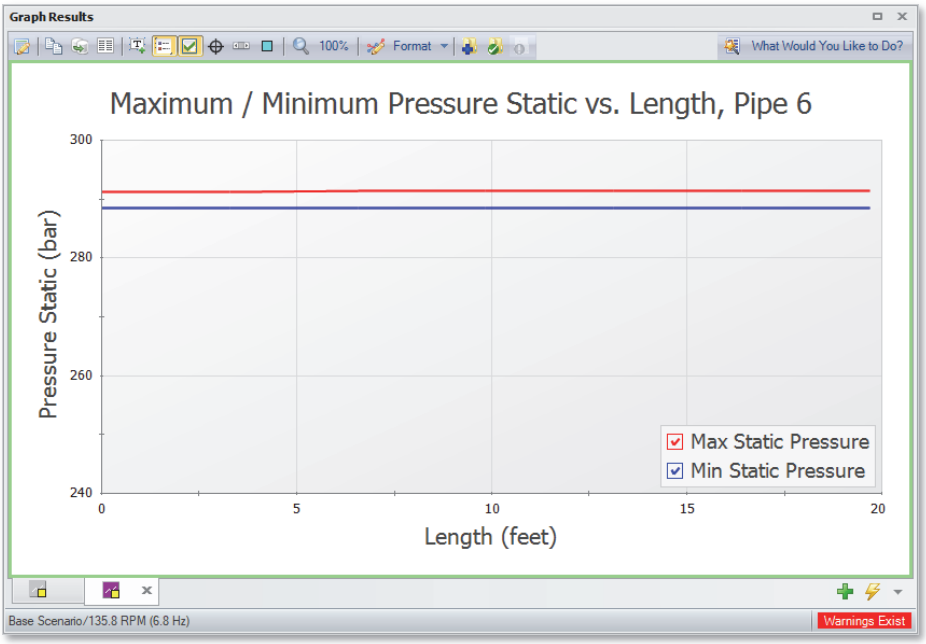


Figure 6.20 Max/Min Pressure Profile of Pipe P6 for 135.8 RPM

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. Select Pressure Stagnation for the Parameter.
4. Use psia for the units.
5. Set the Time Frame to User Specified with a Start Time of 0 and a Stop Time of 2 seconds.
6. Click Generate (see Figure 6.21)

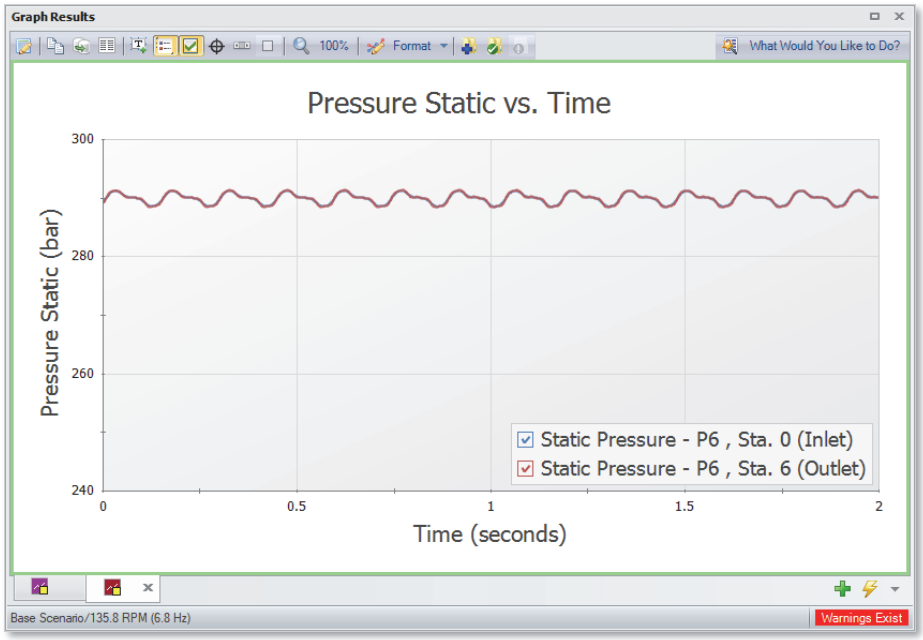


Figure 6.21 Pressure Oscillations at Inlet and Outlet of Pipe P6 for 135.8 RPM

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 222.1 RPM because analysis was performed beforehand to determine that this pump speed causes the largest pressure oscillations. We will show these graphs here. When performing an analysis on your system, all pump speeds that excite the system should be evaluated.

Figure 6.22 shows the Max/Min Pressure Profile for P6 for the pump speed of 222.1 RPM.

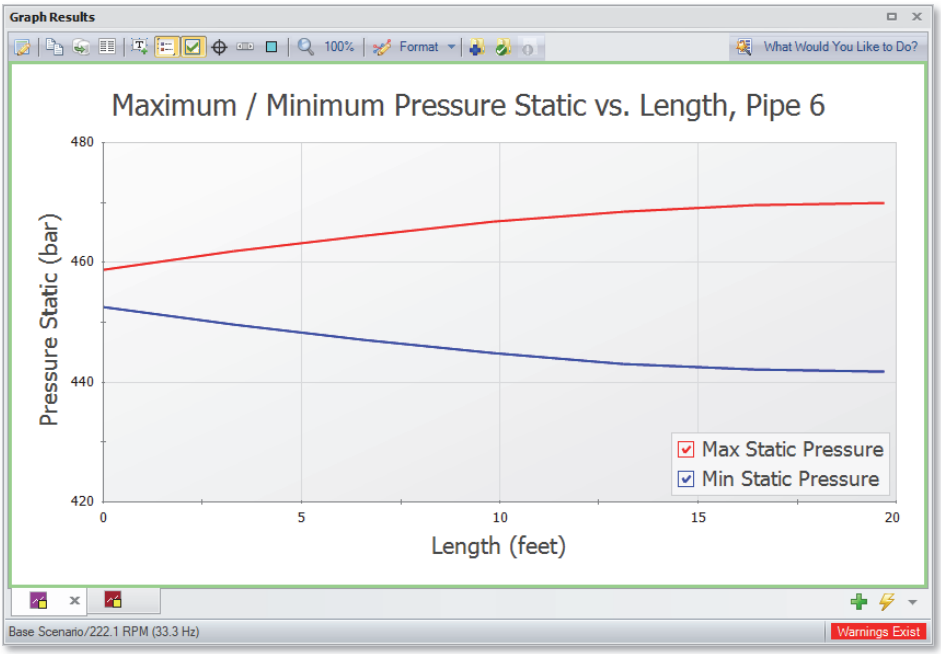


Figure 6.22 Max/Min Pressure Profile of Pipe P6 for 222.1 RPM

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

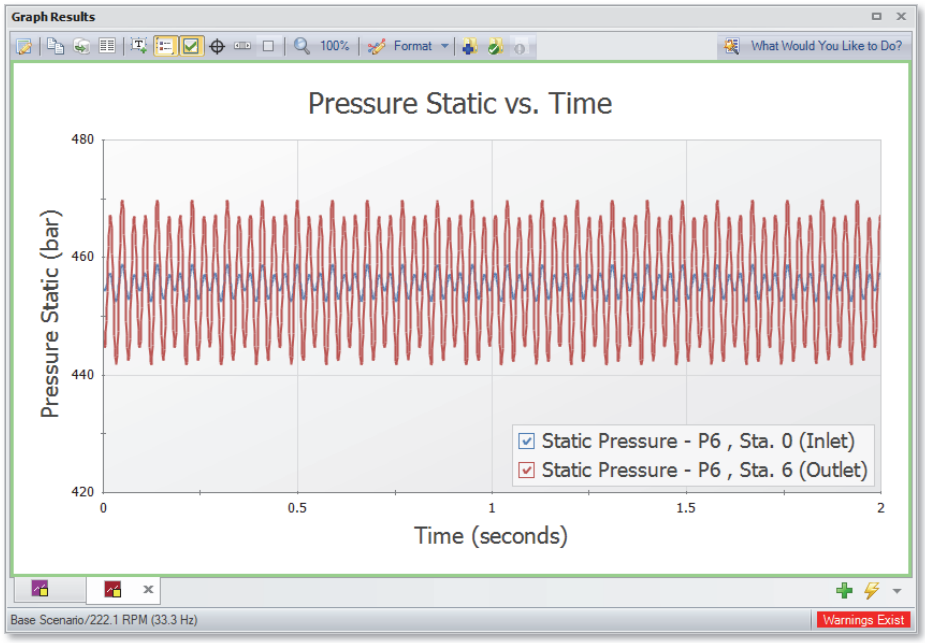


Figure 6.23 Pressure Oscillations at Inlet and Outlet of Pipe P6 for 222.1 RPM

PFA analysis summary

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

CHAPTER 7

Other AFT Impulse Capabilities

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

Microsoft Excel™ data integration

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

Integration with other software and data standards

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

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

Transient cavitation and liquid column separation

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

Four quadrant pump modeling

When pumps can flow or rotate backwards, four quadrant modeling is needed. AFT Impulse offers four quadrant models using the popular Suter method. Twenty-one sets of four quadrant data are provided.

Positive displacement pumps

AFT Impulse can model positive displacement pumps. Typically this is done by modeling the pump as a known flowrate over time. Complicated periodic flowrates from multiple pistons and/or multiple out of phase pumps can also be modeled.

Pumps with viscosity corrections

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

Pumps with variable speed controllers

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

Pumps as turbines (PAT)

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

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.

Vacuum breaker valves

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

Surge tanks

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

Relief valves

The Relief Valve junction allows flexibility in modeling relief valve behavior by defining combinations of opening and closing profiles based on different parameters, such as time or pressure. In addition to set pressure, users can define a blowdown pressure for relief valves that close at pressures lower than 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. Alternatively, 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 entered by the user or automatically set according to the valve set points. In addition, users can define opening and closing rate limits for valves with pressure profiles.

Flow and pressure control valve transients

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

Non-Newtonian fluid modeling

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

Pulp and paper modeling

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

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.

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 behavior of the system. Exceptions are high points which cavitate. If desired, AFT Impulse can model varying elevations along a pipe.

Fitting library

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

Design alerts

Design Alerts can be entered for pipes and then cross-plotted vs. system behavior. A common use is maximum and minimum allowed operating pressure. These can be entered as Design Alert for pipes and then plotted against predicted pressure transients.

Network databases

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

Index

G

H

I

J

L

M

N

O

P

Q

R

S

T

U

V

W

AFT Impulse™ Add-on Modules

SSL

Settling Slurries

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

PFA

Pulsation Frequency Analysis

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

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

Applied Flow Technology

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