AFT Impulse™

Examples - Mertic Units

AFT Impulse Version 9

Waterhammer Modeling in Piping Systems

Published: May 02, 2023

Dynamic solutions for a fluid world ™

Contents

Walk-Through Examples **3**

Beginner Valve Closure 4

Ammonia Transfer with Cavitation 40

Tank Farm with Infinite Pipes 51

Pump Trip with Check Valve 64

Check Valve Modeling 77

Relief Valve Modeling 94

Pump Trip with Accumulator 112

Pump Startup with Event Transients 119

Pump Trip with Backflow - Four Quadrant Modeling 137

Selecting a Pump Four Quadrant Curve 153

Pump with Variable Speed Flow Control 186

Positive Displacement Pump 194

Valve Closure with Pipe Forces 205

Module Walk-Through Examples **215**

Pipe Sizing for Sand Transfer - SSL 215

PD Pulsation Study - PFA 230

Walk-Through Examples

The Impulse Walk-through Help System is designed to get new users trained to use Impulse correctly in the shortest amount of time. If this is your first time to use Impulse, start with the Valve Closure - Beginner Example because it is the most complete. After this example, choose an example from the table below that most closely matches your application. The examples are arranged in order of increasing complexity.

Note: The examples in the Help System are available in both English and Metric units. To work these examples in a different unit system, open the appropriate example help file, or click the link to the appropriate unit system example found at the top of each example.

Example	Fluid	Pipes	Pumps	Description
Beginner Valve Closure	Water	4	-	In-depth discussion of model building in AFT Impulse. Solves a simple valve closure problem.
Ship to Shore Transfer System - With Cavitation	Ammonia	3	-	Three valve closure rates on a fluid transfer system are evaluated with respect to maximum pressure. Cavitation occurs in two cases.
Tank Farm - Use of Infinite Pipes	Gasoline	5	-	A long supply pipeline flows into a gasoline tank farm. The transient occurs when flow is switched from one storage tank to another. Two scenarios compare the results with and without an infinite pipe boundary, and also compares the run times.
Pump Trip With Check Valve	Water	2	1	A system pumping water uphill loses power to the pump. This model predicts the system response and pump speed decay.
Check Valve Modeling	Water	3	1	A check valve needs to be selected to prevent reverse flow trough a pump in the case of a power failure. Different check valve modeling options are explored.
Relief Valve Modeling	Crude Oil	6	1	A relief valve is considered to reduce pressure in a pipeline after an unplanned valve closure and pump trip. A back pressure relief valve and a rupture disk relief valve are compared.
Pump Trip With	Water	3	1	Same system as Pump Trip With

Example	Fluid	Pipes	Pumps	Description
Accumulator				Check Valve, except an accumulator is used to minimize the downsurge due to the pump trip to keep all pressures above atmospheric.
Pump Startup With Event Transients	Water	11	2	Transients from different pump startup scenarios are evaluated. A valve opening transient is initiated based on system events.
Pump Trip With Backflow - Four Quadrant Modeling	Water	13	2	Two pumps supplying cooling towers lose power. Valves close, but not before the pumps start to spin backwards. Four-quadrant pump modeling is used to evaluate the pump reaction and system transients.
Selecting a Pump Four Quadrant Curve	Water	9	1	Provides the engineer with a deeper understanding of how to select four quadrant data sets for a waterhammer analysis, and how to evaluate the transient results predicted by the four quadrant curves.
Pump With Variable Speed Flow Control	Water	8	1	A pump with a variable speed drive for flow control supplies water to two heat exchangers. The flow to one of the exchangers is turned off, and the controlled pump flow rate is reduced by half over five seconds.
Positive Displacement Pump	Water	5	1	Models of a duplex and triplex positive displacement pump
Valve Closure With Pipe Forces	Water	2	-	Determines the transient forces that occur in a gravity drain system that transfers water from one tank to another through a valve in the connecting pipeline.

Beginner Valve Closure

Summary

This example 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.

Topics Covered

- Building a basic model
- Entering pipe and junction data
- Entering transient data
- Graphing output results

Required Knowledge

This example assumes the user has not used AFT Impulse previously. It begins with the most basic elements of laying out the pipes and junctions and solving the transient/waterhammer system hydraulics via the Method of Characteristics (MOC).

Model File

- ***Metric - Beginner Valve Closure.imp***

Step 1. Start AFT Impulse

➢**To start AFT Impulse**, click Start on the Windows taskbar and open AFT Impulse 9. (This refers to the standard menu items created by setup. You may have chosen to specify a different menu item).

As Impulse is started, the Impulse Startup Window, as shown in Figure 1 appears and provides you with several options before you start building a model. You can open a model that was recently worked on or browse to a different Impulse file or the Impulse Example files. If the Impulse SSL or PFA add-on module is owned, it can be activated here. The Unit System can also be specified to use US Only, Metric Only, Both with US Defaults, or Both with Metric Defaults and they can be set as the user default at this time as well. You can also set the working fluid to AFT Standard Water (liquid) or another recently used AFT fluid. You can click the link to Check for Latest Release to go to the AFT website to see if you have the latest release of AFT Impulse. You can also Subscribe to Receive Notifications of when product maintenance releases are available, receive the AFT newsletter, etc. Finally, you can open the PDF Quick Start Guide, Video Tutorials, the AFT Impulse Help File or Example Help File.

Note: If you are working through the Metric Units version of the Examples, be sure to specify either Metric Only or Both with Metric Defaults as the unit sets and check the box to Set As My Default.

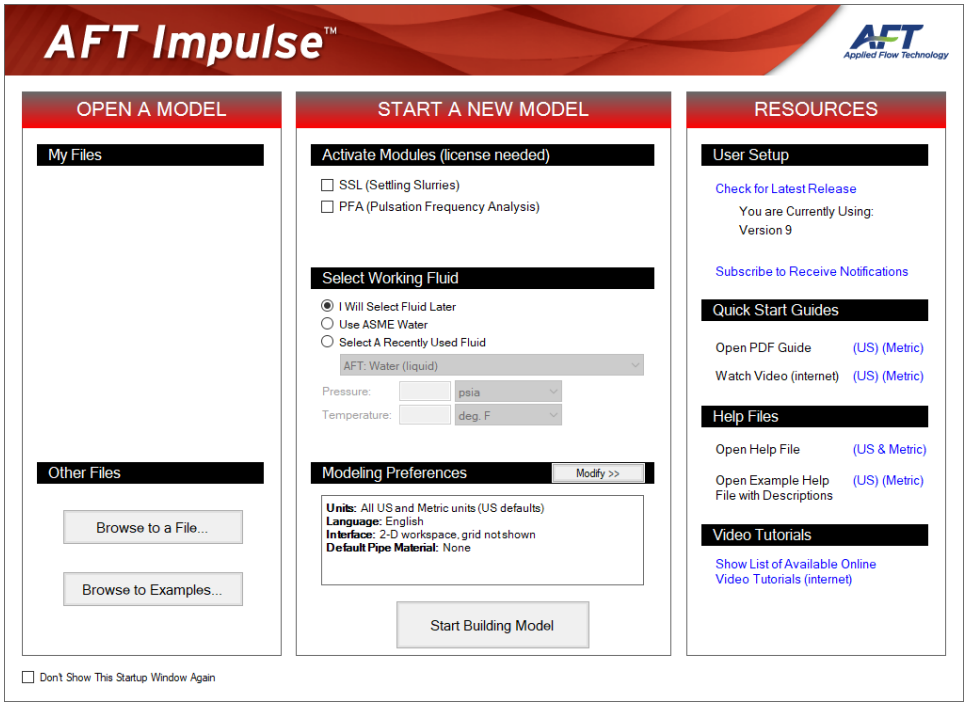


Figure 1: AFT Impulse Startup window

After clicking Start Building Model in the AFT Impulse Startup window, the Workspace window is initially always the active (large) window, as seen in Figure 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.

AFT Impulse supports dual monitor usage. You can click and drag any of the five primary window tabs off of the main 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 Impulse primary tab window bar, simply click the X button in the upper right of the primary window.

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

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 on the right. It is possible to minimize the Quick Access Panel by clicking on the thumbtack pin in the upper right of the Quick Access Panel in order to allow for greater Workspace area.

The Workspace takes up the rest of the window.

You will build your pipe flow model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Pipe Drawing Tool and Annotation Tool. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. The Annotation tool allows you to create annotations and auxiliary graphics.

Below the two drawing tools are twenty-five icons that represent the different types of junctions available in AFT Impulse. Junctions are objects that connect pipes and also influence the pressure or flow behavior of the pipe system. The twenty-five 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.

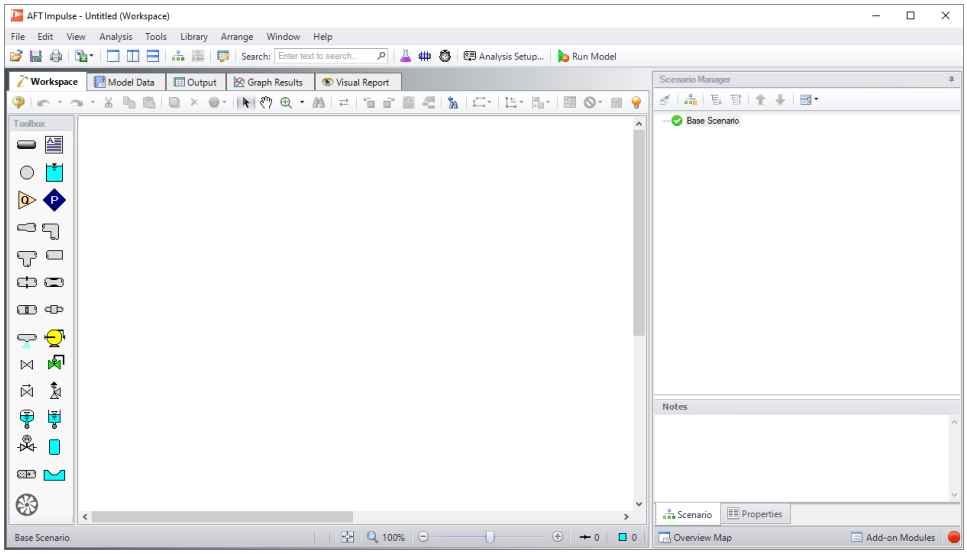


Figure 2: The Workspace window is where the model is built

Step 2. Complete the Analysis Setup

➢**Next, click Analysis Setup on the Toolbar** that runs across the top of the AFT Impulse window. This opens Analysis Setup (see Figure 3). The Analysis Setup contains nine groups (an additional group will display if the PFA module is active). Each group needs to be completed (indicated with a green checkmark next to the group name) before AFT Impulse allows you to run the Solver.

Analysis Setup can also be accessed by clicking on the Status Light on the Quick Access Panel. Once all of the groups in Analysis Setup are defined, the Model Status light in the lower right corner turns from red to green.

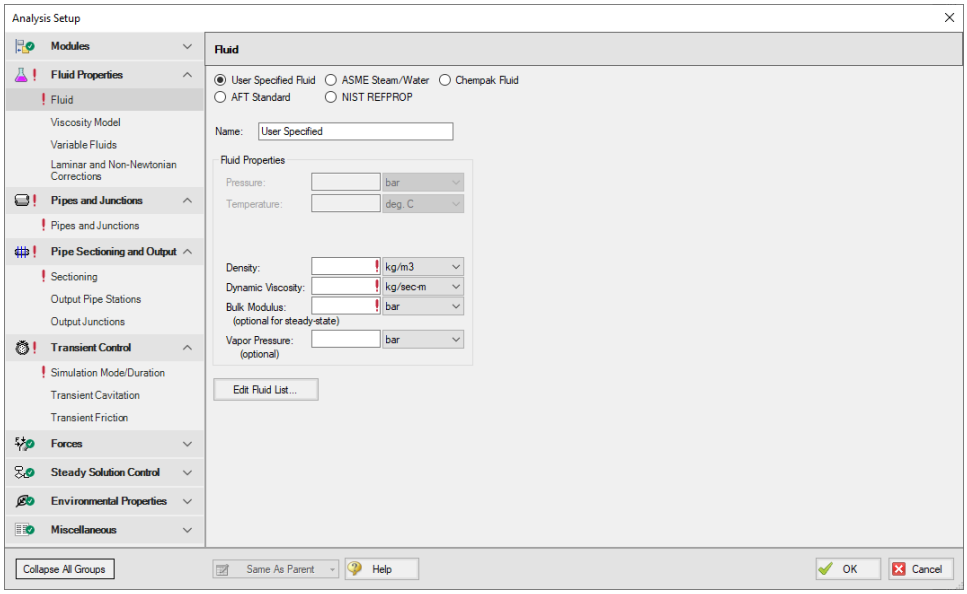


Figure 3: The Analysis Setup tracks the model's status and allows users to specify required parameters

A. Define the Modules Group

The first group, Modules, will always have a green check when you start AFT Impulse because there are no modules activated by default. No further input is required here.

B. Define the Fluid Properties Group

Next is the Fluid Properties group. This group allows you to specify you fluid properties (density, dynamic viscosity, bulk modulus, and vapor pressure), viscosity model, and fluid corrections. Start by clicking on the Fluid item to open the Fluid panel (see Figure 4).

For models with variable fluid properties, the values for density, viscosity and bulk modulus are default fluid properties. You can then enter different property values, if desired, for any pipe in the Pipe Properties window.

You can model the fluid properties in one of five ways.

1. **User Specified Fluid:** This fluid model allows you to directly type in the density, viscosity, bulk modulus, and vapor pressure.
2. **AFT Standard:** This fluid model accesses fluid data from the AFT Standard library. These fluid properties are either temperature dependent or dependent on the solids concentration. Type in the desired condition (e.g., temperature) and the required properties are calculated. Users can add their own fluids to this library. Custom fluids are created by opening the Fluid Library window from the AFT Impulse Library menu or by clicking the Edit Fluid List button at the bottom of the Fluid panel.

3. **ASME Steam/Water:** As its name implies, this fluid model calculates water properties from the built-in ASME Steam Tables. This model is pressure and temperature dependent.
4. **NIST REFPROP:** This fluid model allows you to select a single fluid or create a mixture of fluids from the REFPROP fluid library. These fluid properties are pressure and temperature dependent.
5. **Chempak Fluid:** This fluid model allows you to select a single fluid or create a mixture of fluids from the Chempak fluid library. These fluid properties are pressure and temperature dependent, although some are temperature dependent only. Chempak is an optional add-on to AFT Impulse.

➢**Select AFT Standard,** by clicking the radio button next to the name. Then choose Water (liquid) from the list and click Add to Model. The properties for AFT Standard water are given only as a function of temperature. Enter 21 deg. C in the temperature box.

➢**Check the groups on the left side of Analysis Setup** and you should now see the second group is checked off.

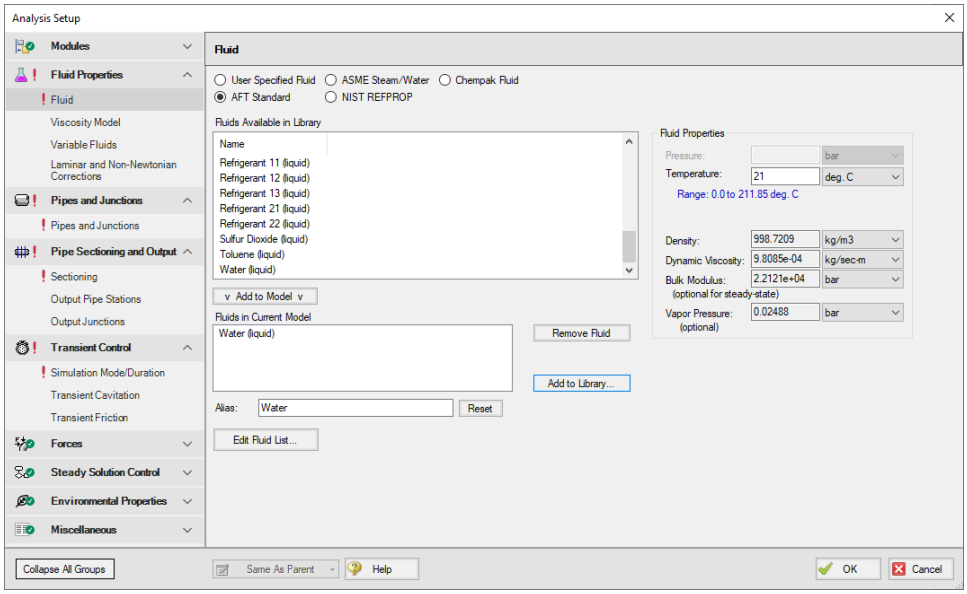


Figure 4: The Fluid panel lets you enter physical properties

On the Viscosity Model panel, users can specify how AFT Impulse will treat viscosity. By default, Newtonian is the viscosity model selected. AFT Impulse offers a variety of non-Newtonian models. However, for this example, the default selection will suffice.

On the Variable Fluids panel, users can specify constant or variable fluid properties. For models with variable fluid properties, the values for density and viscosity are default fluid properties. You can then enter different property values, if desired, for any pipe in the Pipe Properties window. By default, Constant Fluid Properties is selected.

On the Laminar and Non-Newtonian Corrections panel, users can specify the type of corrections that the solver will apply when flow is laminar or if a non-Newtonian viscosity model is selected. The default selections are applicable for most cases, so no input is required for this example.

C. Define the Pipes and Junctions Group

In order to fully define this group, there needs to be pipes and junctions on the Workspace. To lay out the valve closure model, you will place the three reservoir junctions, a valve junction, and a branch junction on the Workspace. Then you will connect the junctions with pipes.

To go back to the Workspace and save the inputs made in Analysis Setup, click OK.

I. Place the first reservoir

➢**To start, drag a reservoir junction from the Toolbox** and drop it on the Workspace. Figure 5 shows the Workspace with one Reservoir.

Objects and ID numbers

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

All pipe and junction objects on the Workspace have an associated ID number. For junctions, this number is, 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 placed on the Workspace will take on the default ID number of 1. You can change this to any desired number greater than zero and up to 99,999.

Editing on the Workspace

Once on the Workspace, junction objects can be moved to new locations and edited with the features on the Edit menu. Cutting, copying, and pasting are all supported. Multiple levels of undo are available for most editing operations.

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.

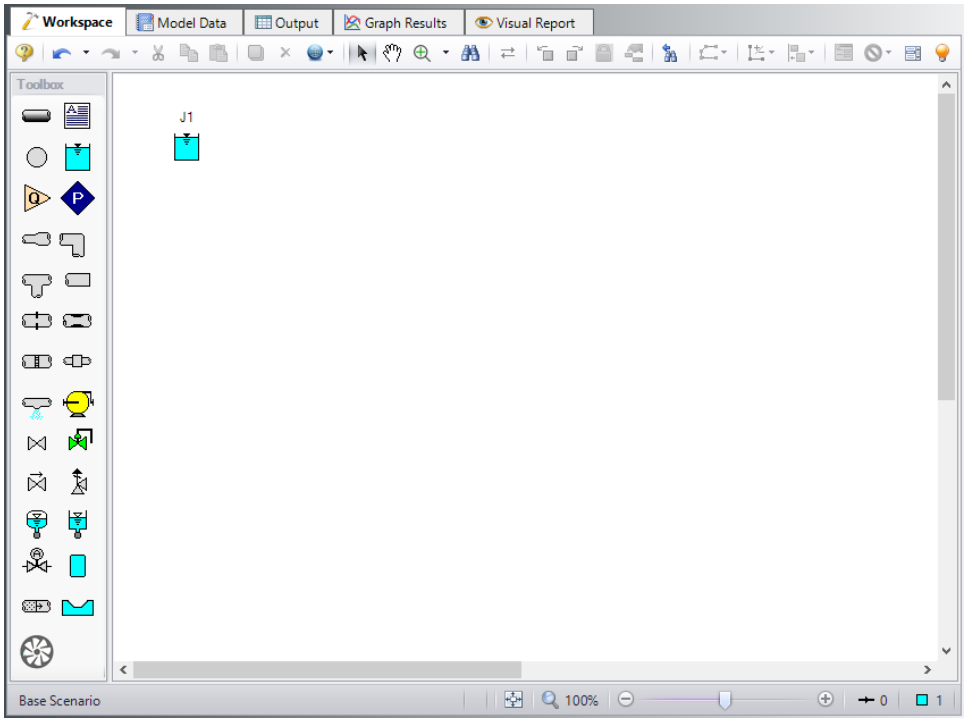


Figure 5: Valve Closure Model with one reservoir placed

II. Place the second reservoir

The second reservoir can be created the same way as the first one or can be derived from the existing reservoir.

➢**To create a second reservoir from the existing one**, select junction J1 by clicking it with the mouse. A red outline will surround the junction to show it is selected. Choose Duplicate from the Edit menu (or use CTRL+D on the keyboard). Your Workspace should appear similar to that shown in Figure 6.

If you like, you can Undo the Duplicate operation and then Redo it to see how these editing features work. To undo an operation, click on the undo button on the Toolbar or choose Undo from the Edit menu. To redo an operation, choose Redo from the Edit menu.

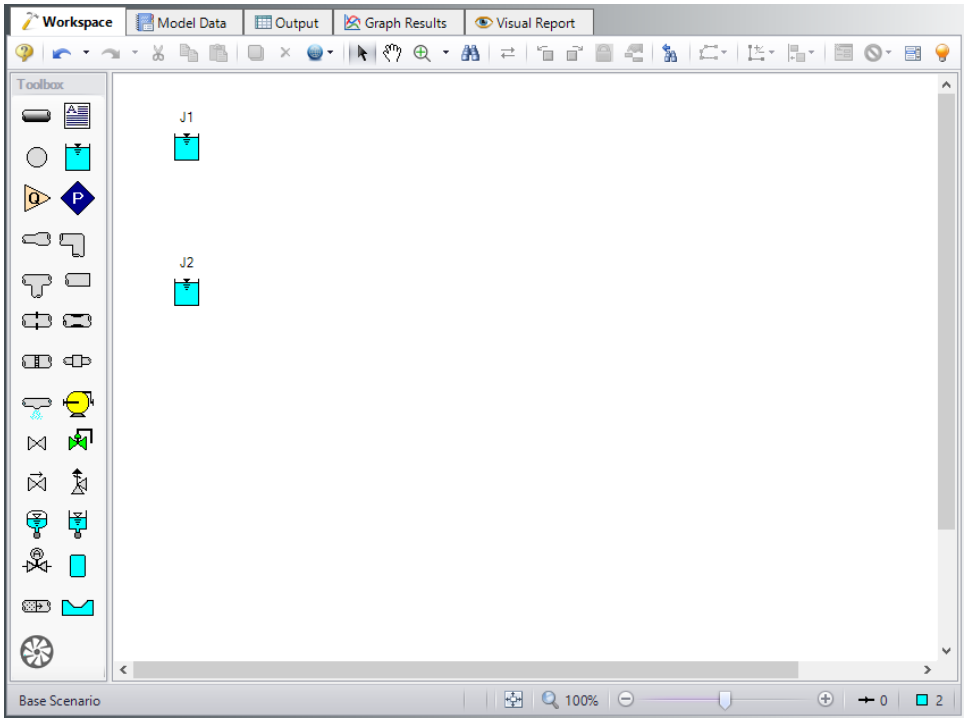


Figure 6: Valve Closure Model with two reservoirs placed

III. Place a Branch junction

➢**To add a Branch junction**, select a Branch from the Toolbox and place it on the Workspace as shown in Figure 7. The Branch will be assigned the default number J3.

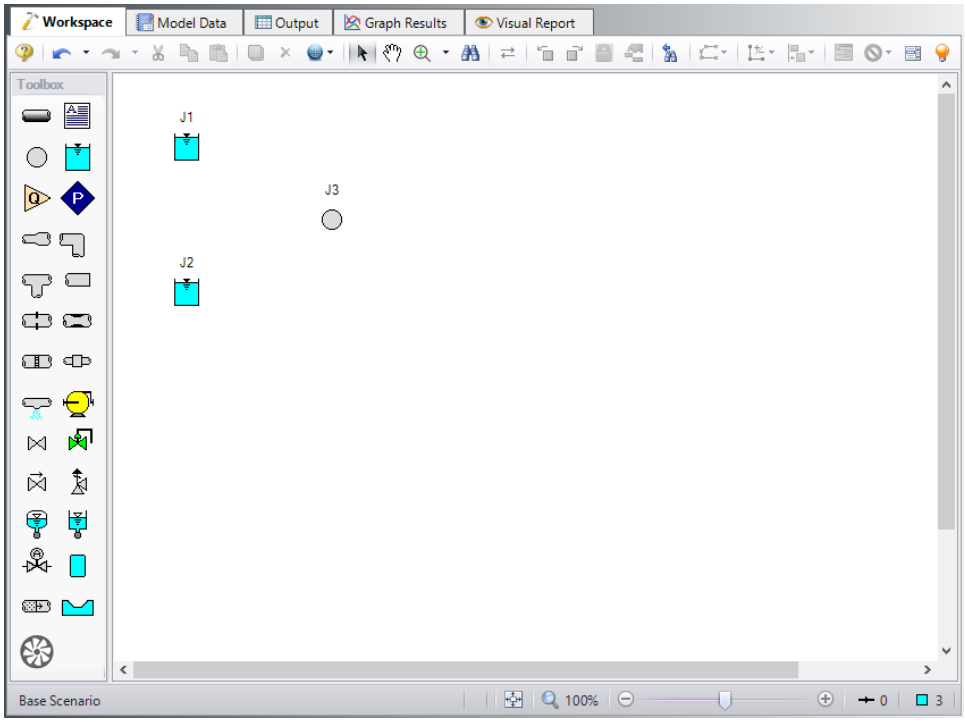


Figure 7: Valve Closure Model with Branch placed

IV. Place a valve and a final reservoir junction

➢**To add a valve junction**, click and drag a Valve junction from the Toolbox and place it on the Workspace as shown in Figure 8. The Valve will be assigned the default number J4. Add a third Reservoir junction, number J5, also shown in Figure 8.

➢**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.

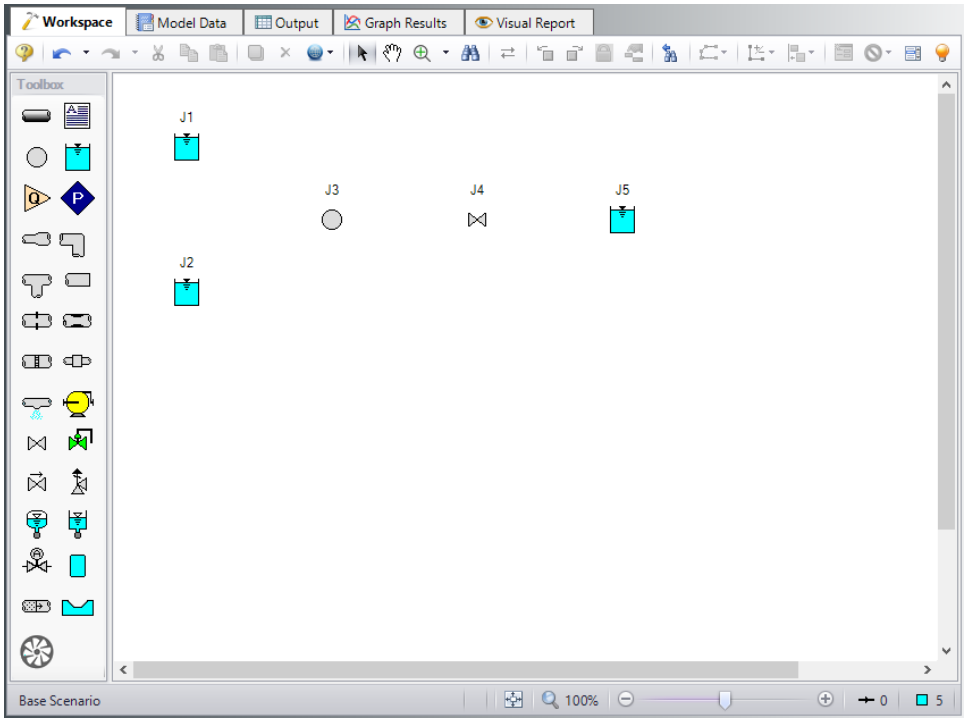


Figure 8: Valve Closure Model with all junctions placed

V. 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. The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe above the junctions, similar to that shown in Figure 9.

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

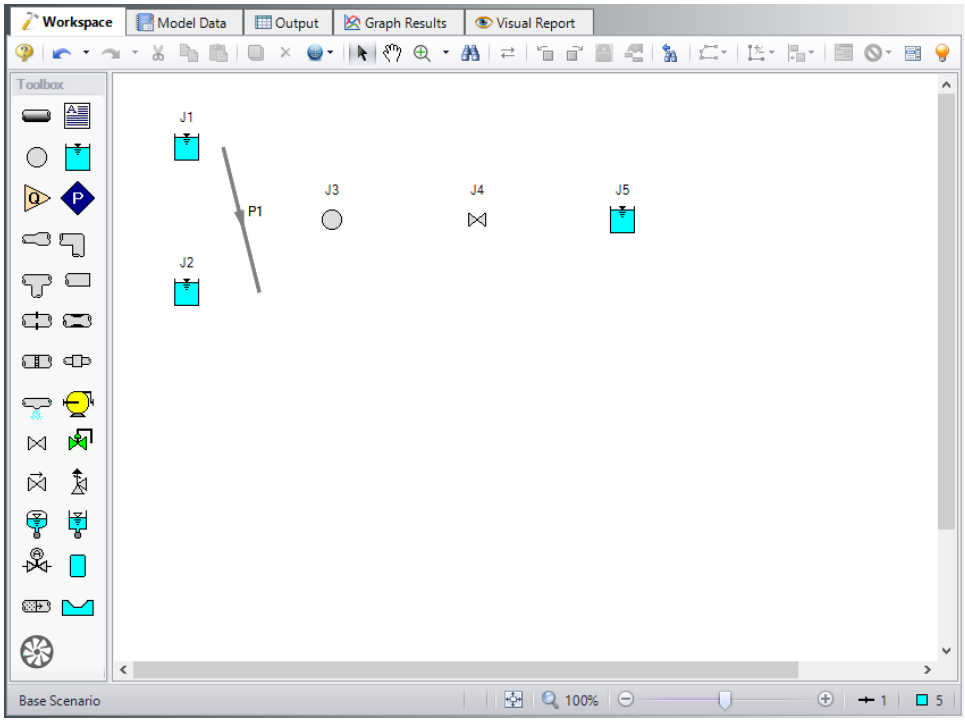


Figure 9: Valve Closure Model with first pipe drawn

➢**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 10). Next, grab the right end-point of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J3 Branch icon (see Figure 11).

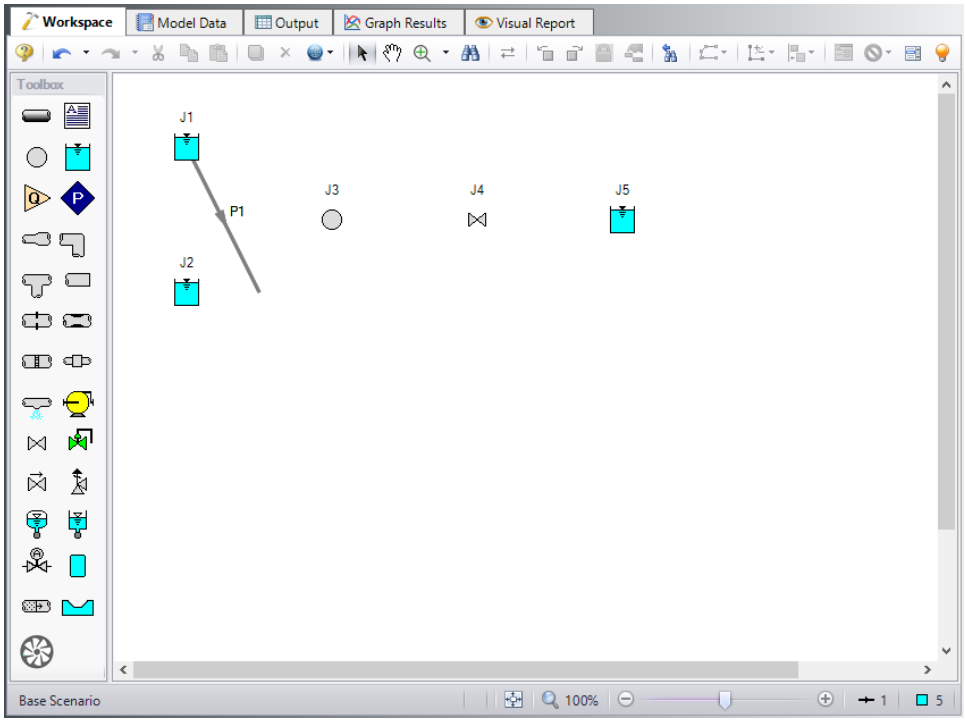


Figure 10: Valve Closure Model with first pipe inlet end connected

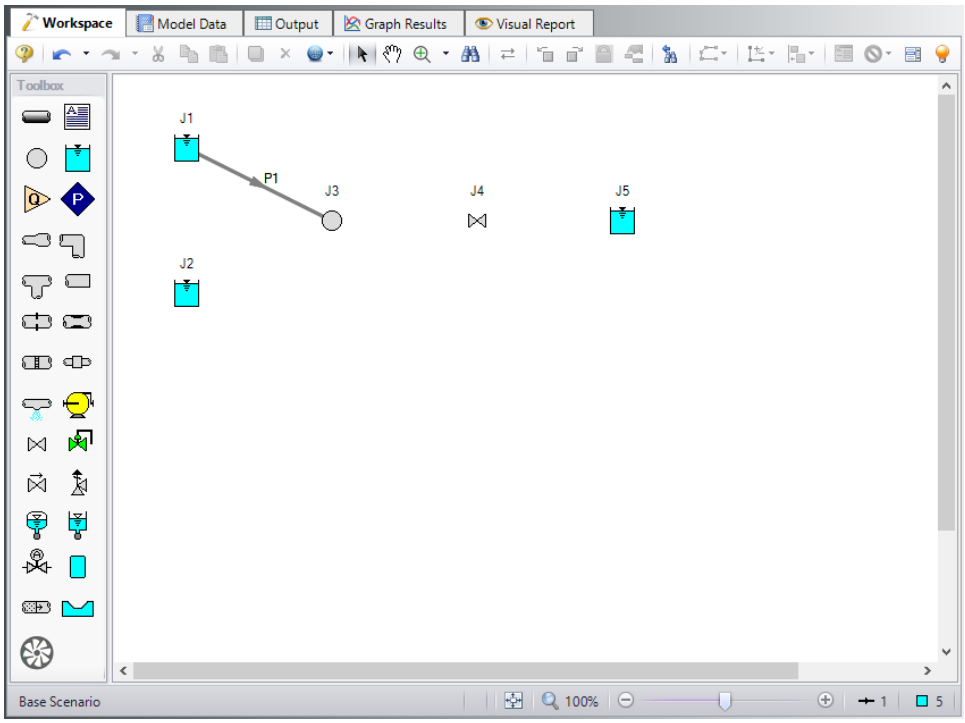


Figure 11: Valve Closure Model with first pipe fully connected

Reference positive flow direction

Located on the pipe is an arrow 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 direction button on the Workspace Toolbar.

The reference positive flow direction indicates which direction is considered positive. If the reference positive direction is the opposite of that obtained by the Solver, the output will show the flow rate as a **negative number**.

VI. 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 cursor on the J2 Reservoir. Press and hold the left mouse button. Stretch the pipe up to the J3 Branch then release the mouse button. Then draw a third pipe from the J3 Branch to the J4 Valve. Finally, draw a fourth pipe from the J4 Valve to the J5 Reservoir. Your model should now look similar to Figure 12.

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

Note: It is generally desirable to lock your objects to the Workspace once they have been placed. This prevents accidental movement and disruption of the connections. Locking also helps prevent accidental deletion of your output once a solution has been completed. You can lock all the objects by choosing Select All from the Edit menu, then selecting Lock Object from the Arrange menu. The Lock Object button on the Workspace Toolbar will appear depressed indicating it is in an enabled state, and will remain so as long as any selected object is locked.

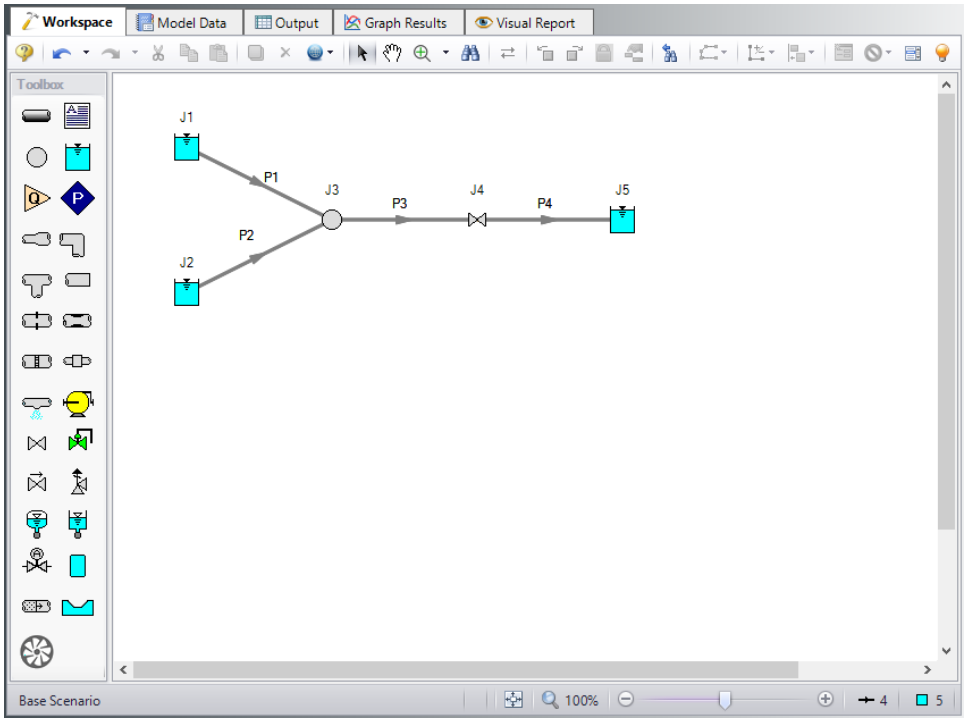


Figure 12: Valve Closure Model with all objects connected

VII. Define the pipes and junctions

To fully define the Pipes and Junctions Group in Analysis Setup, all pipes and junctions must be connected and have the proper input data.

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 floodlight on the Workspace Toolbar (alternatively, you could choose Show Object Status from the View menu). Each time you click the floodlight, 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 model, all the objects' ID numbers will change to red when you turn on Show Object Status.

Undefined Objects window

The Undefined Objects window lists all undefined pipes and junctions and further displays the items that are not yet defined.

VIII. Define Reservoir J1

➢**To define the reservoir J1**, open the J1 Reservoir Properties window (see Figure 13) by double-clicking the J1 icon. By default you will begin on the **Reservoir Model** tab. For Tank Model make sure Infinite Reservoir is selected. Enter a Liquid Surface Elevation of 15 meters. You can assign any unit of length found in the adjacent drop down list box of units.

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

➢**Enter a Liquid Surface Pressure of 0 barG (0 kPa (g))** and with **Pipe Depth** selected, enter 3 meters.

Note: You can specify preferred units for many parameters (such as meters for length) in the User Options from the Tools menu.

You can give the object a name, if desired, by entering it in the **Name** field at the top of the window. In Figure 13, the name of this reservoir is *Supply Tank A*. By default the junction's name is the junction type. The name can be displayed on the Workspace, Visual Report or in the Output.

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

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

The pipe table on the **Pipe Depth & Loss Coefficients** tab allows you to specify entrance and exit loss factors for each pipe connected to the reservoir (in this case there is one). You can enter standard losses by selecting the option buttons at the right. The default selection is the Custom option with loss factors specified as zero. To later change the loss factors, click within the pipe table and enter the loss. You can also specify a depth for the pipe.

The **Transient** tab can be used to specify how the liquid surface elevation changes with time.

The **Optional** tab allows you to enter different types of optional data. You can select whether the junction number, name, or both are displayed on the Workspace. Some junction types also allow you to provide a guess for initial pressure as well as other junction specific data. The junction icon graphic can be changed, as can the size of the icon. Design factors can be entered for most junctions, which are applied to the pressure loss calculations for the junction in order to give additional safety margin to the model.

The **Design Alerts** tab shows the design alerts that are applied to the junction, which ensures that the model is behaving as expected.

The **Notes** tab allows you to enter text describing the junction or documenting any assumptions.

The **Status** tab lists the remaining undefined items for the junction. It should show that All Items Defined.

The highlight feature displays all the required information in the properties window in light blue. The highlight is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window or by pressing the F2 key. The highlight feature can also be turned on or off in the User Options window from the Tools menu. All examples images will be shown with this highlight option turned off.

➢**Click OK.** You may be prompted to display the junction name on the Workspace, if so, click Yes. 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.

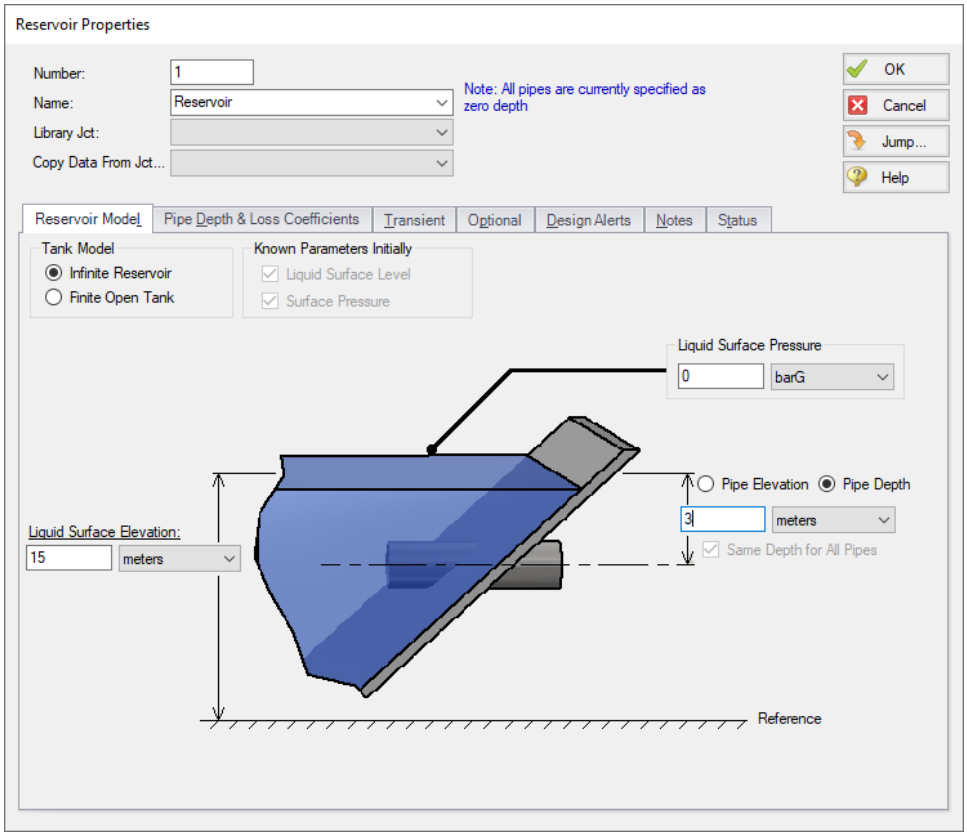


Figure 13: Enter the data for Reservoir J1 in the Reservoir Properties window

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 J1 and hold down the right mouse button. An information box appears, as shown in Figure 14.

Inspecting is a fast way of examining the input (and output if output results are available) for an object.

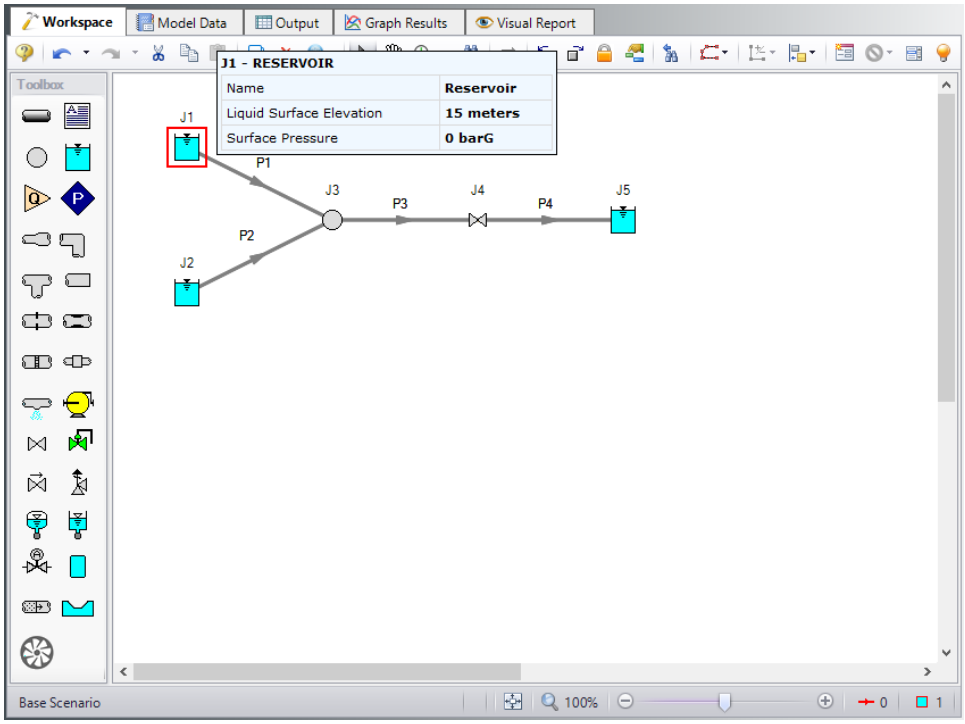


Figure 14: Input data can be inspected from the Workspace by clicking the right mouse button

IX. Define Reservoir J2

➢**Next, open the properties window for reservoir J2** and make sure the Tank Model selection is Infinite Reservoir, then enter a Liquid Surface Elevation of 14 meters, a Liquid Surface Pressure of 0 barG (0 kPa (g)), and with Pipe Depth selected, enter 3 meters. Give the reservoir the name *Supply Tank B*. After you enter the data and click OK you should see the ID number on the Workspace turn black, because the status has changed to defined.

X. Define Branch J3

➢**Open the properties window for branch J3** (see Figure 15). In this window, all three connecting pipes should be displayed in the pipe table area. You could associate loss factors with each pipe by clicking within the pipe table and entering the data. Enter an elevation of 0 meters for the J3 Branch junction (an elevation must be defined for all junctions). You could also click the Optional tab then specify a flow source or sink at the junction; for now leave this as zero. Click OK.

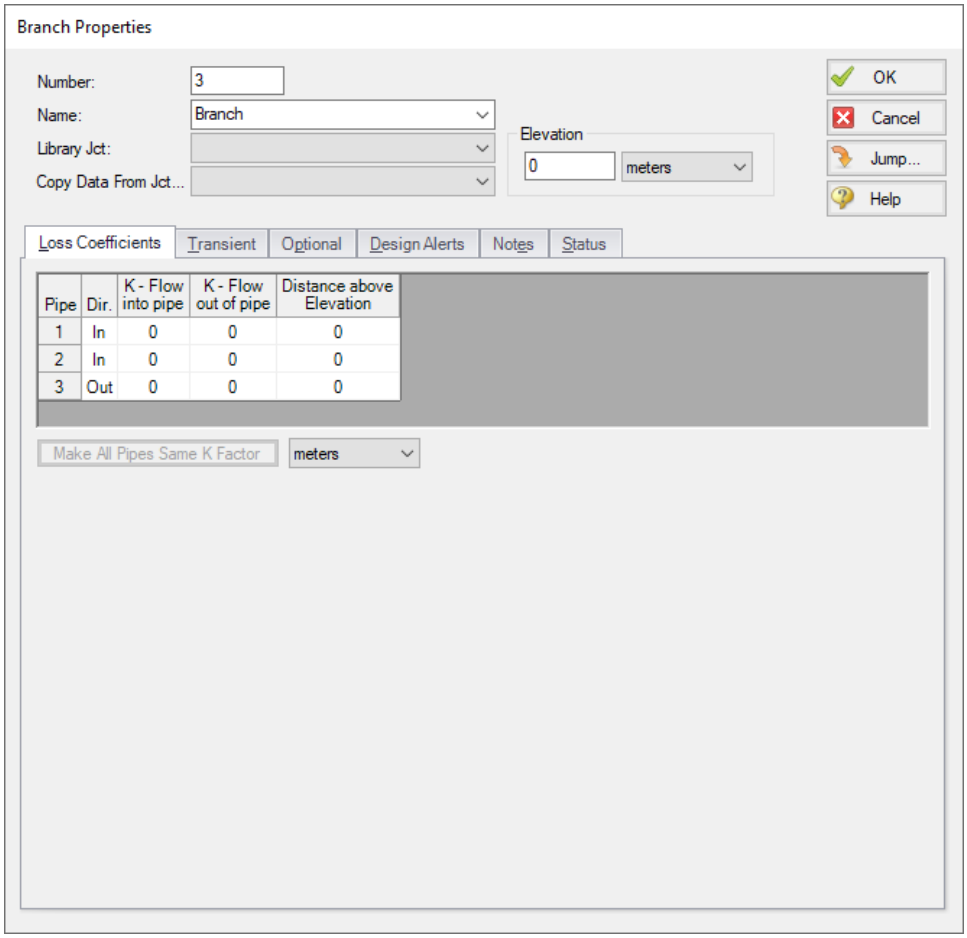


Figure 15: Enter the data for Branch J3 in the Branch Properties window

XI. Define Valve J4

➢**Open the properties window for valve J4** (see Figure 16) and enter an Inlet Elevation of 0 meters.

➢**On the Loss Model tab, make sure the Valve Data Source has User Specified selected.**

➢**For Loss Model, make sure Cv is selected** and enter a Cv value of 1000. This represents the valve's Cv during steady-state.

➢**Click the Transient tab** and enter the data below for Cv (see Figure 17).

Time (seconds)	Cv
0	1000
0.4	400
0.8	100

Time (seconds)	Cv
1	0
2	0

The first data point (Cv = 1000 at time zero) must match the steady-state value. The transient data represents the valve initially open. The valve then gradually closes over a period of one second, and stays closed.

The J4 Valve is the element which causes the transient in this model. The purpose of the model will be to understand how high the pressures can rise during the transient.

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

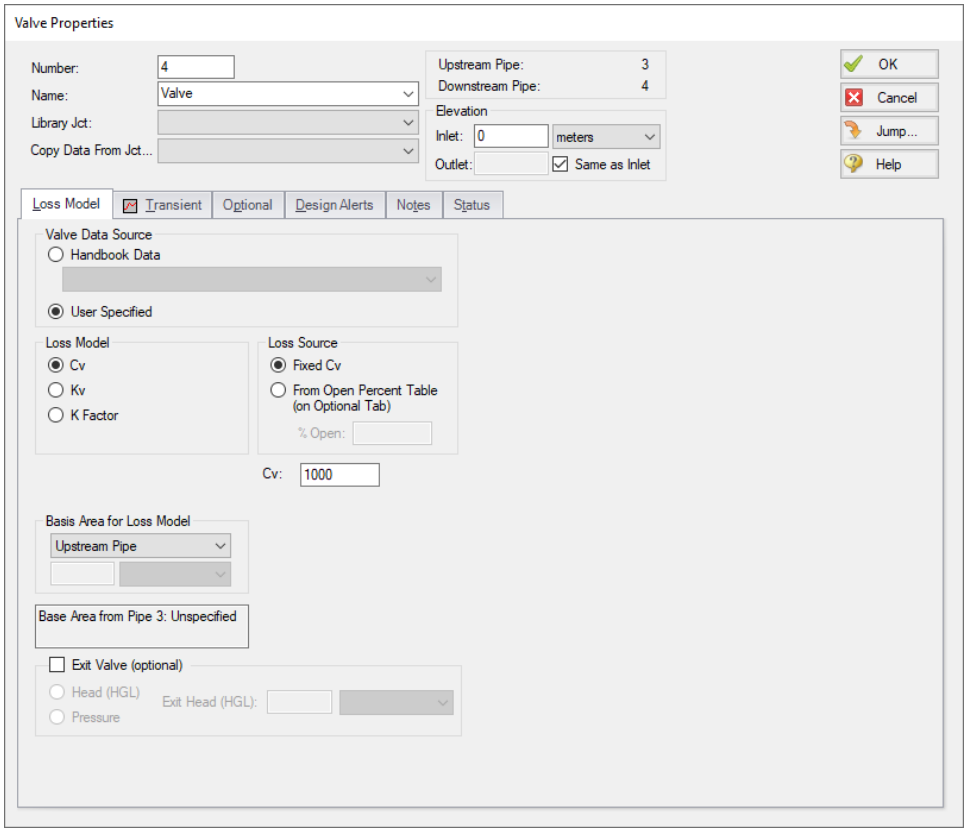


Figure 16: Enter the data for Valve J4 in the Valve Properties window

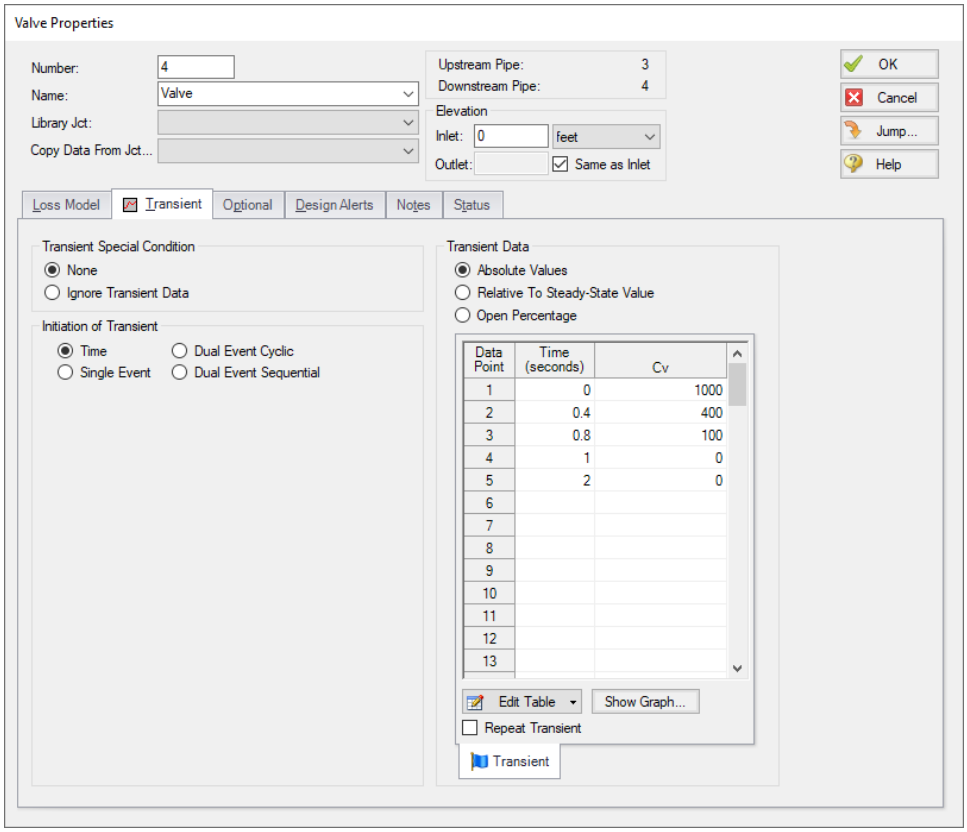


Figure 17: Transient data is entered on the Transient tab on junction Properties windows

XII. Define reservoir J5

➢**Finally, open the properties window for reservoir J5** and make sure the Tank Model selection is Infinite Reservoir, then enter a Liquid Surface Elevation of 1.5 meters, a Liquid Surface Pressure of 0 barG (0kPa (g)), and with Pipe Depth selected, enter 1.5 meters. Give the reservoir the name *Discharge Tank*. Click OK

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

XIII. Define pipe P1

The next step is to define all the pipes. To open the Pipe Properties window, double click the pipe on the Workspace.

➢**First open the Pipe Properties window for Pipe P1** (see Figure 18). For Pipe P1, enter a length of 60 meters and select the Pipe Material as Steel - ANSI, Size of 10 inch, and make sure the Type is STD (schedule 40).

The wavespeed is a very important parameter in a 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 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, Poisson Ratio are built into the pipe material libraries supplied with AFT Impulse, and is automatically obtained when the pipe material, size, and type are chosen. The calculated wavespeed is 1319 meters/sec.

The Pipe Properties window offers control over all important flow system parameters that are related to pipes. Click OK.

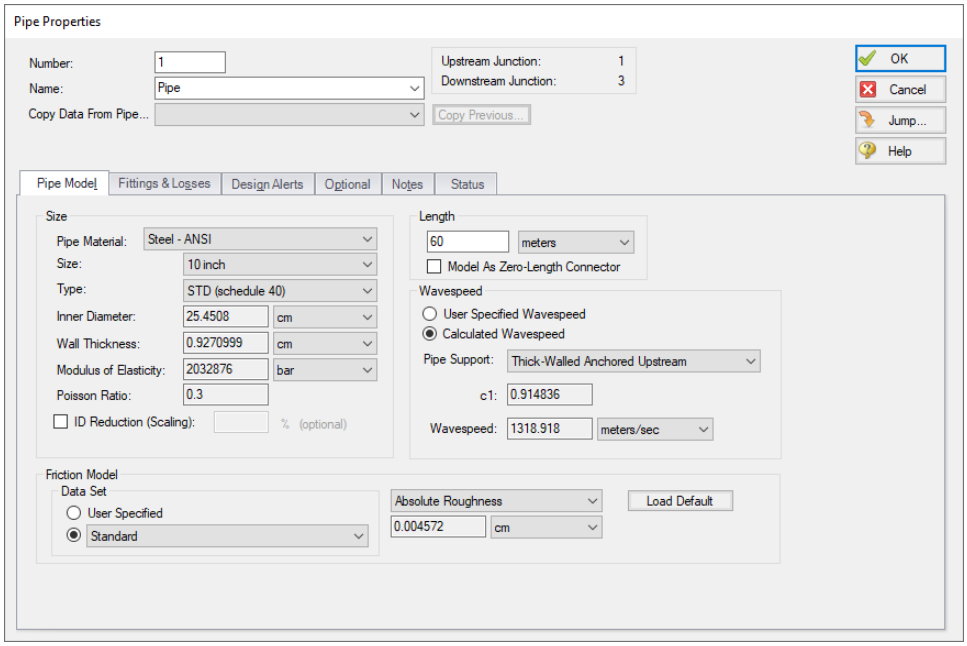


Figure 18: Enter the data for Pipe P1 in the Pipe Properties window

The Inspect feature also works within the Pipe Properties window. To inspect a connected junction, position the mouse pointer on the connected junction's ID number and hold down the right mouse button. This is helpful when you want to quickly check the properties of connecting objects. (You can also use this feature in junction Properties windows for checking connected pipe properties.)

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

XIV. Define pipes P2, P3, and P4

Define pipes P2 - P4 as Steel - ANSI using the sizes and lengths shown below. All pipes should use Thick-Walled Anchored Upstream for the Pipe Support model.

The data for the pipes should be as follows:

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

Pipe	Size (inches)	Length (meters)
2	10	45
3	12	15
4	12	12

After entering the data for all the pipes, the Pipes and Junctions Group should be completed. If it is not, see if the Show Object Status is on. If it is not turned on, select Show Object Status from the View menu or Workspace Toolbar. If the Pipes and Junctions Group in Analysis Setup does not have a green check-mark next to it, see if any of the pipes or junctions have their number displayed in red. If so, you did not enter all the data for that item.

➢**Before running the model, save it 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 19. To change to this window, you can select it from the Primary Window tabs or the Window menu, or by pressing 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, or printed out for review. Figure 20 shows an expanded view of the Transient Data tab from Figure 19. Here all transient input data for the model is shown.

The Model Data window allows access to all Properties windows by double-clicking anywhere in the row associated with the appropriate ID number in the far left column of the table. You may want to try this right now.

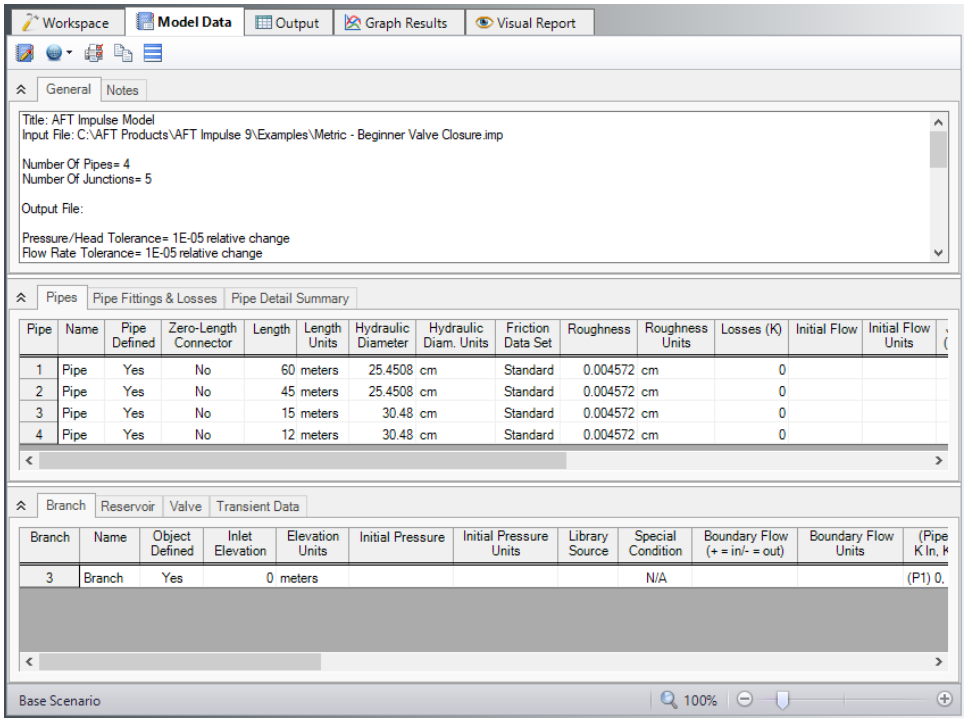


Figure 19: The Model Data window displays pipe and junction data in text format

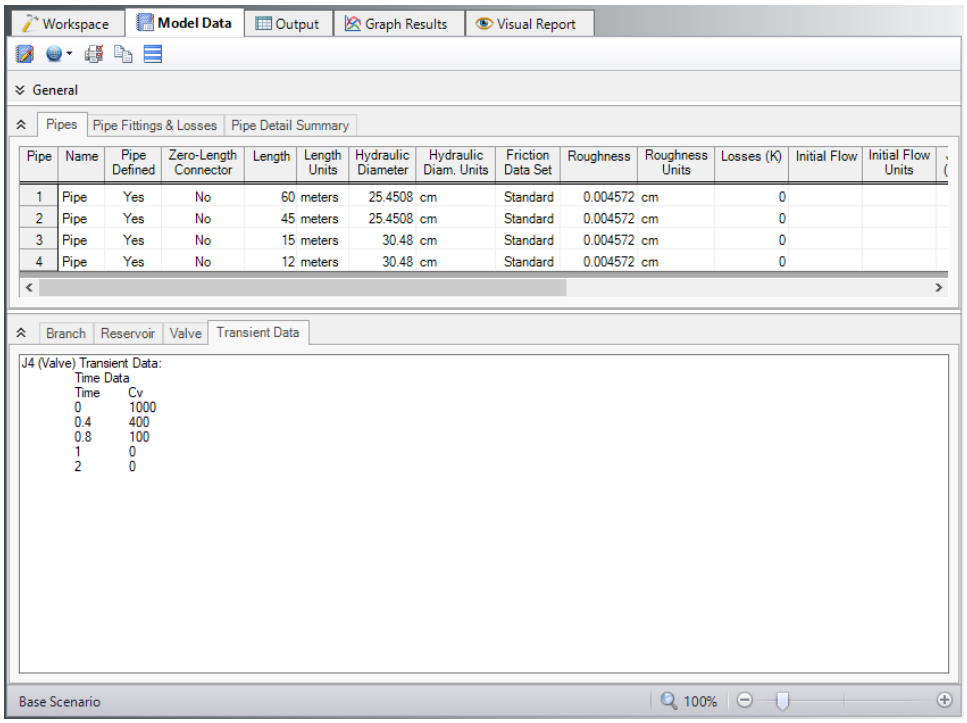


Figure 20: The Transient data tab in the Model Data window displays transient input data

D. Define the Pipe Sectioning and Output Group

After completing the first three Analysis Setup Groups, sufficient information exists to run the model in steady-state. The next two groups are for transient modeling. To run the model in steady-state, open the Analysis menu, select Time Simulation, then choose Steady Only. By doing 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.

The Pipe Sectioning and Output group allows the user to define pipe sectioning and how much information to save to the output file.

I. Define the Sectioning Panel

The only item in the Pipe Sectioning and Output Group in Analysis Setup that needs user input is the Sectioning panel. This panel cannot be defined until the Fluid Properties group and Pipes and Junctions group are defined. The Sectioning panel divides the pipes into computation sections in a manner which is consistent with the Method of Characteristics (MOC).

➢**Open the Sectioning panel** in Analysis Setup (Figure 21). The pipes will automatically be sectioned without user input. 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 *Dt* 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 commonly acceptable to allow up to a 15% uncertainty in wavespeed, though AFT Impulse by default only allows up to 10% variance. 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.

You can customize the Section Pipes search criteria by clicking the arrow next to Advanced Settings. In general, the default search criteria will be sufficient for adequately sectioning the pipes throughout your model.

In this example, make sure the second row that will use two pipe sections in the controlling pipe is selected. The Sectioning Panel and Pipe Sectioning and Output group should now have a green checkmark indicating that they are fully defined.

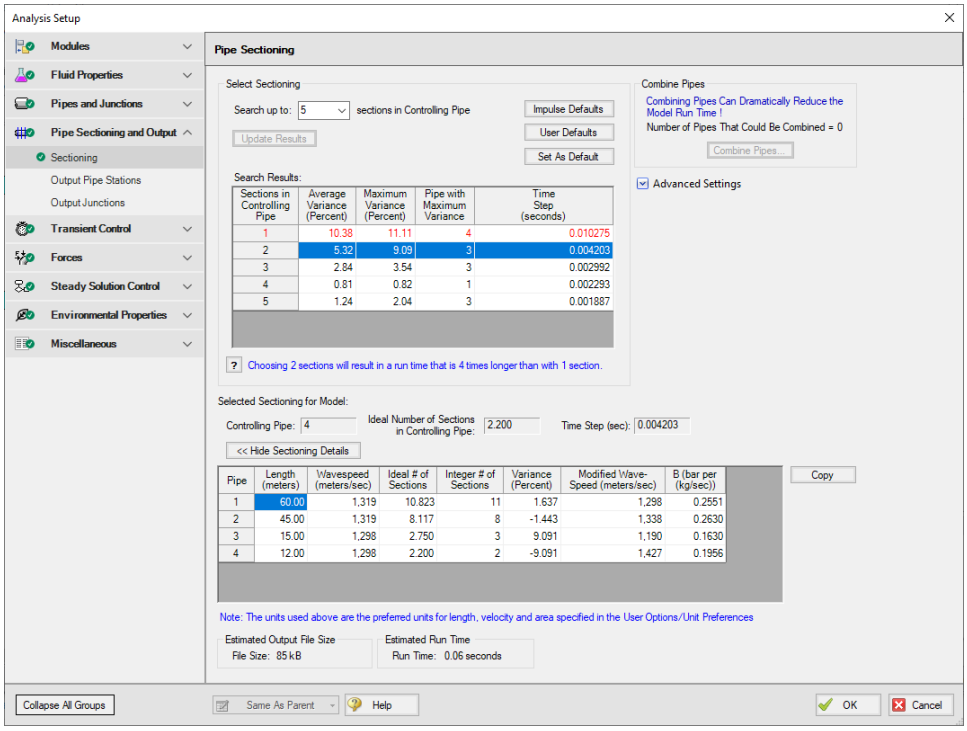


Figure 21: The Sectioning Panel automates the sectioning process and calculates the time step

E. Define the Transient Control Group

The final Analysis Setup group to define is the Transient Control Group. This group allows you to specify the time at which the transient starts and ends, the cavitation model used, and the friction method used.

➢**Open the Simulation Mode/Duration panel** in Analysis Setup (Figure 22). Zero (0) seconds should already be entered for Start Time. Enter two (2) for Stop Time.

The Transient Cavitation panel allows you to enable or disable transient cavitation modeling. It is advised to keep Model Transient Cavitation checked and only disable it for troubleshooting purposes.

Note: The Artificial Transient Detection panel in the Miscellaneous group offers control over how AFT Impulse should respond to artificial transients. Artificial transients are a problem that can sometimes occur when steady-state and initial transient conditions are inconsistent.

At the bottom of the panel 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 77 kB. If the output file does become excessively large, you will want to limit the number of time steps and pipe output written to the output file. The estimated file size and run time may vary on different computers.

➢**Click OK** to accept the current settings now that all groups are defined. The model is ready to be solved.

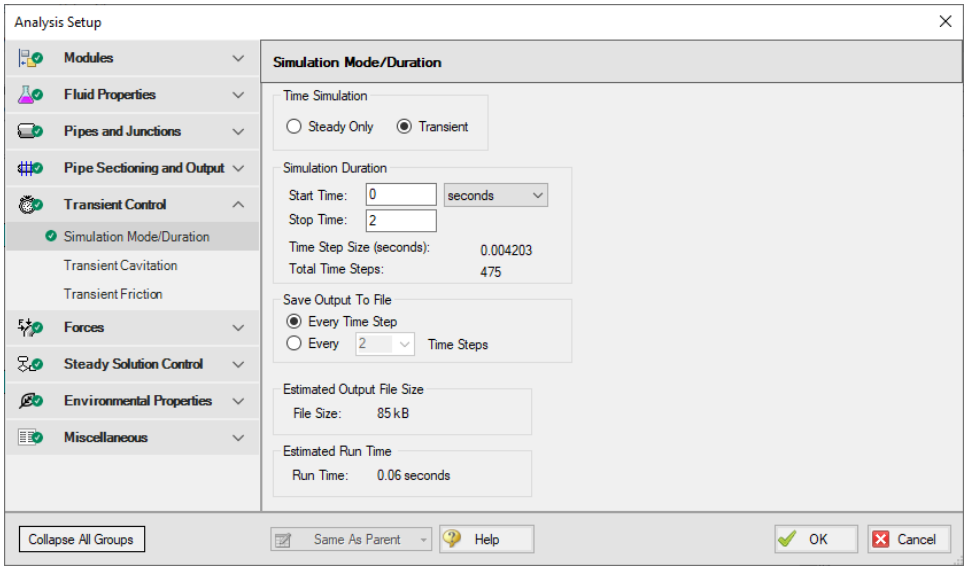


Figure 22: The Simulation Mode/Duration panel offers features to specify the time span for the transient and what output data is written

Step 3. Run the Solver

➢**Click Run Model from the toolbar** or from the Analysis menu. During execution, the Solution Progress window displays (Figure 23). You can use this window to pause or cancel the Solver's activity.

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** 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 the printer.

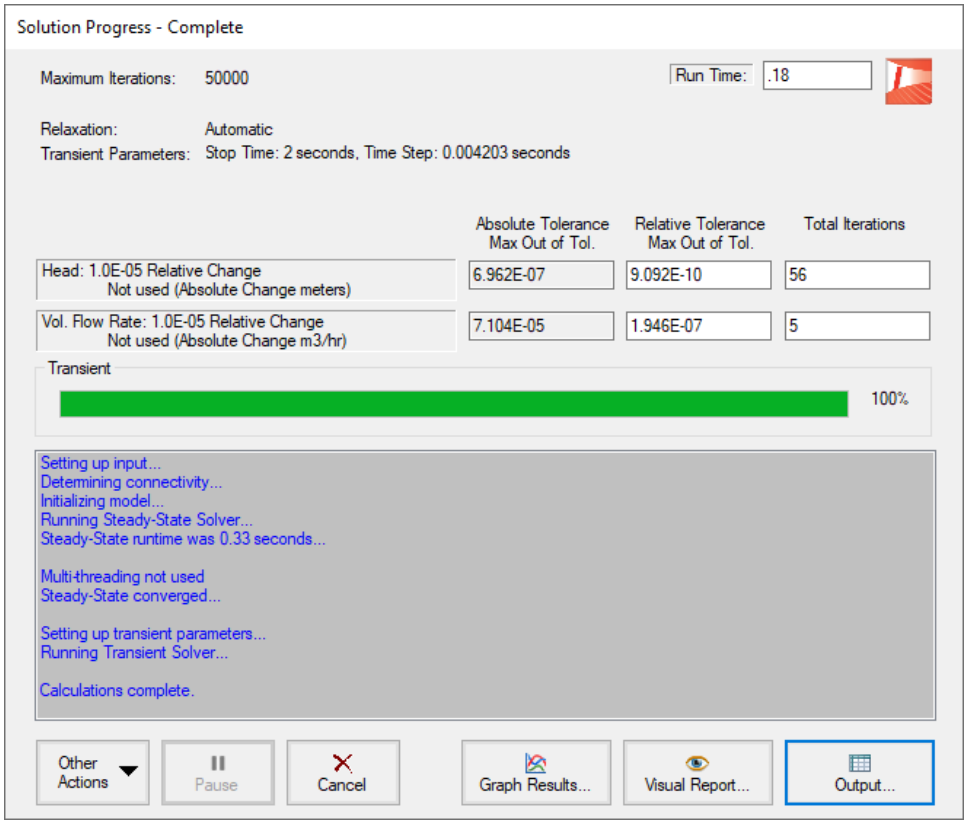


Figure 23: The Solution Progress window displays the state of the simulation

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 model input 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 4. Examine the Output

The Output window (Figure 24) is similar in structure to the Model Data window. Three areas are shown, and you can minimize or enlarge each section by clicking the arrow next to the General, Pipes, and All Junctions tabs or from the View menu. The parameters displayed in the tables can be customized with the Output Control window, from the Tools drop-down menu.

The Output window allows you to review both the steady-state and transient results. You can review the solutions for each time step (i.e., a time history) for which data was written to file. Also, a summary of the

maximum and minimum transient results for each computing station is given on the Transient Max/Min tab in the pipe area. These two data sets are located on the Transient Output tab and Transient Max/Min tab in the pipe area of the Output window (see Figure 25 and Figure 26). Note that in order to display all pipe stations as shown in Figure 25, the model will need to be set to save data for all stations. By default, AFT Impulse will only save the inlet and outlet data points for each pipe. This can be changed on the Output Pipe Stations Panel in the Pipe Sectioning and Output Group in Analysis Setup.

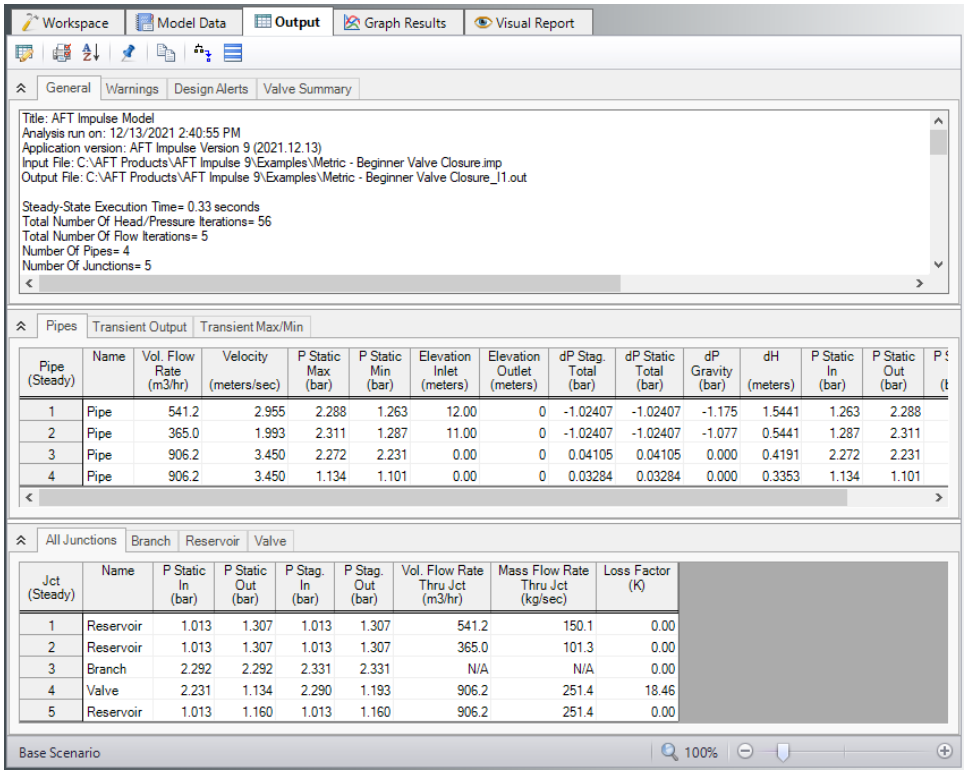
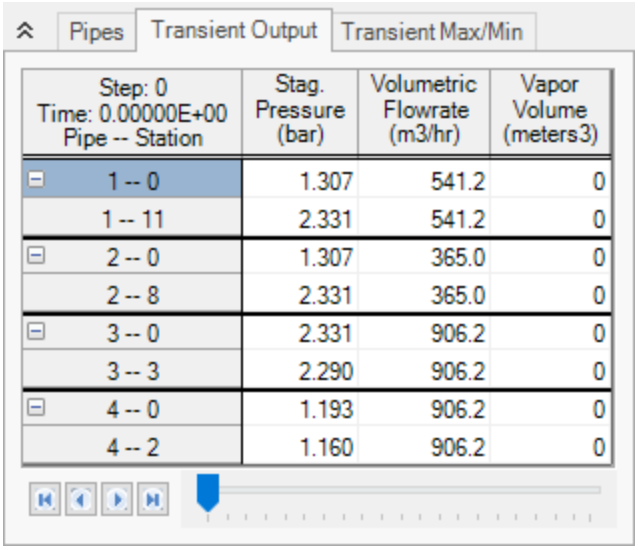
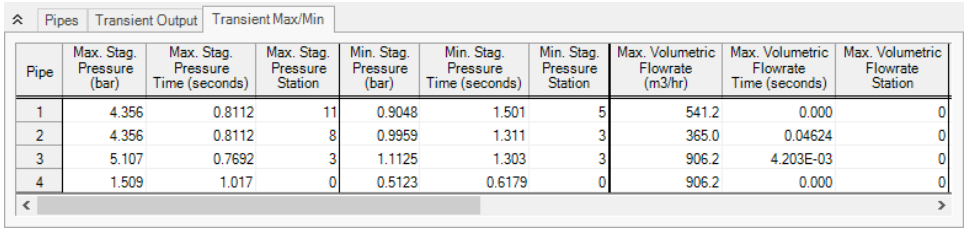


Figure 24: The Output window displays steady and transient output in text form



Step: 0 Time: 0.00000E+00 Pipe -- Station	Stag. Pressure (bar)	Volumetric Flowrate (m3/hr)	Vapor Volume (meters3)
1 -- 0	1.307	541.2	0
1 -- 11	2.331	541.2	0
2 -- 0	1.307	365.0	0
2 -- 8	2.331	365.0	0
3 -- 0	2.331	906.2	0
3 -- 3	2.290	906.2	0
4 -- 0	1.193	906.2	0
4 -- 2	1.160	906.2	0

Figure 25: The Output window displays transient data for each time step



Pipe	Max. Stag. Pressure (bar)	Max. Stag. Pressure Time (seconds)	Max. Stag. Pressure Station	Min. Stag. Pressure (bar)	Min. Stag. Pressure Time (seconds)	Min. Stag. Pressure Station	Max. Volumetric Flowrate (m3/hr)	Max. Volumetric Flowrate Time (seconds)	Max. Volumetric Flowrate Station
1	4.356	0.8112	11	0.9048	1.501	5	541.2	0.000	0
2	4.356	0.8112	8	0.9959	1.311	3	365.0	0.04624	0
3	5.107	0.7692	3	1.1125	1.303	3	906.2	4.203E-03	0
4	1.509	1.017	0	0.5123	0.6179	0	906.2	0.000	0

Figure 26: The Output window displays maximum and minimum transient data

A. Modify the output format

If you selected the default AFT Impulse Output Control, the Pipes table of steady-state results (the tab on the far left in the pipe area) will show volumetric flow rate in the second column with units of m3/hr (cubic meters per hour).

➢**Select Output Control from the Tools menu or the main Toolbar.** On the right side of the Pipes section is the list of currently selected output parameters. Click Volumetric Flow Rate and change the units by clicking the arrow beside the units, and then selecting m3/sec (cubic meters per second) from the drop-down list.

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

➢**Select Output Control from the Tools menu** one more time. The Reorder scroll bar on the far right allows you to reorder parameters in the list. You may also reorder parameters by dragging and dropping the icon just to the left of each parameter within the list of currently selected parameters.

➢**Select the Velocity parameter** and use the Reorder scroll bar to move it up to the top of the parameter list.

➢**Click OK to display the changes to the current results.** You will see in the Pipes table that the first column now contains velocity and the third column contains the volumetric flow rate. The Output Control window allows you to obtain the parameters, units and order you prefer in your output. This flexibility will help you work with AFT Impulse in the way that is most meaningful to you, reducing the possibility of errors.

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

B. Graph the results

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

➢**Browse to the Graph Results window** by clicking the Graph Results tab, choosing it from the Windows menu, or by pressing CTRL+G. The Graph Results window offers full-featured Windows plot preparation.

The Graph Parameters menu will automatically be displayed in the Quick Access Panel on the far right of the Graph Results Window, and be where you are to specify which graphs to generate. On the Transient Pipe tab under Select Pipe Stations, expand P3 pipe stations and double-click Outlet, which is the pipe computing station at the valve inlet. Also add the inlet of pipe P4 which is the valve exit. Select the Graph Parameter as Pressure Static and set the units to barG (see Figure 27).

➢**Click Generate.** The graph shows the static pressure at the valve inlet and outlet over the duration of the simulation (See Figure 28).

You can use the other buttons in the Graph Results window to change the graph appearance and to save and import data for cross-plotting. 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.

Note that the graph guide, located at the top right of the Graph Results window and represented with the What Would You Like to Do? icon, can guide you through the development of your graph. This feature can be hidden by clicking on the icon.

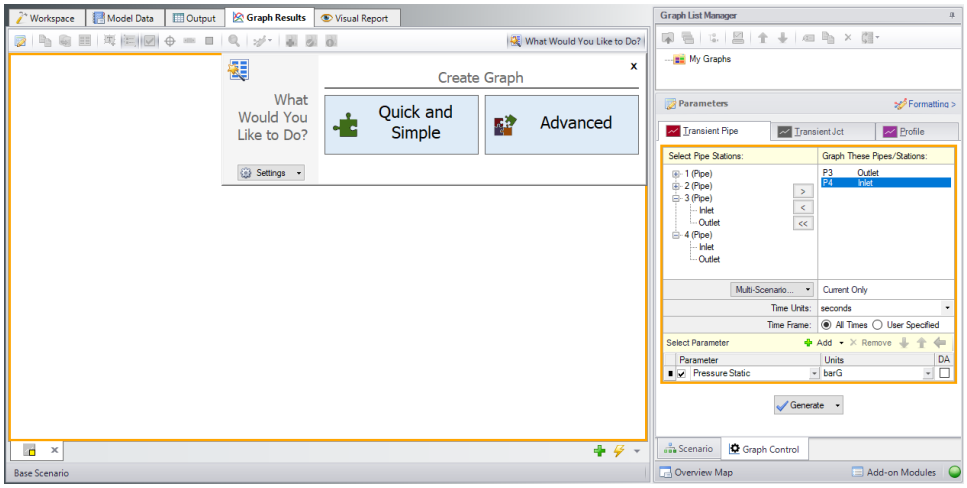


Figure 27: The Select Graph Parameters window controls the Graph Results content

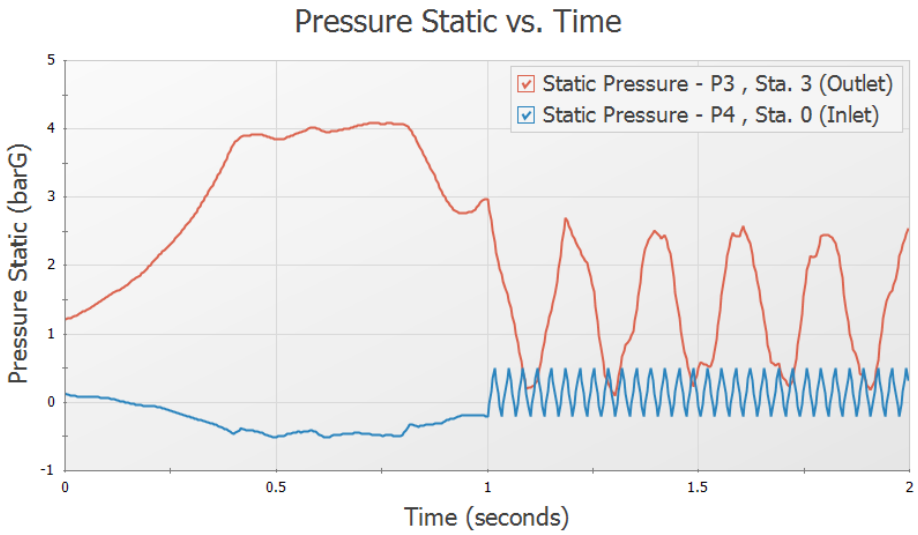


Figure 28: The Graph Results window offers full-featured plot generation. Here the static pressure at valve J4 over time is shown

Further review of the valve graph results in Figure 28 shows that at time zero the difference between the curves is about 1.1 bar (110 kPa). This is the steady-state pressure difference across the valve, which can also be found in the Output window to be 1.097 bar (109.7 kPa).

As time increases one sees that the pressure drop across the valve increases as it closes. Finally 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.

In the Graph Parameters window on the Quick Access Panel click (Clear button). Expand Pipe 1 and select the Inlet of Pipe 1 by double-clicking or clicking the Add button. This is the pipe computing

station at the reservoir J1. Also add the inlet of Pipe 2 which is the reservoir J2. Select the Graph Parameter as Volumetric Flowrate Upstream and set the units to m3/hr. Click Generate. A graph of these stations shows the flow rates out of the two supply reservoirs. Note that after the valve closes, the fluid in reservoir J1 flows to reservoir J2 (see Figure 29).

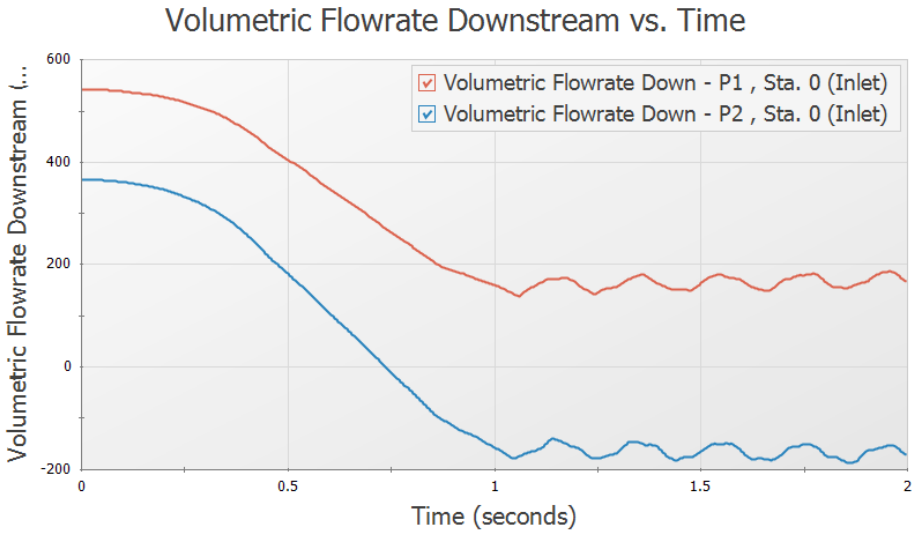


Figure 29: Transient flow rates out of the reservoirs

C. View the Visual Report

➢**Change to the Visual Report window** by choosing it from the Window menu, clicking the Visual Report tab on the toolbar, or by pressing CTRL+I. This window allows you to integrate your text results with the graphic layout of your pipe network. The Visual Report can also animate the transient pipe results in a color animation overlaid on the model.

➢**The Visual Report Control window should open automatically** shown in Figure 30. Default parameters are already selected, but you can modify these as desired. For now, in the Pipe Transient Output area, select Max Pressure Stagnation and Min Pressure Stagnation. Click the Show button. The Visual Report window graphic is generated (see Figure 30).

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 (this has already been done in Figure 31). This window can be printed or copied to the clipboard for import into other Windows graphics programs.

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 under Display Mode at the top of the Visual Report Control window (See Figure 30). 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 pressure stagnation can be seen in Figure 32.

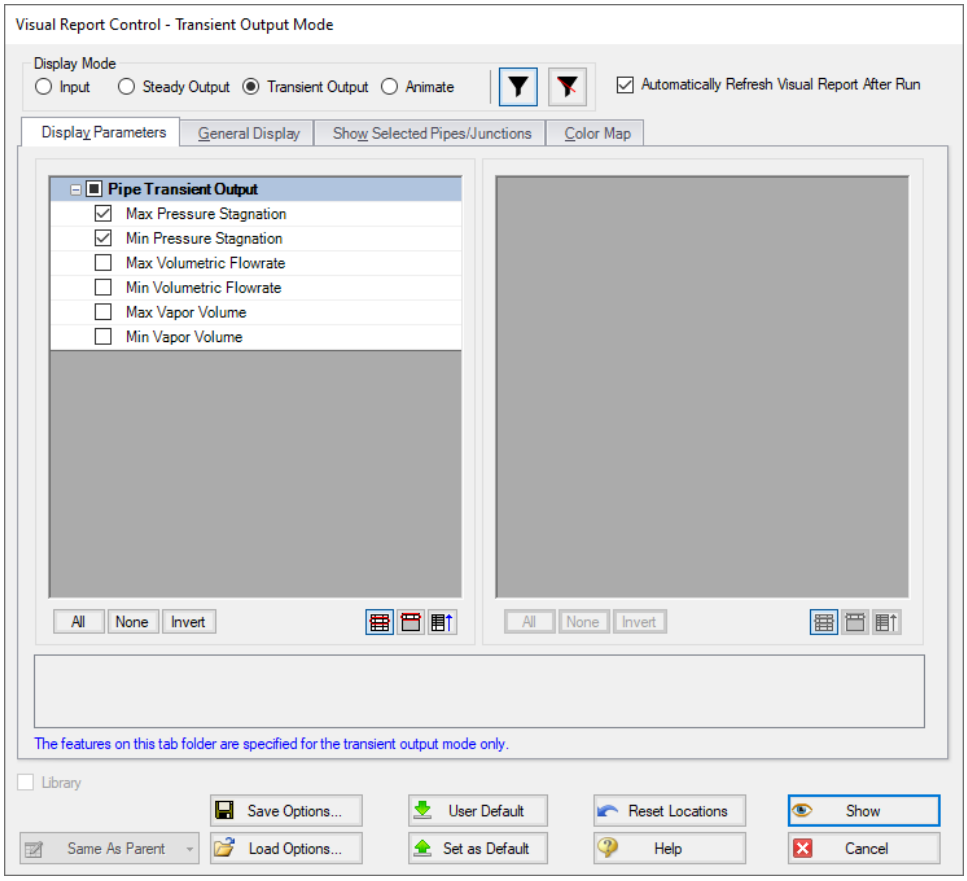


Figure 30: The Visual Report Control window specifies what data to show on the Visual Report window

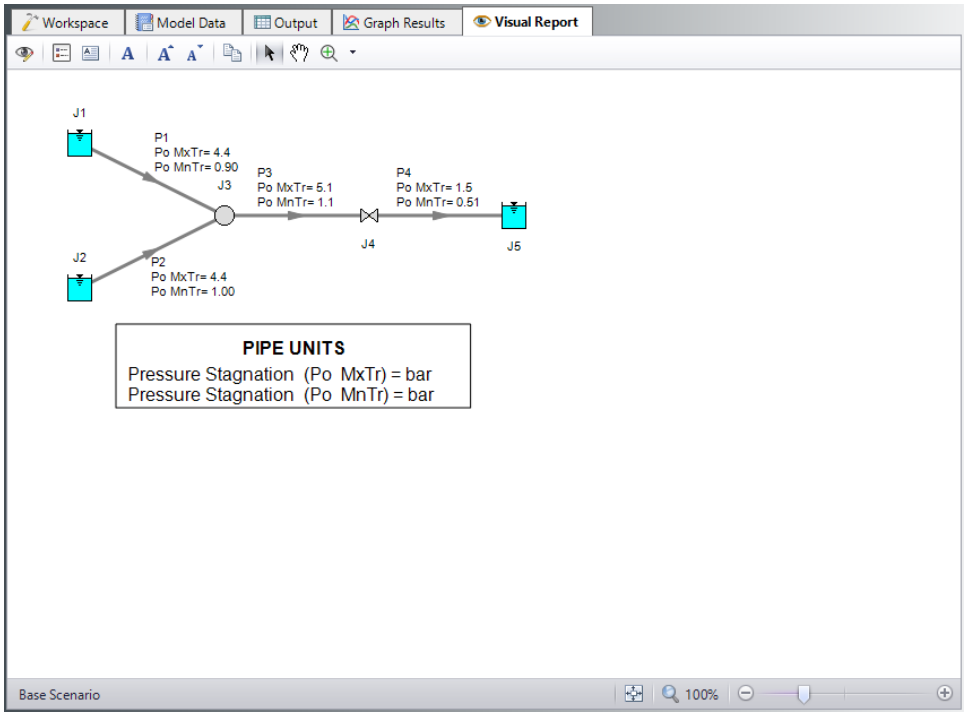


Figure 31: The Visual Report window displays output data on the input schematic. It also can operate in Input Mode where it displays input data

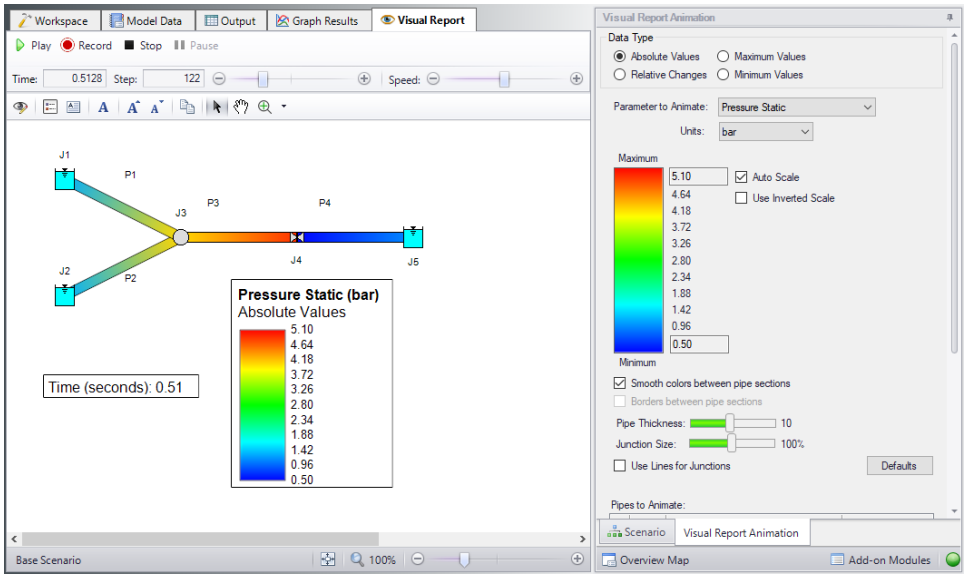


Figure 32: The Visual Report provides an animation feature as an alternate way to view the transient results

Conclusion

You have now used AFT Impulse's five Primary Windows to build and analyze a simple waterhammer model.

Ammonia Transfer with Cavitation

Summary

A ship carrying ammonia in a pressurized holding tank transfers the fluid to an onshore holding tank. The valve controlling flow to the holding tank must be closed as quickly as possible. Three different closing times are to be evaluated, and the resulting maximum pressures need to be determined.

Topics Covered

- Observing effects of varying valve closing time
- Evaluating systems with transient cavitation
- Using Scenario Manager

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Ammonia Transfer with Cavitation.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard
b. Fluid = Ammonia (liquid)
    i. After selecting, click Add to Model
c. Temperature = 24 deg. C

This calculates the fluid properties to use in the model. Notice the high vapor pressure of 9.72 bar (972 kPa). By changing the temperature to 21 or 27 deg. C, it can be seen that vapor pressure is very sensitive to temperature. In such a case, it would be worthwhile to find out if 24 deg. C is the highest expected temperature so that the analysis can be run with the highest potential vapor pressure.

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

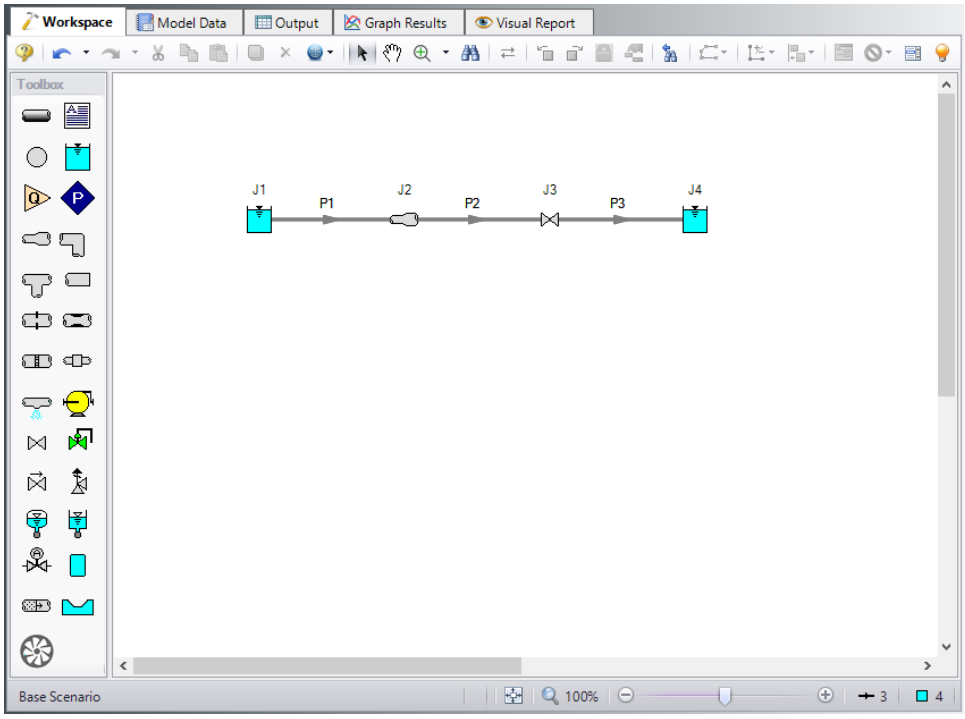


Figure 1: Layout of ammonia transfer system

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe P1
    a. Pipe Material = Steel - ANSI
    b. Size = Use table below
    c. Type = STD (schedule 40)
    d. Friction Model Data Set = Standard
    e. Length = Use table below

Pipe	Size (inches)	Length (meters)
1	8	30
2	10	90
3	10	46

Junction Properties

1. Reservoir J1
    a. Name = Ship - Pressurized Tank
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 12 meters
    d. Liquid Surface Pressure = 17 barG (1700 kPa(g))
    e. Pipe Depth = 12 meters
2. Area Change J2
    a. Inlet Elevation = 0 meters
    b. Type = Abrupt Transition (Cylindrical)
3. Valve J3
    a. Loss Model tab
    b. Inlet Elevation = 0 meters
    c. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Loss Source = Fixed Cv
        iv. Cv = 1000
    d. Transient tab
        i. Transient Special Conditions = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	1000
0.5	0

4. Reservoir J4
    a. Name = Onshore Holding Tank
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 6 meters
    d. Liquid Surface Pressure = 17 barG (1700 kPa(g))
    e. Pipe Depth = 1.5 meters

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and navigate to the Sectioning panel.** When the Sectioning panel is first opened it will automatically search for the best (least number of sections within 10% variance) option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Make sure the second row designating two sections in controlling pipe is selected.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel.**

1. Enter the Stop Time as 5 seconds.
2. Navigate to the Transient Cavitation Panel.
3. Make sure that Model Transient Cavitation is checked and that the selected model is Discrete Vapor Cavity Model (DVCM).
4. Click OK.

Step 6. Create Child Scenarios

In this model we want to see the impact of using different valve closure times. We will create three scenarios to do the comparison. This can be done easily using the Scenario Manager on the Quick Access Panel. Note that the Scenario Manager can also be accessed from the Tools menu.

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 specifications
- Pass changes from a scenario to its variants

Check that the Scenario Manager is active on the Quick Access Panel. It should appear as shown in Figure 2.

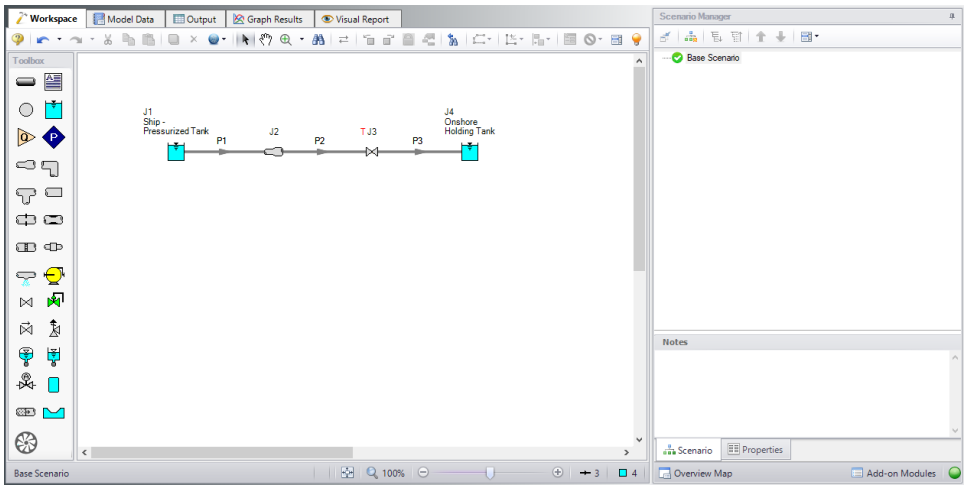


Figure 2: The Scenario Manager window on the Quick Access Panel is used to create and manage model scenarios

Create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on the Scenario Manager on the Quick Access Panel and then selecting the Create Child icon Enter the name *0.5 Second Closure* in the Create Child Scenario window, as shown in Figure 3. Click OK. The new 0.5 Second Closure scenario should now appear in the Scenario Manager on the Quick Access Panel below the Base Scenario.

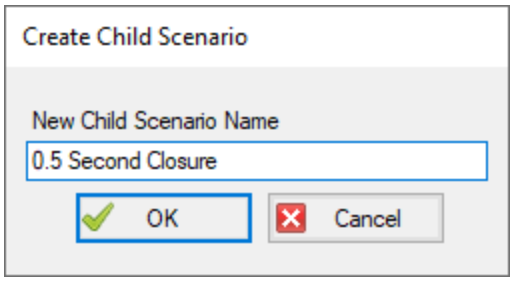


Figure 3: Create Child Scenario window

Select the Base Scenario and create another child and call it *1 Second Closure*. Finally create a third child called *2 Second Closure*. See Figure 4.

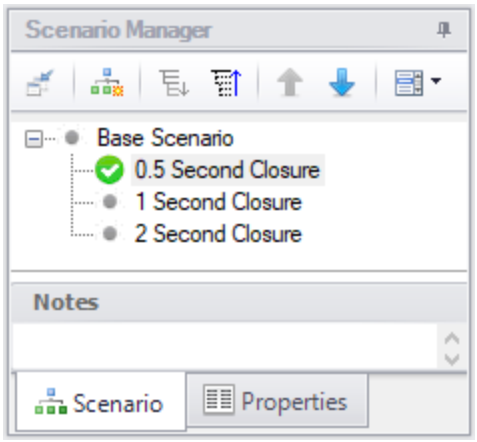


Figure 4: Scenario Manager on the Quick Access Panel with 3 new child scenarios

Set up scenarios

Since the Base Scenario already has the 0.5 second closure time entered, we do not need to modify that child. Load the 1 Second Closure scenario in the list by double clicking the scenario name.

➢**Open the J3 Valve junction** and on the Transient tab change the time of valve closure from 0.5 to 1, then click OK. Select the 2 Second Closure scenario in the list by double-clicking the scenario name in the Scenario Manager. Change the J3 Valve junction here to close at 2 seconds.

Step 7. Run the 0.5 Second Closure scenario

➢**Double-click on the 0.5 Second Closure scenario** in the Scenario Manager on the Quick Access Panel to make it the active scenario. Alternatively, you can select this scenario and then click on the Load Scenario icon to load this scenario.

➢**Click Run Model from the toolbar.** This will open the Solution Progress window, which allows you to watch as the Impulse Solver converges on the answer. This model runs very quickly. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

➢**Go to the Graph Results window** by clicking the Graph Results tab. First create a pressure profile graph as follows:

1. Select the Profile tab in the Graph Control section of the Quick Access Panel.
2. In the Pipes section click All.
3. Make sure Pressure Static is selected in the Parameter section.
4. Change the units to barG.

5. Make sure the boxes for Mx and Mn are checked to display the maximum and minimum values.
6. Click Generate.

Results (shown in Figure 5) indicate that the peak pressure occurs at 120 meters along the transfer line. This corresponds to the valve inlet. Notice that the minimum pressure is flat along almost the entire length of pipe. This value is the vapor pressure, which indicates that the liquid cavitates through most of the length of the pipe.

➢**Save this graph to the Graph List** by selecting Add to Graph List from the toolbar, and give the graph an appropriate name, such as *Static Pressure Profile*. This will allow the graph to be quickly reloaded later in the other scenarios.

➢**Navigate to the Transient Pipe tab on the Quick Access Panel.** Add the outlet station of pipe 2 to the stations to be graphed. Make sure the selected graph parameter is Pressure Static with units of barG. Click Generate. This shows the transient pressures at the valve inlet over time (see Figure 6). Here you can see that the low pressure reaches the vapor pressure repeatedly. Save this graph to the Graph List as well.

From Figure 5 and Figure 6 you can see that the maximum pressure is about 42 barG (4200 kPa(g)). There are several ways to see the actual numerical value. Here is one way:

1. Change to the Output window by selecting the Output tab.
2. In the Pipes section the Transient Max/Min tab should be selected.
3. By default the maximum static pressure is not displayed, but the maximum stagnation pressure is. Typically these values are fairly close. Looking at the Maximum Stagnation Pressure for pipe 2 shows that it is about 42.29 barG (4229 kPa(g)) (see Figure 7). If you want to see the maximum static pressure, you can add it to the Output window using the Output Control window.

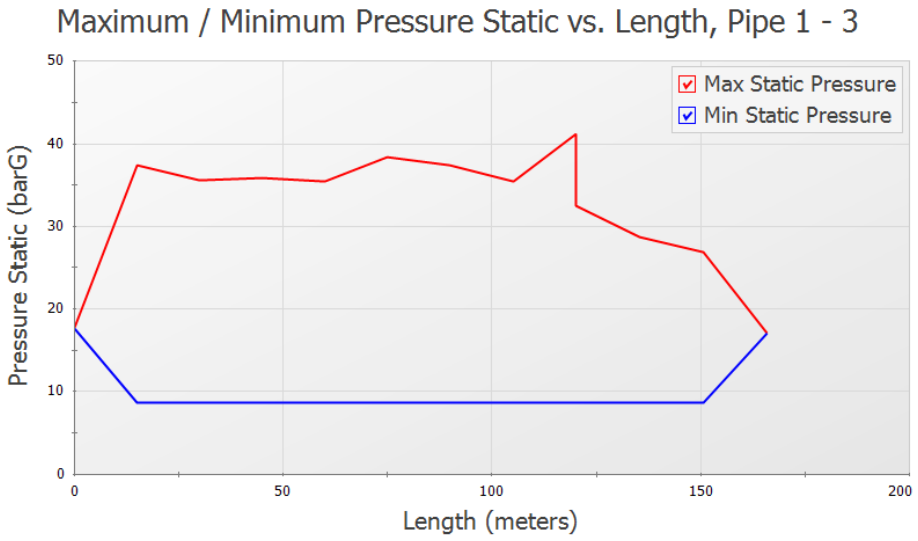


Figure 5: Profile of the maximum and minimum pressures

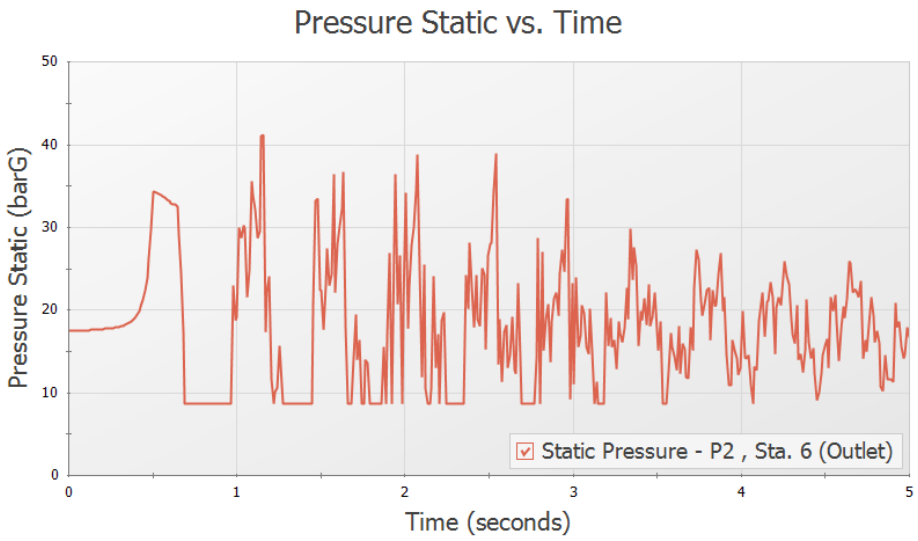


Figure 6: Transient pressures at the valve inlet

Pipes | Transient Output | Transient Max/Min

Pipe	Max. Stag. Pressure (bar)	Max. Stag. Pressure Time (seconds)	Max. Stag. Pressure Station	Min. Stag. Pressure (bar)	Min. Stag. Pressure Time (seconds)	Min. Stag. Pressure Station	Max. Volumetric Flowrate (m3/hr)	Max. Volumetric Flowrate Time (seconds)	Max. Volumetric Flowrate Station	Min. Volumetric Flowrate (m3/hr)	Min. Volumetric Flowrate Time (seconds)	Min. Volumetric Flowrate Station
1	38.47	1.551	1	9.721	1.253	1	453.2	1.050	0	-511.7	1.241	0
2	42.29	1.157	6	9.721	0.7515	0	589.7	1.455	5	-485.0	1.217	1
3	33.48	0.6203	0	9.721	0.4891	0	423.9	0.000	0	-355.8	0.5845	3

Figure 7: Numerical value of maximum pressure can be found in the Output window

Step 9. Run the Second and Third Scenarios

➢**Use Scenario Manager** to load and run the 1 and 2 second closure cases. Use the techniques in Step 8 to find the maximum pressure for these cases. A tabulation of the results is shown in Table 1. If the results for the 2 second case are graphed, one can see that there is no cavitation. The results corresponding to Figure 6 appear much different.

To see the difference in results with and without cavitation, you can turn the transient cavitation model off in the Transient Control window.

Table 1: Max Stagnation pressures with different closure rates

Closure Time (seconds)	Max Pressure (bar)
0.5	42.29
1	41.70
2	25.85

Step 10. Compare DVCM Output with DGCM

As a verification of the cavitation results, you can change the Transient Cavitation model in the Transient Control window to use the Discrete Gas Cavity Model. Do this for the 0.5 Second Closure scenario as follows:

1. Create a child of the 0.5 Second Closure scenario and name it *With DGCM*
2. Open Analysis Setup and go to the Transient Cavitation panel in the Transient Control group
3. Under Model Transient Cavitation change the Model to Discrete Gas Cavity Model
4. Click OK

Run the model, then go to the Graph Results window.

Load the Static Pressure Profile graph by double-clicking the name in the Graph List Manager at the top of the Quick Access Panel. This will load the Pressure Profile with the results for the current scenario. In order to compare the DGCM results to the results using DVCM we can select both scenarios to be graphed using the Multi-scenario graphing feature.

➢**In the Graph Parameters Section of the Quick Access panel** click the Multi-Scenario button to select the scenarios to graph. Check the box next to the 0.5 Second Closure scenario to include it in the graph, and ensure that the With DGCM scenario is still selected, then click OK. Click Generate to update the graph.

Figure 8 shows the multi-scenario graph. Notice that the maximum pressure at the valve is nearly identical for both cavitation models, though there is some deviation in the maximum pressures upstream of the valve.

Repeat the process above to generate a multi-scenario graph of the Valve Inlet Pressure over time that was shown in Figure 6 above. The resulting multi-scenario plot is shown below in Figure 9. Notice that the models produce very similar results for the first pressure spike after cavitation begins at around 1.2 seconds, after which the results begin to diverge. This provides high confidence in the validity of the first pressure spike, as two separate calculation methods produced the same conclusion.

A similar analysis can be performed with the 1 Second Closure scenario, which results in slightly more deviation in the results. Performing this comparison with the 2 Second Closure scenario would produce no differences, as the minimum pressure never reaches vapor pressure in that scenario.

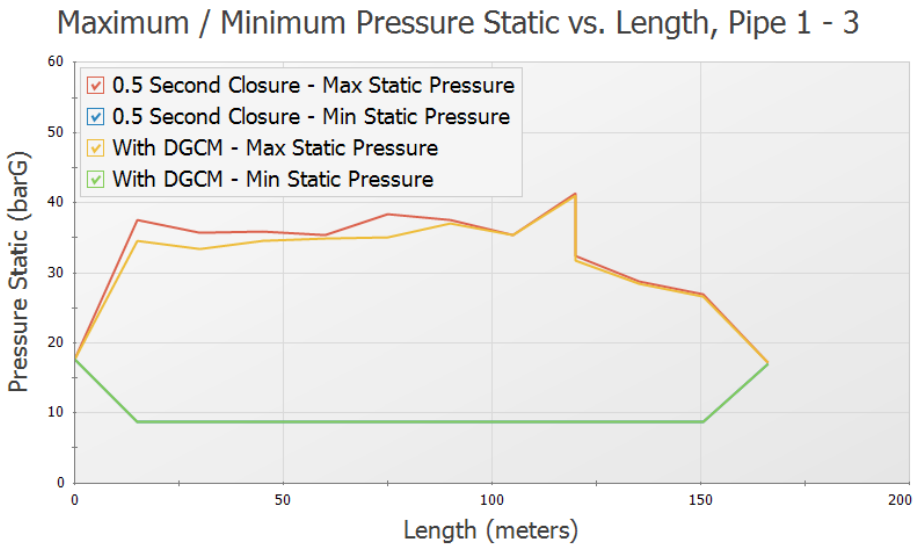


Figure 8: Multi-scenario graph comparing DVCM and DGCM results for the 0.5 Second Closure scenario

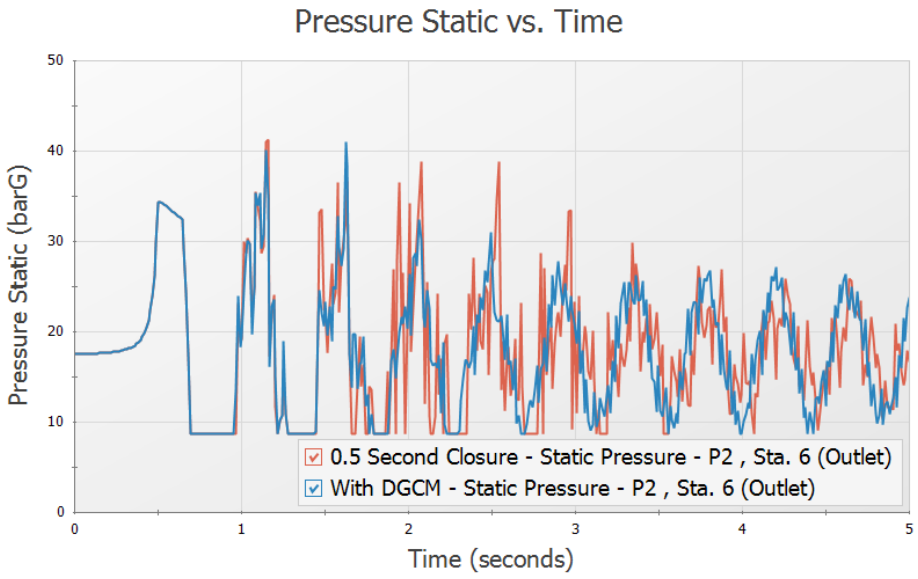


Figure 9: Multi-scenario graph comparing DVCM and DGCM results at the valve inlet for the 0.5 Second Closure scenario

Summary

Table 2 provides a summary of the maximum pressure in each of the scenarios using each of the cavitation models. Lengthening the closure time reduced the maximum pressure with both cavitation models.

Table 2: Max Stagnation pressures with different closure rates

Closure Time (seconds)	Max Pressure DVCM (bar)	Max Pressure DGCM (bar)
0.5	42.29	42.04
1	41.70	41.97
2	25.85	25.85

Tank Farm with Infinite Pipes

Summary

A long supply pipeline flows into a gasoline tank farm. The transient occurs when flow is switched from one storage tank to another. Two scenarios compare the results with and without an infinite pipe boundary, and also compares the run times.

Topics Covered

- Using infinite pipes

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Tank Farm with Infinite Pipes.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = User Specified Fluid
   b. Name = Gasoline

c. Density = 0.759 S.G. water
d. Dynamic Viscosity = 0.4777 centipoise
e. Bulk Modulus = 1034.21 MPa
f. Vapor Pressure = 0.062 MPa

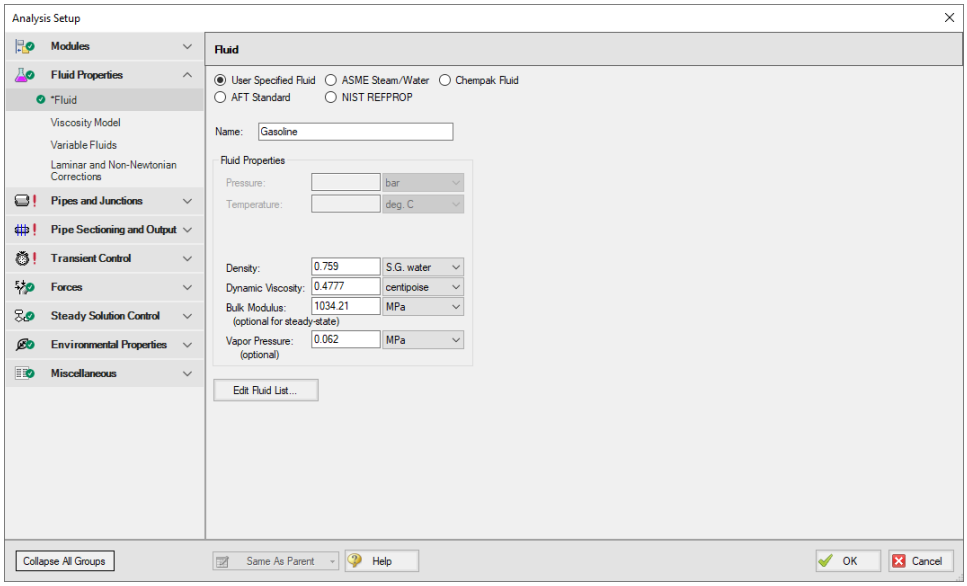


Figure 1: Fluid panel defined for gasoline

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

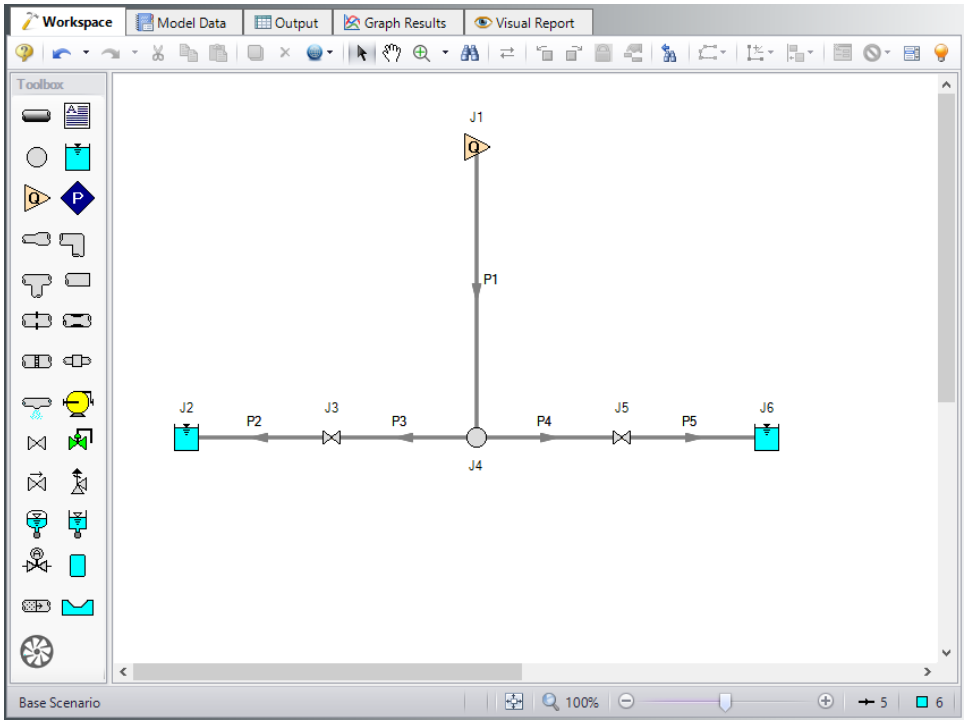


Figure 2: Model lay out on Workspace for the tank farm model

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe size and lengths, as well as junction elevation data are given below. Here we see that the inlet assigned flow junction represents the flow coming from a pumping station that is 50 km away. If the pumps in the pump station interact with the tank farm transients, then the pumps must be modeled in detail. We will find out later that they do not, and thus we can simplify the model by using an assigned flow junction at the pump discharge. The WEB Valve is initially closed. The way we represent this in steady-state is using a Special Condition to close it. As for the transient, the initial data point for Cv must be zero.

Junction Properties

1. Assigned Flow J1
    a. Name = Pump Station Discharge
    b. Elevation = 294 meters
    c. Type = Inflow
    d. Flow Specification = Volumetric Flow Rate
    e. Flow Rate = 13500 barrels/hr
2. Reservoir J2
    a. Name = Desert Petroleum
    b. Liquid Surface Elevation = 314 meters

c. Liquid Surface Pressure = 0 barG (0 kPa(g))
d. Pipe Depth = 12 meters

3. Valve J3
   a. Name = Desert Pet Valve
   b. Elevation = 294 meters
   c. Loss Model tab
      i. Valve Data Source = User Specified
      ii. Loss Model = Cv
      iii. Cv = 15700
   d. Transient tab
      i. Transient Special Condition = None
      ii. Initiation of Transient = Time
      iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	15700
19	15700
19.2	12400
22	12400
22.2	9400
26	9400
28	200
34	0

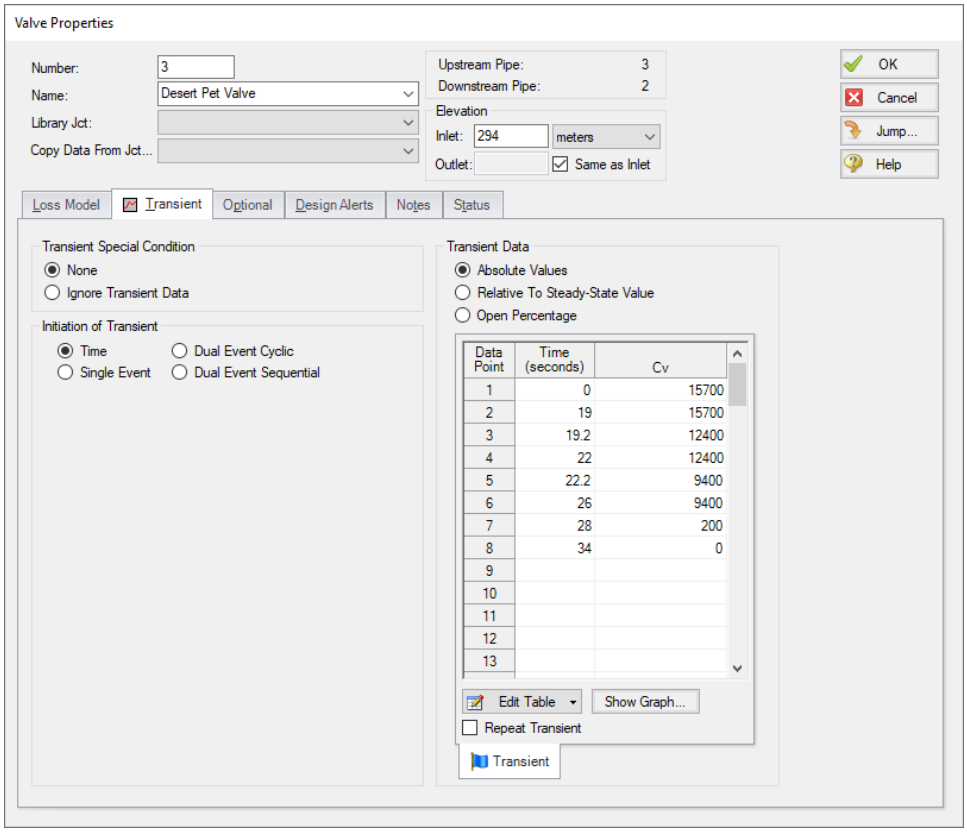


Figure 3: Valve J3 transient data

4. Branch J4
    a. Elevation = 294 meters
5. Valve J5
    a. Name = WEC Valve
    b. Elevation = 294 meters
    c. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Cv = 4000
    d. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	0
10	150

Time (seconds)	Cv
11	800
12	4000
30	4000

e. Optional tab
   i. Special Condition = Closed

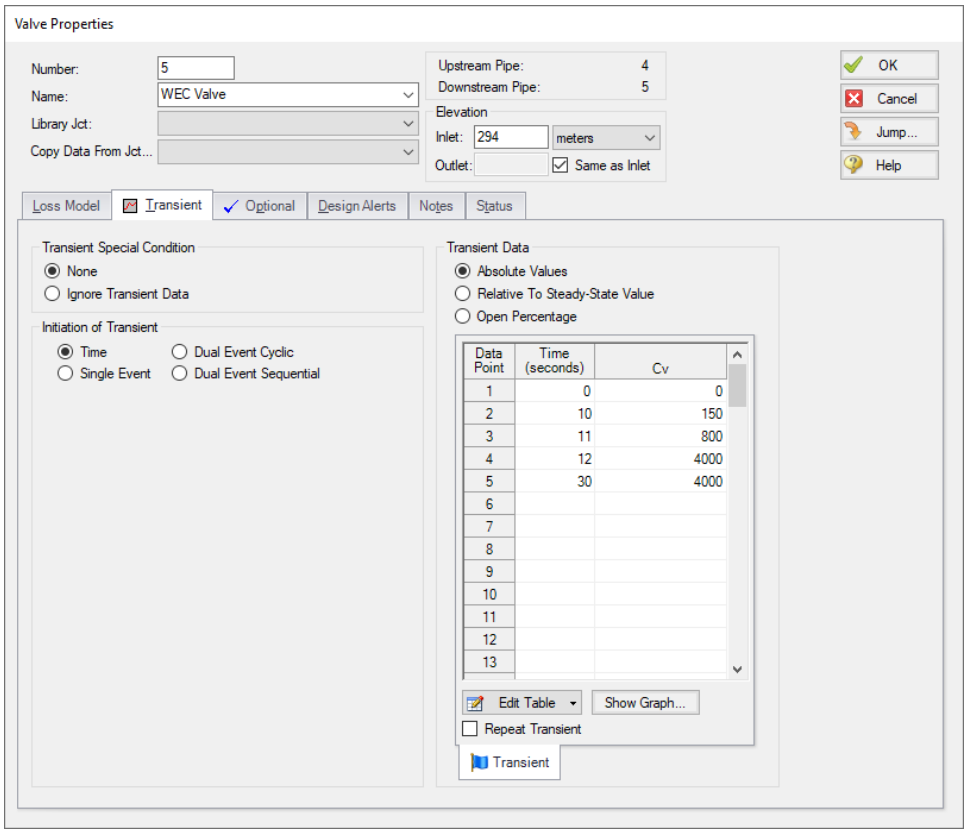


Figure 4: Valve J5 transient data

6. Reservoir J6
   a. Name = West Coast Energy
   b. Tank Model = Infinite Reservoir
   c. Liquid Surface Elevation = 311 meters
   d. Liquid Surface Pressure = 0 barG (0 kPa(g))
   e. Pipe Depth = 0 meters

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Size = Use table below
    c. Type = schedule 40
    d. Friction Model Data Set = Standard
    e. Length = Use table below

Pipe	Size (inches)	Length
1	20	50 km
2	20	91 meters
3	12	61 meters
4	12	9 meters
5	20	305 meters

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel.** Enter the Stop Time as 35 seconds.

Step 6. Create Child Scenarios

In this model we want to see the impact of using an infinite pipe boundary. We will do the comparison between two scenarios. In the Scenario Manager on the Quick Access Panel, create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on the Scenario Manager on the Quick Access Panel and then selecting the Create Child icon. Enter the name *Without Infinite Pipe*. A new scenario will appear below the Base Scenario in the list. Select the Base Scenario and create another child and call it *With Infinite Pipe*.

➢**Double-click the Without Infinite Pipe scenario** in the list to load it as the current scenario, as shown by the green check mark icon, as shown in Figure 5.

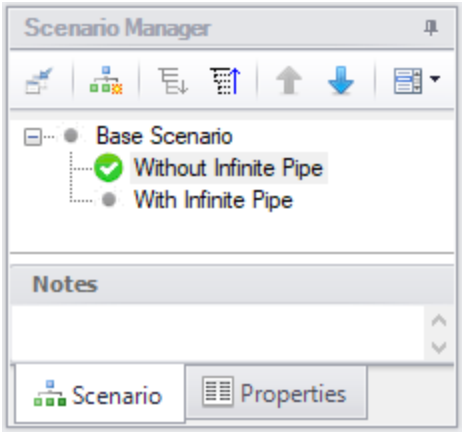


Figure 5: Scenario Manager window with two child scenarios

Step 7. Run the First Scenario

This scenario models pipe P1 as 50 kilometers in length, which is its true length. Select Run Model from the toolbar or Analysis menu. The Waterhammer Assistant may open to inform you that using an Assigned Flow junction may not be appropriate. For our purposes it is appropriate, so click OK to dismiss the caution and to see the Solution Progress. This window allows you to watch the progress of Impulse Steady-State and Transient Solvers. Depending on the speed of your PC, this model may take several minutes to run. Note how long it takes so you can compare that to the other scenario later. Once finished, go to the Graph Results window.

Step 8. Graph the Results

On the Graph Control tab of the Quick Access panel go to the Transient Jct tab. Add valves J3 and J5 to the junctions to be graphed (see Figure 6). Select as Pressure Static Inlet parameter and set the units to barG. Click Generate. This graphs the pressures at the valve inlets over time (see Figure 7). Save this graph to the Graph List by using the Add to Graph List button.

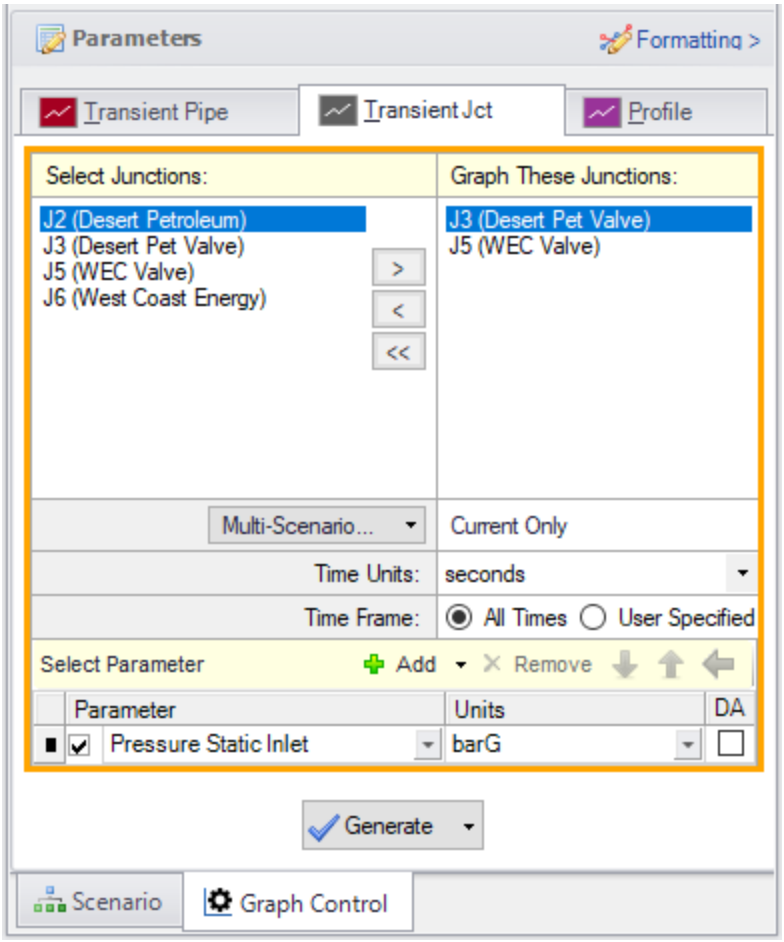


Figure 6: Select Graph Data window specified to plot the pressures at the valve inlets

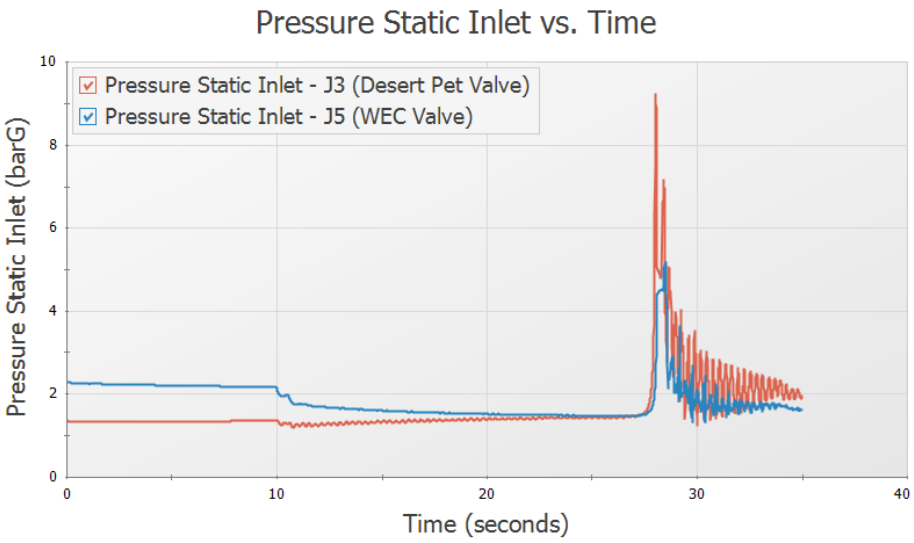


Figure 7: Transient pressures at the valve inlets

Now change the parameter to Vol. Flow Rate Inlet and set the units to barrels/hr. Click Generate. This graph shows the flowrates through the valves to each tank during the transient. Save this graph to the Graph List also.

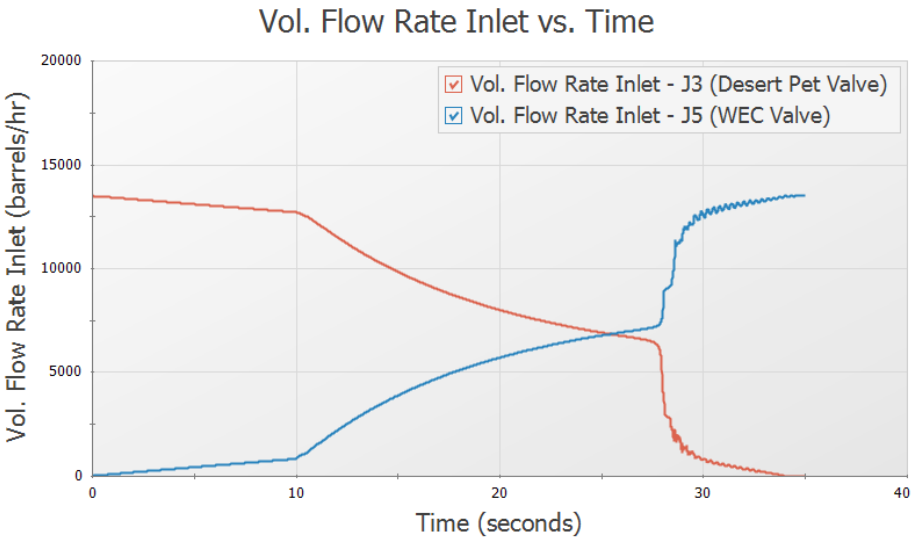


Figure 8: Transient flow rates at the valve inlets

Step 9. Consider an Infinite Pipe

The longest pipe in the system is the 50-kilometer pipe P1. If you open the Section Pipes window you will see it has over 5,000 computing sections, which is over 100 times as many as all the other computing sections combined. Thus the computation time for this model is dominated by pipe P1.

The wavespeed for pipe P1 is 1091 meters/second, and the time frame for the simulation is 35 seconds. During the simulation any transients in the tank farm can propagate 38,185 meters up pipe P1 and thus never reach junction J1. For the J1 boundary to have any impact on the tank farm, it would have to receive the transient wave and then reflect its response back down pipe P1 to the tank farm. Thus the wave would have to travel 100 kilometers (i.e., up the pipe and then back), which would take 91.7 seconds.

The conclusion here is that any transients at the tank farm that take place in less than 91.7 seconds will not be affected by pipe P1 or junction J1. But one cannot neglect pipe P1, because flow is coming in from that pipe and transients are propagating up that pipe. This is an excellent opportunity to use an infinite pipe boundary. Infinite pipe boundaries accept all pressure waves but reflect none. If we use an infinite pipe boundary in this system, we will get virtually identical results for the first 91.7 seconds. If we let the model run longer than this, then the results will differ after that point in time. Since we are not interested in what happens after 91.7 seconds but only what happens up to 35 seconds, we can save a lot of computation time.

Step 10. Specify the Second Scenario

Load the With Infinite Pipe scenario and make the following changes:

1. Open the J1 Assigned Flow junction window.
2. Select the Transient tab.
3. In the Transient Special Conditions section, select No Reflections (Infinite Pipe).
4. Click OK.
5. Open the Pipe Properties window for pipe P1.
6. Change the length to 18 meters. The 18 meters length is somewhat arbitrary, and was chosen because the shortest pipe in the system (P4) is 9 meters long, and the pipe adjacent to the junction using the infinite pipe transient needs to have 2 sections. Thus modeling P1 as 18 meters does not affect the time step.
7. Click OK.
8. Open Analysis Setup and open the Sectioning panel.
9. Notice how pipe P1 now has two sections instead of over 5000.
10. Click OK.

Step 11. Run the Second Scenario

➢**Select Run Model** from the toolbar or Analysis menu. This scenario runs much faster than the first one (about 1/100th as long).

Step 12. Graph the Results

Use the Graph List Manager to create the same graphs as before to compare results to the previous scenario. This can be done by loading the graphs from the Graph List Manager, then selecting both scenarios from the Multi-Scenario button in the Graph Parameters section and regenerating the graph. The results are practically identical for the two scenarios.

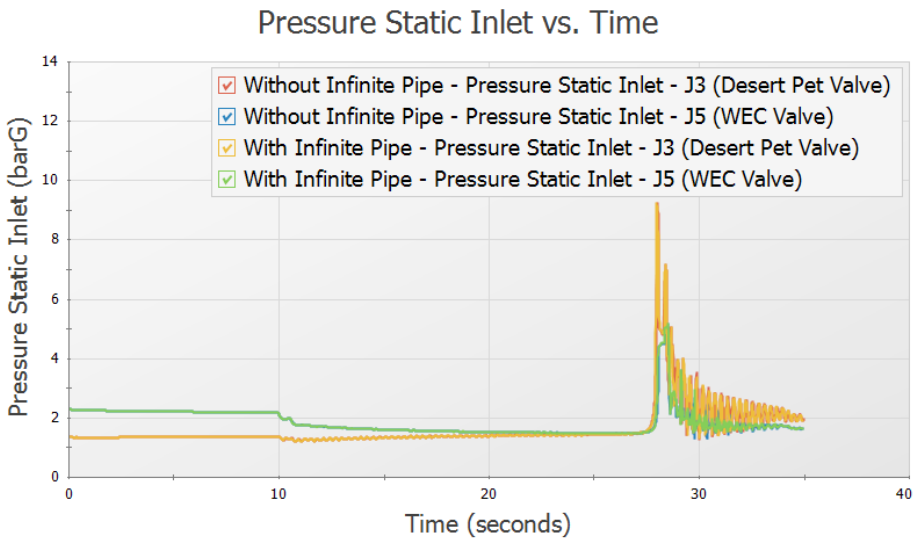


Figure 9: Multi-scenario graph of the transient pressures at the valve inlets

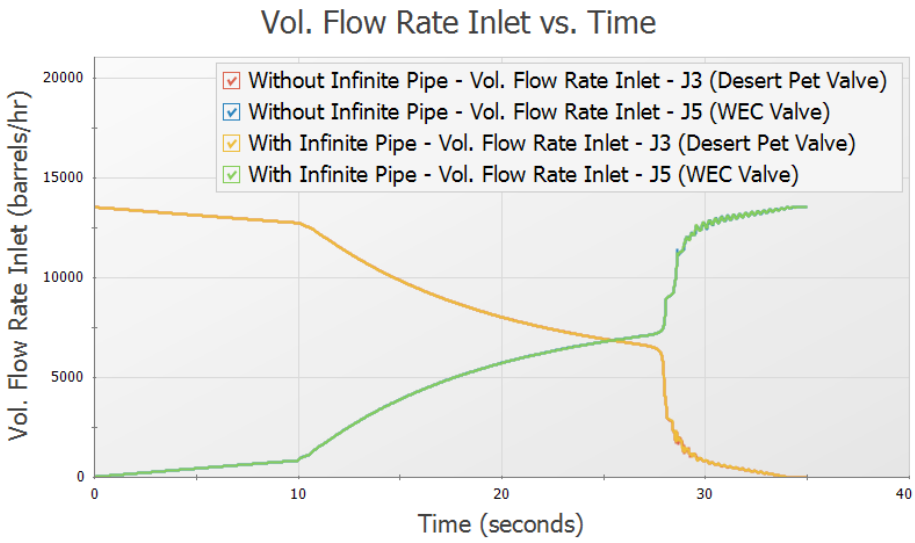


Figure 10: Multi-scenario graph of the transient flow rates at the valve inlets

Conclusion

Using an infinite pipe boundary on the J1 Assigned Flow junction allowed us to obtain the same quality results at the tank farm with 1/100th of the computation time. While in this case the difference was between a few seconds versus a few minutes, a more complicated system could mean the difference between a one-hour run and a one hundred-hour run.

Infinite pipe boundaries can also be employed on the Assigned Pressure junction type. The only difference is that during the steady-state the junction models a known pressure rather than a known flow. During the transient they function identically.

Finally, note that once the transient solution starts, the flow rate from an Assigned Flow junction and the pressure from an Assigned Pressure junction are no longer used. Both the flow rate and pressure at the junction will change over time, in a matter consistent with the fact that no reflections are occurring at that junction.

Pump Trip with Check Valve

Summary

A system pumping water uphill loses power to the pump. A check valve is near the pump discharge. This model predicts the system response and pump speed decay. Uses pump curve and efficiency data.

Topics Covered

- Specifying a pump curve and power curve in the Pump Properties Window
- Using one of the transient pump models which accounts for pump inertia
- Using fittings and losses lumped into a pipe

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Pump Trip with Check Valve.fth***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard
b. Fluid = Water (liquid)
    i. After selecting, click Add to Model
c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

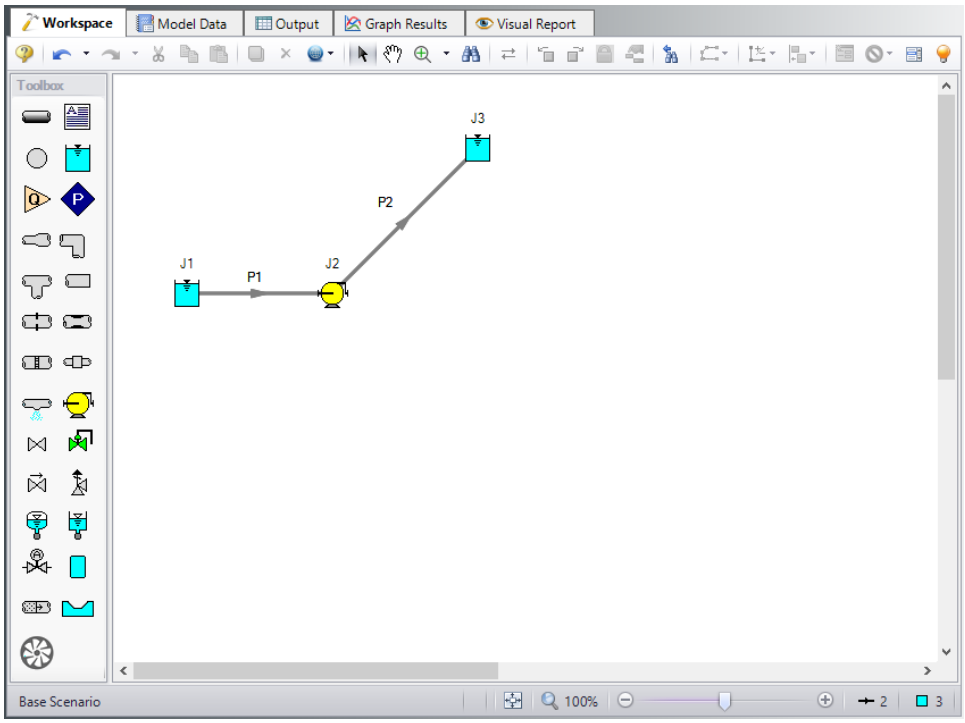


Figure 1: Workspace for pump trip with check valve example

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Junction Properties

1. Reservoir J1
    a. Name = Lower Reservoir
    b. Liquid Surface Elevation = 3 meters

c. Liquid Surface Pressure = 0 barG (0 kPa(g))
d. Pipe Depth = 3 meters

2. Pump J2
   a. Inlet Elevation = 0 meters
   b. Pump Model tab (Figure 2)
      i. Pump Model = Centrifugal (Rotodynamic)
      ii. Performance Curve Used in Simulation = Standard Pump Curve
      iii. Model Combined Check Valve and Pump = Checked
      iv. Forward Velocity to Close = 0 meters/sec
      v. Delta Head to Re-Open = 0 meters
      vi. Rated Pump Speed = 1760 rpm
      vii. Enter Curve Data = (Figure 3)

Volumetric	Head	Power
m3/hr	meters	kW
0	122	19
57	119	25
114	116	45
227	104	97

      viii. Curve Fit Order = 2
   c. Transient tab (Figure 4)
      a. Transient = Trip
      b. Transient Special Condition = None
      c. Initiation of Transient = Single Event
      d. Event Type = Time Absolute
      e. Condition = Greater Than or Equal To
      f. Value = 0 seconds
      g. Total Rotating Inertia = User Estimated
      h. Value = 1.05 kg-m2

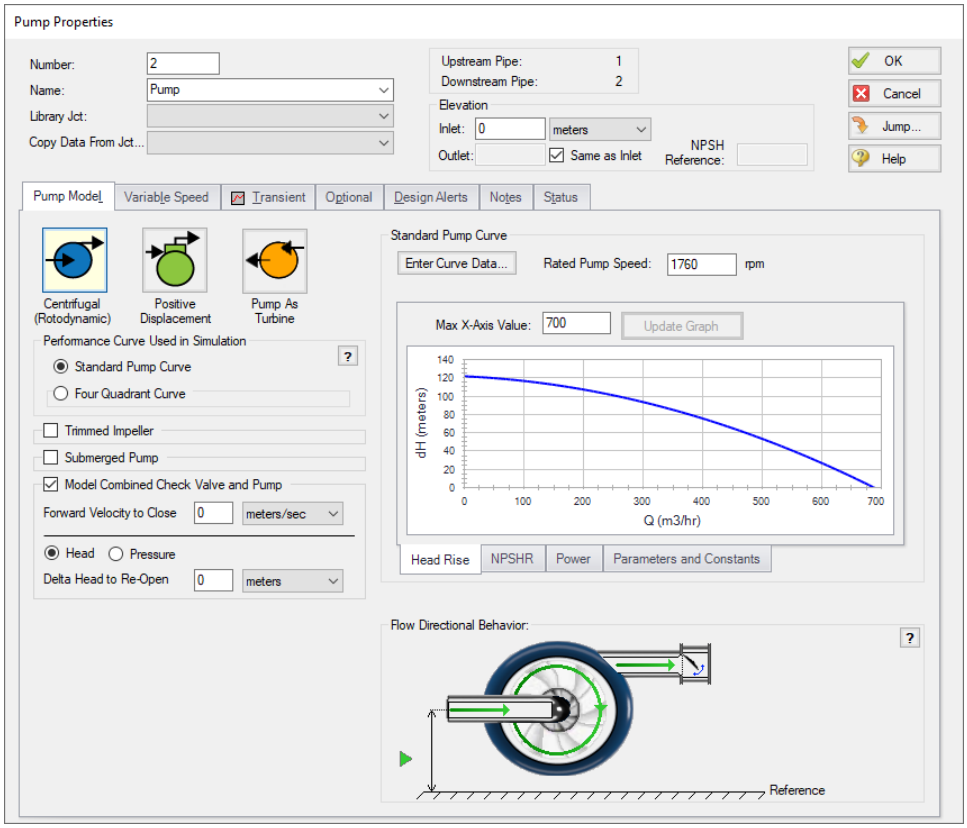


Figure 2: Pump J2 Pump Model tab input

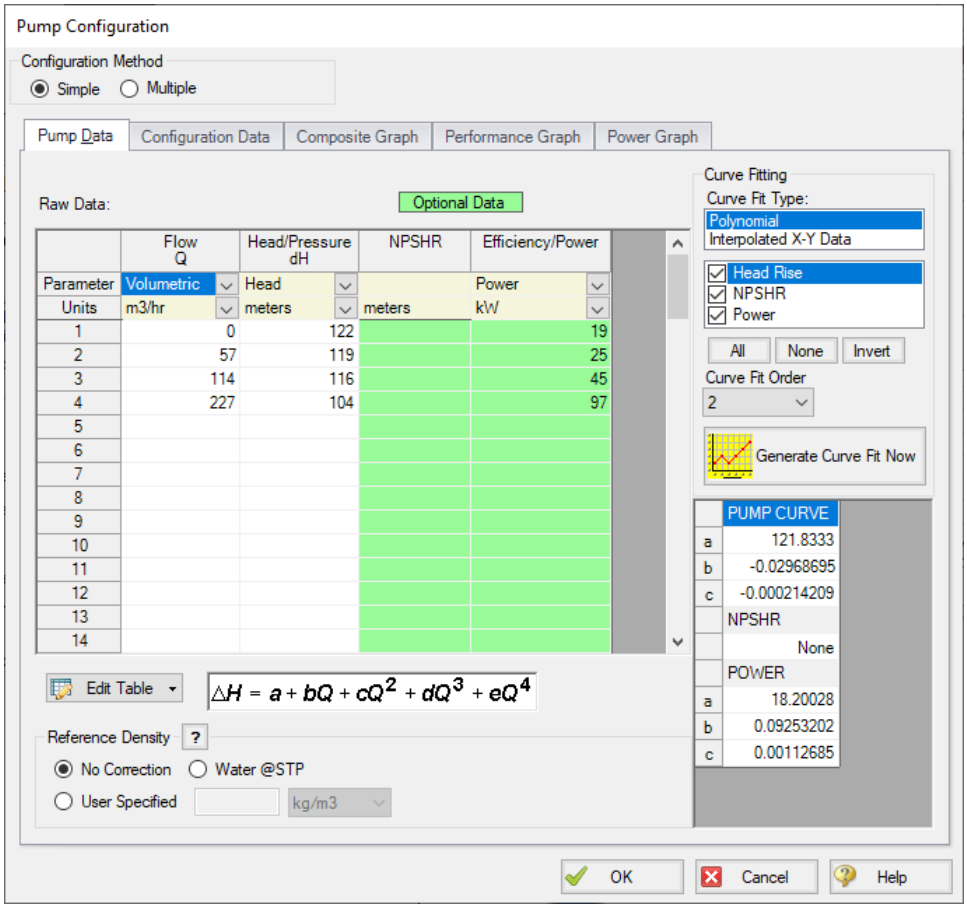


Figure 3: Pump J2 performance data

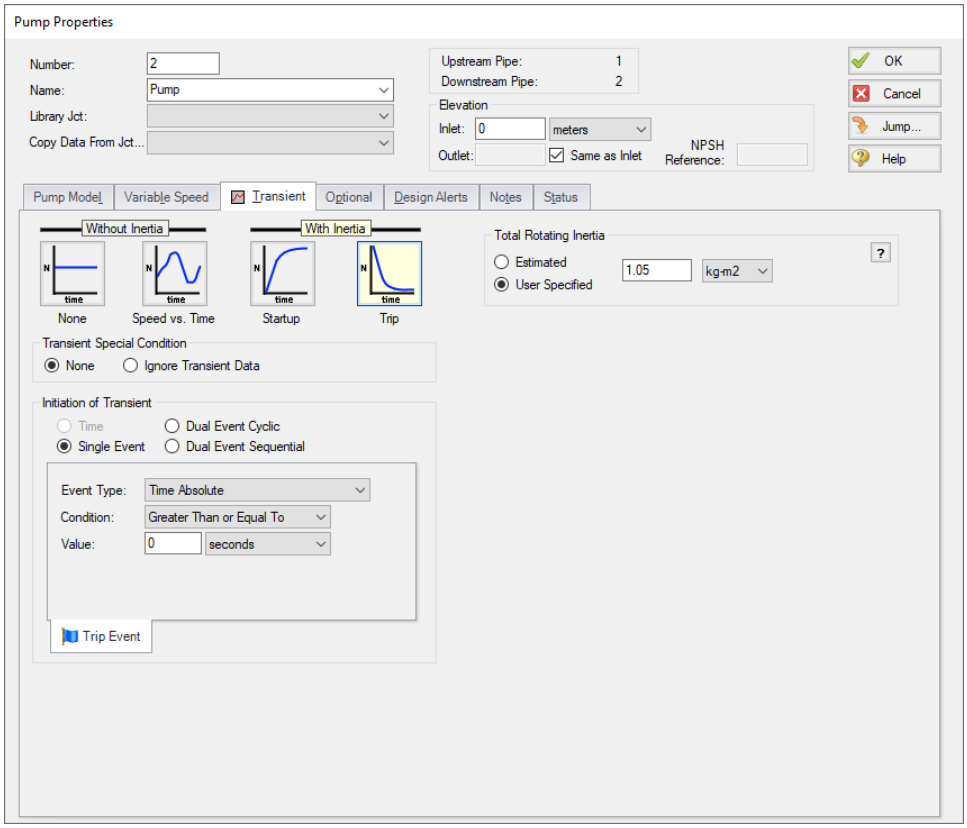


Figure 4: Transient input data for Pump J2

3. Reservoir J3
    a. Name = Upper Reservoir
    b. Liquid Surface Elevation = 61 meters
    c. Liquid Surface Pressure = 0 barG (0 kPa(g))
    d. Pipe Depth = 3 meters

Pipe Properties

1. Pipe P1
    a. Pipe Material = Steel - ANSI
    b. Size = 4 inch
    c. Type = STD (schedule 40)
    d. Friction Model Data Set = Standard
    e. Lengths = Use table below

Pipe	Length (meters)
1	3
2	302

The pipe model also allows for fittings and losses like valves or elbows. Select the Fittings & Losses tab for P2 and type in 25 for the Total K Factor. Now click OK to close the Pipe Properties window and accept your changes.

Note: Unless specified otherwise, pipe elevation is assumed to vary linearly between junctions. Thus the pipe P2 inlet elevation is 0 meters, and the exit elevation is 58 meters (because the J3 reservoir is 3 meters deep). In between the elevation is linear. If another profile is desired, this can be entered on the pipe's Optional tab in the Intermediate Pipe Elevations section.

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 30 seconds for the Stop Time.

All groups should now be complete and the model is ready to run. If all groups in Analysis Setup have a green checkmark then Click OK and proceed. Otherwise, enter the missing information.

Step 6. Run the Model

Click Run Model from the toolbar or from 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 Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

The Output window will display any warnings if they exist. There should not be any warnings here. The Graph Results window will be more useful in understanding the results. Go to the Graph Results window.

First, graph the static pressure.

1. On the Quick Access Panel, on the Transient Pipe tab, add the Pipe 1 Outlet and Pipe 2 Inlet (see Figure 5). These represent the pump suction and discharge locations.
2. For Parameter, select the Pressure Static.

3. Set the units to barG.
4. Click Generate.

The resulting pressure transients are shown in Figure 6. Here one can see that the transient pressure at the pump discharge does not rise above the initial steady-state pressure. This is not always the case.

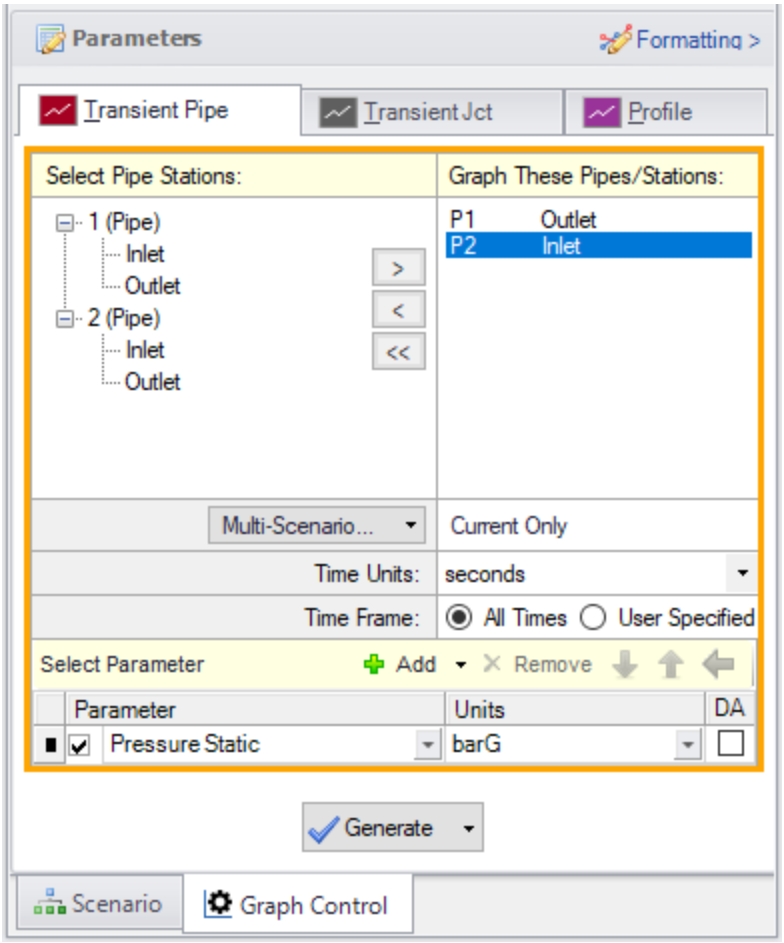


Figure 5: Data selection to view the transient pump suction and discharge pressures

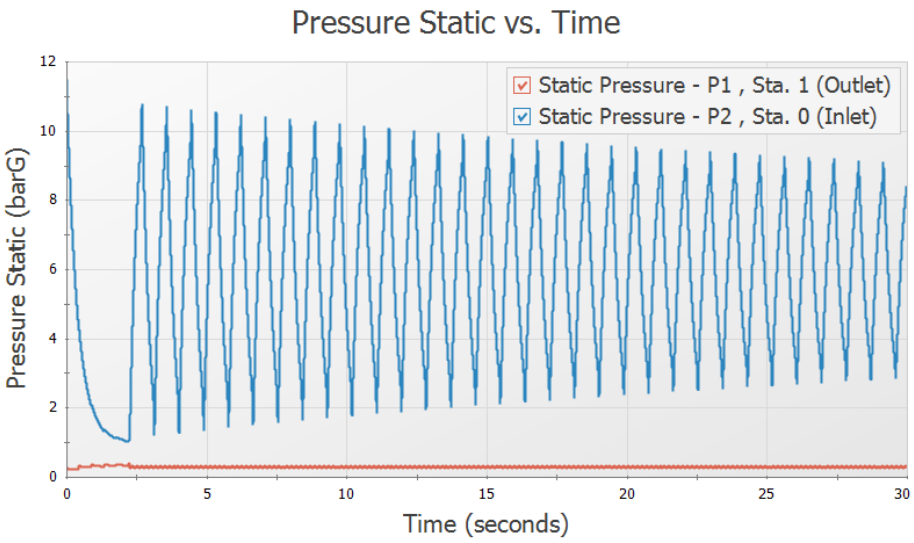


Figure 6: Predicted transient pump suction and discharge pressures

Next, graph the pipeline transient pressure profile

1. On the Quick Access Panel, select the Profile tab (Figure 7).
2. In the Pipes area select All to select both pipes 1 and 2.
3. In the Parameter area make sure Pressure Static is selected.
4. Set the static pressure units to barG.
5. Make sure boxes are checked for Mx and Mn which will plot the maximum and minimum values.
6. Click Generate.

Profile type graphs that show maximum and minimum values can be very helpful. In Figure 8 we see that the maximum transient pressure occurs at the pump discharge, and that the minimum pressure occurs 190-220 meters downstream of the pump. Also note that minimum pressure goes below atmospheric pressure for almost 140 meters of pipe. For some applications this may not be acceptable.

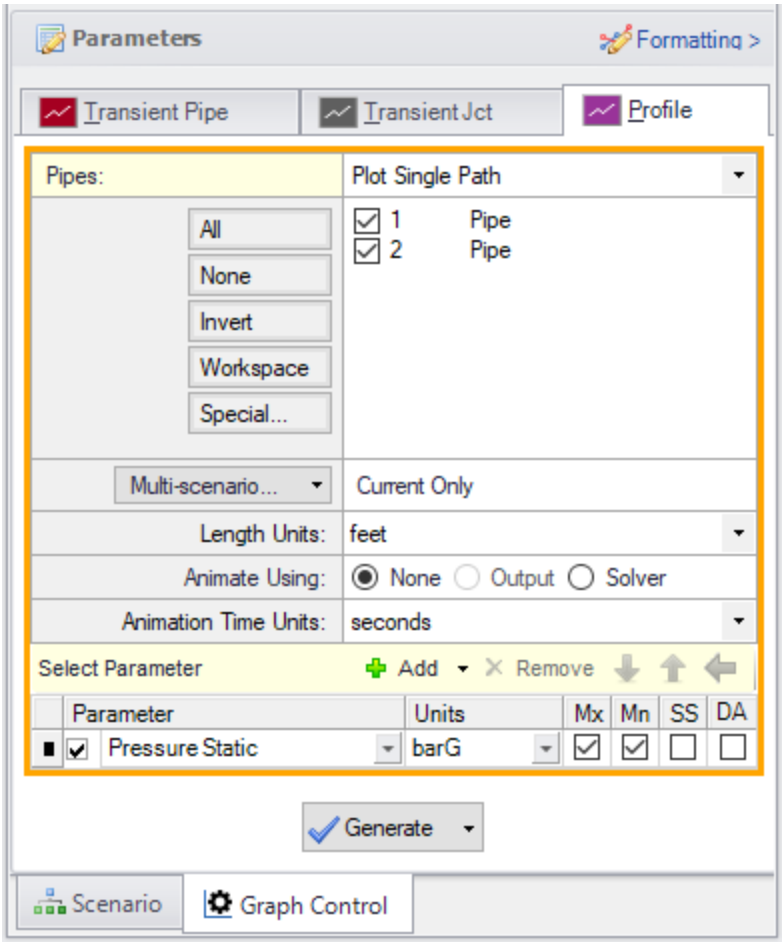


Figure 7: Data selection to view the max/min pressure profile

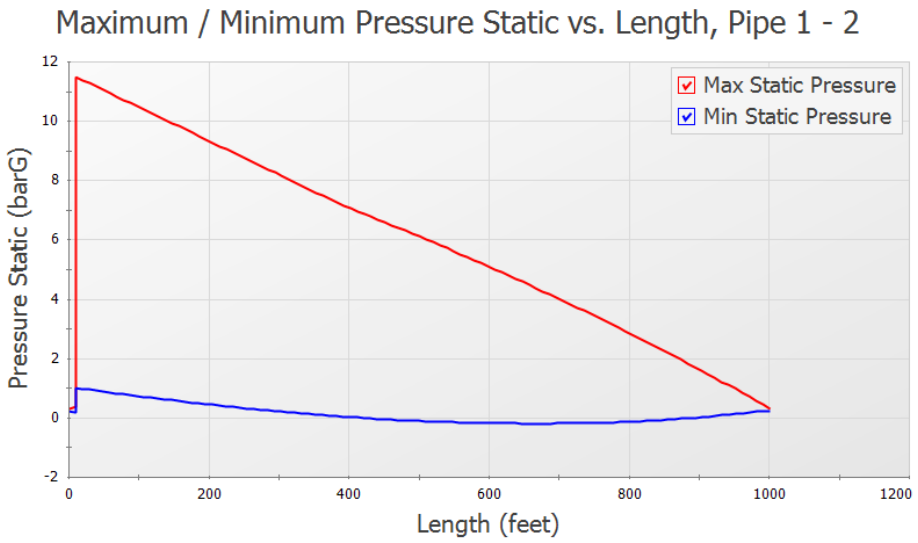


Figure 8: Predicted max/min pressure profiles

Finally, graph the pump speed decay.

1. Select the Transient Jct tab on Quick Access Panel.
2. Add J2 (Pump) to the Graph These Junctions list on the Transient Jct tab (see Figure 9).
3. For the parameter to graph, make sure Speed is selected.
4. Click Generate.

The predicted pump speed is then shown (Figure 10).

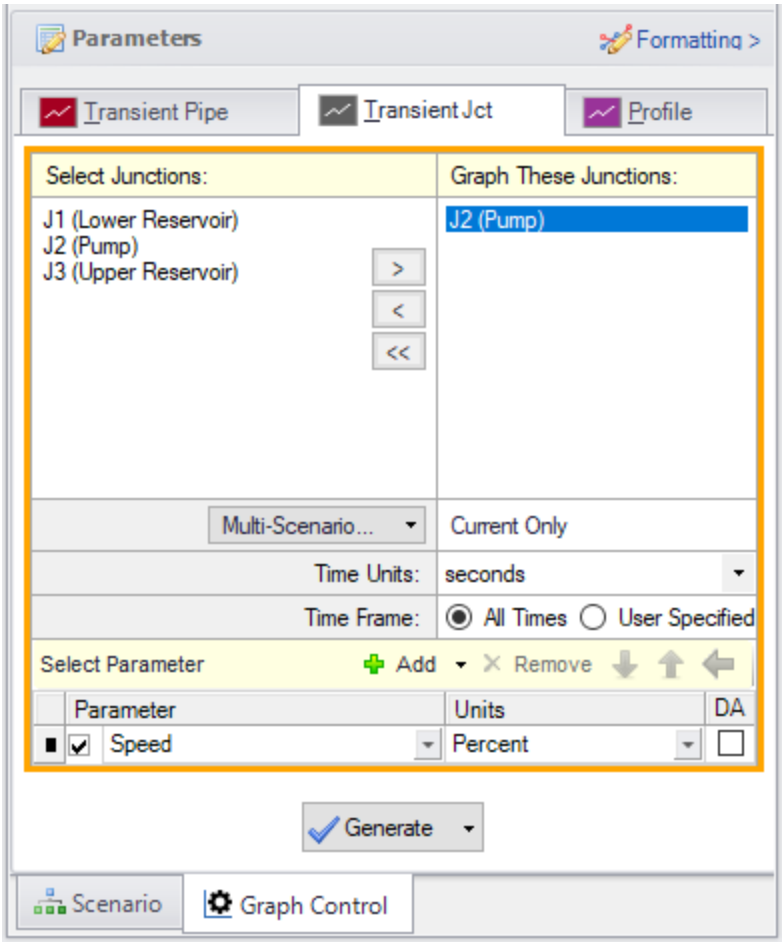


Figure 9: The pump speed is selected on the Transient Jct tab

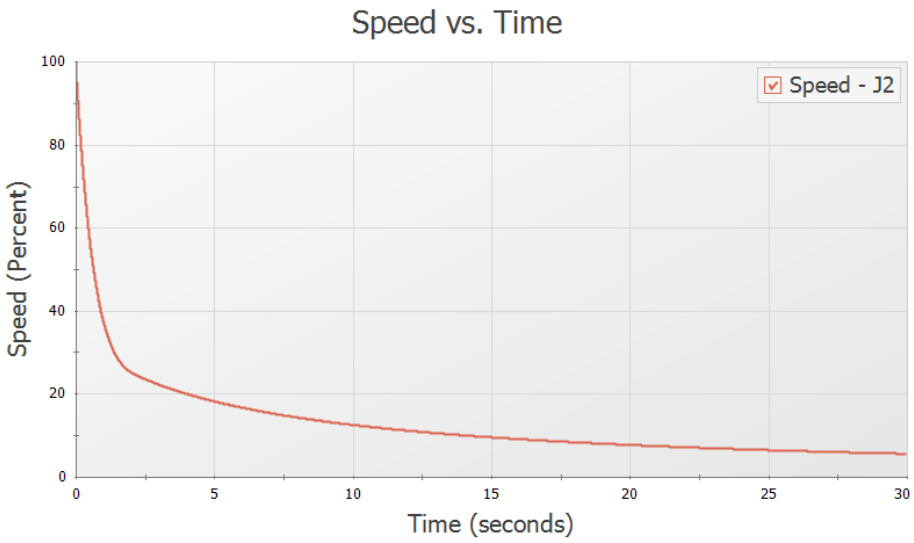


Figure 10: The pump speed decay is shown. The speed decays quickly at first, and then slows down after the check valve closes and the flow goes to zero

Check Valve Modeling

Summary

This example models the closure of the check valve in three ways for a system pumping water uphill loses power to the pump. The check valve is near the pump discharge. Results will be graphed to illustrate pressure on each side of the check valve, and animation features will be used to show the system pressure profile after the valve is closed.

Topics Covered

- Modeling a check valve and fluid velocity at valve closure
- Observing the differences between check valve models
- Creating a system pressure profile animation

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Check Valve Modeling.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard
b. Fluid = Water (liquid)
    i. After selecting, click Add to Model
c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

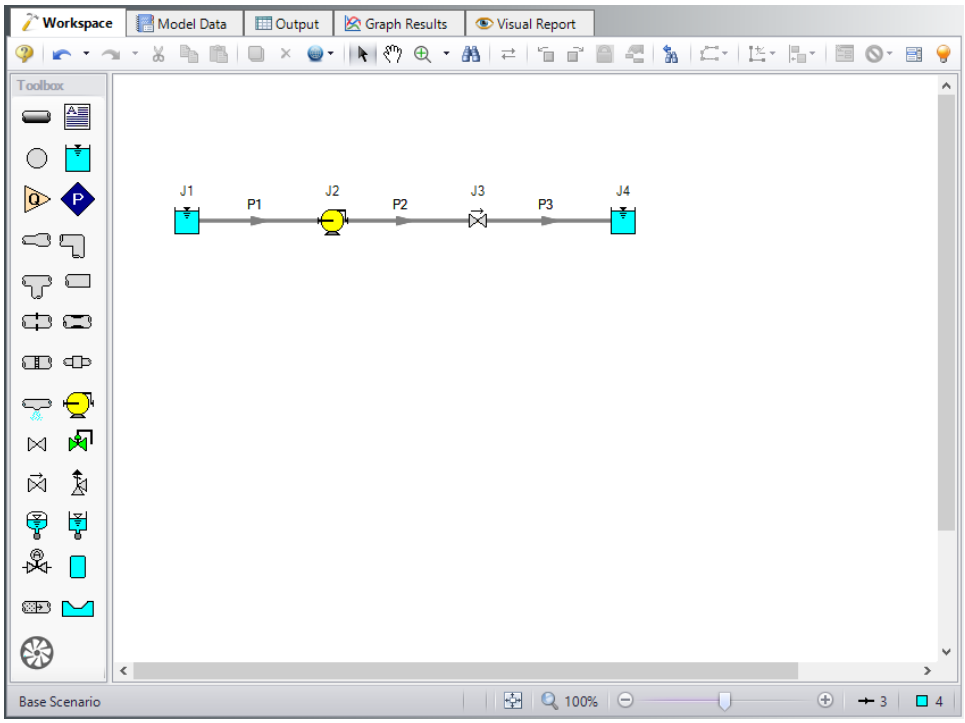


Figure 1: Workspace for Check Valve Example

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

For the first scenario, the simplest User Specified check valve type will be used. The Forward Velocity to Close Valve and Delta Head to Reopen should be obtained either from measured data, or from the data provided by the manufacturer. If no information is known, an estimate of 0 for both parameters may be used to represent an ideal check valve. However, a velocity to close of zero is not realistic, and should not be used for final engineering estimates. A more conservative engineering estimate would be achieved if some reverse flow is allowed through the check valve before it fully closes.

Pipe Properties

1. Pipe P1
    a. Pipe Material = Steel - ANSI
    b. Size = 8 inch
    c. Type = STD (schedule 40)
    d. Friction Model Data Set = Standard
    e. Length = Use table below

Pipe	Length (meters)
1	3
2	3
3	300

Junction Properties

1. Reservoir J1
    a. Name = Lower Reservoir
    b. Liquid Surface Elevation = 3 meters
    c. Liquid Surface Pressure = 10 barG
    d. Pipe Depth = 3 meters
2. Pump J2
    a. Inlet Elevation = 0 meters
    b. Pump Model tab (Figure 2)
        i. Pump Model = Centrifugal (Rotodynamic)
        ii. Performance Curve Used in Simulation = Standard Pump Curve
        iii. Rated Pump Speed = 1760 rpm
        iv. Enter Curve Data = (Figure 3)

Volumetric	Head	Power
Q (m3/hr)	H (meters)	Power (kW)
0	120	27
50	115	35
100	100	45
200	80	60

        v. Curve Fit Order = 2
    c. Transient tab (Figure 4)

a. Transient = Trip
b. Transient Special Condition = None
c. Initiation of Transient = Single Event
d. Event Type = Time Absolute
e. Condition = Greater Than or Equal To
f. Value = 0 seconds
g. Total Rotating Inertia = User Estimated
h. Value = 1 kg-m2

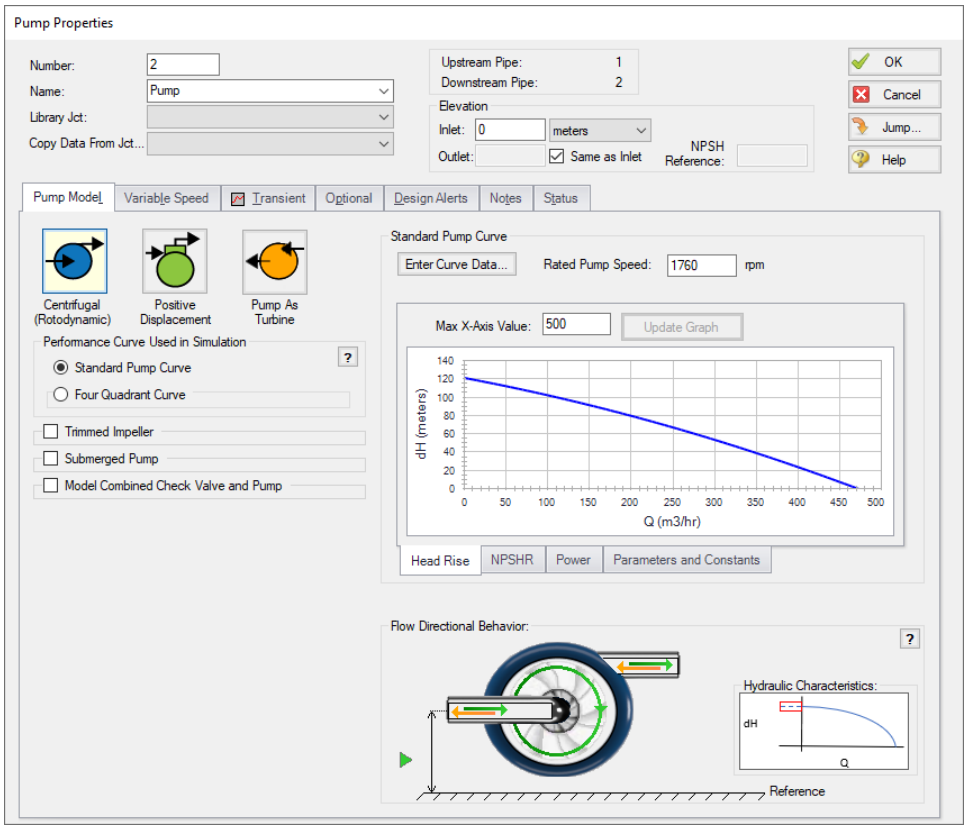


Figure 2: Pump J2 Properties Input

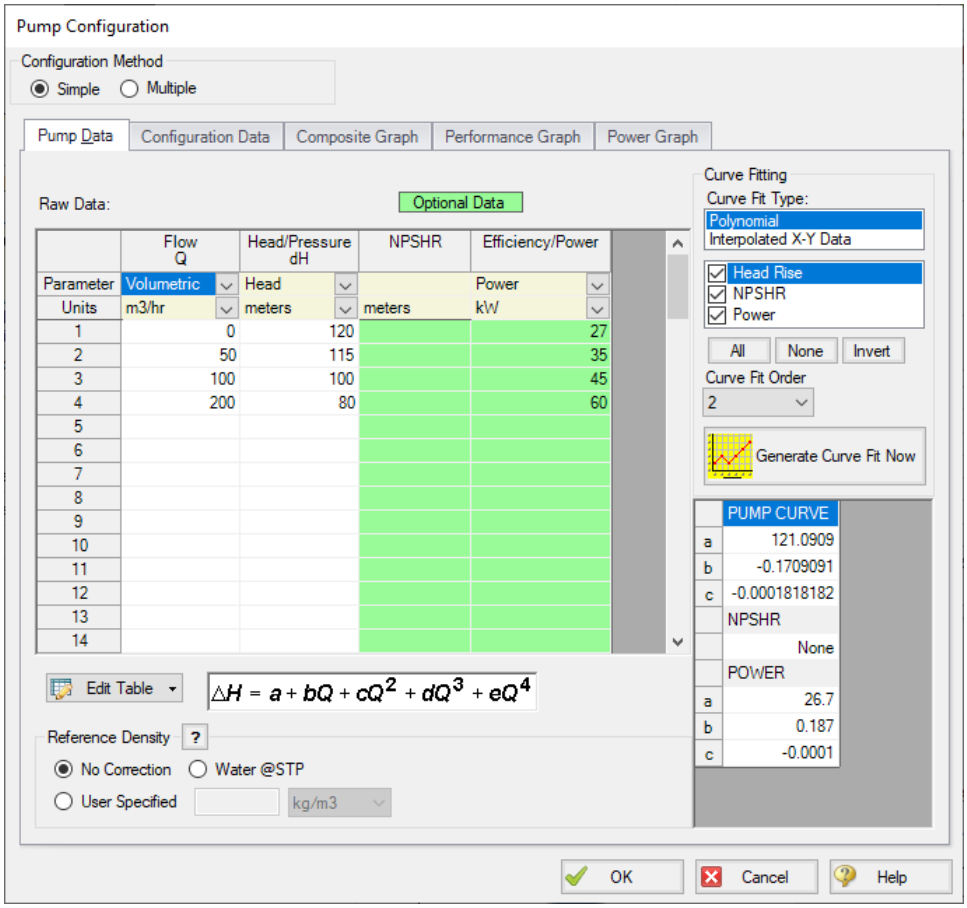


Figure 3: Pump J2 Configuration Data

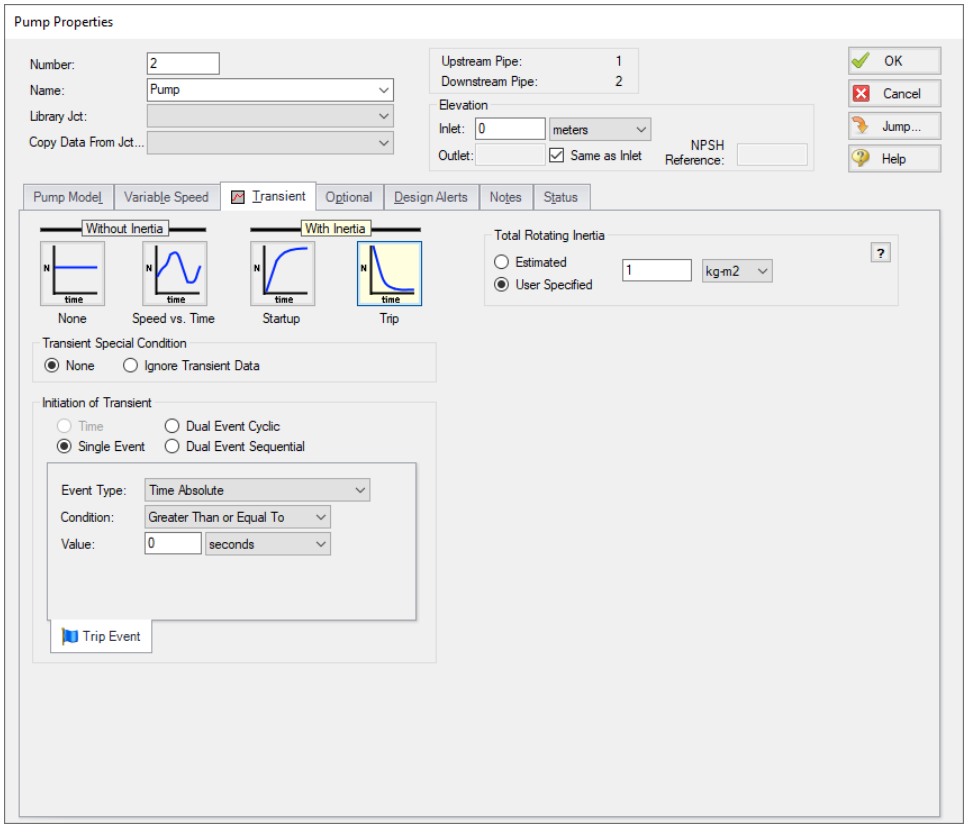


Figure 4: Transient Input Data for Pump J2

3. Check Valve J3
    a. Inlet Elevation = 0 meters
    b. Valve Model = User Specified
    c. Loss Model = Cv
    d. Full Open Cv = 1515
    e. Forward Velocity to Close Valve = -0.05 meters/sec
    f. Delta Pressure/Head to Reopen = Pressure
    g. Delta Pressure = 0 bar
4. Reservoir J4
    a. Name = Upper Reservoir
    b. Liquid Surface Elevation = 90 meters
    c. Liquid Surface Pressure = 10 barG
    d. Pipe Depth = 3 meters

Note: Unless specified otherwise, pipe elevation is assumed to vary linearly between junctions. Therefore, the pipe P2 inlet elevation is 0 meters, and the exit elevation is 80 meters (because the reservoir is 3 meters deep). In between the elevation is linear. If another profile is desired, this can be entered on the pipe's Optional tab in the Intermediate Pipe Elevations section.

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 30 seconds for the Stop Time.

Navigate to the Transient Cavitation panel and select Discrete Gas Cavity Model, the default values that populate will suffice for this example.

All groups should now be complete and the model is ready to run. If all groups in Analysis Setup have a green checkmark then Click OK and proceed. Otherwise, enter the missing information.

Step 6. Run the Model

Click Run Model from the toolbar or from 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 Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

In the Output window there will be a warning shown stating that reverse flow occurs at the pump. If there is a high magnitude of reverse flow at the pump or the reverse flow occurs for a long duration, then we might want to update the pump to use four quadrant data. In this model the magnitude and duration of reverse flow is relatively small, so this warning will be dismissed.

Often when a check valve is present downstream of the pump reverse flow will be negligible and reverse flow warnings can be dismissed; However, it is always good practice to check the volumetric flow rate results at the pump to confirm. The Selecting a Pump Four Quadrant Curve example provides more information on how to perform a sensitivity analysis with four quadrant data if reverse flow is significant at the pump.

Now go to the Graph Results window by clicking the Graph Results tab.

A. Graph the check valve Cv

To confirm the check valve's behavior create a plot of the check valve Cv over time as follows:

1. Select the Transient Jct tab in the Graph Parameter section
2. Add J3 (Check Valve) to be graphed
3. Make sure Check Valve Cv is the Parameter to graph
4. Click Generate

The check valve Cv plot can be seen in Figure 5 below. As is shown in the plot, the check valve closes instantly at about one second, which is the time when the velocity reaches zero at the check valve. The exact closure time can be seen in the Output window by selecting the Event Messages tab in the top third of the window. To model a slower closure a different check valve model will need to be chosen. This will be shown later in the example.

➢**Right click on the graph and select Add Graph to List.** Name this graph *Check Valve Cv*. Your graphing parameters will be saved to the Graph List Manager in the top right corner of the Graph Results Window. This enables you to quickly graph these same parameters in different scenarios. We will create scenarios for comparison later in this example.

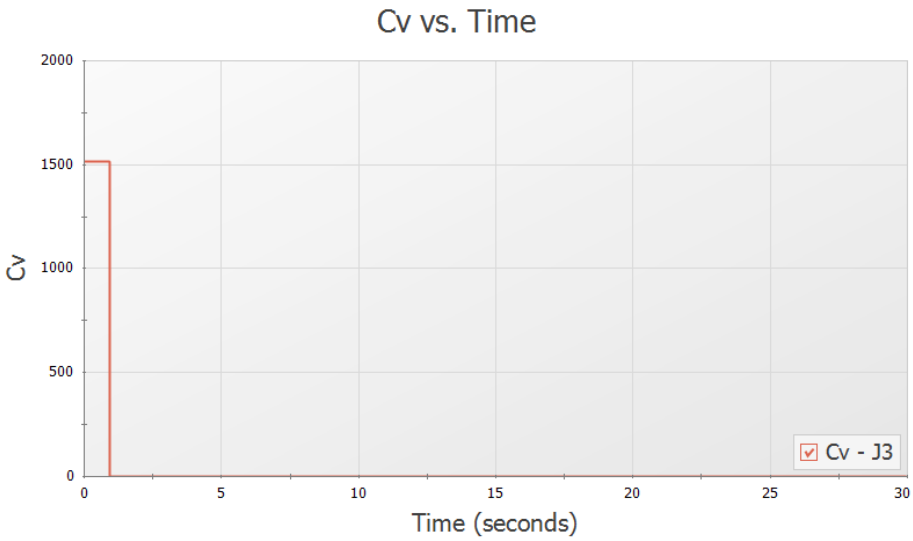


Figure 5: Check Valve Cv vs time using the user specified check valve model

B. Graph the transient pressures at the check valve

1. Still on the Graph Results window, go to the Transient Pipe tab, add the Pipe 2 Outlet and Pipe 3 Inlet. These represent each side of the check valve.
2. Make sure the selected parameter is Pressure Static.
3. Set the units to barG.
4. Click Generate.

The resulting pressure transients are shown in Figure 6. It should be shown that after the valve closes, pressure between the pump and the check valve is brought to equilibrium while pressure after the check

valve will experience water hammer. The maximum and minimum values can be shown by the Show Cross Hair tool in the Graph Results toolbar.

➢**Right click on the graph and select Add Graph to List.** Name this graph *Check Valve Inlet/Outlet Pressure*.

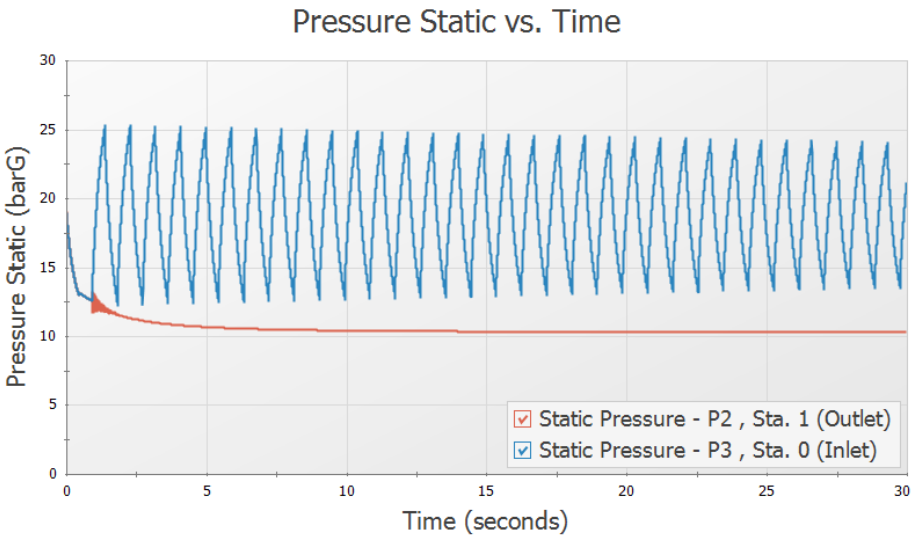


Figure 6: Predicted transient pressures at check valve inlet and outlet

C. Graph the pipeline transient pressure profile

This animation will illustrate the static pressure throughout the pipe system over time.

1. Select the Profile tab on the Quick Access Panel (Figure 7).
2. In the Pipes area click All to select pipes 1-3.
3. Animate Using: Solver
4. In the Parameter area make sure Pressure Static is selected.
5. For units select barG.
6. Click Generate.

Once the graph has been generated, press the Play button on the Graph Results animation controls as is shown in Figure 8 to observe how the pressure profile varies over time. At around 1 second the check valve closes, causing the maximum pressure spike observed in the system. The resulting pressure wave can then be seen moving along the pipe to the reservoir and being reflected back.

Right click on the graph and select Add Graph to List. Name this graph *System Pressure Profile*.

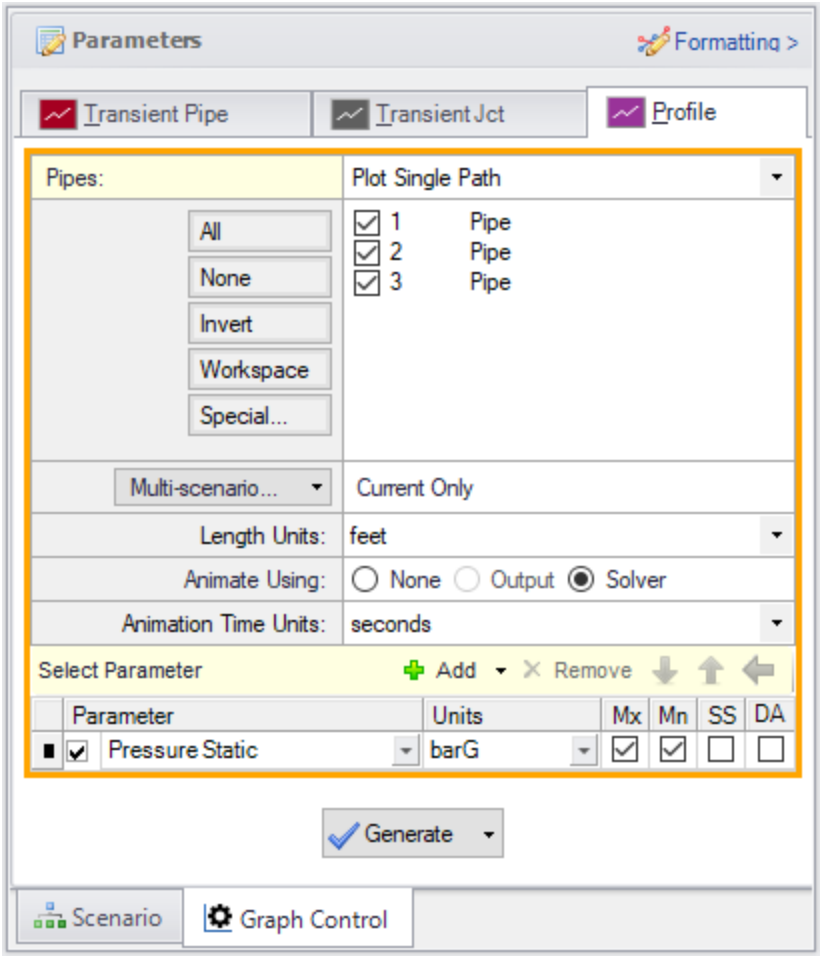


Figure 7: System Pressure Profile Parameters

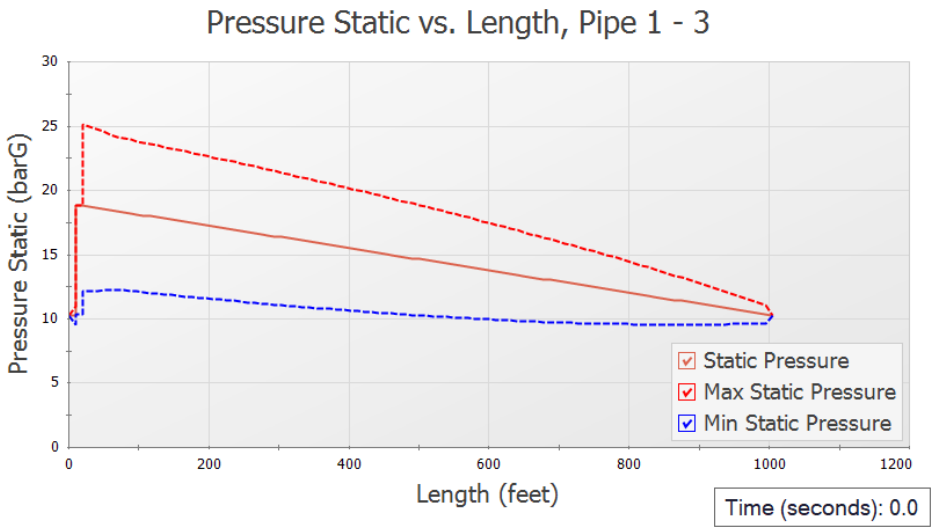


Figure 8: System Pressure Profile Animation Snapshot

Step 8. Create Child Scenarios

Often the forward velocity to close and delta pressure to reopen the check valve are not directly known for the system as was shown above. However, data such as a reverse velocity vs deceleration chart, or information on the valve dimensions may be provided. In that case either the Estimate From Fluid Deceleration or Force Balance check valve model may be more appropriate.

The Estimate From Fluid Deceleration model type for a check valve requires that reopening is not possible. In order to model this scenario accurately, you will need valve performance data for the maximum reverse velocity vs fluid deceleration. If you model a valve with a preset that does not accurately represent your system, this could yield inaccurate results.

Modeling a check valve with the force balance model is ideal for either swing check valves or nozzle/plug check valves where you know the weight and dimensions of your check valve.

We will create scenarios to model each of the check valve types.

Open the Scenario Manager on the Quick Access Panel. Create a child scenario by right-clicking on the Base Scenario and then selecting Create Child. Do this four times, naming each one after the following check valve model types:

1. User Specified
2. Estimate from Fluid Deceleration - Dimensional
3. Estimate from Fluid Deceleration - Non-dimensional
4. Force Balance

Load the Estimate from Fluid Deceleration - Dimensional scenario by double-clicking the scenario name.

Step 9. Define and Run the Estimate from Fluid Deceleration Scenarios

This modeling method is ideal for when you have Valve Performance Data / Fluid Deceleration to help define your check valve, or if you are using a check valve which is geometrically similar to the valves which have deceleration data provided in AFT Impulse (Figure 9). The valves from Ballun's data set were ANSI Class 125 size 8-inch valves. The data sets for Thorley were obtained using 4-inch and 12-inch valves. For more information on the test setups used by Ballun and Thorley see the check valve topic in the main help file.

A. Define the Estimate from Fluid Deceleration - Dimensional Scenario

Open J3 Check Valve properties window. Under Fluid Velocity at Valve Closure select Estimate from Fluid Deceleration (no reopen), as shown in Figure 9.

There are several Valve Performance Data Type options which can be selected. If you have a Deceleration vs Maximum Reverse Velocity chart provided by the valve manufacturer, the User Defined option should be selected, and the data entered. Otherwise, one of the data sets provided by Impulse can be used with either the Non-dimensional or Dimensional option. The dimensional option should be used if the valve in your system is similar in both geometry *and* size to the data set valve type. If your valve has similar geometry but is a *different* size than the valves tested to obtain the data set, then the Non-dimensional option should be used.

The dimensional model is applicable for this example because the check valve being modeled is an 8-inch swing-flex check valve, which is similar in both geometry and size to the Resilient Hinge DN 200 check valve used in the Ballun data set.

Define the check valve as follows:

1. Valve Performance Data Type = Dimensional
2. Valve Type = Resilient Hinge DN 200 (B)

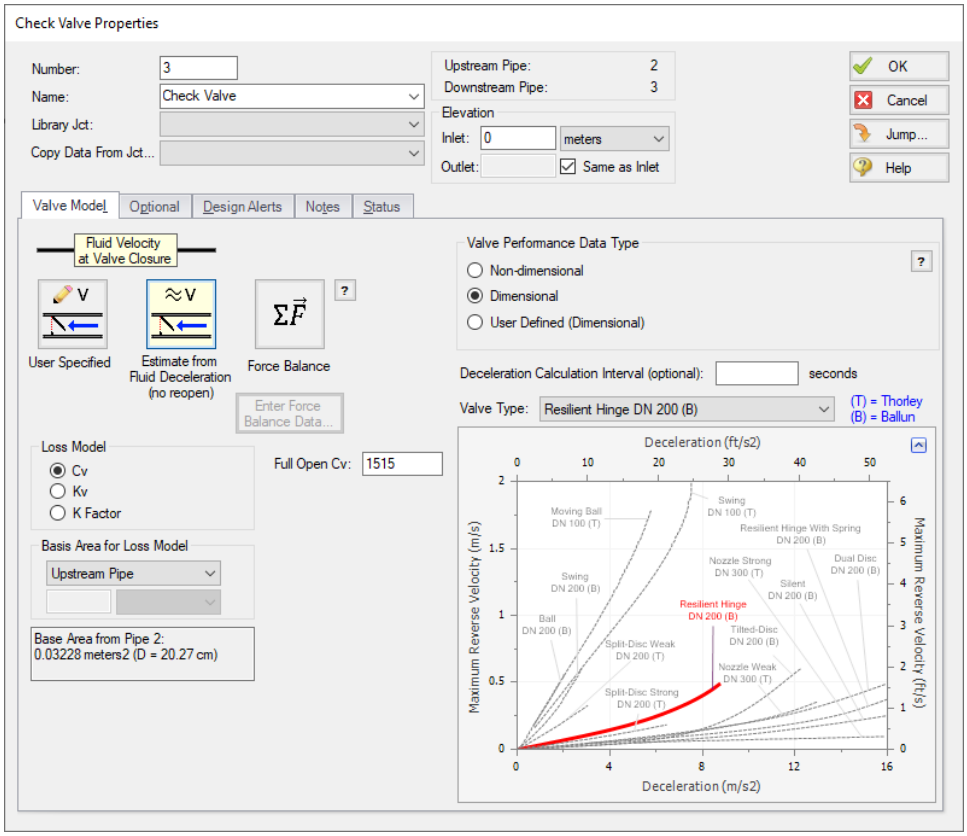


Figure 9: Estimate From Fluid Deceleration Properties

B. Define the Estimate from Fluid Deceleration - Non-dimensional Scenario

Load the Estimate from Fluid Deceleration - Non-dimensional scenario by double-clicking the scenario name in the Scenario Manager.

Define the check valve as follows:

1. Fluid Velocity at Valve Closure = Estimate from Fluid Deceleration (no reopen)
2. Valve Performance Data Type = Non-dimensional
3. Minimum Velocity Required to Fully Open Valve (Vo) = 1.98 meters/sec
4. Valve Type = Resilient Hinge (B)

Note that there is a new parameter which is required to be entered when the non-dimensional valve type is used, the Minimum Velocity Required to Fully Open Valve (Vo). The Vo can be calculated through testing, or by consulting the valve manufacturer. For the purpose of this example the Vo was obtained by using the dimensional and non-dimensional deceleration charts provided by the manufacturer.

C. Run and Compare Results

Run the two Estimate from Fluid Deceleration scenarios and go to the Output window. Note that a warning is once again given for reverse flow at the pump. We will again dismiss this warning as the magnitude of reverse flow is low.

Next go to the Graph Results window. Regenerate each of the graphs for this scenario using the Graph List items by double-clicking the graph names in the Graph List Manager.

Notice that using the given information for the check valve these two models produce practically identical results to the user specified check valve model, since the check valve closes at a reverse velocity of about -0.05 meters/sec in all three scenarios. On the Event Messages by Jct tab in the Output window it can be seen that the forward velocity to close was actually calculated as about -0.05099 meters/sec for the dimensional case and -0.05350 meters/sec for the non-dimensional case. Figure 10 below shows the Event Messages tab for the non-dimensional case.

Similar to the user specified model, the Estimate from Fluid Deceleration valve model closes the valve instantaneously once the calculated forward velocity to close has been reached. However, unlike the User Specified valve model, the fluid deceleration option will never allow the check valve to reopen, so this option should be used carefully.

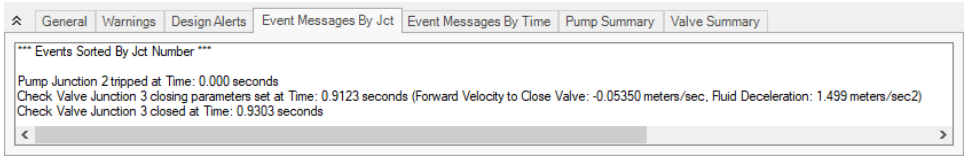


Figure 10: The Event Messages tab provides additional information on the fluid deceleration calculations at the check valve. The non-dimensional case is shown here

Step 10. Define the Force Balance Scenario

This modeling method is useful when you have information on the dimensions of the check valve in your system. This information may be available from the valve manufacturer, though in many cases it must be approximated. You can model swing check valves or translating nozzle check valves with the Force Balance method.

For this example we will use approximations of the valve dimensions from a schematic provided by the manufacturer to model the 8-inch Resilient Hinge swing check valve.

A. Enter the Check Valve Properties Data

Double click on J3 Check Valve.

1. Fluid Velocity at Valve Closure = Force Balance
2. Enter Force Balance Data =
    a. Valve Type = Swing Check
    b. Distance to CG (L) = 16.4 cm
    c. Area (A) = 697 cm2
    d. Weight (submerged, W) = 91.8 N

e. Inertia (I) = 0.353 kg-m2
f. Mass of Displaced Fluid = Sphere (disk diameter, recommended)

Note: Equations to approximate the submerged weight and inertia of the disk can be found in the AFT Impulse Help file, accessed from the ? buttons.

e. θ when Fully Open = 10 deg
f. θ when Fully Closed = 45 deg
g. Valve Loss vs. Position (θ) =

θ (deg)	Cv
10	1515
45	0

Note: For this example the Valve Loss vs. Position (θ) profile was assumed to be linear. For a more realistic force balance detailed information should be obtained and entered for the Cv profile from the valve manufacturer.

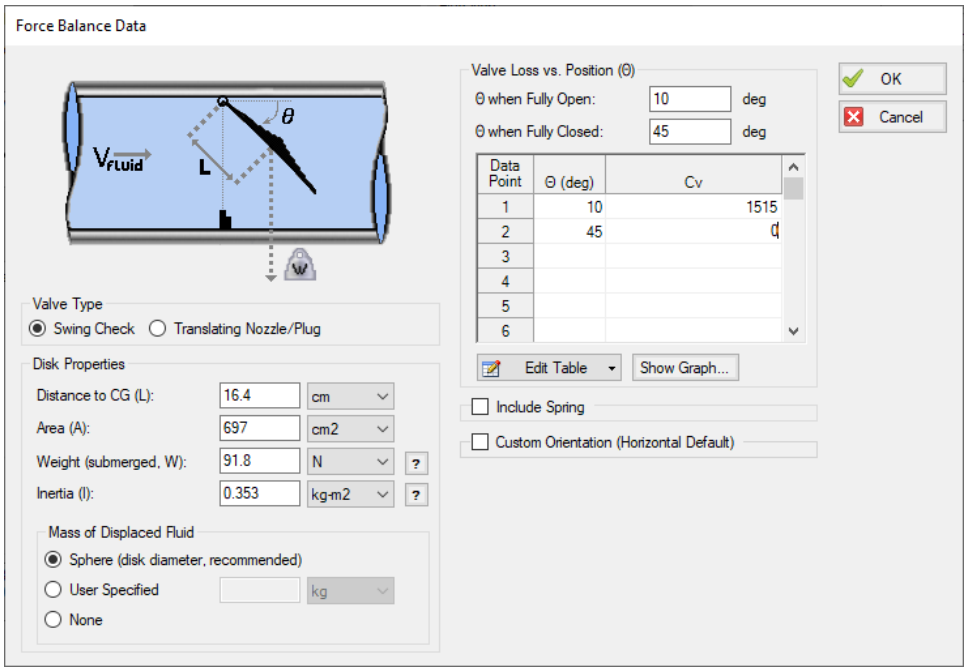


Figure 11: Force Balance Data

B. Run and Compare Results

Run the model and go to the Graph Results window.

Regenerate the pressure Check Valve Inlet/Outlet Pressure graph, as can be seen in Figure 12. Compared to the other three scenarios, the force balance check valve closure produces slightly larger pressure oscillations, with the amplitude of the oscillations increased by about 1 bar.

It can be seen from the check valve Cv graph (Figure 13) which displays the first 1.5 seconds of the run, that the check valve does not close over a single time step like in the previous scenarios, but instead closes rapidly over 1 second. Using a realistic Cv profile for the force balance input will allow for the most realistic calculation of this closing profile.

A comparison of the velocity at the check valves over the first 1.5 seconds can be seen in Figure 14. From the velocity graph it can be seen that the force balance check valve calculated a maximum reverse velocity of about -0.092 meters/sec. The higher reverse velocity when using the force balance model can likely be attributed to the assumptions made when entering the geometry data for the check valve. Some of the primary assumptions made for this example were that the Cv vs open percent profile was linear, and that the check valve disk was a perfect cylinder.

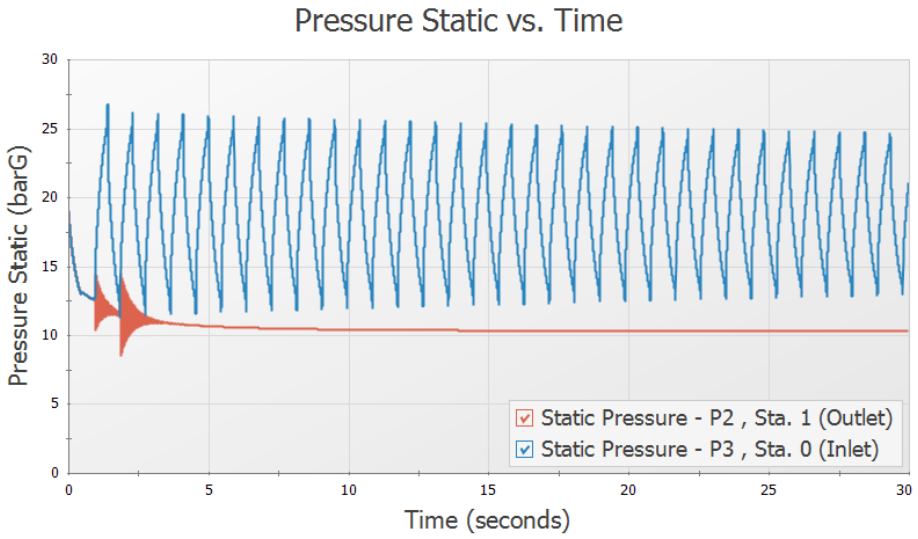


Figure 12: Pressure at check valve inlet/outlet for Force Balance Model

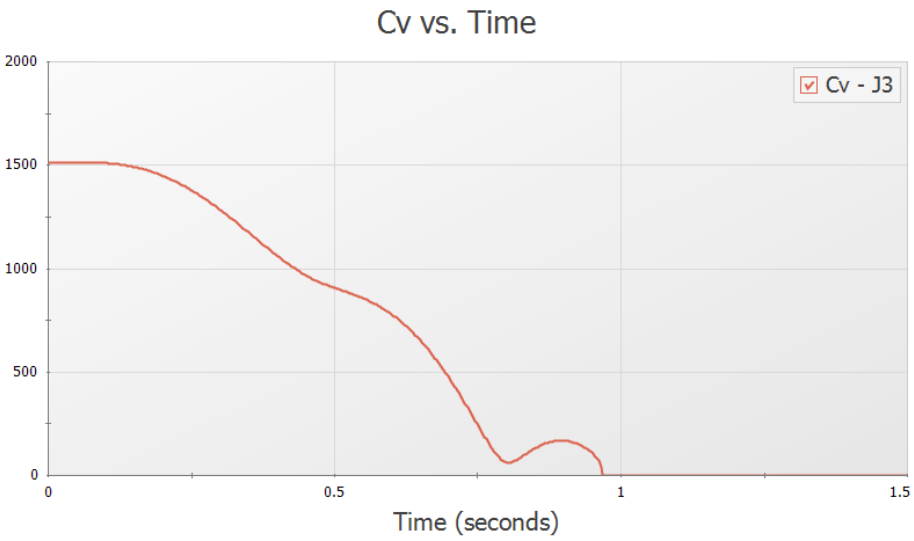


Figure 13: Check Valve Cv for Force Balance Model

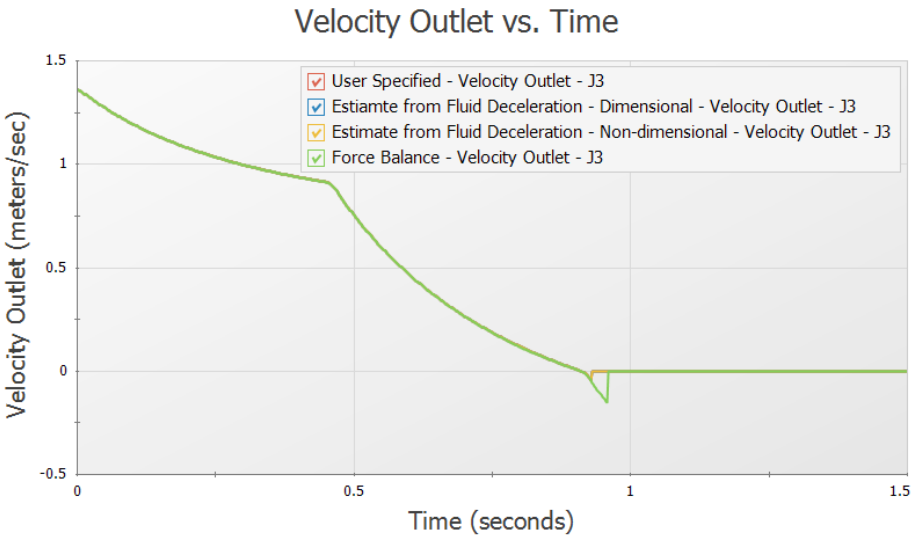


Figure 14: Velocity at the check valve outlet for each of the check valve types plotted for the first 3 seconds

Summary

Check Valves can be modeled in multiple ways depending on the information you have available to you. They can be modeled as combined with the pump outlet, with user specified fluid velocity, with estimated fluid velocity from fluid deceleration data, or with a force balance calculation. With enough reliable data, you should be able to get similar results with any modeling method.

Relief Valve Modeling

Summary

A system pumping crude oil over a long distance experiences a sudden valve closure. The pump is not able to stop before the valve is completely closed. A relief valve will be considered for surge mitigation.

Topics Covered

- Modeling relief valve profiles during pressure surges
- Examining the effect of valve rate limits on a system
- Observing differences between relief valve profiles

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model Files

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

- ***Metric - Relief Valve Modeling.imp***
- ***Relief Valve Modeling Pump.dat***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = User Specified Fluid
b. Name = Crude Oil
c. Density = 0.85 S.G. water
d. Dynamic Viscosity = 4 centipose
e. Bulk Modulus = 13800 bar
f. Vapor Pressure = 0.69 bar

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

We will first evaluate the system without a relief valve to see if the maximum pressure of 25 bar is exceeded.

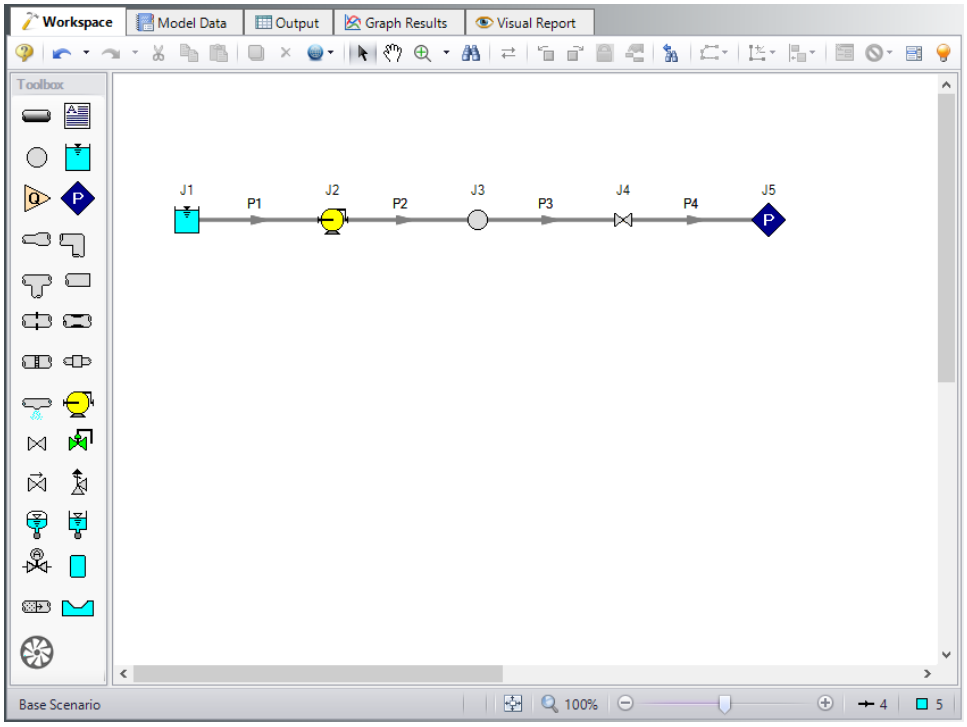


Figure 1: Workspace with no relief valve added

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Size = 24 inch
    c. Type = STD (schedule 20)
    d. Friction Model Data Set = Standard
    e. Length = Use table below

Pipe	Length (meters)
1	6
2	3000
3	6
4	6

Junction Properties

To define the pump, a pre-built library will be used. We will need to first connect the library as follows

a. From the Library menu select Library Manager.
b. At the bottom, click Add Existing Library.
c. Browse to the ***Relief Valve Modeling.dat*** library file in the Impulse Examples folder and click OK. The library should now appear in the Library Browser under the name Relief Valve Modeling. Make sure the box is checked next to this library.
d. Click OK to exit the Library Manager.

Now proceed with defining the junctions.

1. J1 Reservoir
    a. Tank Model = Infinite Reservoir
    b. Liquid Surface Elevation = 3 meters
    c. Liquid Surface Pressure = 10 bar
    d. Pipe Depth = 3 meters
2. J2 Pump
    a. Library Jct = Metric - Crude Oil Pump
    b. Inlet Elevation = 0 meters

Note: The Model Combined Check Valve and Pump Option is used in this example to simplify the model. For higher accuracy it is generally recommended to model a check valve at the pump discharge using a Check Valve junction instead. See the Check Valve Modeling example for more information.

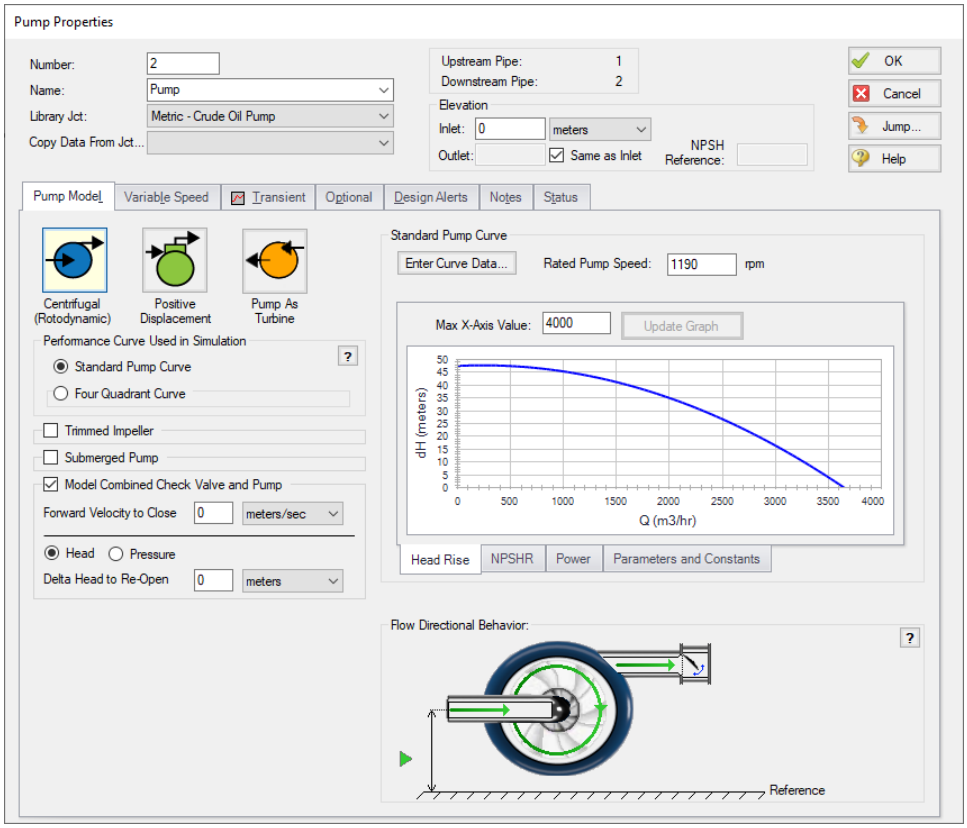


Figure 2: Pump Model tab with pump information from library

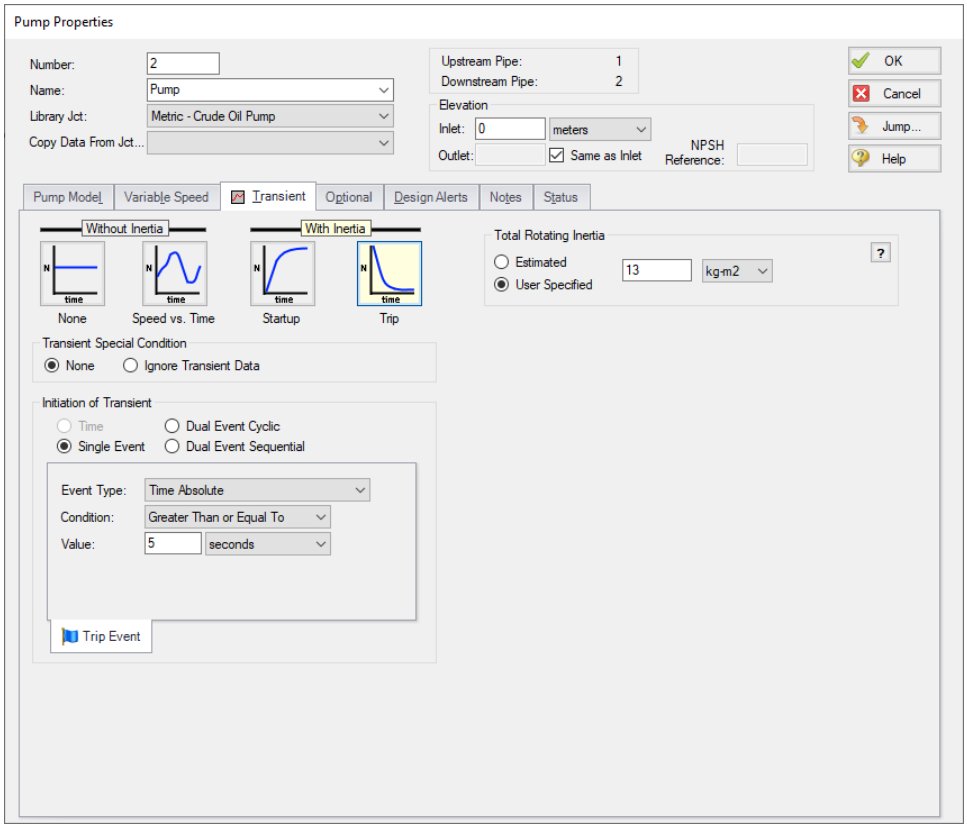


Figure 3: Pump Properties Transient Data

3. J3 Branch
    a. Elevation = 0 meters
4. J4 Valve
    a. Inlet Elevation = 0 meters
    b. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Cv = 10000
    c. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	10000
1	2000
2	0

Time (seconds)	Cv
30	0

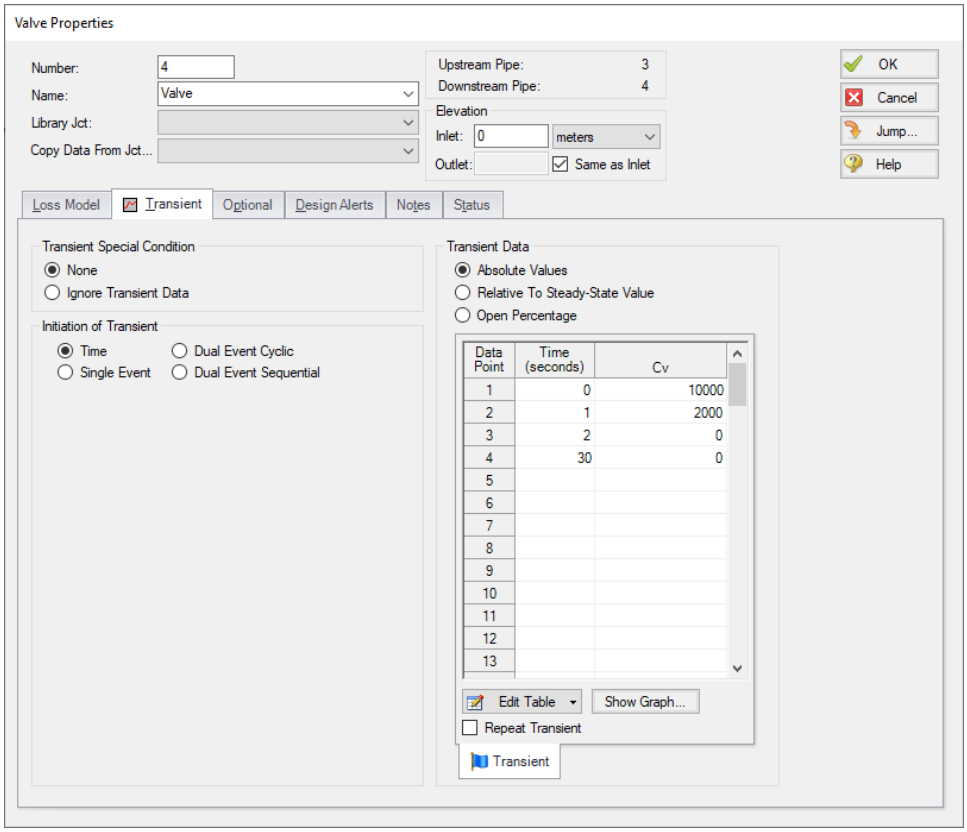


Figure 4: Valve Properties Transient Data

5. J5 Assigned Pressure
    a. Elevation = 0 meters
    b. Pressure = 12 bar
    c. Pressure Specification = Stagnation

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 30 seconds for the Stop Time.

All groups should now be complete and the model is ready to run. If all groups in Analysis Setup have a green checkmark then Click OK and proceed. Otherwise, enter the missing information.

Step 6. Run the Model

Click Run Model from the toolbar or from 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 Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

The maximum pressure requirement for the line is 25 bar. A profile graph should be created to review the maximum pressure along the line as follows:

1. Go to the Workspace and select all of the pipes
2. Right-click the Workspace and select Generate Graph for Pipes > Pressure Static
3. Click Add Graph to List and name it *Static Pressure Profile*

The maximum/minimum pressure profile graph can be seen in Figure 5. The pressure goes above the maximum pressure requirement for the majority of the line. To decrease the maximum pressure we will add a relief valve to the line near to the closing valve, which is the source of the pressure surge.

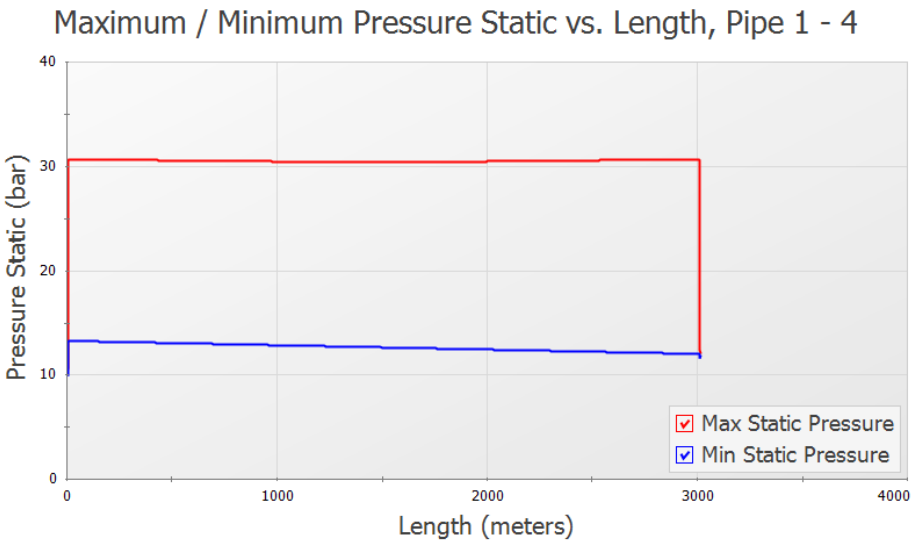


Figure 5: Maximum/minimum pressure profile along the pipeline

Step 8. Create Child Scenarios

We will create two scenarios to compare the model with and without a relief valve.

➢**In the Scenario Manager right-click the Base Scenario** and click Create Child. Name the scenario *No Relief Valve*. Repeat this process to create a second scenario, *Passive Relief Valve*.

Now we will add the relief valve to the Passive Relief Valve scenario. Make sure the relief valve scenario is active in the Workspace by double-clicking the scenario name, then add a relief valve line as shown in Figure 6.

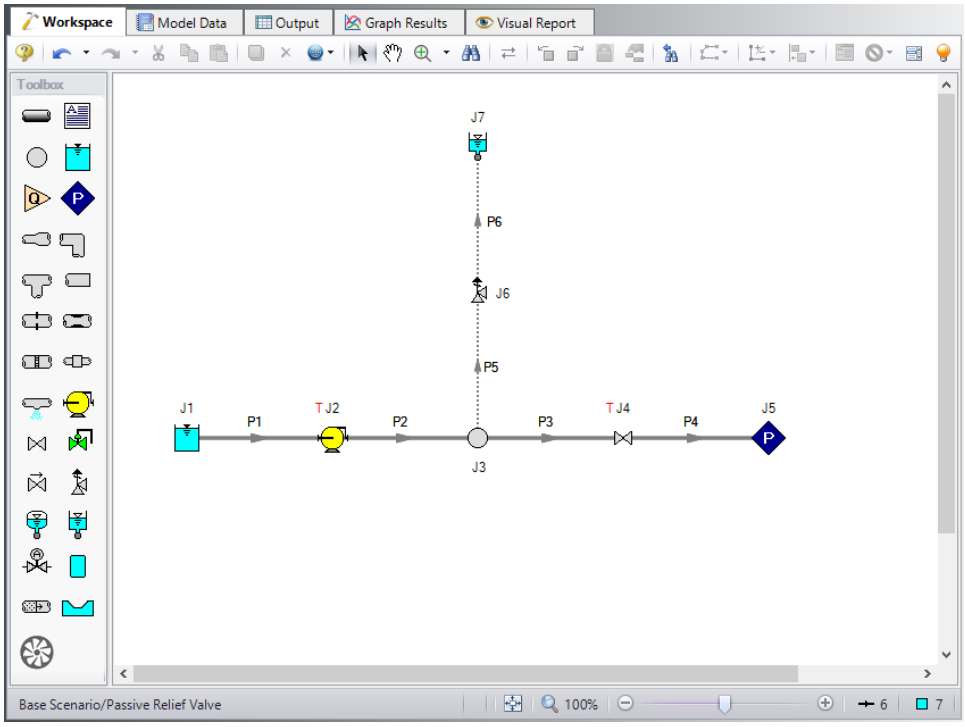


Figure 6: Workspace with relief valve and vessel added to the line

The crude oil will need to be contained in a closed vessel at the exit of the relief valve. For this example we will assume that the transient is fast, thus the pressure in the closed vessel will not change during the transient. The surge tank junction will be used to represent the closed, pressurized vessel. If the pressure in the closed vessel were to change significantly during the transient, then the surge tank would not be a good representation of this.

Define the additional junctions and pipes as follows.

Pipe Properties

3. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Size = 12 inch
    c. Type = STD
    d. Length = 6 meters

Junction Properties

1. J6 Relief Valve
    a. Inlet Elevation = 0 meters
    b. Valve Model tab

a. Valve Model = Internal
b. Profiles = Passive
c. Pressure Type = Pressure
d. Constant Backpressure (Balanced) = Checked
e. Overpressure = 22.5 bar
f. Set Pressure = 20 bar
g. Blowdown Pressure = 15 bar

c. Loss Model tab
   a. Loss Model = Cv (Variable)
   b. Variable Data = Linear based on setpoints
   c. Full Open Cv = 500

Note: The Ignore Relief Valve option on the Optional tab can be used to keep the relief valve closed. This is useful for running cases with no relief valve in the system, or where the relief valve fails to open without deleting the relief valve junction.

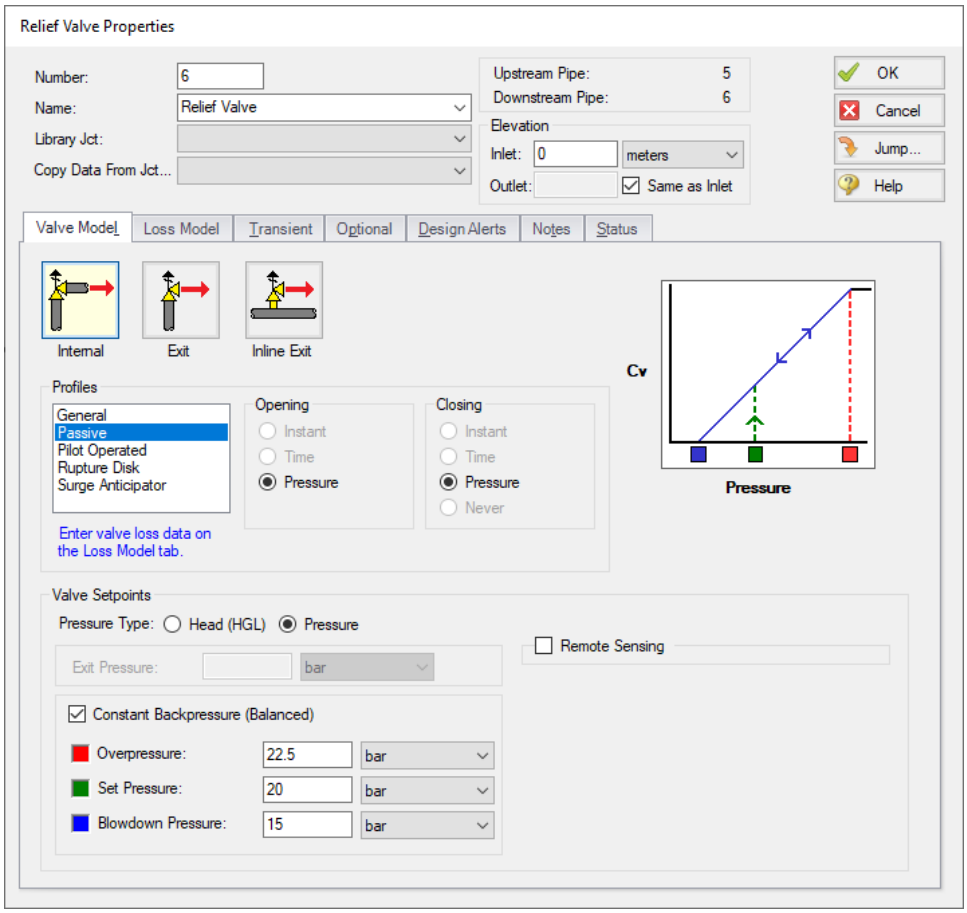


Figure 7: Relief Valve Model Properties

2. J7 Surge Tank
    a. Cross-Sectional Area = Constant
    b. Value = 100 meters2
    c. Initial State of Liquid Height = Liquid Height Known (Possible Inflow/Outflow)
    d. Surface Pressure = 11 bar
    e. Initial Liquid Height = 0 meters
    f. Junction Elevation = 0 meters

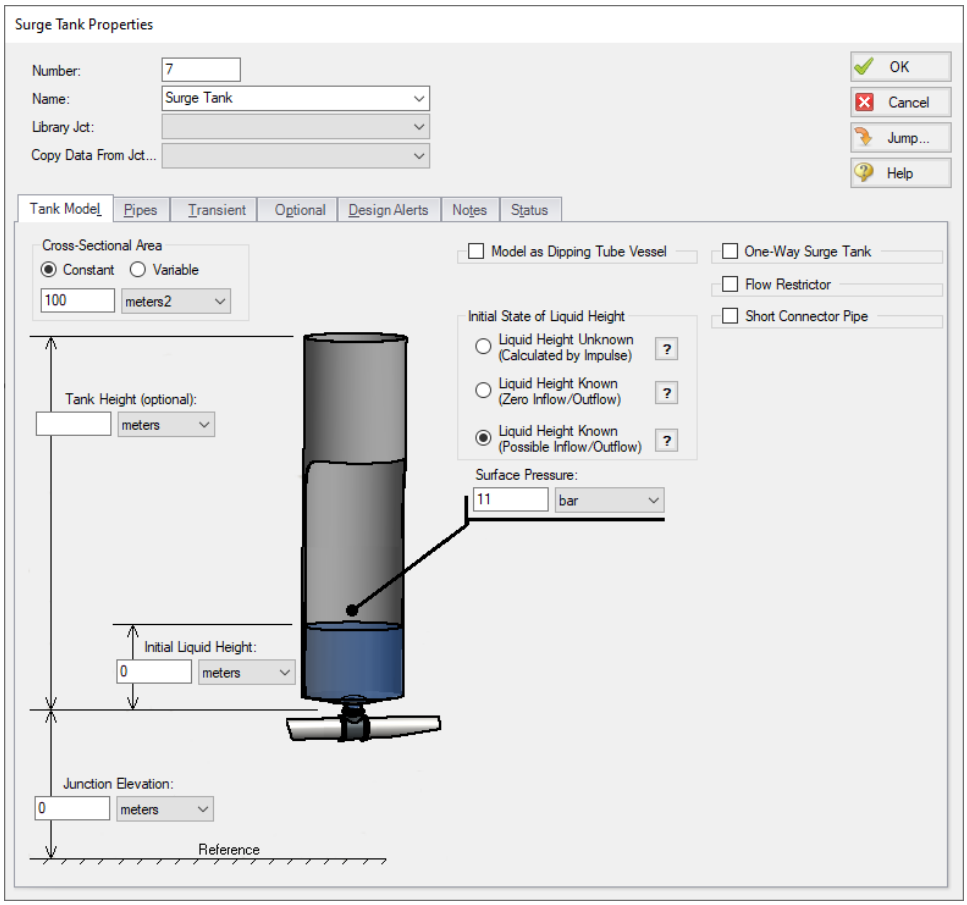


Figure 8: Surge Tank Properties

Step 9. Define the Pipe Sectioning and Output Group

New pipes were added to the model, so the model needs to be sectioned again.

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 10. Run the Model

Click Run Model from the toolbar or from the Analysis menu. When complete, click the Graph Results button at the bottom of the Solution Progress window.

Step 11. Examine the Output

Reload the Static Pressure Profile graph item by double-clicking the graph item in the Graph List Manager. The pressure profile should now appear as shown in Figure 9 below.

With the relief valve the maximum pressure reached in the pipeline is about 21.6 bar, which is below the required maximum pressure of 25 bar.

Now create a Transient Jct graph for the relief valve, and select Cv as the parameter to be graphed (Figure 10). Save this graph as *Relief Valve Cv* in the Graph List Manager.

Taking a closer look at the graph it can be seen that the valve opens over about 0.2 seconds, and closes over about 2.3 seconds(time from maximum Cv to zero). This is by design - the relief valve should open quickly to reduce pressure, but close more gradually to avoid creating a pressure wave. When defining the relief valve for this model we did not take into account the actual closing time that will occur at the relief valve. We will add rate limits to the model to better account for the closing time at the valve.

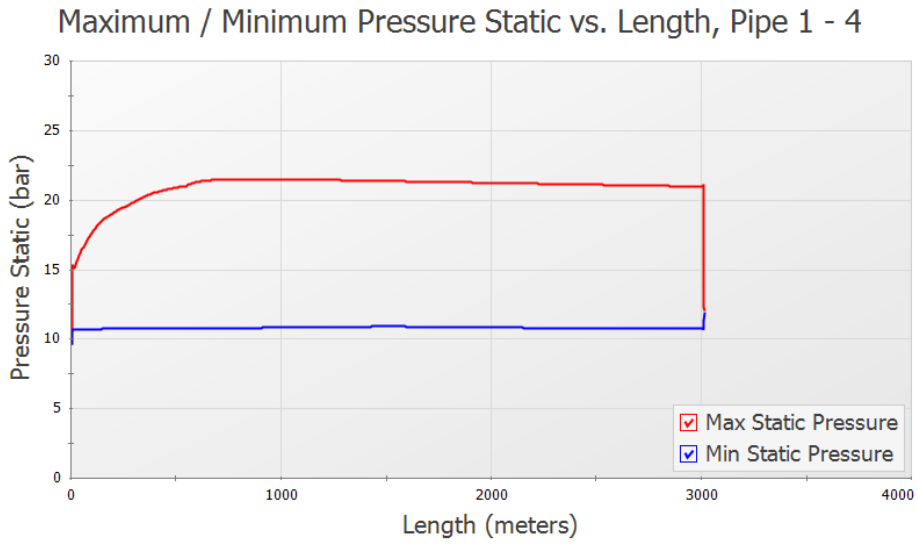


Figure 9: Maximum/minimum pressure profile along the pipeline with a passive relief valve

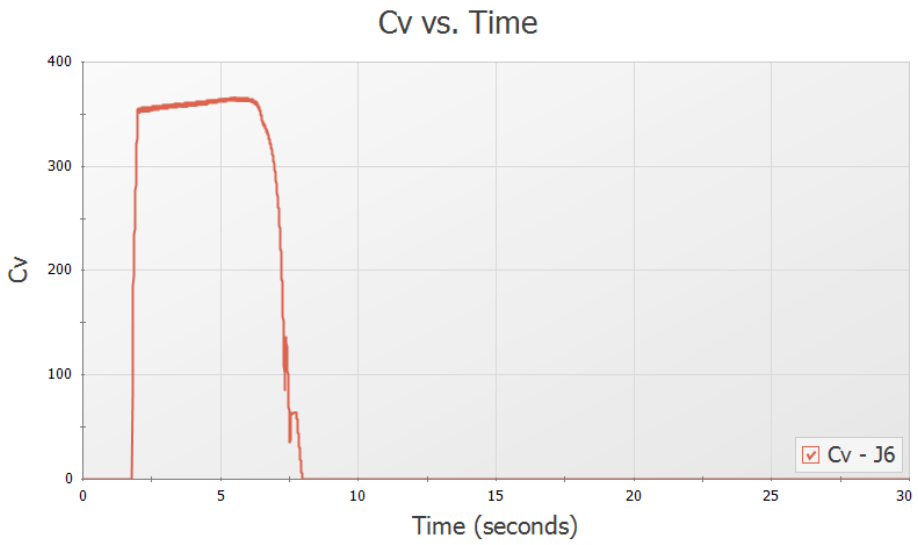


Figure 10: Relief valve Cv vs time

Step 12. Add Rate Limits to the Relief Valve

A relief valve's rate limit represents the rate at which the valve can open and close. If no rate limit is defined for the relief valve, it is possible for the valve to open/close nearly instantaneously. An instant opening for a relief valve is idealized, as the high pressures can be instantly alleviated in the pipeline once the valve setpoint is reached. Instant closure of the relief valve would represent the worst case scenario, as the rapid closure would produce the maximum pressure rise at the relief valve. In reality it is not physically possible for the relief valve to close in one time step. In addition, relief valve manufacturer's often design the valves to close slowly to avoid surge events that could occur due to a sudden closure. Adding rate limits allows for a more realistic representation of how fast the valve position can change over time.

Add rate limits to the relief valve as follows:

1. In the Workspace window, under Scenario Manager, create a new child scenario under the Passive Relief Valve scenario named *With Rate Limits*, and load it as the active scenario.
2. Open the properties window for Relief Valve J6.
3. On the Loss Model tab check the boxes next to both Apply Closing Limit and Apply Opening Limit (Figure 11)
4. Add a Max **Closing** Cv change of 150 and a Max **Opening** Cv change of 1500
5. Click OK
6. Run the Model, click on Graph Results
7. Open your saved graph Relief Valve Cv under Graph List Manager
8. In the Graph Parameters section, click the Multi-scenario button, then select the With Rate Limits and Passive Relief Valve scenarios.

9. Click Generate to create the graph as shown in Figure 12.
10. Repeat steps 7 and 8 above for the Static Pressure Profile graph, as shown in Figure 13.

In Figure 12 it can be seen that though the relief valve still opens in about 0.2 seconds, the relief valve now closes at a slower rate of about 3.3 seconds.

From Figure 13 it can be seen that the slower opening speed results in a higher maximum pressure in the pipeline since the relief valve takes longer to fully open. The longer closing time practically eliminates surge waves caused by the closure of the relief valve, which causes a slightly lower minimum pressure along the pipeline.

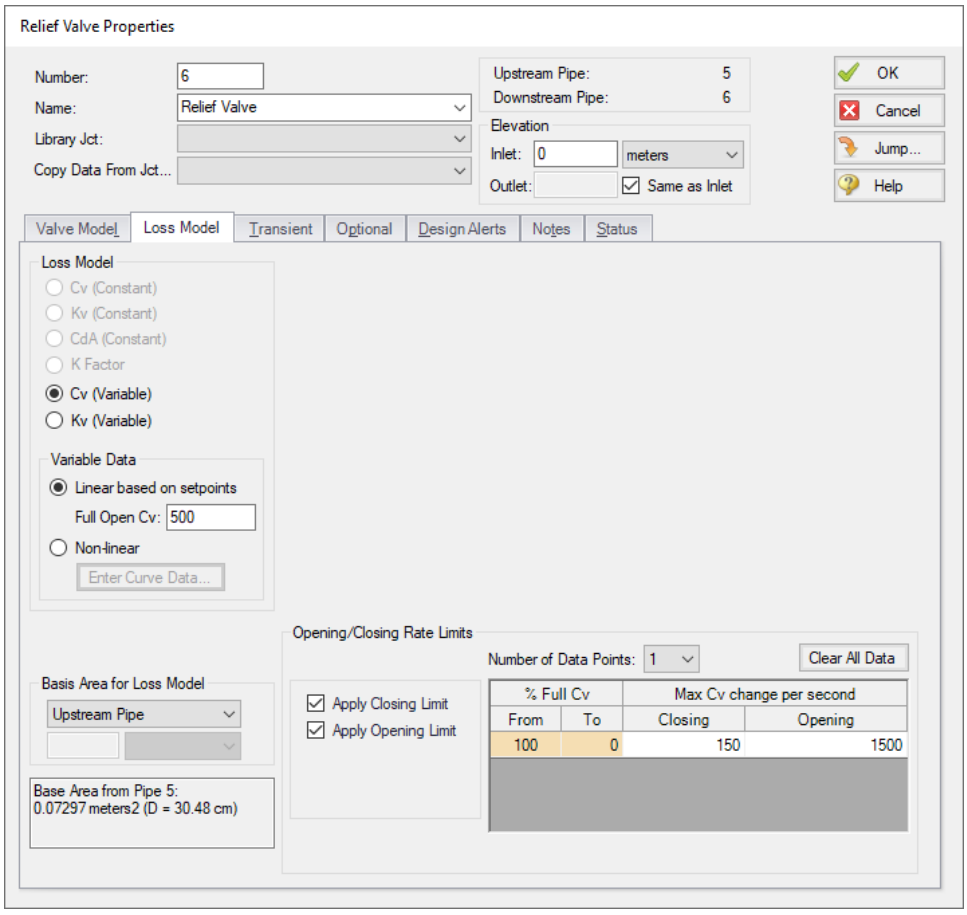


Figure 11: Relief Valve Loss Model tab with opening and closing rate limits defined

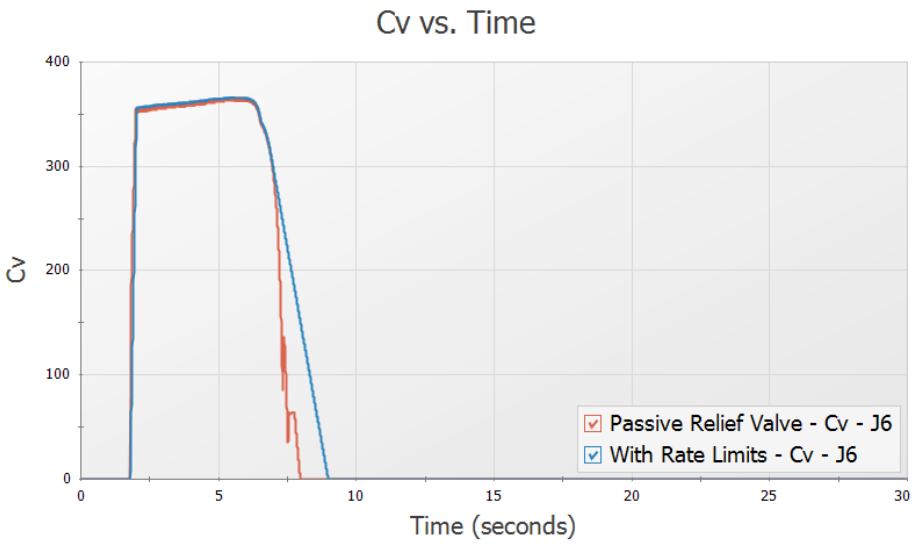


Figure 12: Relief valve Cv profile with and without rate limits

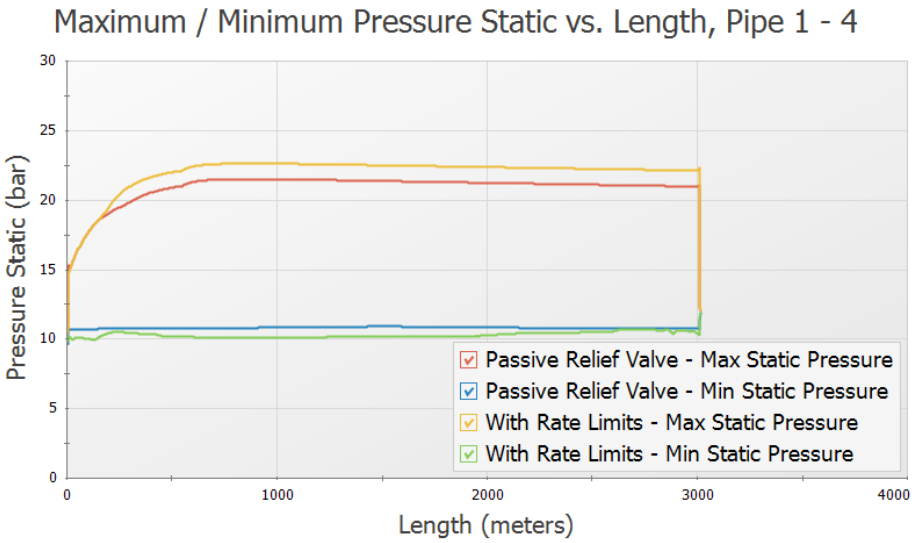


Figure 13: Comparison of maximum and minimum pressures along the pipeline when the passive relief valve is defined with and without rate limits

Step 13. Rupture Disk

We are now going to compare the passive valve above to a rupture disk relief valve. A rupture disk will burst open when the set pressure is met, and will remain open after that. The rupture disk will still relieve

to a closed tank to keep the crude oil contained.

Update the model as follows:

1. In the Workspace window, under Scenario Manager, right click on the Passive Relief Valve scenario and choose Clone Without Children. Name the new scenario *Rupture Disk*.
2. Double click on the Relief Valve J6.
3. In the Profiles list select Rupture Disk (Figure 14)
4. Set Pressure (dP) = 10 bar

Note: The rupture disk will open on a pressure difference, unlike the passive relief valve which was defined using a fixed setpoint at which the relief valve would open. This is because the passive relief valve modeled previously had a mechanism to maintain a constant back-pressure on the valve, which is not used with the rupture disk. The pressure difference setpoint for the rupture disk was found by subtracting the outlet tank pressure from the desired opening pressure.

5. In the Loss Model tab, Cv (Constant) = 300
6. Click OK to close the window.

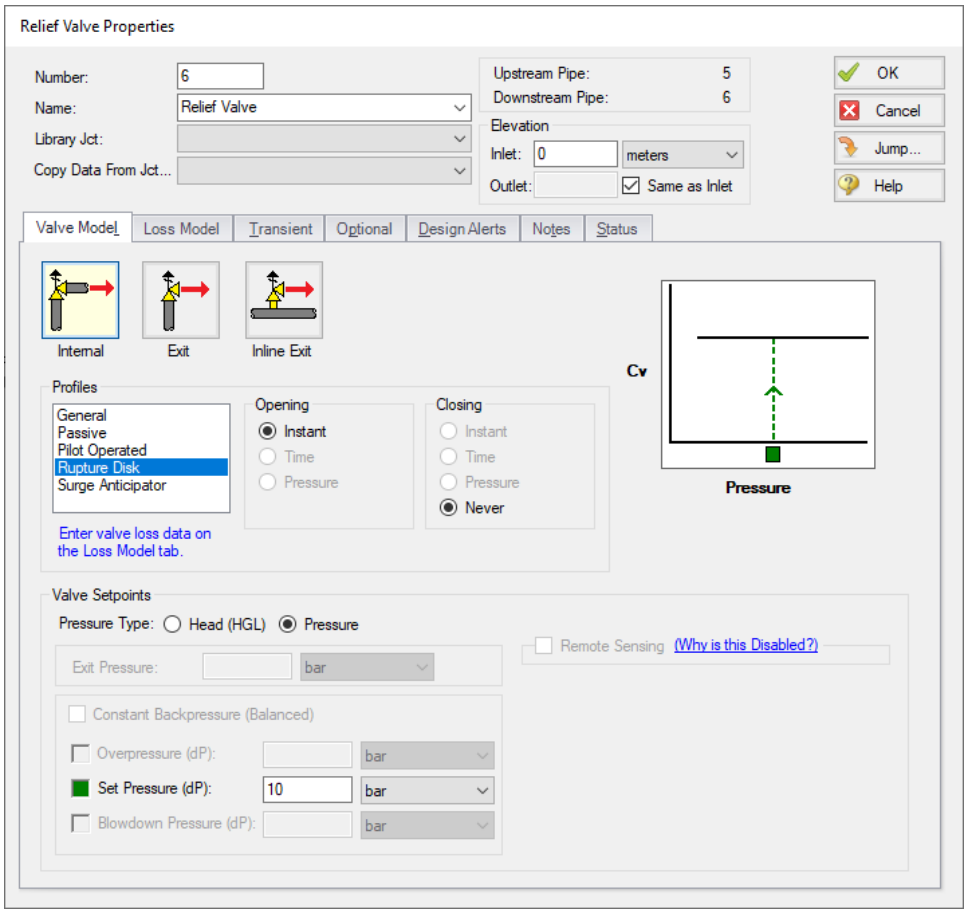


Figure 14: Inline Exit Rupture Disk Properties

The scenario is now complete and ready to be run.

➢**Run the Model and select Graph Results.**

Load the Relief Valve Cv graph from the Graph List Manager, as is shown in Figure 15. Unlike the passive relief valve type, the rupture disk opens instantly, and remains open, as can be seen in the graph.

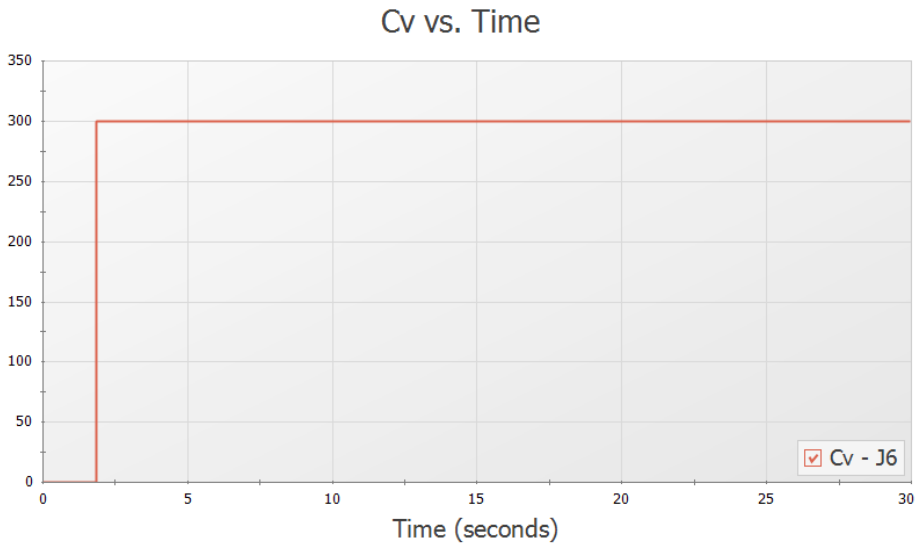


Figure 15: Relief valve Cv over time for the rupture disk relief valve

Next we want to compare the pressure results for each of the relief valve types.

1. Reload the Static Pressure Profile graph.
2. In the Graph Parameters area in the Quick Access Panel click the Multi-scenario dropdown and choose Select Scenarios
3. Select both the Rupture Disk and With Rate Limits scenarios, then click OK
4. Click Generate to incorporate the additional scenario (Figure 16)

The rupture disk generally maintains a lower maximum pressure in the pipeline, and overall reaches a lower minimum pressure. This makes sense considering that the rupture disk is allowed to open instantaneously (over one time step), thus the pressure in the pipeline is lowered faster for that scenario. In reality the rupture disk will not open instantaneously, but will still likely open faster than a passive relief valve type.

A lower minimum pressure occurs with the rupture disk because the rupture disk never closes.

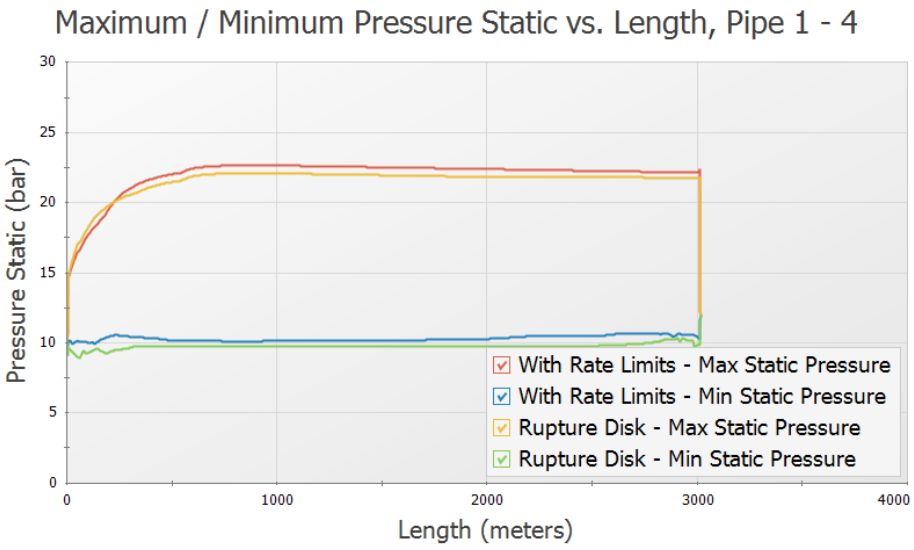


Figure 16: Comparison of maximum and minimum pressures with the passive and rupture disk relief valve types

Summary

Relief Valves provide pressure relief to systems during pressure surges. Impulse can model relief valves as an internal valve model, exit valve model, or inline exit valve model. An internal valve model connects two pipes acting as a gate providing relief from elsewhere in the system when necessary. An exit valve essentially models a spray nozzle at the end of a pipeline. An inline exit valve provides relief to the atmosphere along the length of some piping. These valve models can be profiled in a number of ways. In this example we worked with graphing results with a passive profile and rupture disk profile.

Pump Trip with Accumulator

Summary

This example shows the effect of adding a gas accumulator to reduce the down-surge after a pump trip.

Topics Covered

- Using accumulator junctions

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

This model is based off of the Pump Trip with Check Valve example and model, and that example should be worked before this one.

Model File

- ***Metric - Pump Trip with Check Valve.imp***
- ***Metric - Pump Trip with Accumulator.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Open the Model

In reviewing the Pump Trip with Check Valve example it is observed that the pressure drops below atmospheric pressure along about 140 meters of pipe. This example addresses the case when this is not acceptable. To try to limit the low pressures, an accumulator will be sized and located.

From the AFT Impulse examples folder make a copy of the ***Metric - Pump Trip with Check Valve.imp*** model file and save it to a new location. This model will be used as a starting point.

Step 3. Define the Pipes and Junctions Group

The Pipes and Junctions group is already defined, but some modifications are warranted. The model will be altered to resemble Figure 1.

Note: Instead of adding a new pipe to the Pump Trip model in order to add the accumulator, you can instead hold down Shift, then drag and drop the accumulator junction on the pipe to Split the pipe. This will allow you to add the junction, define the new length for each of the sections of pipe, and define which section of pipe the Fittings & Losses should be placed in.

Split pipe P2 into two pipes with a Gas Accumulator junction. For Pipe A, set the length to 15. Select Pipe B under Move Fittings & Losses to. Rearrange the junctions on the workspace until is resembles to figure below.

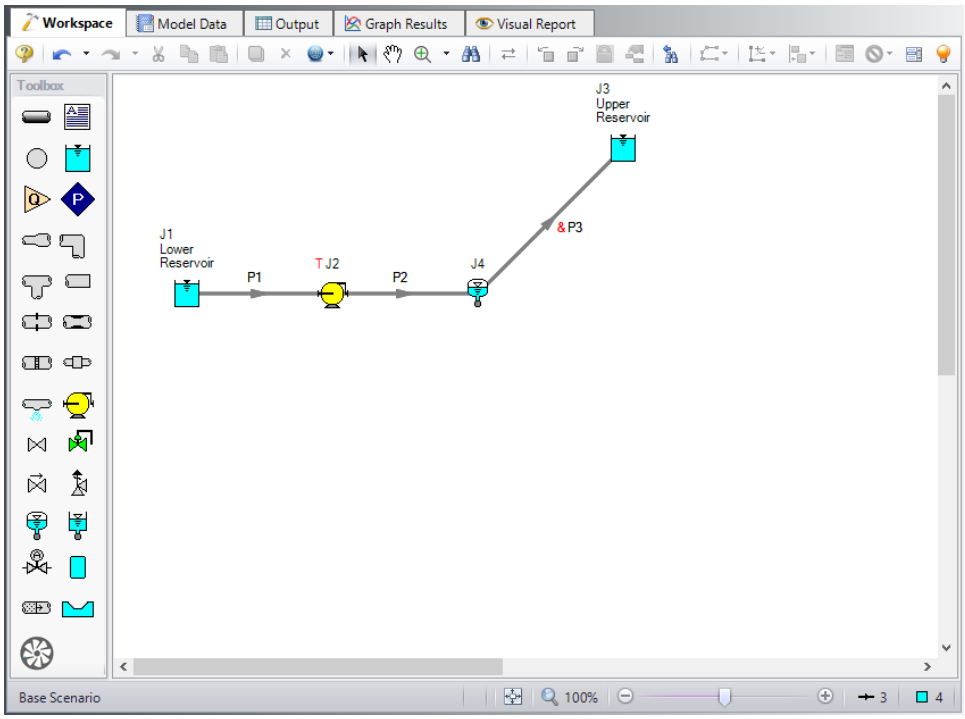


Figure 1: Workspace for Pump Trip With Accumulator Example

Sizing and locating accumulators is very much a trial and error method. The typical assumption is that the accumulator should be located as near the source of the transient as possible. In developing this example, the accumulator was first placed 3 meters from the pump suction. Initial gas volumes of 0.007 meters3 to 0.28 meters3 were used. The accumulator was effective in keeping the pressures in the discharge pipe above atmospheric, but the suction pipe, which previously had all positive pressures, now had sub-atmospheric pressure.

Therefore the accumulator was moved further away from the pump to see if this could be avoided. It was therefore first located at 200 meters along pipe P2 because that was the point of minimum pressure in the Pump Trip With Check Valve example. Thus pipe P2 was 200 meters long and P3 was 102 meters

long and the pump yielded acceptable positive pressures. The accumulator was then moved closer to the pump, and ultimately it was discovered that placing the accumulator 15 meters downstream of the pump gave acceptable pressures on both the pump suction and discharge piping. If this were a real system, this would have the added benefit of locating the accumulator near the pump, supposedly an accessible location rather than halfway up the side of a hill.

The system is in place but now we need to enter the input data for the Gas Accumulator junction. Double-click the junction and enter the following data in the properties window.

Junction Properties

1. Gas Accumulator J4
    a. Tank Geometry = Unchecked
    b. Initial Gas Pressure = Calculate From Steady State
    c. Initial Gas Volume = Known Volume
    d. Value = 0.014 meters3
    e. Polytropic Constant in Transient = 1.2
    f. Max/Min Gas Volume = Unchecked
    g. Flow Restrictor = Unchecked
    h. Short Connector Pipe = Checked
    i. Friction Factor = 0.018
    j. Diameter = 10 centimeters
    k. Pipe Length = 1 meters
    l. Specify Elevation Change = Checked
    m. Elevation Change = 1 meter
    n. Junction Elevation = 3 meters

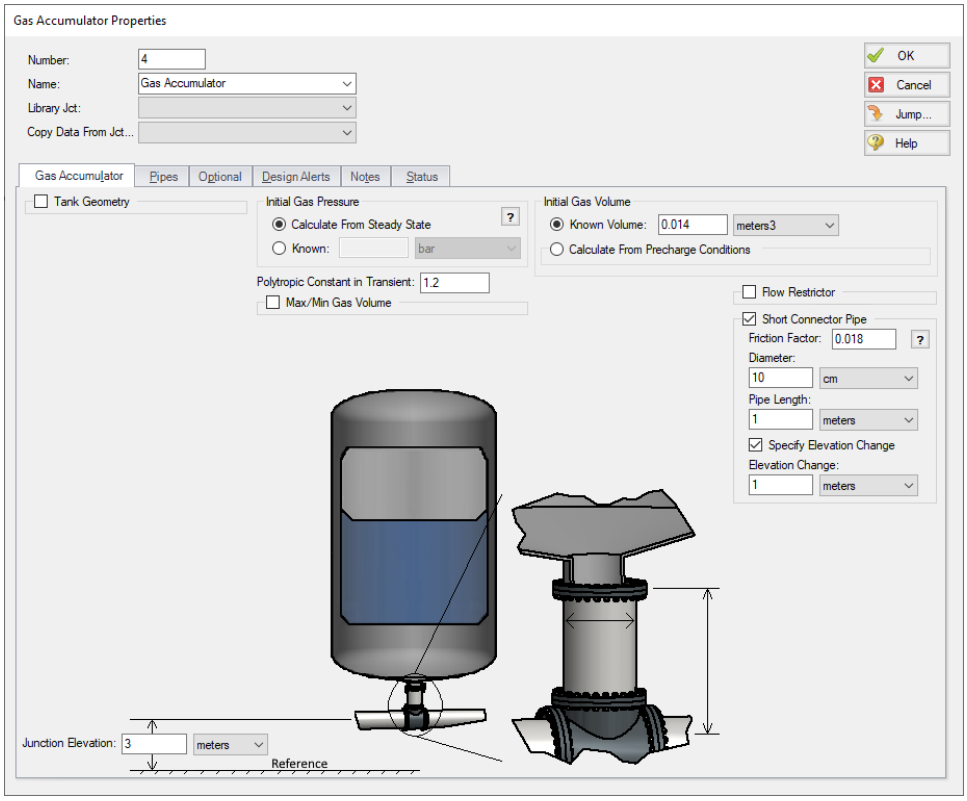


Figure 2: Gas Accumulator window with data used for example

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Run the Model

Click Run Model from the toolbar or from 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.

Step 6. Examine the Output

Graph the static pressure profile for all three pipes. This is shown in Figure 3. The pressure transient at the pump suction and discharge are shown in Figure 4. Note that the maximum pump discharge pressure has increased slightly above 12.5 barG (1250 kPa (g)), the value without the accumulator. For interest, it should be mentioned that an initial accumulator volume of 0.014 meters3 was also used. It successfully kept all minimum pressures above atmospheric, but caused the pump discharge pressure to rise up to about 13 barG (1300 kPa(g)). Although accumulators are often thought to be pressure reduction devices, they can also increase peak pressures. The pump speed decay is shown in Figure 5. Note how the pump speed does not decrease as quickly as in the Pump Trip with Check Valve example. The gas volume in the accumulator is shown in Figure 6.

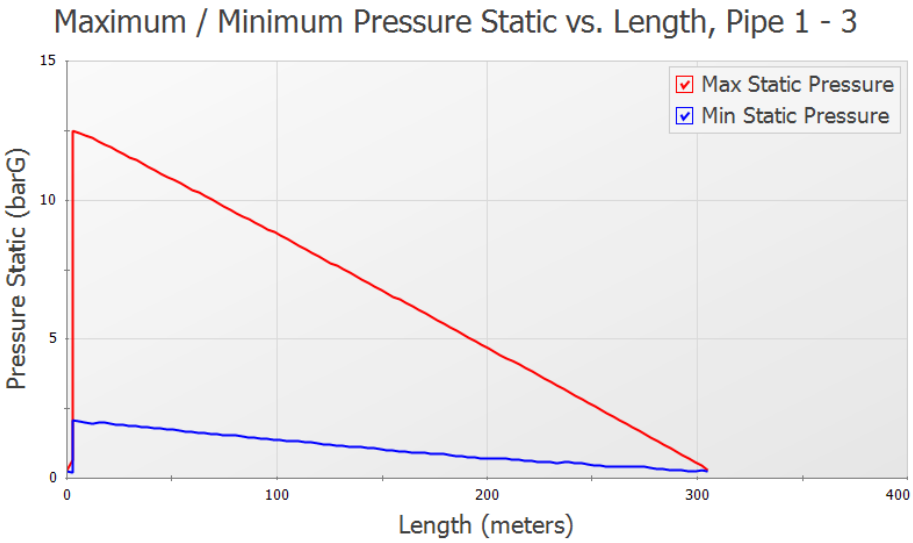


Figure 3: Maximum and minimum pressure profile with the accumulator

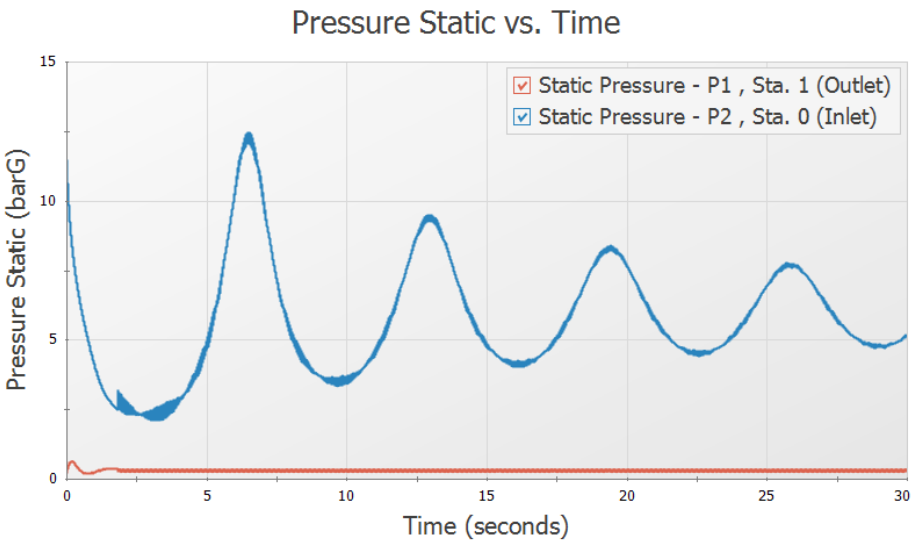


Figure 4: Transient pressures at pump suction and discharge

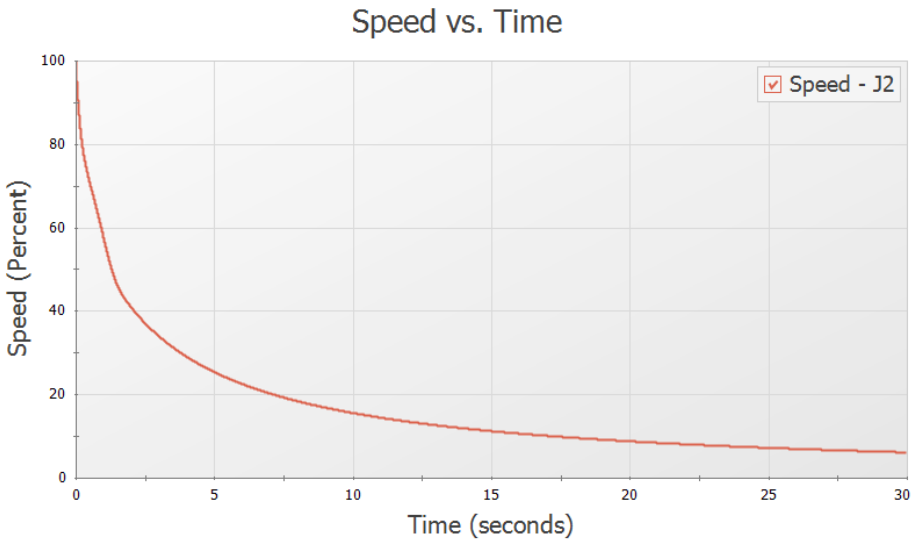


Figure 5: Pump speed decay

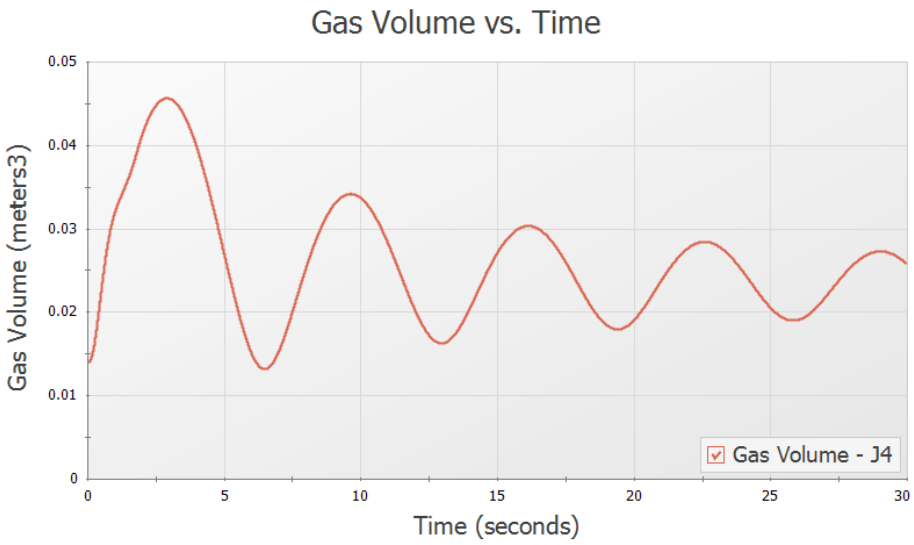


Figure 6: Gas accumulator volume over time

Pump Startup with Event Transients

Summary

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

Topics Covered

- Starting one or more pumps
- Using steady-state and transient special condition features
- Using event transients
- Using Scenario Manager
- Creating Graph Sets
- Using animation features

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

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

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 16 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

The Add/Remove Segments button on the Toolbar can be used to create the bends in pipes on the Work-space. Note that these segments are only visual, and have no affect on the calculations.

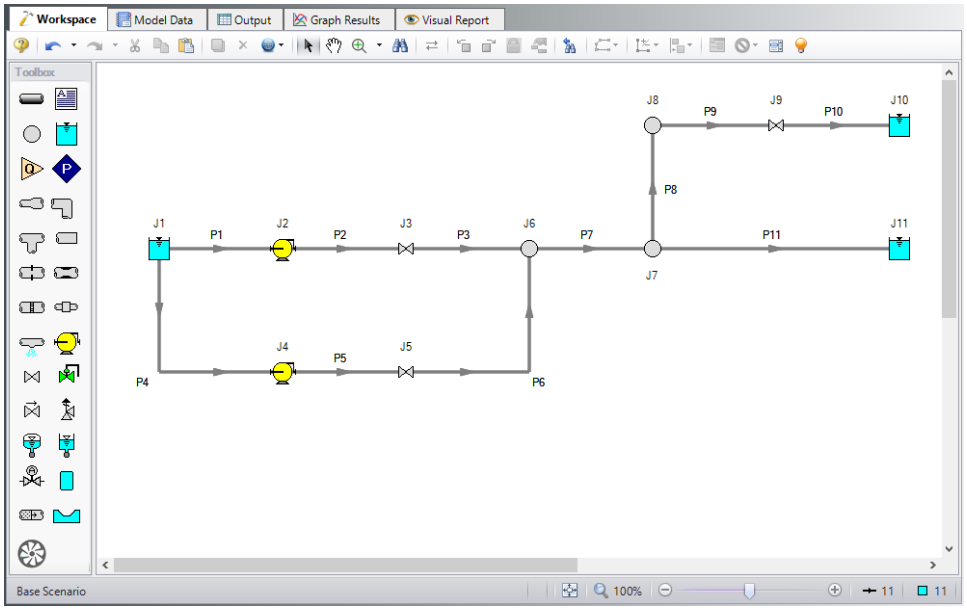


Figure 1: Layout of water transfer system

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

For Valve J9, this valve will use what is called an event transient. This means that the time zero on the Transient tab is with respect to some condition occurring in the system. If the condition is never reached, the valve transient is never initiated. Here we want the valve to open when the pressure at J7 is sufficient to cause the water to flow into the J10 reservoir. Flow will occur when the pressure at J7 reaches 3.10

barG (310 kPa(g)) or greater. The pressure at J7 is the same as that at the outlet of pipe P7, so we will use pipe P7 as the location of the event initiation.

Pipe Properties

1. Pipe Model tab
    a. Name = Use table below
    b. Pipe Material = Steel - ANSI
    c. Size = Use table below
    d. Type = Use table below
    e. Friction Model Data Set = Standard
    f. Length = Use table below

Pipe	Name	Size	Type	Length (meters)
1	Suction Pipe Pump #1	10 inch	STD (schedule 40)	15
2	Discharge Pipe Pump #1	10 inch	STD (schedule 40)	6
3	Pipe	10 inch	STD (schedule 40)	6
4	Suction Pipe Pump #2	10 inch	STD (schedule 40)	18
5	Discharge Pipe Pump #2	10 inch	STD (schedule 40)	6
6	Pipe	10 inch	STD (schedule 40)	6
7	Pipe	16 inch	STD (schedule 30)	30
8	Line to Process #1 Tank	12 inch	STD	76
9	Line to Process #1 Tank	12 inch	STD	61
10	Line to Process #1 Tank	12 inch	STD	366
11	Line to Process #2 Tank	12 inch	STD	1524

Junction Properties

1. Reservoir J1
    a. Name = Supply Reservoir
    b. Tank Model = Infinite Reservoir

c. Liquid Surface Elevation = 6 meters
d. Liquid Surface Pressure = 0 barG (0 kPa (g))
e. Pipe Depth = 6 meters

2. Pumps J2 & J4
   a. J2 Name = Transfer Pump #1
   b. J4 Name = Transfer Pump #2
   c. Inlet Elevation = 0 meters
   d. Pump Model tab
      i. Pump Model = Centrifugal (Rotodynamic)
      ii. Performance Curve Used in Simulation = Standard Pump Curve
      iii. Enter Curve Data =

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

      iv. Curve Fit Order = 2
   e. Configuration Data tab
      i. End of Curve Flow Rate = 900 m3/hr
   f. Optional tab
      i. Special Condition = Pump Off With Flow Through
   g. Transient tab
      i. Transient = Speed vs. Time
      ii. Transient Special Condition = None
      iii. Initiation of Transient = Time
      iv. Transient Data = Absolute Values

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

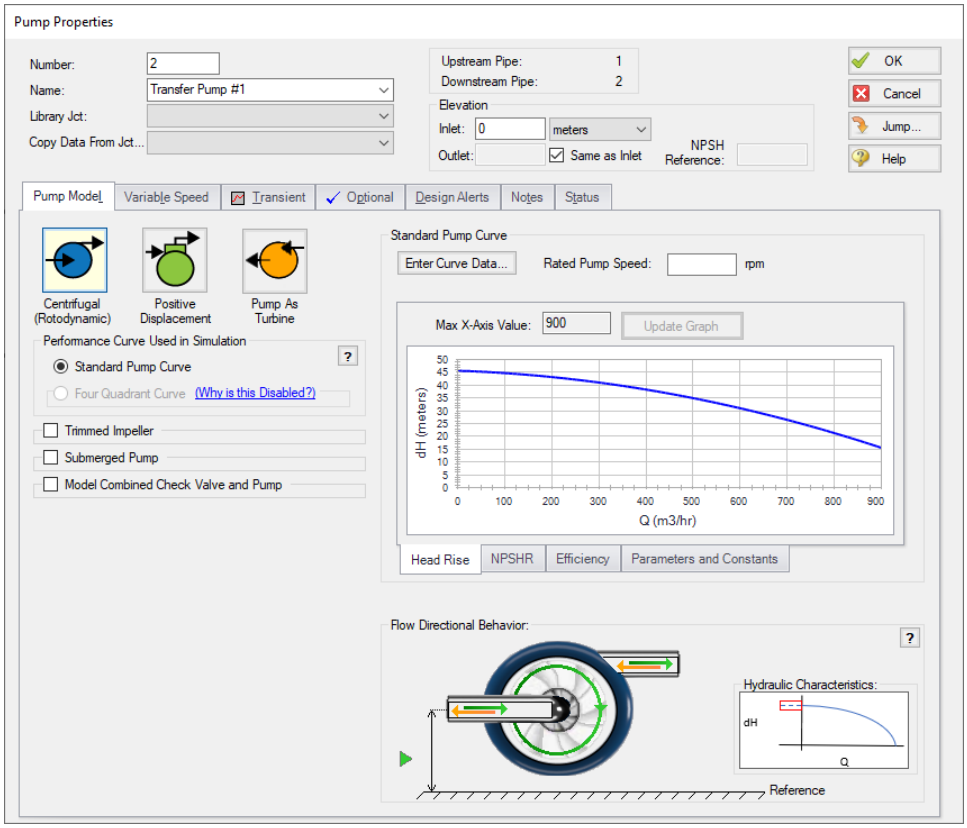


Figure 2: Setting up Pump Model tab for pump J2

3. Valves J3 & J5
    a. J3 Name = Valve #1
    b. J6 Name = Valve #2
    c. Inlet Elevation = 0 meters
    d. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Loss Source = Fixed Cv
        iv. Cv = 1000
    e. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	0
1	800

Time (seconds)	Cv
2	1000
10	1000

f. Optional tab
    i. Special Condition = Closed

4. Branch J6 - J8
    a. Elevation = 0 meters
5. Valve J9
    a. Name = Valve to Process #1 Tank
    b. Inlet Elevation = 0 meters
    c. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Loss Source = Fixed Cv
        iv. Cv = 500
    d. Optional tab
        i. Special Condition = Closed
    e. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Single Event
        iii. Event Type = Pressure Stagnation at Pipe
        iv. Condition = Greater Than
        v. Value = 3.1 barG
        vi. Pipe = 7 (Pipe) Outlet
        vii. Transient Data = Absolute Values

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

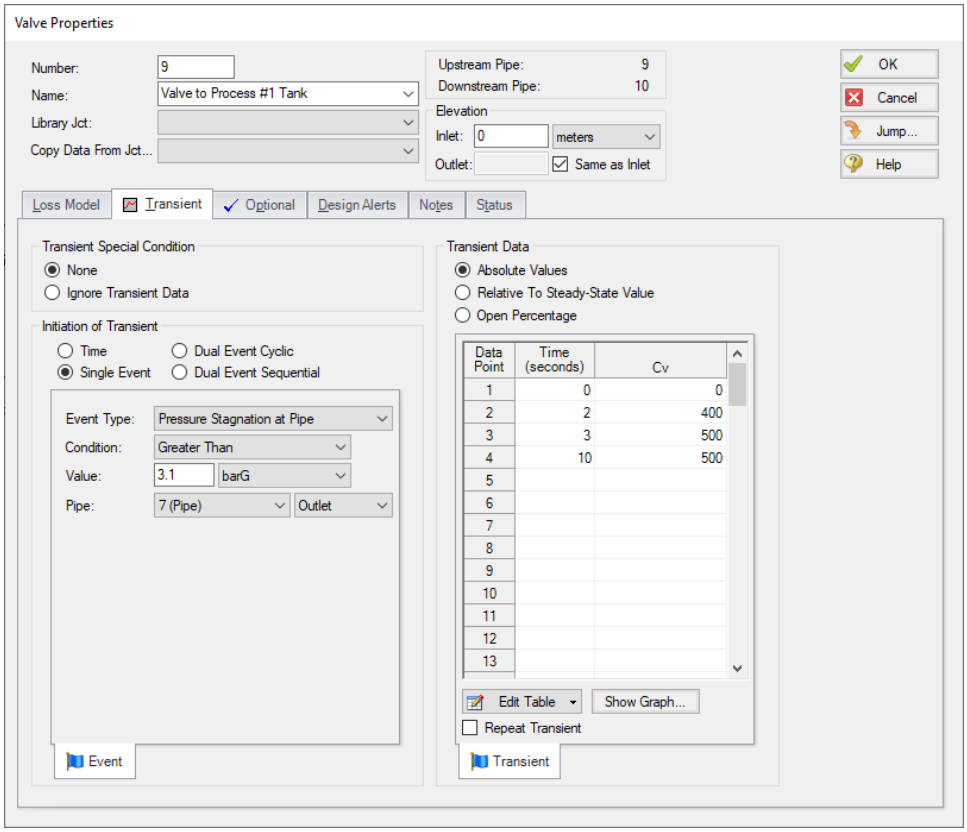


Figure 3: Specifying an event transient for valve junction

6. Reservoir J10
    a. Name = Process #1 Tank
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 30 meters
    d. Liquid Surface Pressure = 0 barG (0 kPa (g))
    e. Pipe Depth = 3 meters
7. Reservoir J11
    a. Name = Process #2 Tank
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 3 meters
    d. Liquid Surface Pressure = 0 barG (0 kPa (g))
    e. Pipe Depth = 3 meters

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Navigate to the **Output Pipe Stations** panel. To minimize run time and output file size, the default is set to Inlet and Outlet for all pipes. This will cause only the inlet and outlet station of each pipe to be saved to the output file. Data can be saved for all pipe stations or only selected stations. The selected stations are shown on the Stations In Output column (see Figure 4). With the list next to Change Selected Pipes To set to All Stations, click the All button, then click Change Selected Pipes To. This will save all pipe station data for all pipes, which will be useful later for animation purposes.

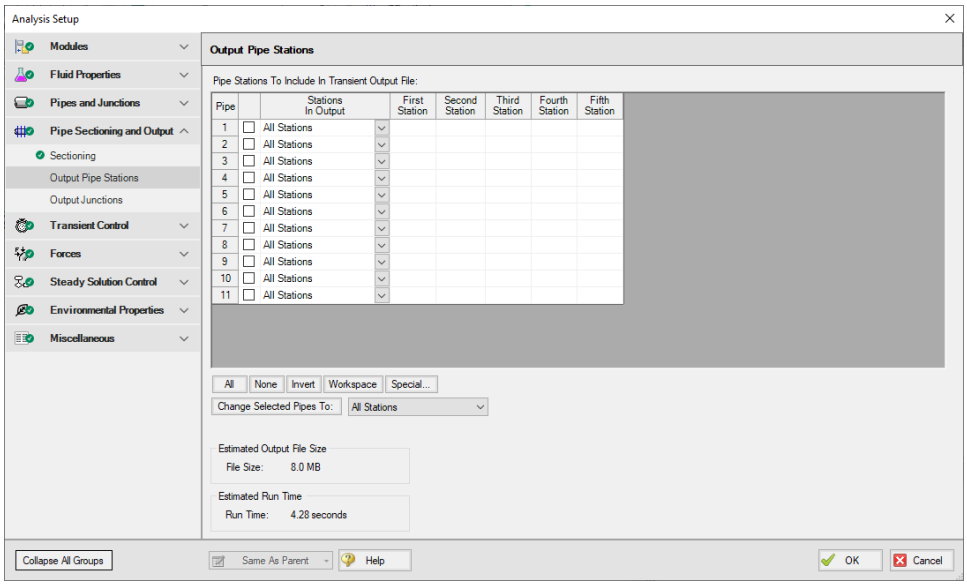


Figure 4: Transient Control window for water transfer system

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 10 seconds for the Stop Time.

Go to the **Transient Cavitation** panel and clear the check box for Model Transient Cavitation (we are going to ignore this until it becomes clear we need to model cavitation).

Step 6. Create Child Scenarios

In this model we want to evaluate three pump startup cases:

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

We will create three scenarios to model the three cases. Open up the Scenario Manager on the Quick Access Panel. Create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on the Scenario Manager on the Quick Access Panel and then selecting the Create Child icon. Enter the name *Two Pump Start* in the Create Child Scenario window, and click OK. The new Two Pump Start scenario should now appear in the Scenario Manager on the Quick Access Panel below the Base Scenario. Select the Base Scenario and create another child and call it *One Pump Start*. Finally create a third child called *One Pump Start with One Running*. The children scenarios should now be displayed as shown in Figure 5.

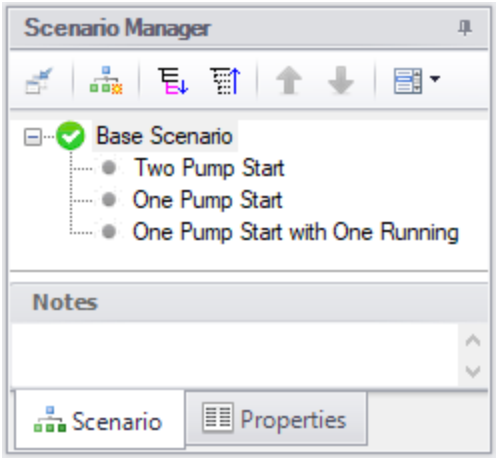


Figure 5: Scenario Manager in Quick Access Panel with three child scenarios

Since the Base Scenario already has been setup with two pumps starting, we do not need to modify the Two Pump Start scenario.

Load the One Pump Start scenario by double-clicking the name in the Scenario Manager. Here we want to start only pump J2 and keep pump J4 off. Since pump J4 and valve J5 are off during the transient, if we delete their transient input data then they will stay turned off. Alternatively, we can specify that their transient data be ignored. The second option is what we will use here.

➢**Open the J4 Pump window** and on the Transient tab in the Transient Special Condition area choose the Ignore Transient Data option and click OK. Do the same with the J6 Valve. This scenario is now completed.

➢**Load the Child Scenario** One Pump Start with One Running.

Here we want pump J2 to be running at 100% speed during the steady-state and transient, and to start pump J4. Open the properties window for J2 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. Since this pump will run at 100% speed during the transient, select the Transient Data tab and set the Transient Special Condition to Ignore Transient Data.

We also need to change the J3 Valve junction Special Condition to None, and the Transient Special Condition to Ignore Transient Data.

The third scenario is now complete.

Step 7. Run the First Scenario

➢**Load the Two Pump Start scenario** in the Scenario Manager. Select Run Model. After completion click Graph Results at the bottom of the Solution Progress window.

Step 8. Examine the Output

First let's look at a pressure profile.

1. Select the Profile tab on the Quick Access Panel, as shown in Figure 6.
2. In the Pipes section click None.
3. Select pipes 1, 2, 3, 7 and 11.
4. Select Pressure Static as the Parameter to plot.
5. Change the units to barG.
6. Make sure the boxes for Mx and Mn are checked as these are the maximum and minimum values.
7. Click Generate.

Results (shown in Figure 7) indicate that the peak pressure occurs at the pump discharge, and that the minimum pressure is above atmospheric at all times.

To easily recreate this graph for the other scenarios, you can file this graph into the Graph List Manager at the top of the Quick Access Panel on the Graph Control tab. After the graph has been generated, click the Add Graph to List icon on the Graph Results Toolbar (or by simply right-clicking on the graph itself and choosing Add Graph to List). Give the graph a name, such as *Pressure Profile Pump #1 to Process Tank #2*, then click OK in order to file the graph into the default My Graphs folder within the Graph List Manager.

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

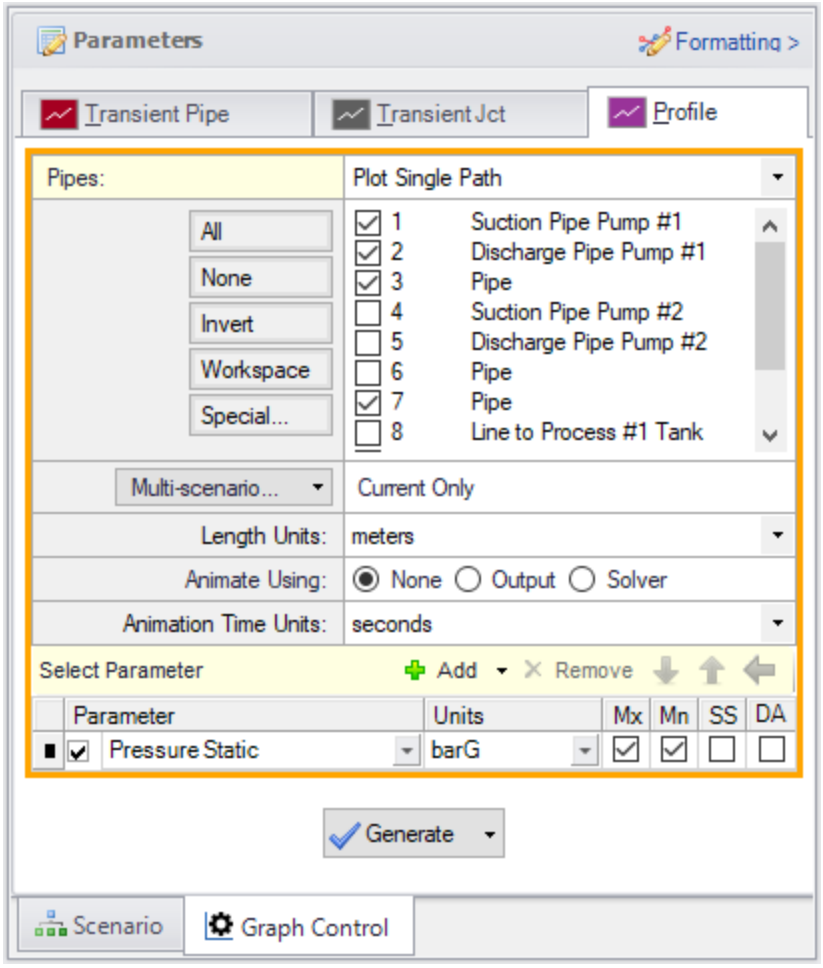


Figure 6: Select Graph Data for creating a pressure profile

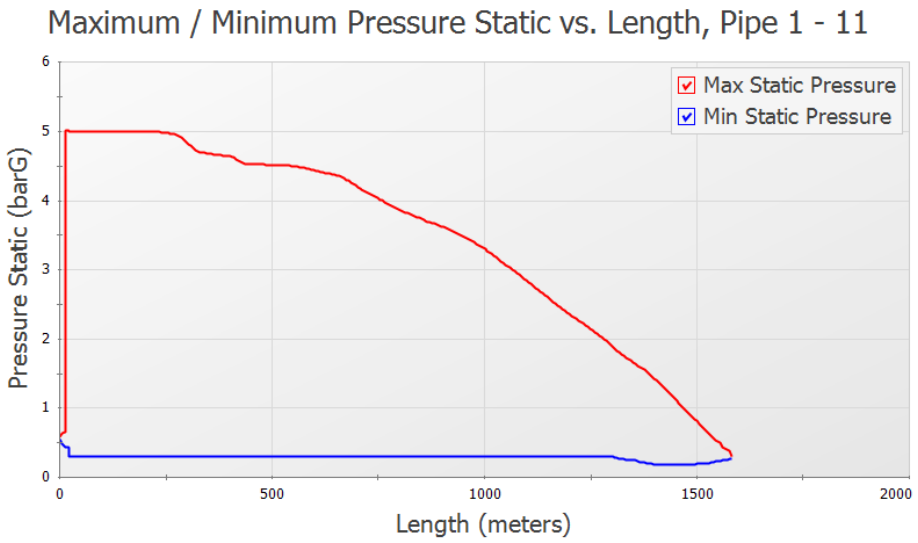


Figure 7: Profile of the maximum and minimum pressures through Pump #1 to Process #2 Tank for Two Pump Start scenario

Since the maximum pressure is at the pump discharge, it is of interest here to plot the pressure vs. time at the pump. Go to the Transient Jct tab on the Quick Access Panel. Add the two pumps to the Graph These Junctions section. Select the Graph Parameter as Pressure Static Outlet and set the units to barG. Click Generate. As would be expected, both pressure transients are very similar. Results are shown in Figure 8. Click the Add Graph to List button on the Graph Results Toolbar (or by right-clicking on the graph and choosing Add Graph to List) and type the name *Pump Discharge Pressures* to save this graph to the Graph List Manager as well.

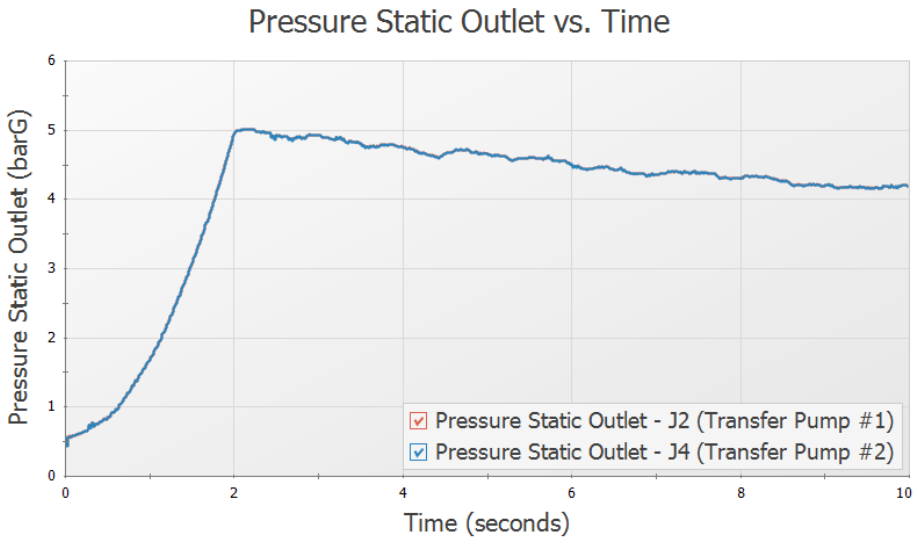


Figure 8: Pressure transient at both pump discharge locations for Two Pump Start scenario

The flow rates through the pumps are also of general interest, and can also be plotted as shown in Figure 9 by changing the parameter being graphed to Vol. Flow Rate Outlet. Add this graph to the Graph List Manager as well and name it *Pump Discharge Flow Rates*.

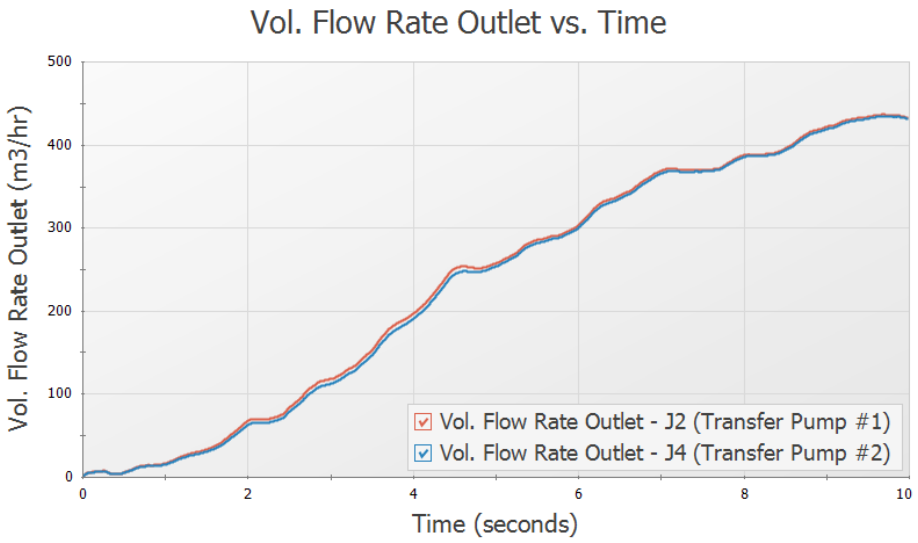


Figure 9: Flow rate transient at both pump discharge locations for Two Pump Start scenario

The flow rates into the two process tanks are of interest. On the Transient Pipe tab add the outlet of pipes P10 and P11. Select to plot the Volumetric Flowrate, click Generate, and then add this graph to the Graph List Manager with the name *Process Tank Flow Rates*, then click OK. Results are shown in Figure 10.

It should be noted that the graphs in Figure 8 and Figure 9 could also be generated by using the Transient Pipe tab to graph the static pressure and volumetric flow rate at the inlet of pipes P2 and P5.

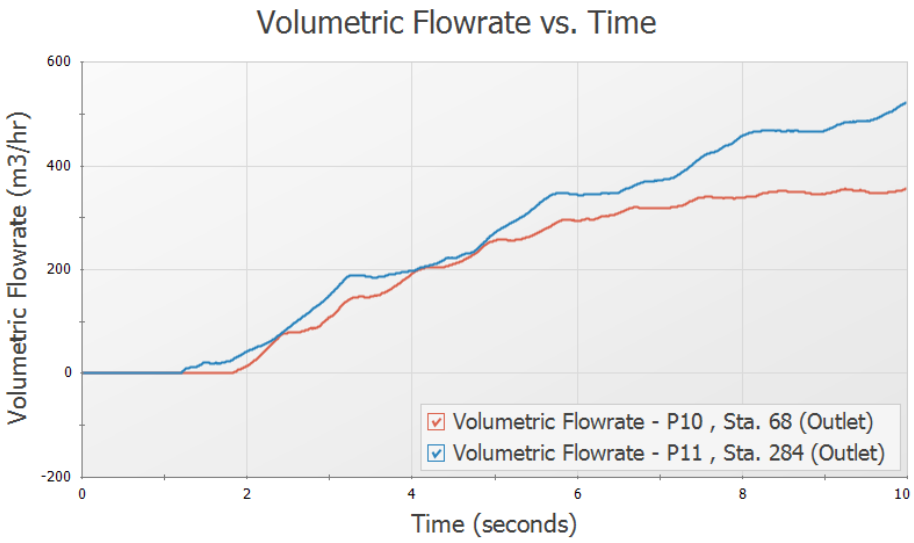


Figure 10: Flow rate transient at process tanks for Two Pump Start scenario

Step 9. Animate the Results

➢**Load the pressure profile graph** Pressure Profile Pump #1 to Process Tank #2 that was created in Step 8 from the Graph List Manager, and make the following updates, as shown in Figure 11.

1. For Animate Using, select Output. This is where changing the output to store output data for all stations in Step 4 comes into play.
2. Uncheck the boxes for Mx and Mn so that the maximum and minimum values will NOT be displayed.

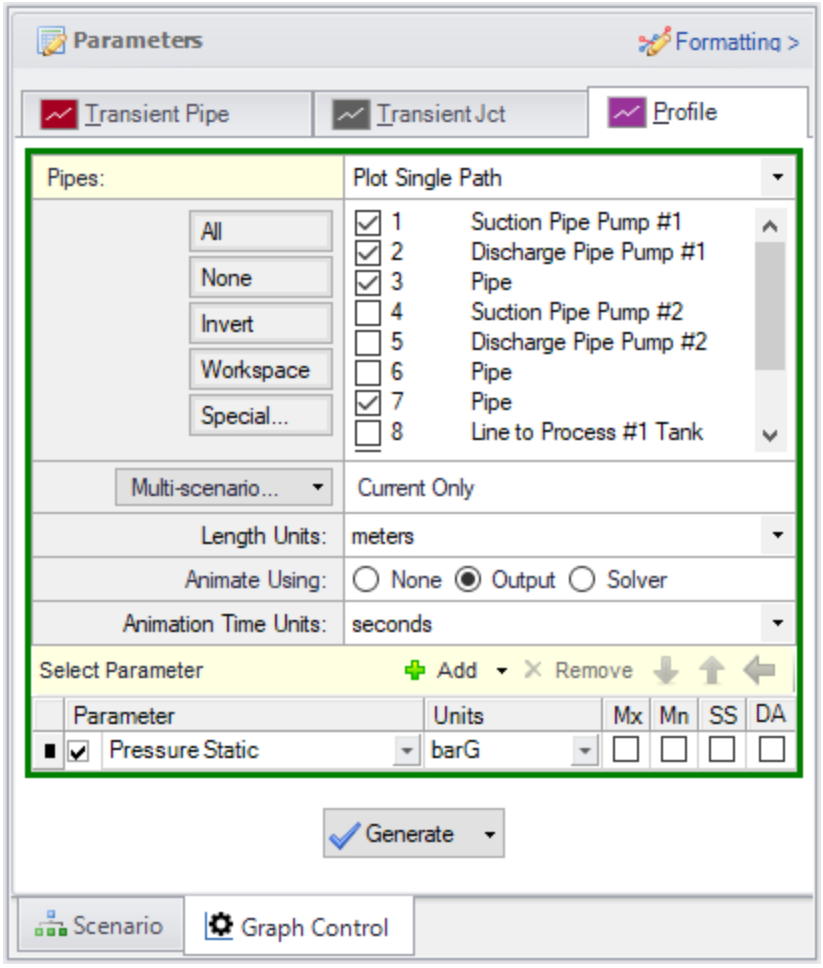


Figure 11: Selecting animation in Graph Results for Two Pump Start scenario

To use the Time Animation feature in conjunction with the Use Output File option, which we are using here, all pipe stations need to be saved in Transient Control (Step 5). In addition, it is frequently best to save all time step data as well (from the Save Output to File area in the Transient Control window). We selected both of these. This animation option allows you to start, pause, record, restart animation and also allows you to slowly move forward and backward in the animation by changing one time step at a time.

Another animation option in Figure 11 is Solver. This option actually re-runs the Transient Solver to generate the data for animation, and thus does not need to read it from the output file. Therefore, it does not require all of the data to be saved in Transient Control. This will reduce the run time of the initial simulation run, but will not have the advantage of being able to pause the animation and backup time step by time step to go back to the time where significant spikes start to appear.

➢**Click Generate.** Additional animation control features appear on the Graph Results window (Figure 12). Press the Play button and watch the pressure waves move.

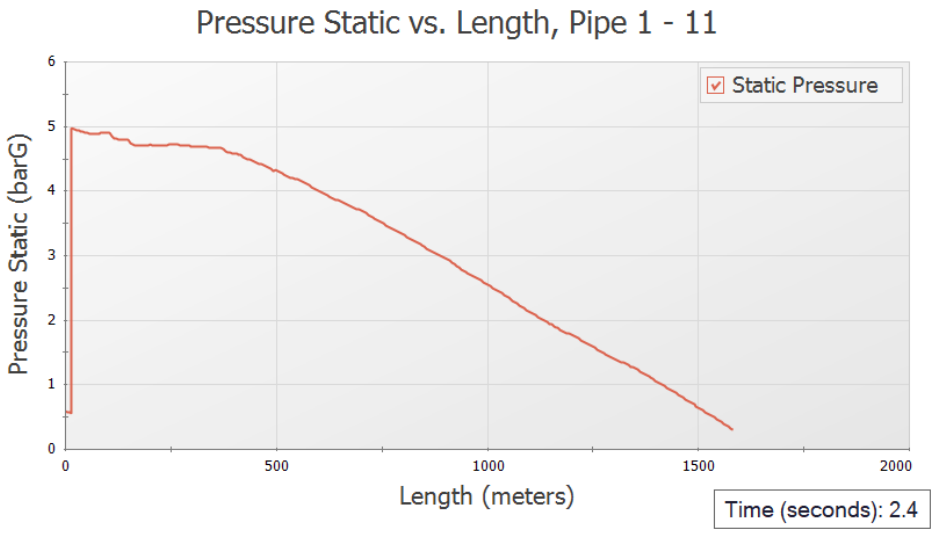


Figure 12: Animating output in Graph Results for Two Pump Start scenario

There is also animation available in the Visual Report, which can be useful to see how the flow paths interact with each other.

➢**Go to the Visual Report window** and the Visual Report Control should automatically open. Set the Display Mode at the top of the window to Animate, then choose the settings as follows. (Figure 13)

1. Display Parameters tab
    a. Data Type = Absolute Values
    b. Parameter to Animate = Pressure Static
    c. Units = barG
    d. Smooth colors between pipe sections = Checked
2. General Display tab
    a. Show Analysis Title = Unchecked
    b. Show Input Filename = Unchecked

Click Show. The Visual Report window should now appear similar to Figure 14. Note that the animation toolbar again appears, similar to when using Graph Animation. Additionally the Visual Report Animation tab appears in the Quick Access Panel providing options from Visual Report Control window for adjusting the animated parameters and pipes. It is recommended to save data for All Stations when creating a Visual Report Animation, but it is not required to do so. The Pipes to Animate section shows what data is being used to create the animation.

➢**Press the play button** to watch the pressure transients in the system.

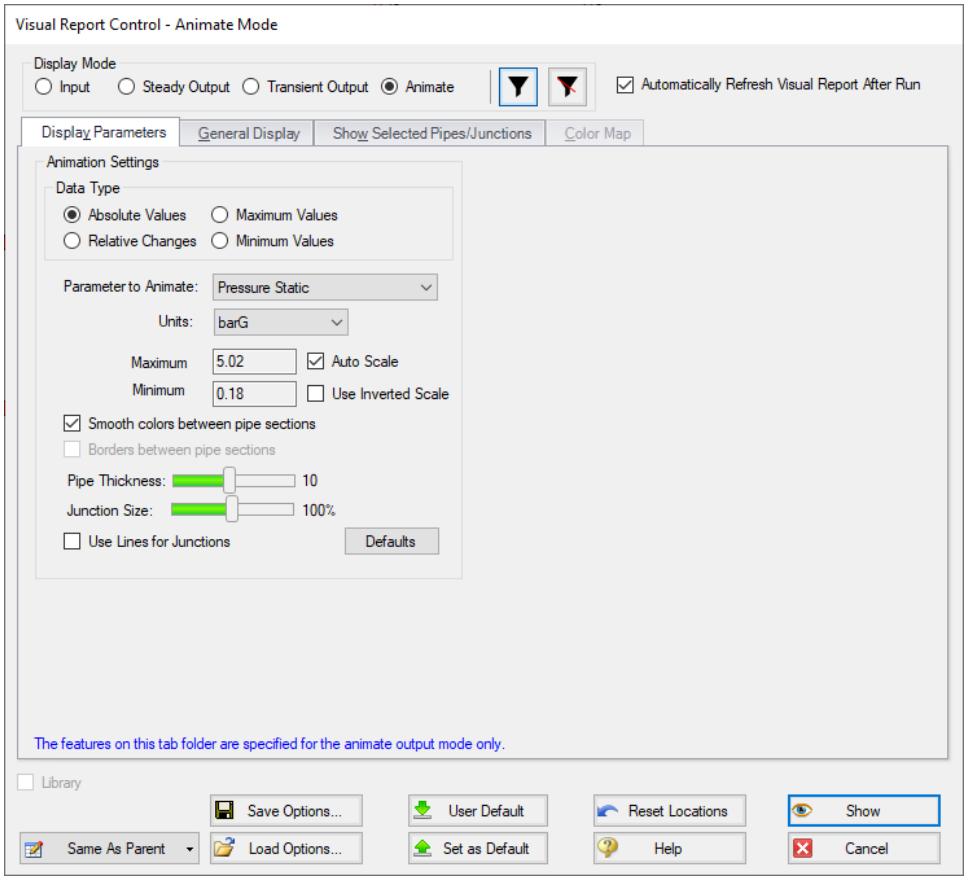


Figure 13: Visual Report Control settings for Visual Report Animation of static pressure

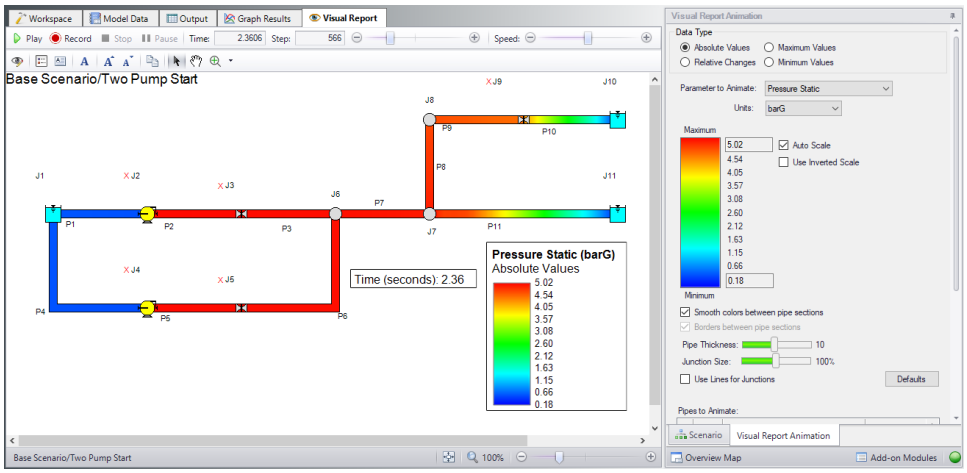


Figure 14: Visual Report Animation enabled in Visual Report window

Step 10. Run the Other Scenarios and Graph the Results

Using Scenario manager load the other two scenarios and run them.

In the One Pump Start With One Running scenario, it should 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 J4. This warning message indicates that backwards flow occurred at the pump, and insufficient information was available to perform the head calculations. 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 opening profile to prevent reverse flow to the pump. This is only acceptable if the changes are physically accurate for the system. Otherwise, it will be necessary to choose an appropriate four quadrant data set for the pump to better predict the head at reverse flow conditions. See the Pump Trip With Backflow - Four Quadrant Modeling example for more information on specifying the pump using four quadrant data.

Note: Whenever you see caution, warning, or critical warning messages in general, you can click on the message itself, then hit the F1 button on the keyboard to pull up the help system in order to read a description of what the message means.

Results

Create graphs similar to Figure 6 through Figure 9 for each of the scenarios by double-clicking on the graph names in the Graph List Manager. This will reveal that the maximum pressure for the One Pump Start scenario occurs at the pump discharge location, similar to the first scenario, but in the One Pump Start With One Running scenario the maximum pressure occurs in pipe P11. Also the pressure drops below atmospheric, but does not reach the vapor pressure of water.

The numerical maximum pressure can be found in the Output window of each scenario and is summarized in Table 1. For these cases the maximum pressures do not significantly differ from each other.

Table 1: Summary of maximum static pressure for the three cases

Case	Max Pressure (barG)
Two Pumps Start	5.0156
One Pump Start	4.9306
One Pump Start with One Running	5.0646

Pump Trip with Backflow - Four Quadrant Modeling

Summary

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

- Specifying a pump curve and power curve in the Pump Properties window
- Using one of the transient pump models which accounts for pump inertia
- Using a four quadrant data set for the pump performance curve
- Using four quadrant data to model reverse flow and reverse rotation in the pump

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

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

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
   a. Fluid Library = AFT Standard
   b. Fluid = Water (liquid)
      i. After selecting, click Add to Model
   c. Temperature = 38 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

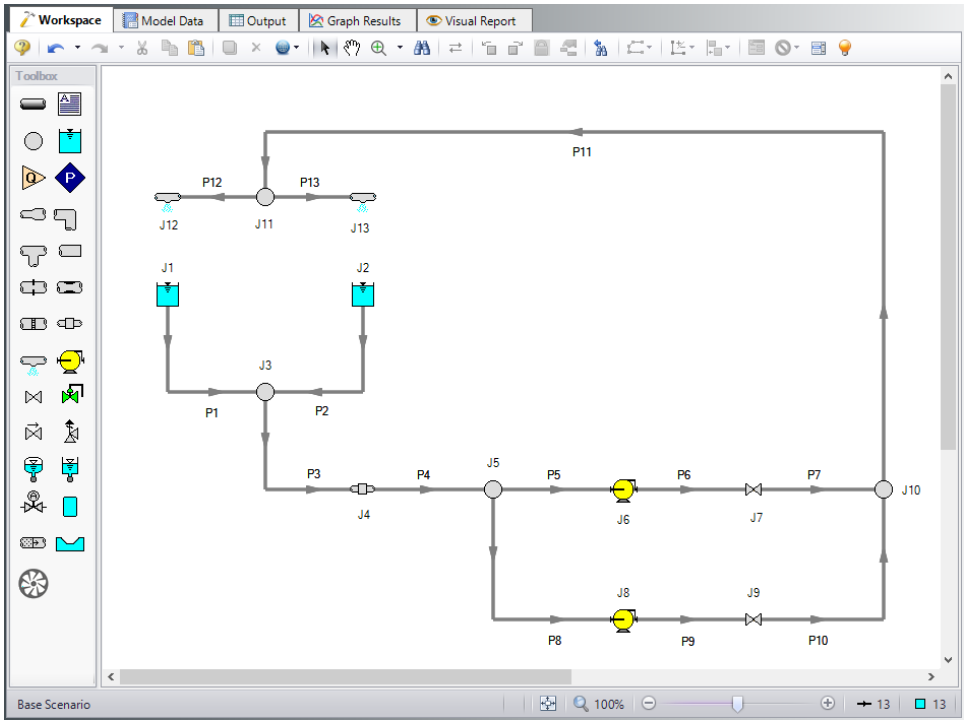


Figure 1: Layout of cooling system

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Model tab
    a. Name = Use table below
    b. Pipe Material = Steel - ANSI
    c. Size = Use table below
    d. Type = Use table below
    e. Friction Model Data Set = Standard
    f. Length = Use table below

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

Junction Properties

1. Reservoir J1 & J2
    a. J1 Name = CT #1 Basin
    b. J2 Name = CT #2 Basin
    c. Tank Model = Infinite Reservoir

d. Liquid Surface Elevation = 12 meters
e. Liquid Surface Pressure = 0 barG (0 kPa (g))
f. Pipe Depth = 1 meter

2. Branch J3
   a. Elevation = 11 meters
3. General Component J4
   a. Name = Surface Condenser
   b. Inlet Elevation = 0 meters
   c. Loss Model = Resistance Curve
   d. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	0
4542	6.7
9084	26.8

   e. Curve Fit Order = 2
4. Branch J5 and J10
   a. Elevation = 0 meters
5. Pumps J6 and J8
   a. J6 Name = CWP-1
   b. J8 Name = CWP-2
   c. Inlet Elevation = 0 meters
   d. Pump Model tab
      i. Pump Model = Centrifugal (Rotodynamic)
      ii. Performance Curve Used in Simulation = Standard Pump Curve
      iii. Rated Pump Speed = 1790 rpm
      iv. Enter Curve Data =

Parameter	Volumetric	Head	Power
Units	m3/hr	meters	kW
1	0	34.1	78.3
2	454	32.9	96.9
3	908	31.7	119.3
4	1363	29.9	141.6
5	1817	27.4	156.5
6	2271	23.8	164.0
7	2725	18.3	156.5

v. Curve Fit Order = 2

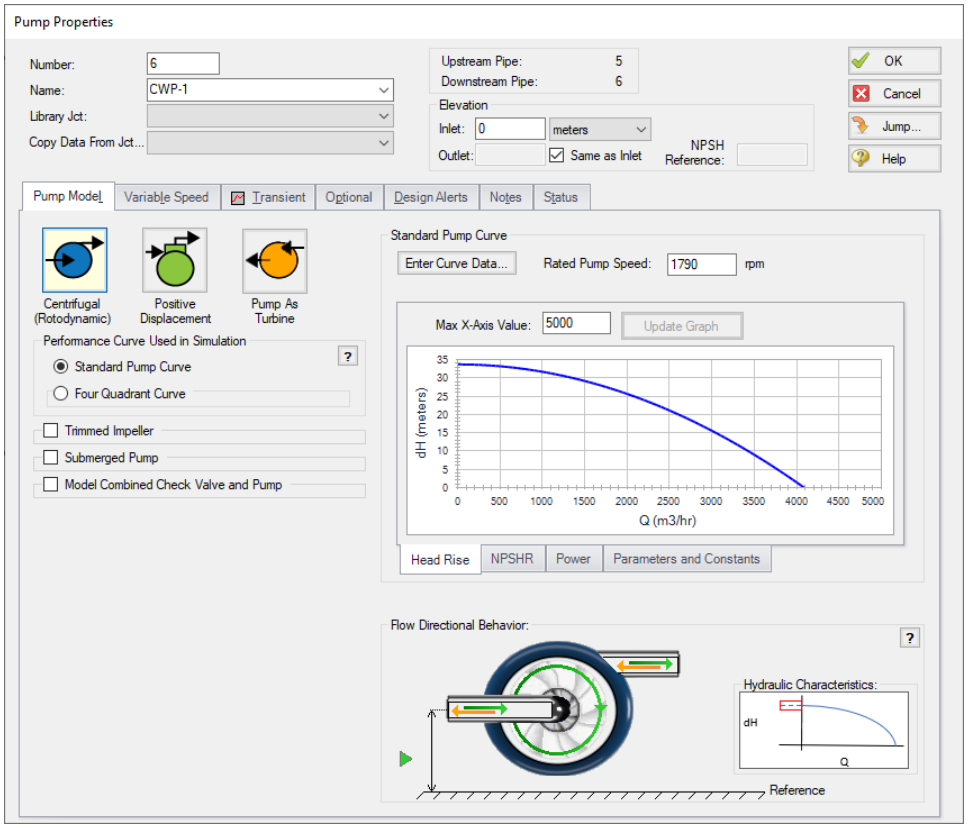


Figure 2: Pump Model tab in the Pump Properties window for J21 with a Standard Pump Curve

v. Transient tab
   i. Transient = Trip
   ii. Transient Special Condition = None
   iii. Initiation of Transient = Single Event
   iv. Event Type = Time Absolute
   v. Condition = Greater Than or Equal To
   vi. Value = 0 seconds
   vii. Total Rotating Inertia = User Specified
   viii. Value = 5.9 kg-m2

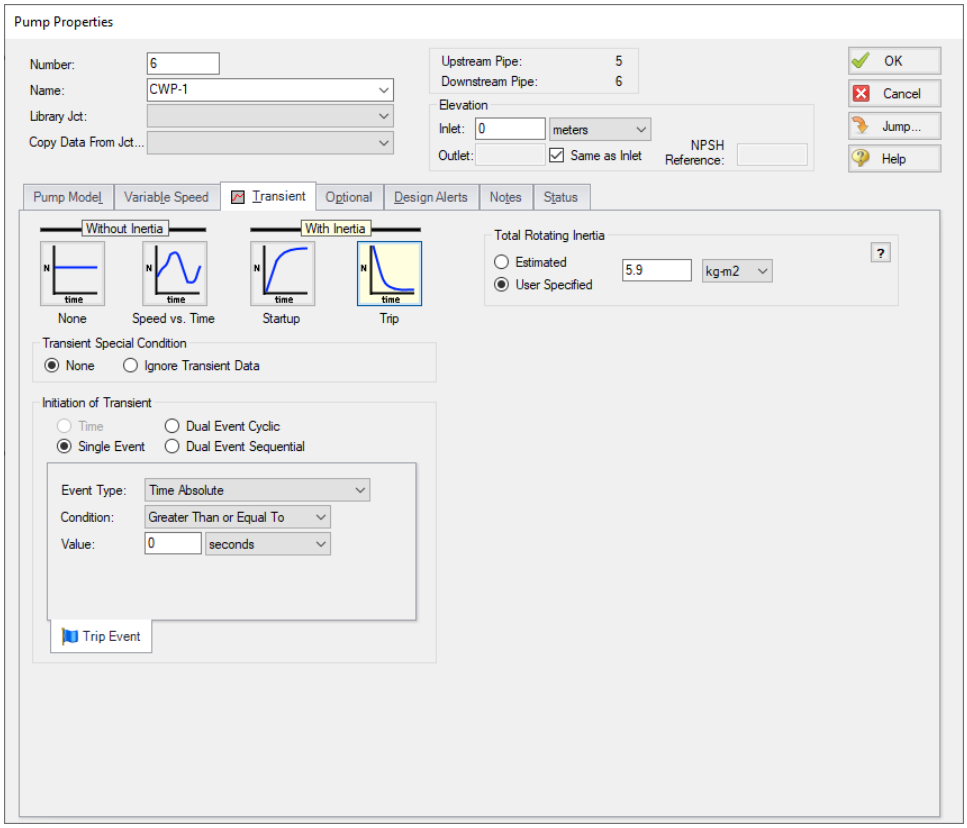


Figure 3: Pump Transient Data tab

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. We will come back later and compare these results to the Four Quadrant option.

6. Valves J7 & J9
    a. J7 Name = Valve #1
    b. J9 Name = Valve #2
    c. Inlet Elevation = 0 meters
    d. Loss Model tab
        i. Valve Data Source = User Specified
        ii. Loss Model = Cv
        iii. Loss Source = Fixed Cv
        iv. Cv = 10000
    e. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

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

7. Branch J11
    a. Elevation = 15 meters
8. Spray Discharges J12 & J13
    a. J12 Name = CT #1
    b. J13 Name = CT #2
    c. Elevation = 24 meters
    d. Loss Model = Cd Spray (Discharge Coefficient)
    e. Geometry = Spray Nozzle
    f. Exit Properties = Pressure
    g. Exit Pressure = 0 barG (0 kPa (g))
    h. Cd (Discharge Coefficient) = 0.6
    i. Discharge Flow Area = 0.93 meters2

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 15 seconds for the Stop Time.

To minimize run time and output file size, select to Save Output to File Every 10 Time Steps

Step 6. Run the Model

Click Run Model from the toolbar or from 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 Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

For this model several warnings and cautions will be shown in the output. A warning is given for both pumps stating that the pump experienced reverse flow and was using a Standard Pump Curve, meaning that the zero flow head value was used to predict head and power at reverse flow. Generally if the reverse flow is low in magnitude and duration using the zero flow head will be a good assumption and this error can be dismissed. For the purpose of this example we will compare the results from this scenario to the results using a four quadrant curve to examine the impact on the model.

There is also a caution given that the end of curve flow was exceeded in this model. Note that in this model we did not explicitly define the end of curve flow, so the last flowrate entered (2,725 m3/hr) was assumed to be the end of curve flow rate. The pump head and power curve fits will only be reliable within the range of data entered by the user, thus predictions at flowrates above this range will be inaccurate, particularly when the pump is operating at full speed. Operating beyond the end of curve is less of a concern when the pump is operating at lower speeds, as is seen for the pump in this example. However, it is still prudent to verify that the effect of operating past the end of curve is low by using a four quadrant curve.

The speed of the two pumps should decay similarly. Figure 4 shows the speed decay for CWP-1. Here the pump speed decays quickly at first, then begins to slow after the discharge valve closes at about 5 seconds.

The transient pressure at the pump suction and valve #1 outlet is graphed in Figure 5, with the volumetric flow rate at these points on the secondary axis.

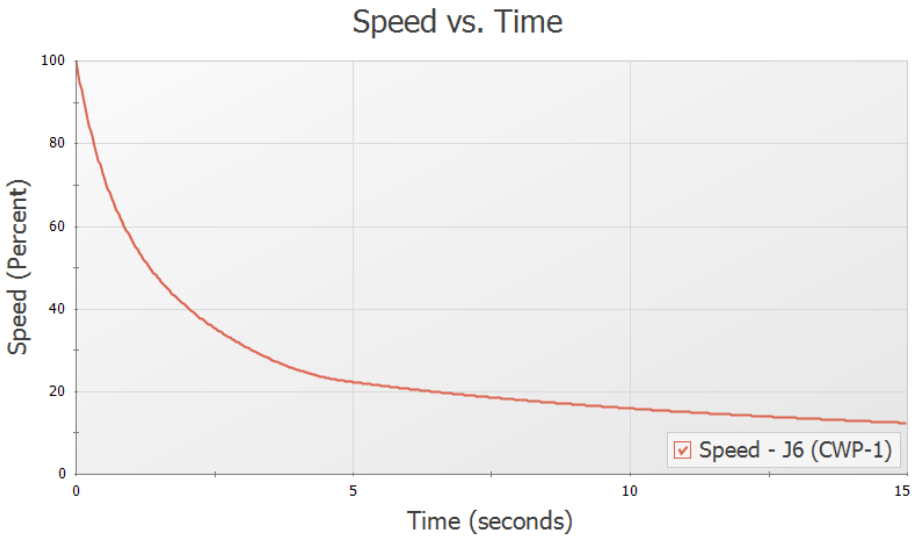


Figure 4: Pump speed decay for CWP-1

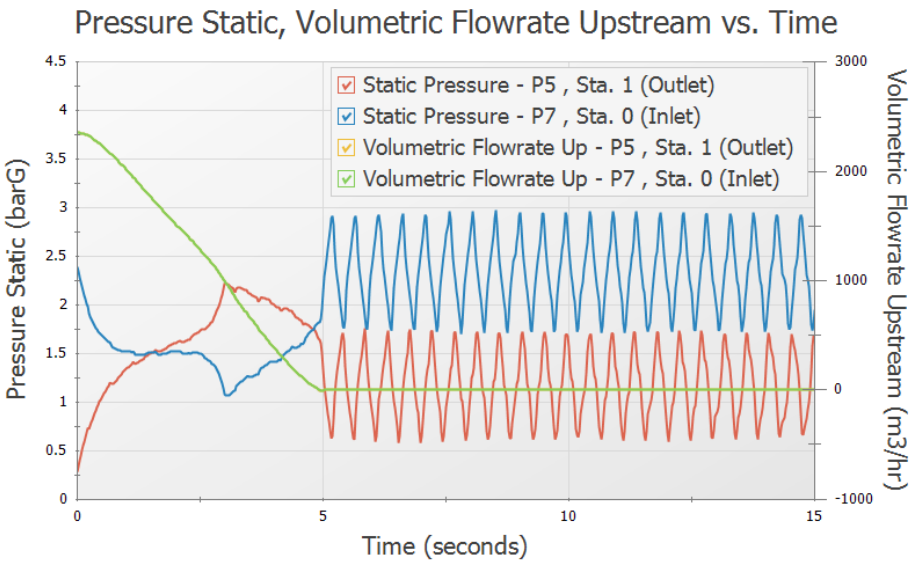


Figure 5: Transient pressures and volumetric flowrates at pump suction and discharge, and valve outlet

Step 8. Create Child 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.

Note: Only one four quadrant data set is chosen for this example. In an actual four quadrant analysis it is recommended to compare multiple four quadrant data sets to ensure that the chosen data set is a good fit for the pump being modeled. See the Selecting a Pump Four Quadrant Curve example for more information.

To do this, click Create Child 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.

Step 9. Define the Four Quadrant BEP Scenario

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

1. Performance Curve Used in Simulation = Four Quadrant Curve
2. Four Quadrant Curve = User Selected
3. Click Specify Model
4. Click Suggest a Data Set
    a. 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.
5. Flow/Head Source = Calculated from Standard Pump Curve
    a. Calculated from Standard Pump Curve allows most of the data to automatically populate from the Pump Properties window. In this case, the pre-defined data is accurate for the pump.
6. Click Calculate
    a. The Estimated Specific Speed in this case is 132.4 m3/s units, and the Suggested Data Set is N_S = 147 m3/s units.
7. Click Select Suggested Data Set and Close
    a. The N_S = 147 m3/s units data set will now be selected, and the dimensionalized curve will be displayed against the previously entered Standard Pump Curve. (Figure 6)

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

---

8. Dimensional Reference Point (DRP) = Best Efficiency Point
    h. We will select the Steady State Operating Point option later for the third scenario.
9. Click OK on this window, then OK on the Pump Properties window to accept the changes.
10. Repeat the above steps for the other pump junction.
11. The required input for this scenario is now complete.

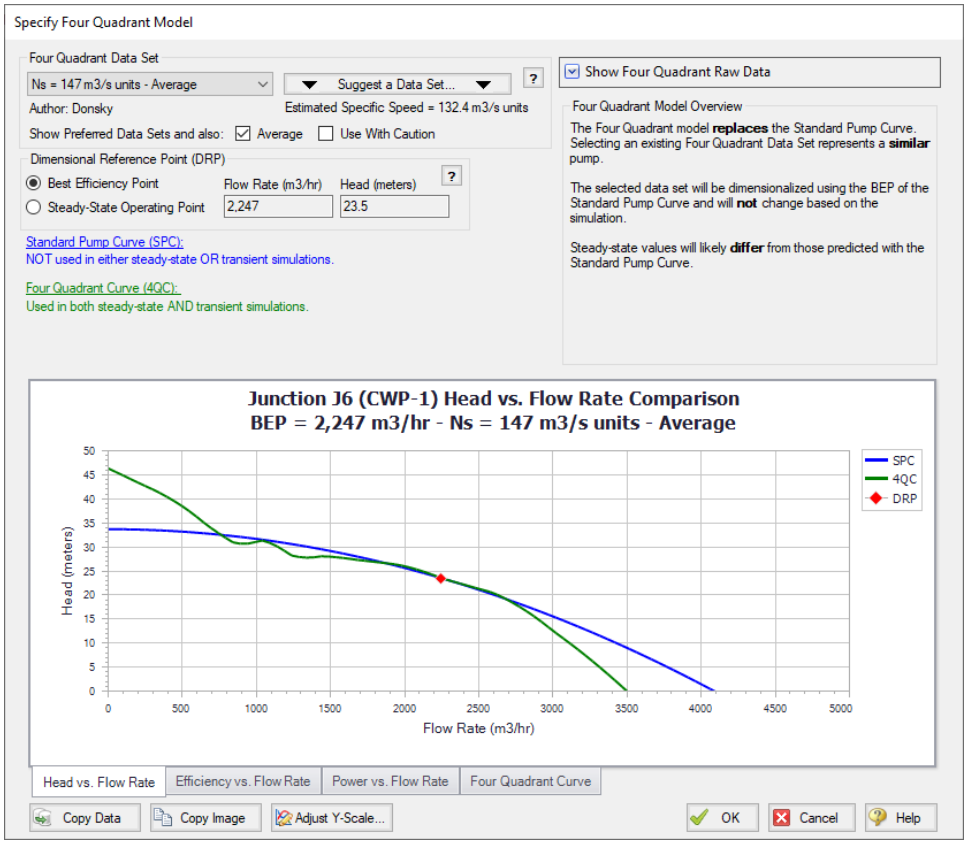


Figure 6: The Define Four Quadrant Model window can be used to select and view options for the dimensionalized four quadrant data set

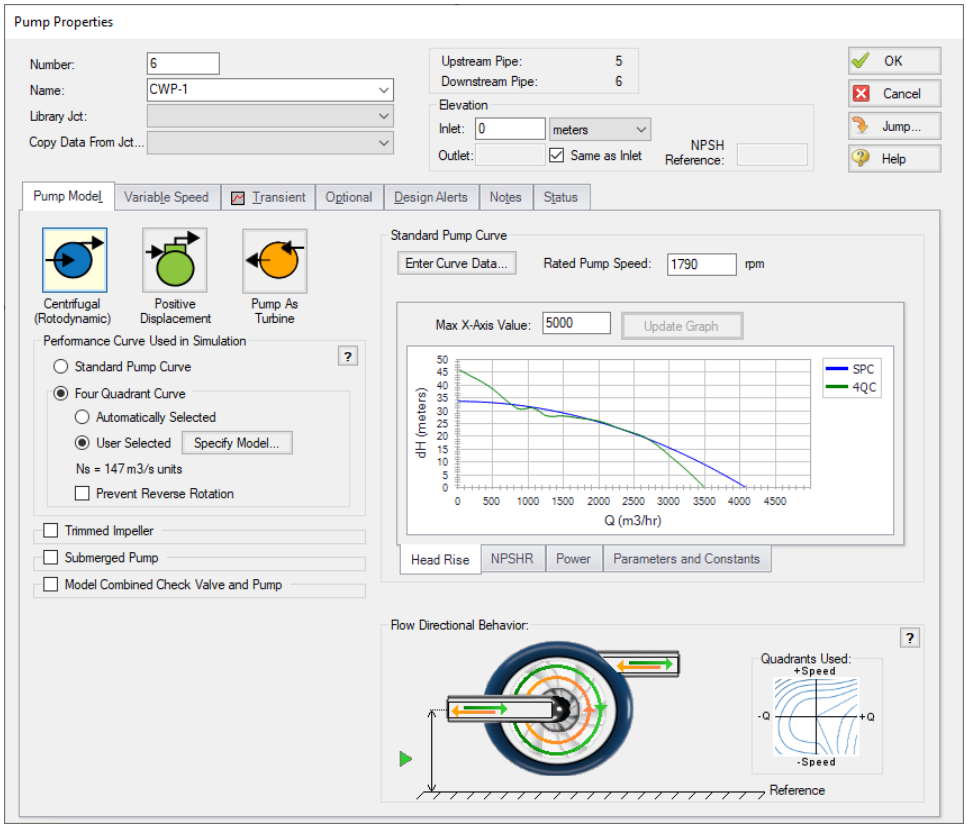


Figure 7: The Pump Model tab defined using the Four Quadrant Curve. Note that the Flow Directional Behavior section shows the pump behavior for both forwards and reverse flow and speed possibilities

Step 10. Define the Four Quadrant SSOP Scenario

Clone the Four Quadrant BEP scenario by right-clicking the scenario name in the Scenario Manager and choosing Clone Without Children. Name this cloned scenario *Four Quadrant SSOP*. For both pumps, open the Pump Properties window and click Specify Model under Four Quadrant Model. 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 11. Run the Model and Examine the Output

Using the Scenario Manager load each of the four quadrant scenarios and run them. In comparing the transient results for the two four quadrant scenarios, there are no major visible differences. Figure 8 shows a multi-scenario graph with the pressures and volumetric flowrates for the Standard Pump Curve, Four Quadrant BEP and Four Quadrant SSOP scenarios. The two four quadrant scenarios produce results that are practically indistinguishable from the Standard Pump Curve results.

Comparison between the four quadrant graphs and standard pump curve graph reveals that the results are similar, though there is a visible difference of about 0.1 barG between the maximum and minimum

points in the four quadrant scenario graphs and the standard pump curve scenario graph. These differences can be attributed to the larger flow oscillations that occur when the pump head is fixed to the zero flow head for all values of reverse flow.

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 2,362 m3/hr, while the steady state flow rate for the BEP scenario is 2,365 m3/hr. This results from the fact that the steady state operating point option forces the four quadrant curve to provide the same results as the manufacturer curve during the steady state, while the BEP option does not. However, since the pump operates close to BEP in this scenario, the difference in steady state operating point is minor when compared to the Standard Pump Curve scenario.

While the difference in steady state and transient results for the two dimensional points is minor in this model, it may become more pronounced in other cases. This choice of dimensional reference points can result in 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 Selecting a Pump Four Quadrant Curve example, as well as the main Help file for AFT Impulse.

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

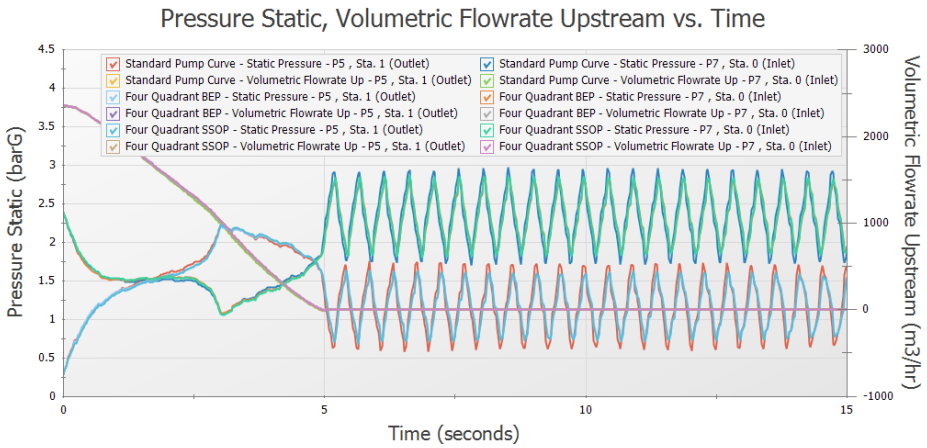


Figure 8: Comparison of Pressure and volumetric flow rate transients for the Standard Pump Curve, Four Quadrant BEP and Four Quadrant SSOP scenarios

Step 12. Adjust the Model

Consider the case where the valves at the discharge of the pumps cannot be closed in five seconds, but instead will close fully at twelve 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. The pumps will already inherit the four quadrant model from the Four Quadrant BEP scenario, but the valves need adjustment. For both valves, open the Valve Properties window, then edit the Transient Data table found on the Transient tab to reflect the following closure profile.

Time (seconds)	Cv
0	10000
5	1000
12	0
30	0

Since the valve closure takes a longer time, it will be useful to extend the run time.

➢**Open Analysis Setup and navigate to the Simulation Mode/Duration panel** and change the Stop Time to 30 seconds, then click OK.

Step 13. Run the Model

Click Run Model from the toolbar or from the Analysis menu. When complete, click the Graph Results button at the bottom of the Solution Progress window

Step 14. Examine the Output

The pressure and flow rate results are shown in Figure 9. Note that as expected, a significant amount of reverse flow occurs at the pump from about 6 to 12 seconds, requiring data beyond the standard data which is typically used for forward flow with forward rotation at the pump. The 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 10. Note how the pump speed decreases at a faster rate than Figure 4. Additionally, at about 10 seconds the pump begins to experience reverse rotation. This means that from 5 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.

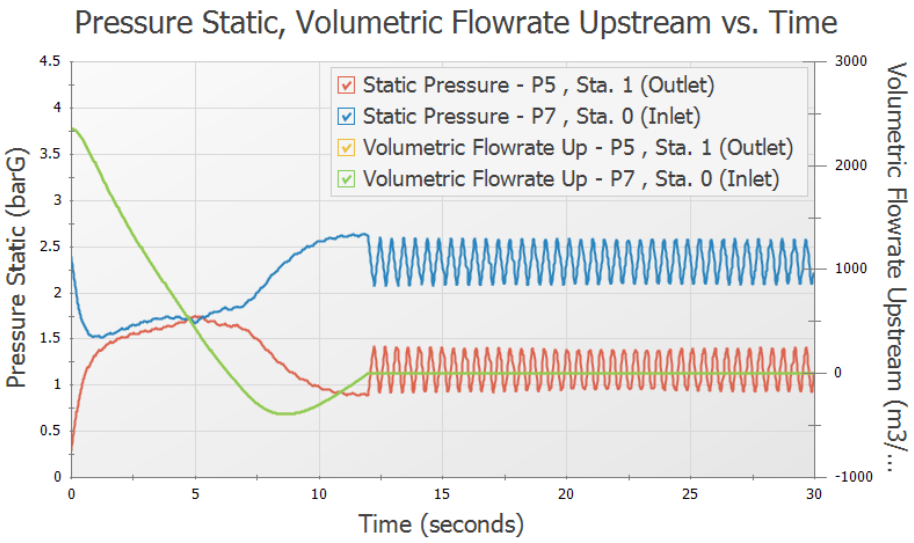


Figure 9: Predicted transient pump suction and discharge pressures and volumetric flowrates with a 12 second valve closure rate

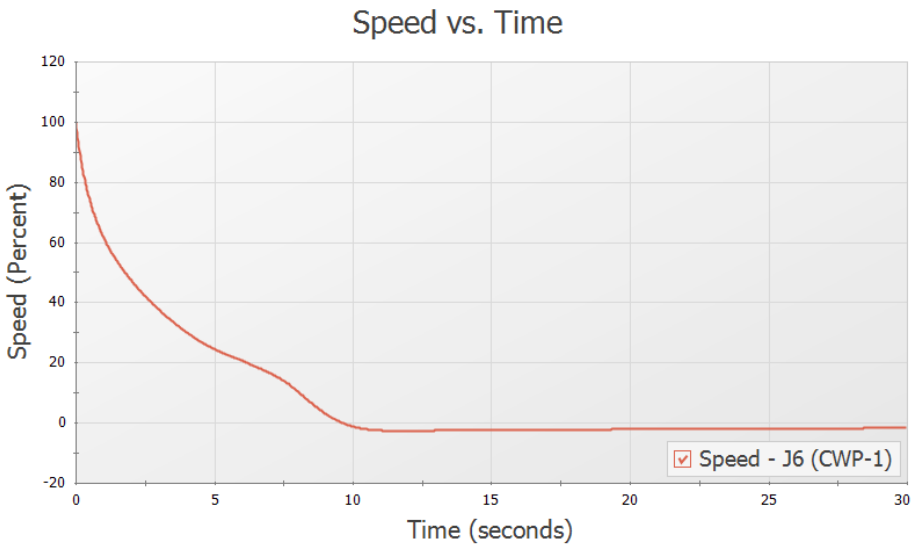


Figure 10: Pump speed decay when the valves are modified to close fully at 12 seconds

Conclusion

For the valve closure of five seconds only a minimal amount of reverse flow was experienced at the pump. As a result, the standard pump curve was sufficient, since use of four quadrant data to accurately

model this reverse flow had only a small impact on the results. Additionally, the choices of using the BEP or Steady State Operating Point to dimensionalize the four quadrant data were shown to be very similar in the transient, with minor differences in the steady state operating point. Changing the valve to close fully at twelve 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.

Selecting a Pump Four Quadrant Curve

Summary

During waterhammer analysis of pumping systems there is potential for the pump to operate outside of its normal operating region where flow, head, and speed are positive. It may be required to use a four quadrant pump curve in order to better predict the pump's behavior when operating outside of this normal operating region. As manufacturers rarely provide four quadrant data for the pump being used, a published four quadrant data set in dimensionless form is often chosen instead and dimensionalized to approximate the behavior of the actual pump being used in the system. For many cases the choice of four quadrant curve has a low impact on the waterhammer analysis results; however, this will not always be the case. This example is intended to provide the engineer with a deeper understanding of how to select four quadrant data sets for a waterhammer analysis, and how to evaluate the transient results predicted by the four quadrant curves.

This example considers a water transfer system for which several pump trip/valve closure cases are being modeled. There is a high possibility for reverse flow through the pump and reverse rotation of the pump impeller as the water is being pumped uphill, thus four quadrant data sets will be considered to model the behavior of the pump when operating with reverse flow and/or reverse rotation.

For those more interested in a quick how-to discussion and do not need in-depth discussion, consult the Overview of performing a sensitivity study for four quadrant analysis topic in Step 2 below.

Topics Covered

- Selecting a four quadrant data set to analyze cases with reverse flow and/or reverse rotation at the pump
- Performing sensitivity studies to assess four quadrant data selection
- Comparison of the Best Efficiency Point (BEP) and Steady State Operating Point (SSOP) dimensional reference point options

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

This example expands on the topics introduced in the Pump Trip with Backflow - Four Quadrant Modeling example. It is recommended to work that example first as an introduction to basic four quadrant modeling concepts before working this example.

Nomenclature

As this is an advanced topic, a brief list of abbreviations is given here for reference.

BEP	Best Efficiency Point of the standard pump curve
DRP	Dimensional Reference point, reference point used to create a four quadrant curve for a chosen four quadrant data set
Ns	Specific speed for the pump
SPC	Standard pump curve – sometimes called here the Manufacturer Curve, this is the head and efficiency (or power) vs. flow rate curve for the actual pump in your system
SSOP	Initial steady state operating point for the actual pump (from the manufacturer's curve and system curve)
4QBEP	Four quadrant pump curve created with reference to the BEP
4QSSOP	Four quadrant pump curve created with reference to the SSOP

Model Files

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

- ***Metric - Selecting a Pump Four Quadrant Curve - Initial.imp***
- ***Metric - Selecting a Pump Four Quadrant Curve - Final.imp***
- ***10 Sec Valve Opening.txt***
- ***10 Sec Valve Closing.txt***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Open the Model

Water is being provided to two different processes, represented by Tank A and Tank B, as can be seen in Figure 1. The system is designed to provide water to one tank at a time.

In this transfer system several pump trip scenarios will need to be analyzed which have potential to experience back flow through the pump before the emergency shut-down (ESD) valve fully closes. Different four quadrant data sets will be compared against the manufacturer pump curve data for each of these cases to help the engineer understand the sensitivity to four quadrant data set selection and the range of potential predictions that result. The cases are as follows:

1. Unplanned pump trip after the flow has been transitioned from Tank A to Tank B
   a. The pump operates far below BEP initially (about 60% BEP). Minimal reverse flow occurs at the pump.
2. Unplanned pump trip after the flow has been transitioned from Tank B to Tank A
   a. The pump operates above BEP initially (about 113% BEP). Sustained reverse flow and reverse rotation occur at the pump.
3. Planned pump trip and Valve B closure after Valve A fails open
   a. The pump operated at BEP initially. Sustained reverse flow and rotation occur at the pump.

Note: No pump startup cases are considered for this example, but the procedure is applicable to those situations as well if reverse flow does not occur during the initial steady state solution.

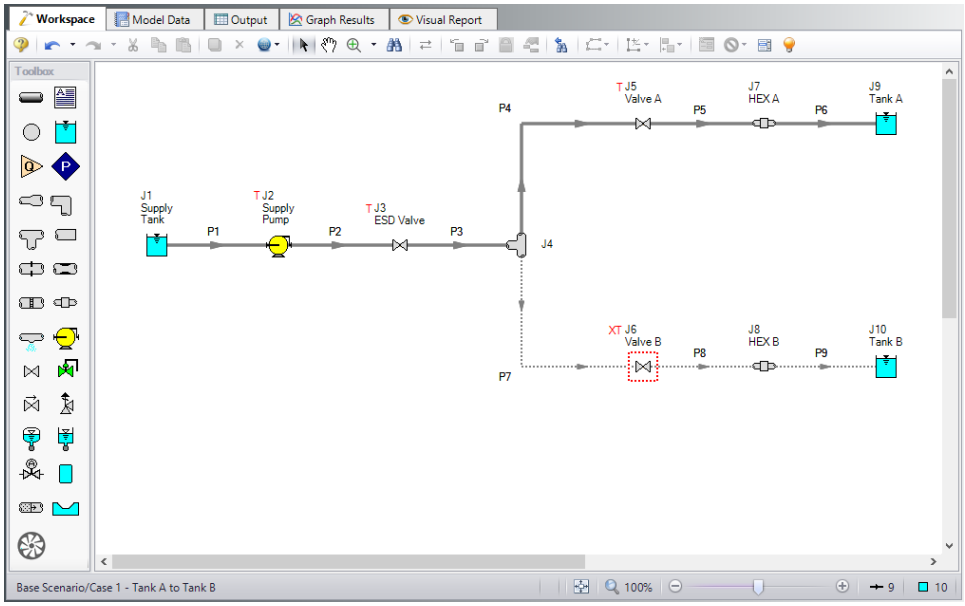


Figure 1: Workspace for water transfer system

We will start with a given model file that is fully defined for the transient run using the manufacturer's curve. Navigate to the Examples folder installed with AFT Impulse, and open the model file named ***Metric - Selecting a Pump Four Quadrant Curve - Initial.imp*** to the Case 1 - Tank A to Tank B scenario.

Note that we have four junctions with transients defined, as is denoted by the red T in the Workspace. These transients occur as two events. For the first event a planned switch from Tank A to Tank B occurs at time zero. Valve A will begin closing over ten seconds, following the transient shown in Figure 2 below. Valve B will begin opening at the same time over ten seconds following the reverse profile of that shown for Valve A. The second event happens twenty seconds into the transient run (ten seconds after the tank transition finishes), at which time an unplanned pump trip occurs and the ESD valve begins to close over seven seconds. The pump is modeled as a pump trip with inertia.

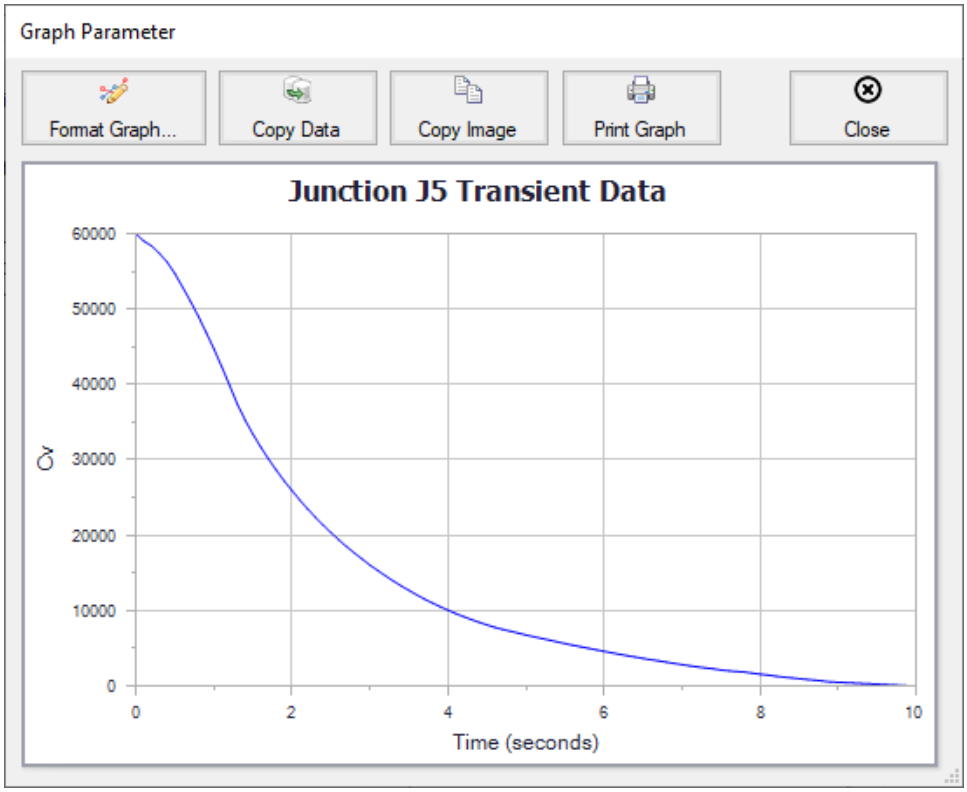


Figure 2: Valve closure transient for Valve A

Step 3. Understand the Need for Sensitivity Studies for Four Quadrant Analysis

When the Standard Pump Curve is used (i.e., not four quadrant data) and reverse flow occurs at a pump, AFT Impulse uses the zero flow value for head and power. On a graph of head vs flow this would be represented as a horizontal line for negative flows. This may be a good estimate for negative flow that either has a small magnitude or happens for a short duration, but using a four quadrant data set is often better for high magnitude or sustained reverse flow, as this method introduces less uncertainty than just assuming that head and power would remain constant. Different four quadrant data sets can produce widely varied results, so choosing the four quadrant data set which best fits the pump being modeled is important.

Considerations for Choosing a Four Quadrant Curve

During the four quadrant data set selection process, there are several factors that will need to be accounted for. A brief background for each metric is provided below:

1. **Pump Specific Speed**: It is typically good practice to select a four quadrant data set based on the specific speed of the pump, as pumps with similar specific speeds will often behave similarly. However, the four quadrant data set with the closest reported specific speed is not always the best choice to represent the pump being analyzed for multiple reasons, including uncertainty in the

accuracy of the reported specific speed for some four quadrant data sets. Thus, comparing multiple four quadrant data sets at specific speeds close to the specific speed of the pump being analyzed is advised. Note that in Impulse, each four quadrant data set is labeled as Preferred, Average, or Use With Caution, indicating the apparent quality of the four quadrant data set and the accuracy of the reported specific speed. When possible, the Preferred data sets should be selected for use in the model, as these sets provide the most trustworthy data. Average curves may be used if no Preferred data sets are available at similar specific speeds to the pump being used. The Use With Caution data sets should only be used when there are no Preferred or Average data sets that have a similar specific speed to the pump in use, or if the user has previous experience with using that data set which gives them confidence in that data set.

2. **Comparison to Manufacturer's Curve**: The manufacturer's curve will always provide the most accurate prediction of the pump's behavior while the pump is operating in the normal pumping zone of operation (i.e. flow in the positive direction, positive head, positive pump speed), and while the pump is operating within the range of flows that were tested by the manufacturer. Therefore it is reasonable to assume that a four quadrant data set which closely matches the manufacturer's curve while in the normal zone of operation will provide more accurate results than a four quadrant curve that shows larger differences from the manufacturer's curve in the normal pumping zone of operation. This comparison to the manufacturer's curve should be done for both the initial steady state operating point and the final operating point after all transients have died out. For this example we will refer to the final operating point after the transients have died out as the final steady state operating point. The reasoning for this will be discussed further throughout the example.
3. **Dimensional Reference Point (DRP)**: Four quadrant data is usually published as a dimensionless set of data to allow it to be used for analysis with pumps besides the test pump. When dimensionalizing the data set for a pump, either the BEP or the SSOP may be used as the dimensional reference point. Many published four quadrant data sets use the BEP as the DRP to non-dimensionalize the data, so it seems logical that the BEP should be used as the DRP for the reverse transformation. However, if the pump is operating far from the BEP in the steady state, the initial steady state operating point predicted using the BEP as a DRP may be very different from the SSOP predicted from the manufacturer's curve. In that case using the SSOP as the DRP is useful to prevent differences from the steady state results using the manufacturer's curve. However, using the SSOP as the DRP often causes large differences between the manufacturer's curve and the four quadrant curve. In summary, there is often a trade-off between how well the four quadrant curve matches the manufacturer's curve for the steady state and final steady state, so it is always advised to test both DRP options. Note that if the simulation begins with a significantly incorrect initial flowrate, this will impact all subsequent results in a way that may be difficult to assess.

Procedure for Comparing Four Quadrant Data Sets

1. To perform the sensitivity study explained above the following steps can be taken:
2. Create a child scenario of the scenario that is defined using the manufacturer's curve
3. Open the Pump Properties window and select Four Quadrant Curve as the Performance Curve Used in Simulation
4. Choose User Selected and click Specify Model to launch the Specify Four Quadrant Model window

5. Choose the desired four quadrant data set from the drop-down list at the top of the window, and select Best Efficiency Point as the Dimensional Reference Point.
6. Select OK to accept the changes
7. Clone the current scenario and change the Dimensional Reference Point to the Steady-State Operating Point in the cloned scenario
8. Repeat steps 1 - 6 for each data set to be analyzed
9. Run all of the newly created scenarios and compare the transient results. Note that frequently results will not differ greatly among scenarios. In that case, results are not very sensitive to the specific choice of four quadrant curve. This makes the analyst's job easier.
10. In order to be most conservative, use the worst case predictions from among the various four quadrant scenarios for design purposes.
11. If the worst case predictions lead to excessive and/or expensive implications in design and/or operations, take a closer look at results. Compare each four quadrant pump curve to the manufacturer's curve at the initial steady state and, if applicable, the final steady state operating points to determine which scenario(s) appears to give the most accurate results. Criteria used to decide the most reliable four quadrant curve(s) include:
    a. Similarity of specific speed, Ns – when making this comparison, be sure you have an accurate Ns value for your pump (see discussion below for more on this)
    b. Overall agreement between the manufacturer's curve (Standard Pump Curve) and dimensionalized four quadrant curve for head and power vs. flow rate in the normal zone of pump operation (Impulse displays these curves for you – review the comparisons)
    c. Give more weight to the four quadrant curves which show closer agreement within the range of flow rates that the manufacturer tested the pump at (i.e. the range of flow rates that the pump curve is graphed for on the specification sheet)
    d. Give more weight to four quadrant curves which generate transient results similar to the Standard Pump Curve during time frames when the pump is operating in the normal zone of operation
    e. Give more weight to four quadrant curves Impulse categorizes as Preferred
    f. Good agreement in initial steady-state flow rates between SPC and four quadrant curve
    g. Good agreement in final steady-state flow rates between SPC and four quadrant curve – this not as important as agreement in initial steady-state flow rates, but should be considered

Note that there are several reasons why the engineer may choose to skip Step 10 above. One reason would be if the transient results for the different scenarios show only minor differences as noted in step 8 above. In that case it would be unnecessary to determine which scenario is most accurate, as any scenario will produce similar results. Another reason would be if the engineer's analysis involves a project where highly conservative results are required, such as for a nuclear system. In that case the engineer could choose to simply use the most extreme results for the analysis for their design purposes. Note this means that if different scenarios produce the worst-case pressures in different areas of the model, then the engineer may want to use the most extreme results (e.g., highest or lowest pressure) regardless of scenario, rather than only using the results from one scenario.

Step 3. Run the Model with the Manufacturer's Curve

➢**Click Run Model from the Toolbar or Analysis menu,** then proceed to the Output window.

Notice that two warnings are given for this scenario, one indicating that negative head rise was calculated at the pump, and one indicating that reverse flow occurred at the pump. It will be easier to discuss these warnings by reviewing the transient head rise and flow rate at the pump.

➢**Go to the Graph Results window.** In the Graph List Manager on the Quick Access Panel double click the Head Rise and Vol Flow at Pump Outlet graph list item to load the graph shown in Figure 3. You should see that the flow rate remains positive for the majority of the transient, with minor reverse flow occurring after 27 seconds, never reaching a magnitude higher than 30 gal/min. Graphing the pump head rise shows that the head rise reaches zero at about 21 seconds, and is fixed at zero until around 27 seconds. The low magnitude of reverse flow would suggest that a four quadrant curve may not be needed for use in this model. However, the head rise results introduce a large amount of uncertainty into the results, since AFT Impulse artificially fixes the head rise to zero. Using a four quadrant curve will allow Impulse to estimate the negative head rise which would occur at the actual pump.

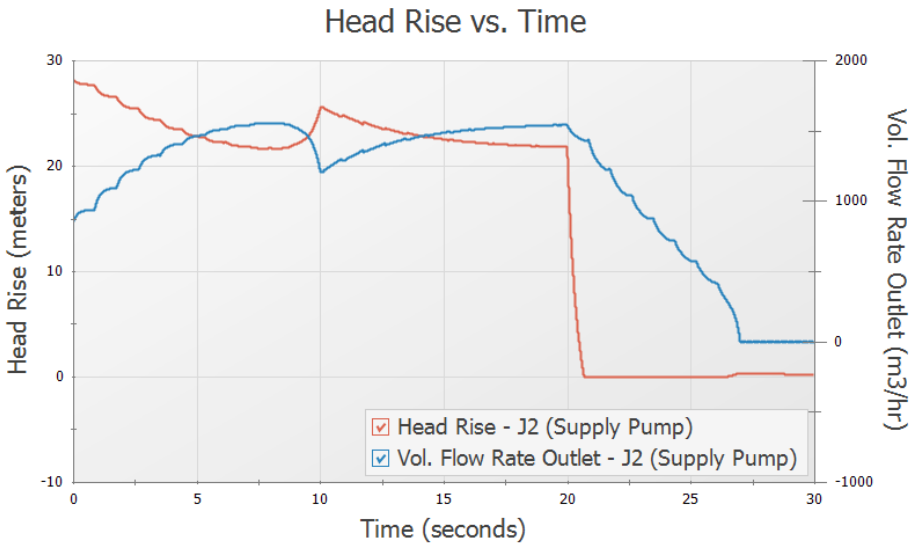


Figure 3: Pump head rise and volumetric flow rate when using the manufacturer's curve

➢**Return to the Output window** and go to the Pump Summary tab (Figure 4). The BEP was calculated based on the user-specified power curve to be 1,389 m3/hr. Note that in this case the pump is operating at 854.2 m3/hr during the steady state, which is 61.51% of BEP. This will be an important consideration when choosing a dimensional reference point for the four quadrant data set, as was discussed previously.

General | Warnings | Design Alerts | Event Messages By Jct | Event Messages By Time | Pump Summary | Valve Summary

Jct (Steady)	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Supply Pump	854.2	237.2	2.771	28.27	79.24	100.0	82.9	1,389	61.51	101.1	N/A

Figure 4: Steady State results for the pump calculated using the manufacturer's curve

Step 4. Choose Four Quadrant Data Sets to Analyze

A. Calculating or Estimating Specific Speed

From the first consideration above, the first step for selecting the four quadrant curve will be to determine the specific speed of the pump and choose a four quadrant data set accordingly. AFT Impulse has a built-in tool to estimate the specific speed for the pump on the Specify Four Quadrant Model window if the specific speed is not available from the manufacturer.

➢**Create a child scenario of Case 1 - Tank A to Tank B named *Manufacturer's Curve***, this scenario will hold the results that were just obtained. Create another child of the Case 1 - Tank A to Tank B scenario named *Four Quadrant Curve*. Next, open the Pump Properties window. Select Four Quadrant Curve, choose User Specified for the type, then click Specify Model. By clicking Suggest a Data Set towards the top of the window, we can see that Impulse has estimated a specific speed of 51.5 metric units based on the displayed inputs (Figure 5). Looking at the specification sheet for this pump given by the manufacturer, the specific speed is given as 32.0 metric units. In this case the discrepancy between Impulse's calculations and the manufacturer's calculations is due to several assumptions that were made in calculating each value.

The first assumption involves the impeller size for the pump. ANSI/HI 14.3.1.3.1-2019 states that the specific speed for any pump should be calculated for the pump at the maximum impeller size. Impulse will assume by default that the pump is untrimmed. However, for this example the impeller was trimmed. The specific speed estimate in Impulse can be improved by entering the trimmed and untrimmed impeller sizes, allowing Impulse to adjust the head and flow values using the affinity laws.

➢**Select Trimmed under Impeller Diameter**, and enter 508 mm and 472 mm as the maximum and Standard Pump Curve diameters, respectively. Click Calculate and the estimated specific speed should now be listed as 47.9 metric units, as shown in Figure 6.

Another assumption contributing to this discrepancy involves the flow rate used for the calculation. ANSI/HI 14.3.1.3.1-2019 states that the specific speed should be calculated using the total flow through the pump. However, the standard does note that there is an alternate definition in industry which defines the specific speed as the flow rate per impeller eye. For double suction pumps (such as the pump in this example) this definition produces a specific speed which is reduced by 1/sqr(2), or approximately 0.707 times the total flow definition. If the specific speed value provided by the manufacturer (32.0) is divided by this factor the resulting specific speed of 45.3 matches much more closely with Impulse's estimate of 47.9 that was just calculated, which confirms that the specific speed reported by the manufacturer was for the flow per impeller eye. In Impulse it is recommended to use the definition of specific speed based on the total flow, as this is generally more consistent with the specific speed definition used for the four quadrant data sets available in Impulse.

➢**Choose Select Suggested Data Set and Close** to accept the suggested 41.8 m3/sec units data set, which has the closest specific speed to the calculated specific speed chosen above.

Note: In cases like this where the specific speed has been provided by the manufacturer the value provided by the manufacturer will typically be more accurate. However, the engineer should verify whether the manufacturer has reported the specific speed based on the total flow or flow per impeller eye.

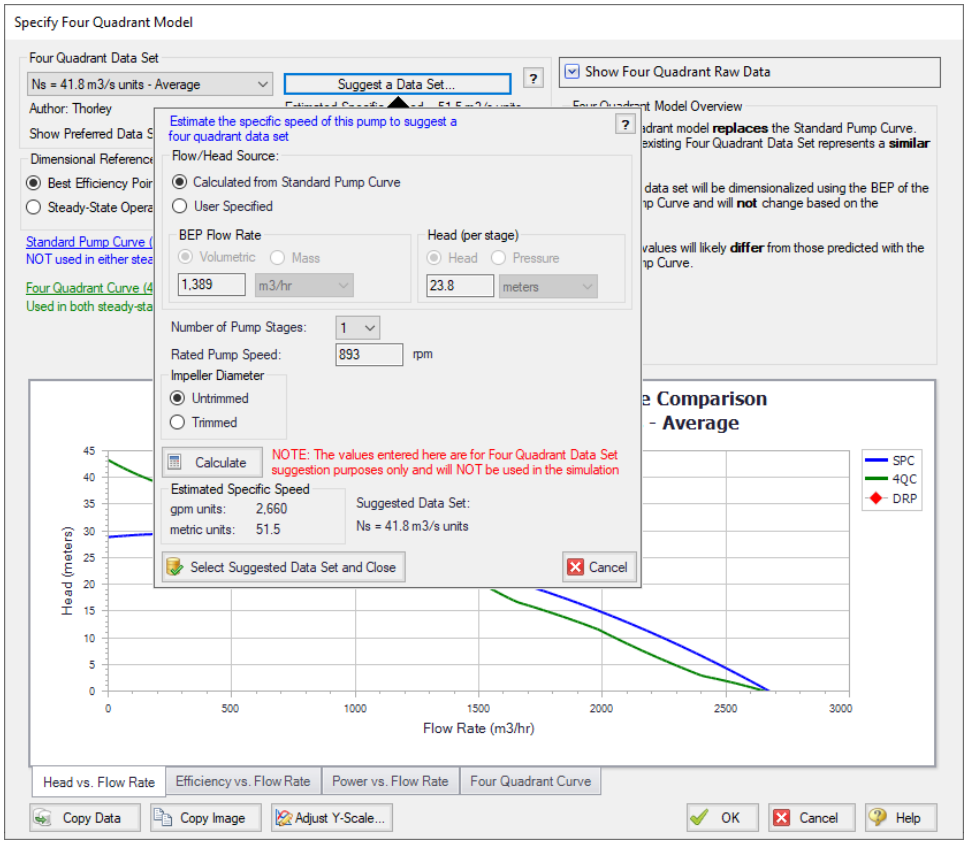


Figure 5: Impulse can estimate the specific speed and suggest the nearest four quadrant data set in the Specify Four Quadrant window. Suggested Data sets will by default only be Preferred or Average

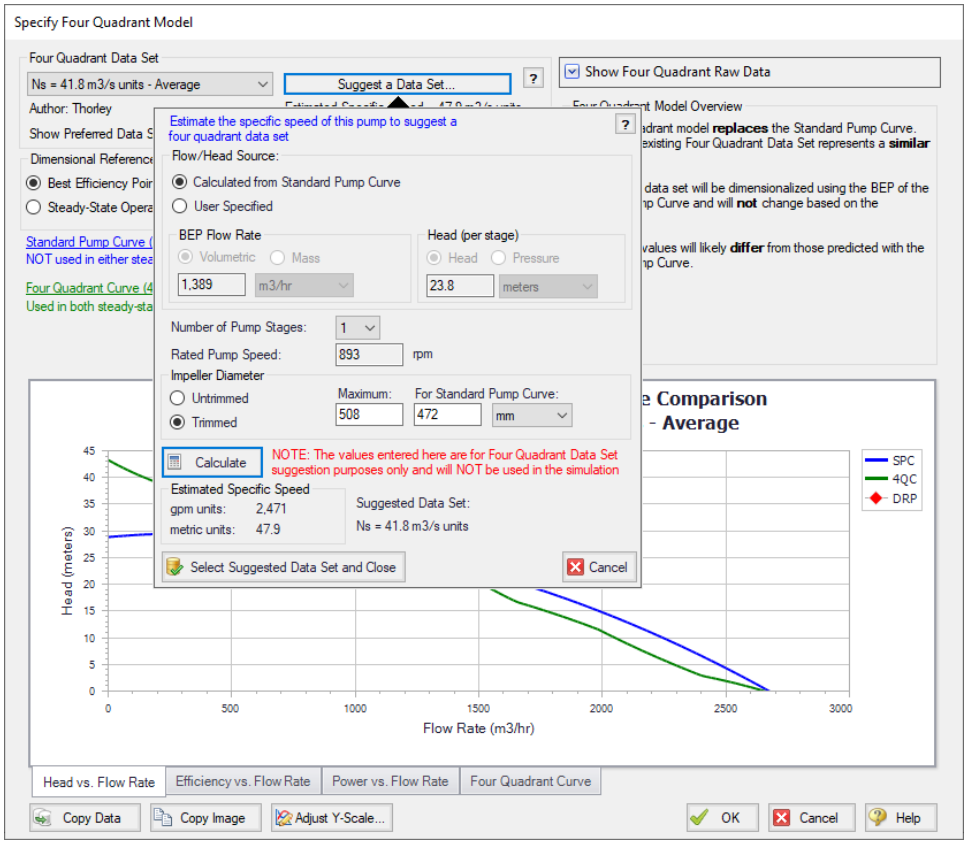


Figure 6: Impulse can account for trimmed impeller pumps using the affinity laws to adjust the flow rate and head for calculations, as shown above

B. Setting up Four Quadrant Scenarios

We just determined that the four quadrant data set with the closest specific speed is the 41.8 m3/sec data set. This data set is Average, so it is an acceptable candidate for use. As discussed above, it would also be prudent to test several other data sets with similar specific speeds. The next closest Preferred or Average data sets are the 37.4 m3/sec and 64 m3/sec data sets, so they will also be considered for use in this model.

Each of the selected data sets should now be compared to the manufacturer's curve with the data set dimensionalized at the BEP and the SSOP. From here on we will refer to four quadrant curves that have been dimensionalized using the BEP as 4QBEP curves, and four quadrant curves that have been dimensionalized using the SSOP as 4QSSOP curves for convenience.

Create the scenarios for the selected four quadrant data sets as follows:

1. In the current Four Quadrant Curve Scenario check that BEP is selected for the DRP (dimensional reference point). Note that the head, efficiency, and power curves versus flow can be seen plotted against the Standard Pump Curve (referred to as the manufacturer's curve for this example) at the bottom of this window, as shown below for head in Figure 7 and power in Figure 8.

Note: The raw data from the graphs shown in the Specify Four Quadrant Model window (Figure 7 and Figure 8) can be copied using the Copy Data button at the bottom of the window to compare this data for all scenarios in Microsoft Excel.

2. Select OK to accept the current settings.
3. Rename the Four Quadrant Curve scenario in the Scenario Manager as Ns 41.8 4QBEP.
4. Clone the Ns 41.8 4QBEP scenario to create a Ns 41.8 4QSSOP scenario, then change the Dimensional Reference Point in the cloned scenario to the Steady-State Operating Point and change the Sample OP value to 854, as can be seen in Figure 9.
5. Create cloned scenarios for the 37.4 m3/sec and 64 m3/sec four quadrant data sets, changing the four quadrant data set and dimensional reference points as needed. When finished the Scenario Manager should appear similar to Figure 10.
6. Run all of the four quadrant scenarios so that they have results.

Note: A batch run can be used to speed up the process of running multiple scenarios. This can be set up from the File menu by selecting Start Batch Run, then selecting all of the scenarios to be run and clicking Start Run.

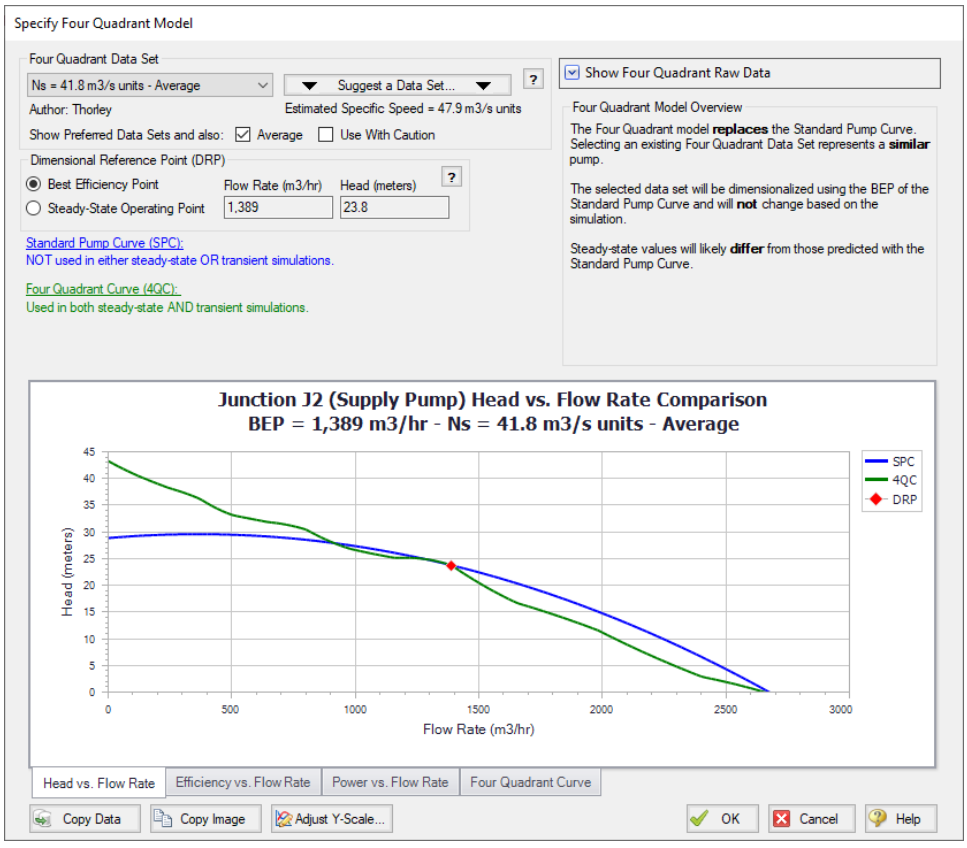


Figure 7: Specify Four Quadrant window showing the Head vs Flow data for the Ns = 41.8 data set dimensionalized using the BEP

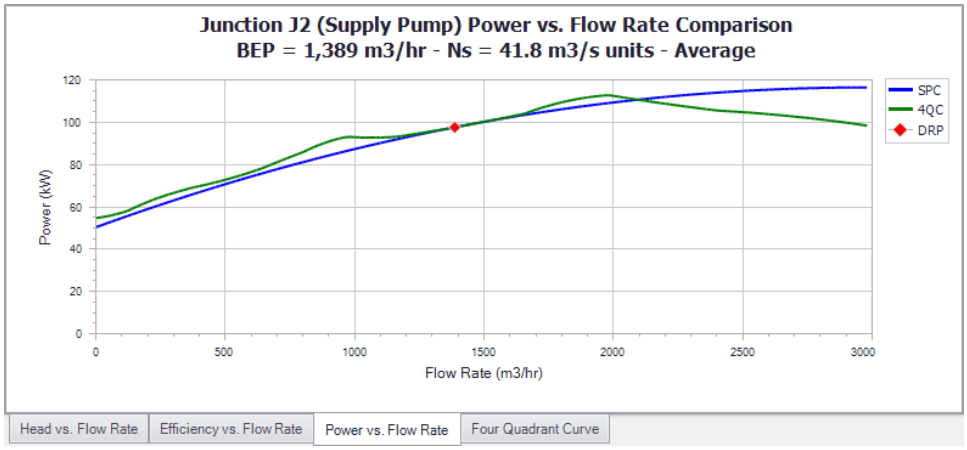


Figure 8: Power vs Flow graph from the Specify Four Quadrant window for the Ns = 41.8 m3/s units data set dimensionalized using the BEP

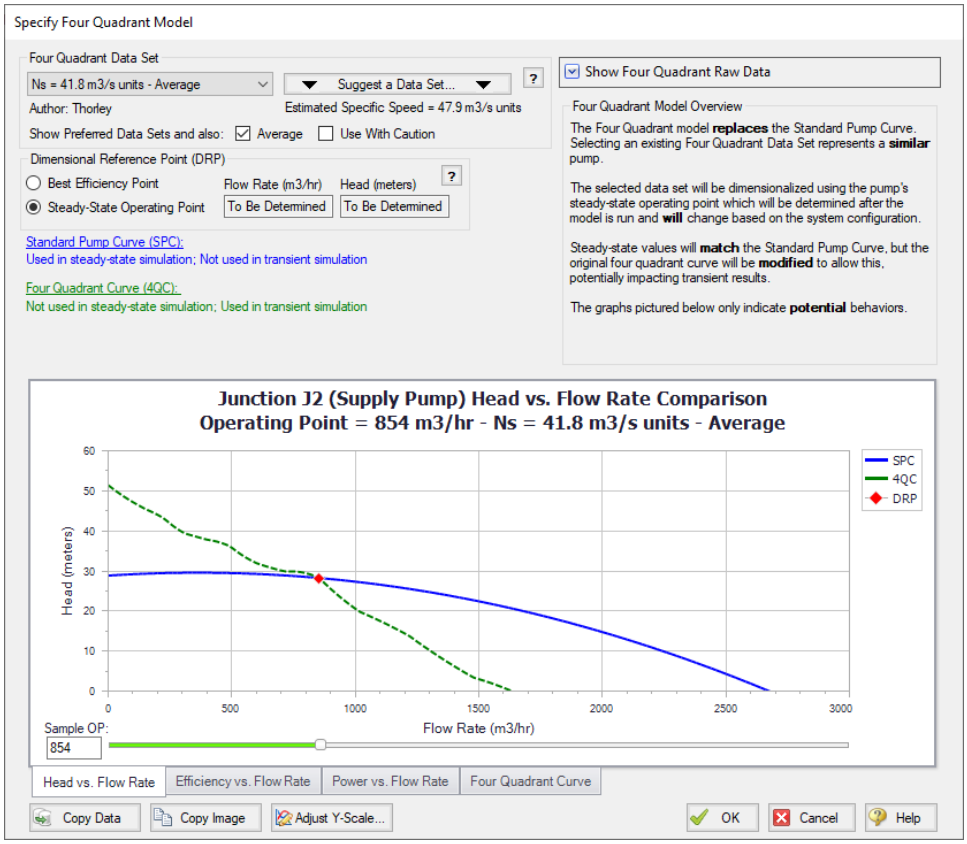


Figure 9: Specify Four Quadrant window showing the Head vs Flow data for the Ns = 41.8 data set dimensionalized using the SSOP, which is 854 m3/hr for this scenario

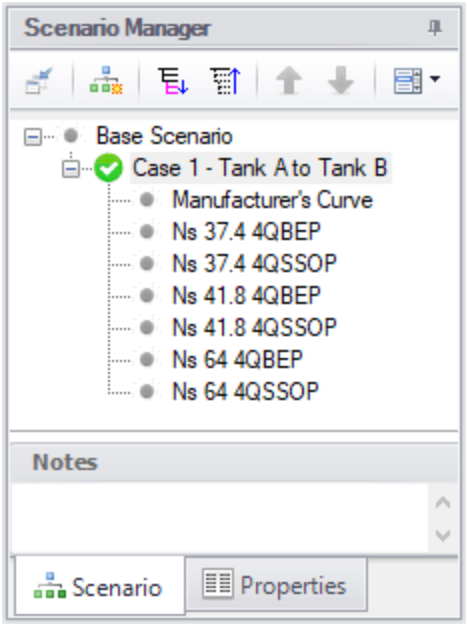


Figure 10: Scenario Manager with the scenarios created for each of the four quadrant data sets dimensionalized at the BEP and SSOP

C. Comparing Four Quadrant Pump Curves

If the engineer's goal is simply to perform the most conservative analysis possible in the shortest amount of time, then comparing the pump curves as will be done below would be unnecessary. Instead, the engineer should simply proceed to compare the transient results for all four quadrant scenarios and use the maximum/minimum results from all cases, as was discussed at the beginning of this example.

Additionally, the below comparison may prove to be unnecessary if the transient results show negligible differences. However, we will compare the manufacturer's curve and four quadrant pump curves here to be thorough.

Finding the actual initial and final steady state flow rates

For the next step in the sensitivity analysis we will compare all of the pump curves to the manufacturer's curve at both the initial steady state operating point and the final steady state operating point, as was mentioned at the beginning of this example. There are two transient events that occur in this model, so in a sense we have two final operating points, one after each event. The final operating point for Event 1 will be when the flow has fully transitioned to Tank B, but Event 2 has not yet begun. The final operating point for Event 2 will be when the ESD valve and Valve A have fully closed, preventing flow in the system. We need to compare the four quadrant curves for the final operating point after Event 1, since the operating point at this time will affect the results for Event 2. After Event 2 the system will be at zero flow and no transients will occur, so this is not a point of interest for comparison.

The initial and final steady state flow rates can be found by running the manufacturer's curve scenario, as was done earlier in the example. In this case we already saw that the steady state operating point for the pump was 854 m3/hr (see Figure 4, actual flow rate). To find the actual final steady state after the

transient effects have completely dispersed create a new scenario using the manufacturer's curve for the pump, and adjust the model in the workspace to reflect the final steady state. This scenario can then be run using Time Simulation – Steady Only from the Analysis menu, as we are only interested in determining the steady results. This predicts a flow rate of about 1560 m3/hr for the final steady state operating point.

Checking the dimensionalized curves

Each individual four quadrant curve can be compared to the manufacturer's curve within the Specify Four Quadrant Model window in Impulse, as was seen above in Figure 7 and Figure 8 for the 41.8 m3/s units data set. Each four quadrant curve will intersect the manufacturer's curve at the dimensional reference point, which is shown in red (1,389 m3/hr, see Figure 4 BEP). For the head, power, and efficiency curves we will want to check how similar the shapes of the curves are and how close they are to each other. Focus should be given in particular to how well the curves match near the initial and final steady state flow rates, as this will indicate how different those operating points will be.

When the BEP is used as the dimensional reference point, there is only one possible effective head and power curve for each four quadrant data set, as the BEP will not change when the system changes. However, if the steady state operating point is chosen then different effective head and power curves for each four quadrant data set will result depending on where the manufacturer's curve operates during the initial steady state. Use the slider or the text box on the graph in the Specify Four Quadrant Model window to make sure that the graph is displaying the four quadrant curve for the actual steady state operating point before using the graph for comparison. The 41.8 four quadrant data set dimensionalized at the steady state operating point identified above is shown in Figure 9.

If any four quadrant data set shows a substantial discrepancy for the initial or final steady state operating point, then it may be prudent to consider substituting this data set for another, in particular if the data set you are inspecting is marked as Average or Use With Caution.

Pump vs System Curve Comparison

To more easily compare the curves, the operating points for the head curves are shown using a multi-scenario pump vs. system curve that was created in Fathom 12 (Figure 11). The power curves are graphed in Figure 12 using Microsoft Excel. Table 1 listing the operating points shown in Figure 11 is also provided below for convenience. An engineer would not typically need to do this, but it is shown here to help reinforce the concepts discussed.

Each of the 4QBEP head curves cross the manufacturer's curve at the BEP, which is 1,389 m3/hr for this pump as was seen in the Pump Summary in the Output window (see Figure 4, BEP). The SSOP curves cross the manufacturer's curve at the actual operating point (854 m3/hr, see Figure 4 volumetric flow rate). Thus, the steady state flow rates will be identical between the 4QSSOP scenarios and the original scenario using the manufacturer's curve, unlike the 4QBEP curves. This distinction between the 4QSSOP and 4QBEP curves can be important because if the initial steady-state flow differs significantly from the actual pump you are modeling, the entire simulation will be impacted. Indeed, something like a sudden valve closure which happens early in the transient can result in very different peak pressure predictions. For example, the BEP curve that operates furthest from the manufacturer's curve for the initial steady-state is the 37.4 4QBEP curve which operates at about 610 m3/hr and 26 meters and is 240 m3/hr and 2 meters below the operating point with the manufacturer's curve.

We should also compare the final system curve to the pump curves. The final steady state will impact any transients that may occur later on in the simulation or after the end of the simulation, such as a sudden valve closure. In this case the Final System Curve represents the steady state after the first event (Valve

A closing and Valve B opening), but before the second event (the unplanned pump tripping while the ESD valve closes). Thus any large differences in the final steady state will impact the subsequent pump trip. It can be seen from Figure 11 below that each of the 4QBEP curves show deviations from the manufacturer's curve at the Final System Curve, but the 4QSSOP curves show a much larger deviation than the 4QBEP curves. The 41.8 4QSSOP curve differs by 430 m3/hr and 5 meters from the predicted final operating point, which could have large consequences on the peak pressures.

In Figure 12 we can see similar trends to those observed for the power curves. At the initial steady state the 4QSSOP curves match the manufacturer's curve perfectly, while the 4QBEP curves deviate by various amounts. Near the final steady state flow the 4QSSOP curves show significantly more deviation than the 4QBEP curves. This is likely due to the fact that the initial steady state flow is far from the final steady state flow.

The 41.8 4QBEP curve seems to show the least overall deviation from the manufacturer's curve in terms of both head and power based on these qualitative observations. However, using a 4QBEP curve will produce different steady state results than the manufacturer's curve, which could be problematic. While the 4QSSOP curves match the steady state results perfectly, the results in the transient would likely provide less trustworthy results based on the large deviations they show from the manufacturer's curve at the final steady state system curve. To test this conclusion let's compare the transient results from all six four quadrant curve scenarios to the transient results using the manufacturer's curve.

Table 1: Pump head, flow, and discharge velocity at the initial and final steady state operating points as can be seen in Figure 11. The Manufacturer's Curve shows the correct results, while the others are an artifact of the selected 4Q curve

	Initial Steady-State			Final Steady-State		
Pump Curve	**Flow (m3/hr)**	**Head (meters)**	**Velocity (meters/sec)**	**Flow (m3/hr)**	**Head (meters)**	**Velocity (meters/sec)**
Manufacturer's Curve	854.2	28.27	1.264	1,560	21.67	2.308
37.4 4QBEP	612.5	26.22	0.906	1,611	22.36	2.383
37.4 4QSSOP	854.2	28.27	1.264	1,255	17.96	1.857
41.8 4QBEP	882.4	28.55	1.305	1,490	20.75	2.205
41.8 4QSSOP	854.2	28.27	1.264	1,130	16.66	1.672
64 4QBEP	992.4	29.73	1.468	1,518	21.11	2.246
64 4QSSOP	854.2	28.27	1.264	1,157	16.93	1.711

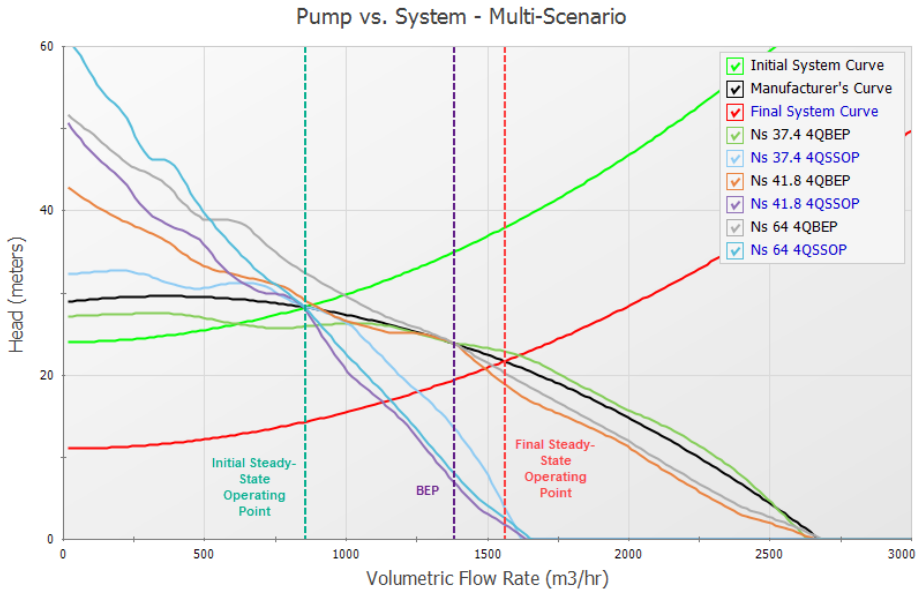


Figure 11: Pump vs. System Curve plot showing the manufacturer provided pump curve compared to pump curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at both the initial and final steady state conditions in Case 1

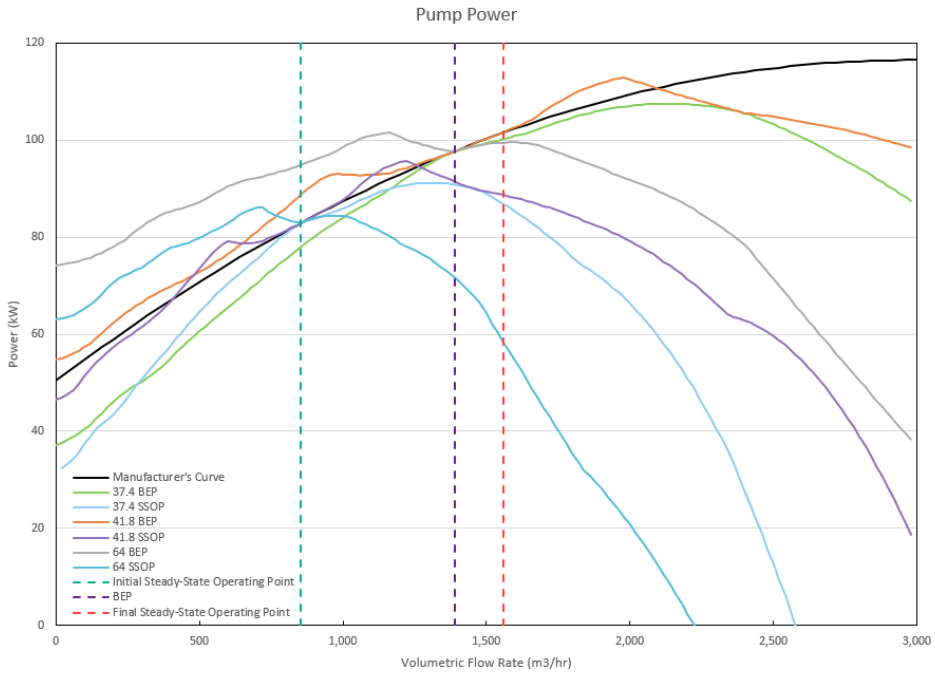


Figure 12: Pump Power Curve plot showing the manufacturer provided power curve compared to power curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at both the initial and final steady state conditions in Case 1

Step 5. Compare the Transient Results

We now want to compare the results to see how sensitive the results are to the choice of pump curve used for the simulation.

First go to the Output window for any of the four quadrant scenarios. Note that when any of the four quadrant scenarios are run, several information items will be given in the Warnings tab of the Output window. These messages are meant to inform the user of the potential implications that the four quadrant curves may have for the model, which this example explains in detail.

Now go to the Graph Results window and generate a graph of the pump speed and the velocity at the pump outlet, as can be seen in Figure 13 and Figure 14 below. From Figure 13 we can see that the pump does not reach zero speed during the transient simulation, meaning that the pump does not experience reverse rotation. In Figure 14 the velocity results at the outlet of the pump are compared. Recall from Step 3 that reverse flow occurs at the pump at around 27 seconds, though the magnitude of reverse flow was minor. In addition, during Step 3 we found that negative head rise would occur at the pump starting at 21 seconds. Since the pump operates at positive speed, positive flow and positive head for the first 21 seconds of the transient, the manufacturer's curve will produce the most accurate results during this time. Therefore, the four quadrant curve results can be compared directly to the manufacturer's curve results to determine the error in the four quadrant curve results up to 21 seconds into the transient run. After 21 seconds one of the four quadrant curves will likely give more accurate results.

For this case there is a clear difference between the initial steady state velocity (at time zero) using the manufacturer's curve and the steady-state velocities using the 37.4 4QBEP and 64 4QBEP curves.

However, as the tank transition causes the flow to increase at the pump, the velocity results for the 4QBEP curves become closer to the results from the manufacturer's curve, while the 4QSSOP curves diverge. Notice that at 20 seconds when the pump trip and ESD valve closure transients begin, the 4QSSOP curves now show a large difference from the manufacturer's curve. The 41.8 4QSSOP curve shows the largest difference from the manufacturer's curve, with a difference of about 0.7 meters/sec. The initial velocities in Figure 14 should and do match those shown in the Figure 11 (and Table 1) steady-state results.

The static pressure at the ESD valve outlet is shown in Figure 15, with the maximum and minimum pressures detailed in Table 2. We can see that the maximum and minimum pressures at the ESD valve occur once the ESD valve has fully closed after 27 seconds. The 4QSSOP pressure results experience significantly decreased peak pressures and do not experience the low pressure peaks that are seen at about 27 and 28.5 seconds for the other Case 1 scenarios.

For example, consider the 41.8 4QSSOP results, which we noted showed the largest difference in the velocity results when compared to the manufacturer's curve at time 20 seconds. The 41.8 4QSSOP curve has a maximum pressure of about 12.7 bar, as compared to the 16.2 bar peak pressure for the manufacturer's curve (See Table 2 below). At about 27 seconds a low pressure of 5.7 bar is calculated using the manufacturer's curve, while the 41.8 4QSSOP curve remains at a pressure of 10 bar.

This difference in results makes sense based on the large differences from the manufacturer's curve at the final system curve, as were shown in Figure 11. Note for completeness that the Figure 11 final steady state values would compare most precisely to the Impulse results if the transient was run longer and the unplanned pump trip at 20 seconds did not occur. In other words, the waterhammer transients need to all die out for Impulse to find the true, final steady state. We can see in Figure 14 at 20 seconds that the transient velocity has roughly but not completely steadied out.

Overall for Case 1 where negligible reverse flow occurs at the pump, we can see that the large differences observed between the 4QSSOP curves and the manufacturer's curve at the final steady state resulted in large variation in the results, especially for the pressures at the ESD valve outlet. The differences in the steady state results for the 4QBEP curves did not result in significant error in the transient results for this case due to the fact that the valve closures during the first transient event were not sensitive to the steady state results. In this case one of the 4QBEP curves would be the best choice out of the four quadrant curves analyzed as Case 1 is much more sensitive to the final steady state conditions (i.e., at 20 seconds before the pump trips) than it is to the initial steady state.

Table 2: Maximum and minimum static pressures at ESD valve and time at which the pressures occur for Case 1. Note that the results for the manufacturer's curve are suspect after 21 seconds per the discussion above

	Max. Static Pressure	Max. Static Pressure Time	Min. Static Pressure	Min Static Pressure Time
Pump Curve	**bar**	**seconds**	**bar**	**seconds**
Manufacturer's Curve	17.78	10.84	5.62	27.01
37.4 4QBEP	17.54	10.84	6.57	27.01
37.4 4QSSOP	15.69	10.84	8.70	20.85
41.8 4QBEP	17.45	10.84	6.75	27.01
41.8 4QSSOP	14.61	10.84	9.36	20.85
64 4QBEP	17.78	10.84	5.90	27.01
64 4QSSOP	14.96	10.84	8.93	27.01

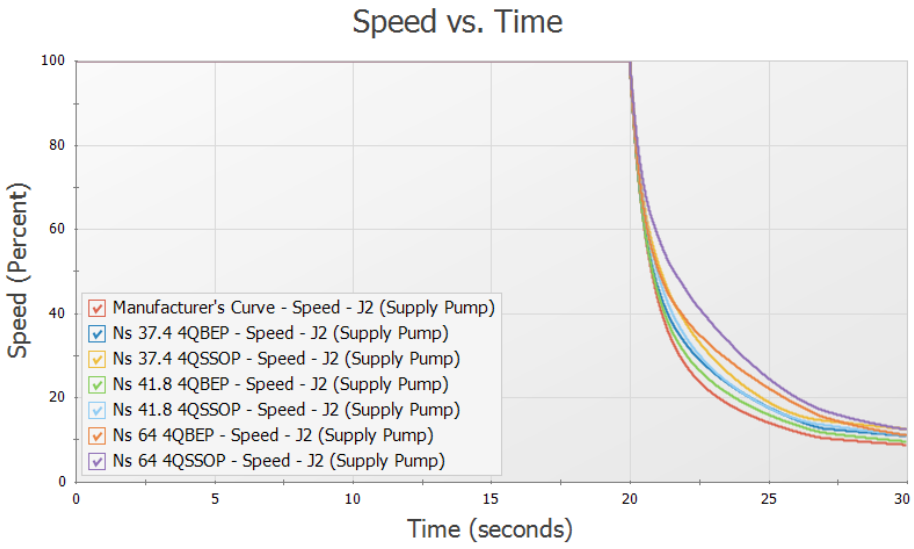


Figure 13: Pump speed vs time for Case 1 using the manufacturer's curve and each of the four quadrant curve scenarios

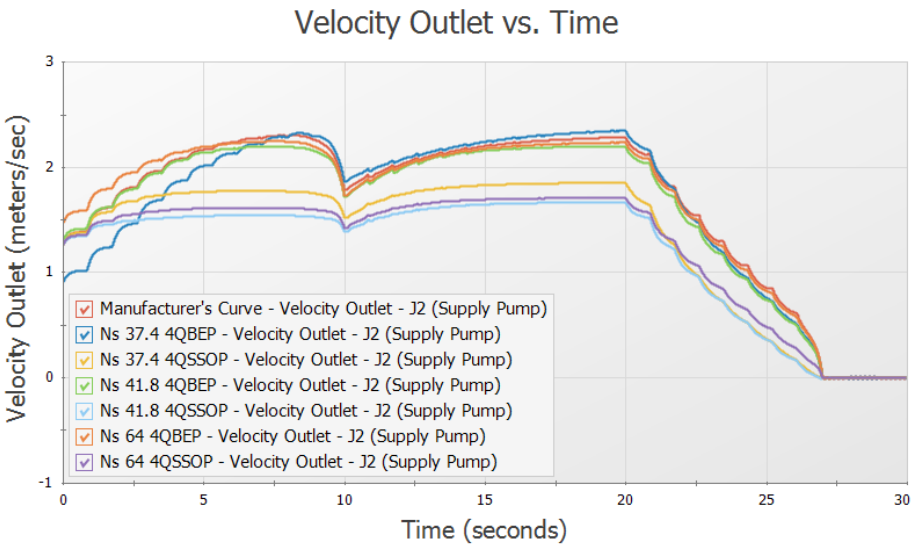


Figure 14: Velocity at pump outlet for Case 1 using the manufacturer's curve and each of the four quadrant curve scenarios

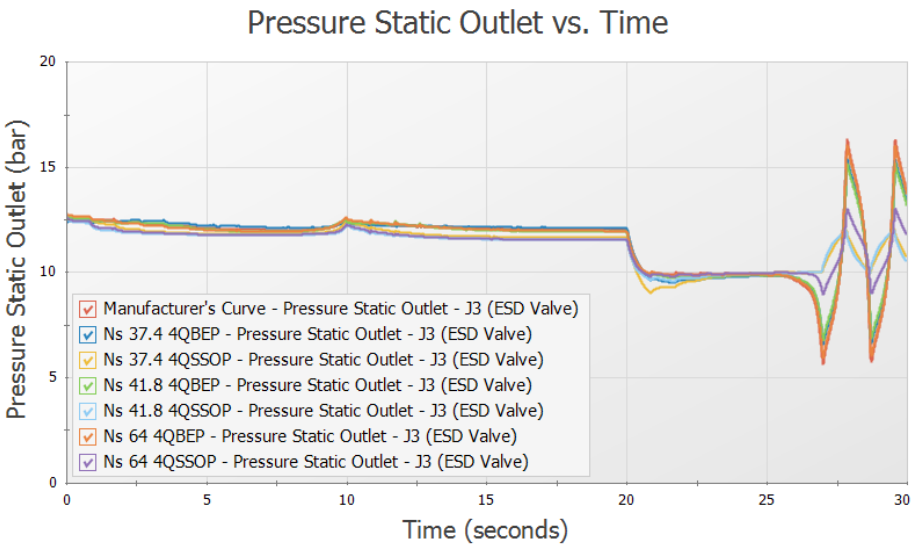


Figure 15: Static pressure at ESD Valve outlet for Case 1 using the manufacturer's curve and each of the four quadrant curve scenarios

Step 6. Case 2 - Tank B to Tank A

Case 1 operated in the normal pumping zone of operation (positive speed, head and flow) for the majority of the run, so four quadrant data was only required for the analysis of the results when the pump exper-

ienced negative head. Case 2 and Case 3 may experience more extensive negative flow, speed, and/or head at the pump requiring further use of four quadrant methods.

A. Create Scenarios for Case 2

For Case 2 the system starts with flow to Tank B during the steady state. The flow then transitions from Tank B to Tank A over ten seconds. At twenty seconds into the transient, the pump trips and the ESD valve closes, as occurred in Case 1.

Let's create scenarios for Case 2 and repeat the analysis above. Do the following:

1. Right-click the Case 1 - Tank A to Tank B scenario and select Clone With Children. Rename the scenario to Case 2 - Tank B to Tank A
2. Open the Valve Properties window for Valve A (J5). We will need to set this valve to begin closed, and open over ten seconds.
3. On the Optional tab set the Special Condition to Closed
4. Go to the Transient tab. Under the Transient Data menu select Import from File from the drop-down menu, then browse to ***10 Sec Valve Opening.txt*** in the Impulse Examples folder
5. Select Tab as the Separator, then click OK. The data should now start with Cv = 0 at time 0, and open to Cv = 6000 at 10 seconds
6. Now open the Valve Properties window for Valve B. This valve will need to begin open, then close over 10 seconds.
7. On the Optional tab set the Special Condition to None
8. Go to the Transient tab. Under the Transient Data menu select Import from File from the drop-down menu, then browse to ***10 Sec Valve Closing.txt*** in the Impulse Examples folder
9. Select Tab as the Separator, then click OK. The data should now start with Cv = 60000 at time 0, and close to Cv = 0 at 10 seconds. The Cv vs. time data should be the same as shown earlier in Figure 2 for Case 1, Valve A.

B. Compare Four Quadrant Curves to Manufacturer's Curve

Now that the scenario is fully defined, let's examine the results for the model using the manufacturer's curve.

➢**Run the Case 2 - Tank B to Tank A scenario** from the shortcut on the toolbar, or from the Analysis menu, and go to the Output window.

There will be several warnings visible for this scenario. A warning is given for reverse flow at the pump. If we were to graph the flow rate at the pump as was done for Case 1 we would see that a significant magnitude of reverse flow occurs towards the end of the transient run, thus it would be wise to use four quadrant data for this case. This is discussed further in Step 6-C.

Warnings are also given that the static pressure is lower than the vapor pressure at several pipe stations. This warning is given because cavitation modeling was disabled in the Transient Control window in order to decrease the run time for this model. When cavitation modeling is disabled the pressure is allowed to drop below an absolute pressure of zero. Though this is not realistic, we will leave cavitation modeling disabled to simplify the model for the purpose of this example – as well as to better be able to see the impact on low pressures from different pump curve choices. This can be done as the cavitation in this model is

minor and has a low impact on the pressure results. When modeling an actual system the cavitation modeling should be turned on in Transient Control in order to obtain more realistic results.

➢**Go to the Pump Summary tab.** For this case the pump is operating at 1559 m3/hr, which is 112% of the BEP (Figure 16).

General | Warnings | Design Alerts | Event Messages By Jct | Event Messages By Time | Pump Summary | Valve Summary

Jct (Steady)	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Supply Pump	1,560	433.1	2.125	21.68	90.60	100.0	101.6	1,389	112.3	101.1	N/A

Figure 16: Steady State results for the pump calculated using the manufacturer's curve for Case 2

As was done in Step 4-C for Case 1, let's compare the manufacturer's pump curve to the dimensionalized four quadrant curves. It will again be useful to compare the pump curves to both the initial steady state, and the final steady state for the system after the tank transition and before the pump trips. Note that the final steady state for Case 2 will be equivalent to the initial steady state for Case 1, which means the final steady state operating point for Case 2 occurs at a flow rate of 854 m3/hr.

A multi-scenario pump vs. system curve graph was generated in AFT Fathom for Case 2, as can be seen in Figure 17, with the operating points from Figure 17 shown in Table 3. The power curves for each of the scenarios can be seen in Figure 18.

In this case the pump operates much closer to BEP, resulting in fewer differences between the 4QSSOP curves and the 4QBEP curves. All of the four quadrant pump head curves intersect the initial steady state system curve close to the manufacturer's curve, though the 4QBEP curves do show minor differences in the steady state operating point. At the final system curve there is not a clear trend when comparing the 4QSSOP and 4QBEP curves. Instead the curves are grouped based on the specific speed, rather than dimensional operating point. Both 41.8 curves appear to be closer to the final steady state operating point than the other curves.

Another point of interest is that the 37.4 4QSSOP curve falls below the final system curve at low flowrates. This means that the 37.4 4QSSOP curve predicts the pump provides insufficient head to overcome the system resistance at this time, even though the manufacturer's curve shows otherwise.

The comparison of the power curves again shows similar trends. In this case it is difficult to determine qualitatively which four quadrant data set best matches the manufacturer curve, though the 64 curves clearly show the most deviations.

If looking for the most reliable four quadrant curve in this case, one of the 41.8 data sets should be chosen due to the closer match that the 41.8 curves show at the final steady state for the head and power curves. If the system is sensitive to the initial steady state then the 41.8 4QSSOP curve should be chosen.

Table 3: Pump head, flow, and discharge velocity at the initial and final steady state operating points as can be seen in Figure 17

	Initial Steady-State			Final Steady-State		
Pump Curve	**Flow (m3/hr)**	**Head (meters)**	**Velocity (meters/sec)**	**Flow (m3/hr)**	**Head (meters)**	**Velocity (meters/sec)**
Manufacturer's Curve	1,560	21.68	2.307	854.4	28.27	1.264
37.4 4QBEP	1,610	22.37	2.382	612.3	26.22	0.906
37.4 4QSSOP	1,560	21.68	2.307	422.8	25.08	0.626
41.8 4QBEP	1,490	20.76	2.204	882.6	28.54	1.306
41.8 4QSSOP	1,560	21.68	2.307	852.8	28.25	1.262
64 4QBEP	1,518	21.12	2.245	992.7	29.73	1.469
64 4QSSOP	1,560	21.68	2.307	968.3	29.45	1.433

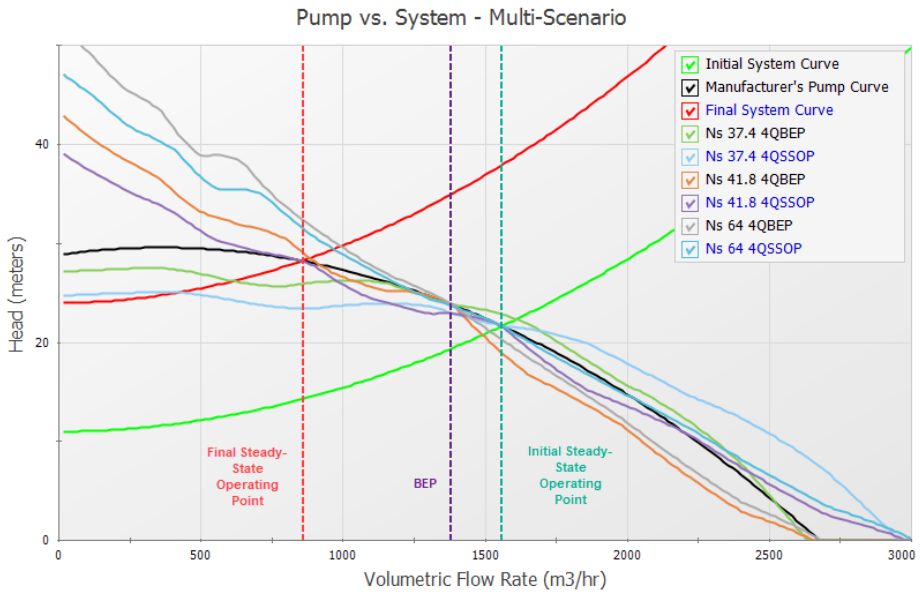


Figure 17: Pump vs. System Curve plot showing the manufacturer provided pump curve compared to pump curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at both the initial and final steady state conditions in Case 2

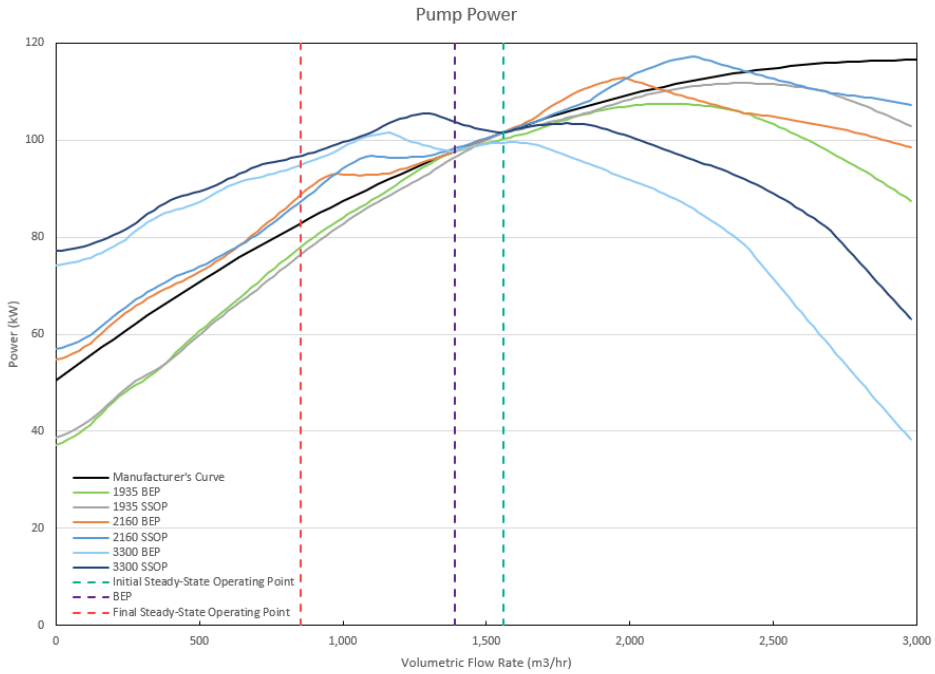


Figure 18: Pump Power Curve plot showing the manufacturer provided power curve compared to power curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at both the initial and final steady state conditions in Case 2

C. Examine Transient Results for Case 2

➢**Run each of the four quadrant scenarios** using the batch run feature from the file menu, then proceed to the Graph Results window.

From the pump speed and velocity graphs in Figure 19 and Figure 20 it can be seen that the pump will operate in the normal pumping zone of operation with both forwards flow and positive speed up until about 10 seconds into the transient. Depending on the pump curve being used, the pump experiences some reverse flow between 10 seconds and 27 seconds, though most (but not all) of the pump curve results return to forward flow before reversing again. From the pump speed curve we can see that all of the four quadrant curves predict that the pump will begin to operate at negative speeds (reverse rotation) between 24 - 26 seconds. This means that for the time period between 24 - 27 seconds the pump will likely operate with both reverse flow and rotation. There is a high level of uncertainty when using the manufacturer's curve due to the magnitude and duration of reverse flow and reverse rotation, so the manufacturer's curve should not be used as a measure of accuracy once the flow becomes negative (at about 10 seconds), and especially during negative rotation.

Note: When the Standard Pump Curve option is used to model a pump in Impulse, the speed will not be allowed to go below 0%, similar to how the head rise was limited to a minimum of 0 in Case 1 when the SPC was used.

Further examining the velocity at the pump we can see that there is less deviation in the initial velocity for each of the curves. In this case this is due to the pump operating closer to the BEP than Case 1. However, the distribution of velocities is much broader once the tank transition has finished at around 10 seconds, reflecting the broader distribution we observed in the pump head curves. A maximum difference of 1.5 meters/second occurs at 20 seconds into the run between the 64 4QBEP and 37.4 4QSSOP curves. If a different transient than a pump trip was modeled at 20 seconds, for example a near instant valve closure of the ESD valve, then wildly different pressure transients would result.

Figure 21 shows the pressure results at the outlet of the ESD valve for the pump trip case, while Table 4 details the maximum and minimum pressures shown in the graph. The range of peak pressures predicted using the four quadrant curves at 27 seconds has been reduced from about 4.2 bar to 4 bar likely as a result of the four quadrant curves being more similar when the pump operates closer to BEP. The highest peak pressure of 27.1 bar is predicted by the 37.4 4QSSOP curve, and the lowest peak pressure of 23.1 bar is predicted by the 64 4QBEP curve.

Zooming in on the minimum pressures that occur at about 28 seconds and 30 seconds into the run (shown in Figure 22) shows that the 41.8 pump curves and the 64.0 pump curves predict a minimum pressure above the vapor pressure (0.018 bar), while the manufacturer's curve and 37.4 pump curves predict that the pressure will drop to vapor pressure (The Figure 22 results actually shows these pressures as negative absolute pressures because transient cavitation is being neglected here for exploratory purposes. If this were a real analysis, other runs with transient cavitation models enabled would be highly recommended.) This range of pressure results highlights another reason why the choice of four quadrant curve to be used is important in waterhammer analysis, as this result will impact the decision of what protective equipment may be required for the system.

Be aware that the ESD valve pressures do not represent the overall maximum/minimum pressures for the system in this case. Many of the Case 2 scenarios actually reach vapor pressure at the outlet of Valve B, even the scenarios which did not drop to vapor pressure at the ESD valve. In an actual analysis the engineer would need to assess the impact of cavitation on the model and consider mitigation such as alternate valve closure times/profiles or installing protective equipment, though this will not be discussed further in this example.

To be conservative the engineer could simply choose the extreme case, which would be the 37.4 4QSSOP scenario. For higher accuracy results further comparison of the four quadrant data sets to the manufacturer’s curve could be done as was described in Step 6-B. From step 6-B the two 41.8 curves were determined to be the best fit. The 41.8 4QSSOP pump curve may be the better option since it gave more extreme results than the 41.8 4QBEP pump curve in this case.

Table 4: Maximum and minimum static pressures at ESD valve and time at which the pressures occur for Case 2 as shown in Figure 21 and Figure 22 below. Note that minimum pressures below absolute zero are not physically possible but are shown for comparison purposes

	Max. Static Pressure	Max. Static Pressure Time	Min. Static Pressure	Min Static Pressure Time
Pump Curve	**bar**	**seconds**	**bar**	**seconds**
Manufacturer's Curve	26.19	27.02	-1.512	27.88
37.4 4QBEP	25.56	27.02	-0.891	27.88
37.4 4QSSOP	27.15	27.01	-2.446	27.88
41.8 4QBEP	23.37	27.03	-0.924	10.00
41.8 4QSSOP	24.19	27.02	-0.919	10.00
64 4QBEP	23.37	27.03	-1.215	10.00
64 4QSSOP	23.80	27.03	-1.150	10.00

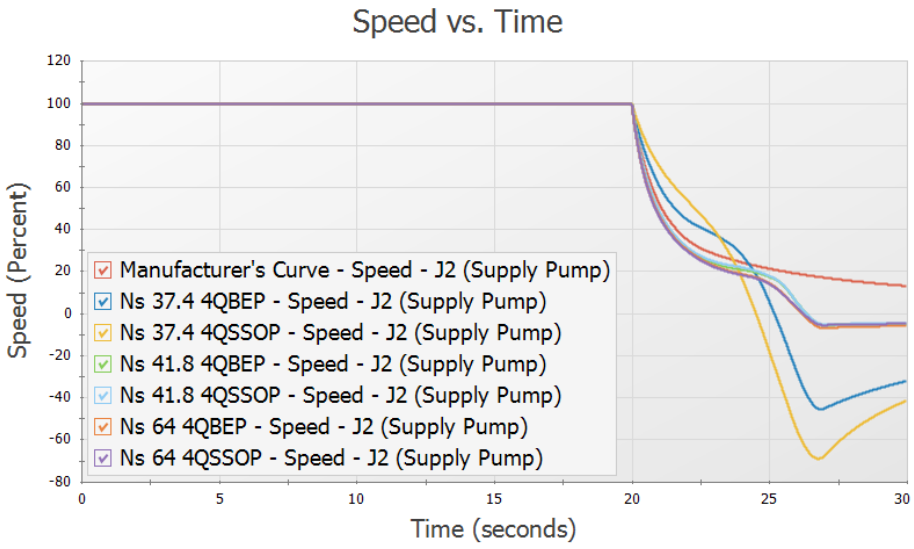


Figure 19: Pump speed vs time for Case 2 using the manufacturer's curve and each of the four quadrant curve scenarios

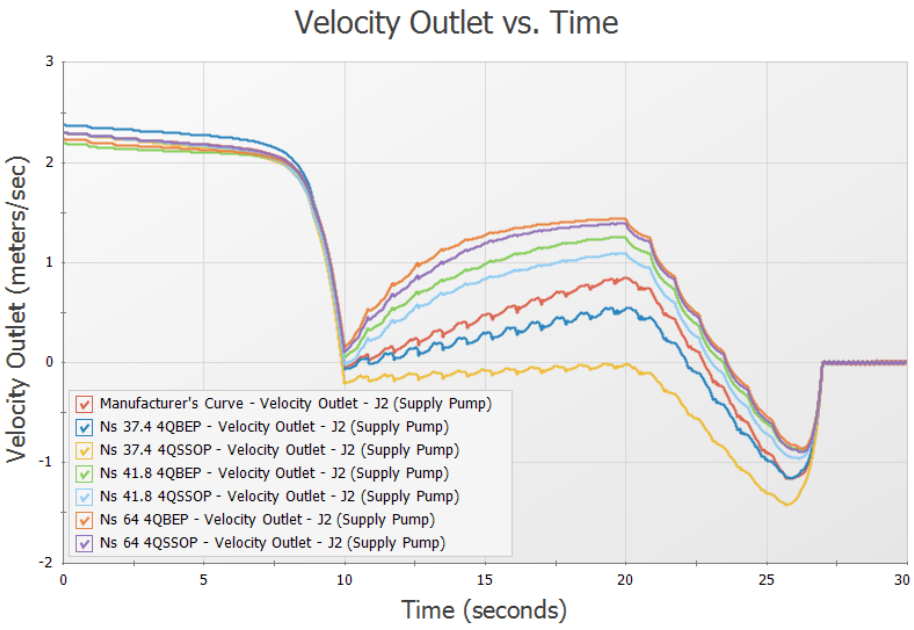


Figure 20: Velocity at pump outlet for Case 2 using the manufacturer's curve and each of the four quadrant curve scenarios

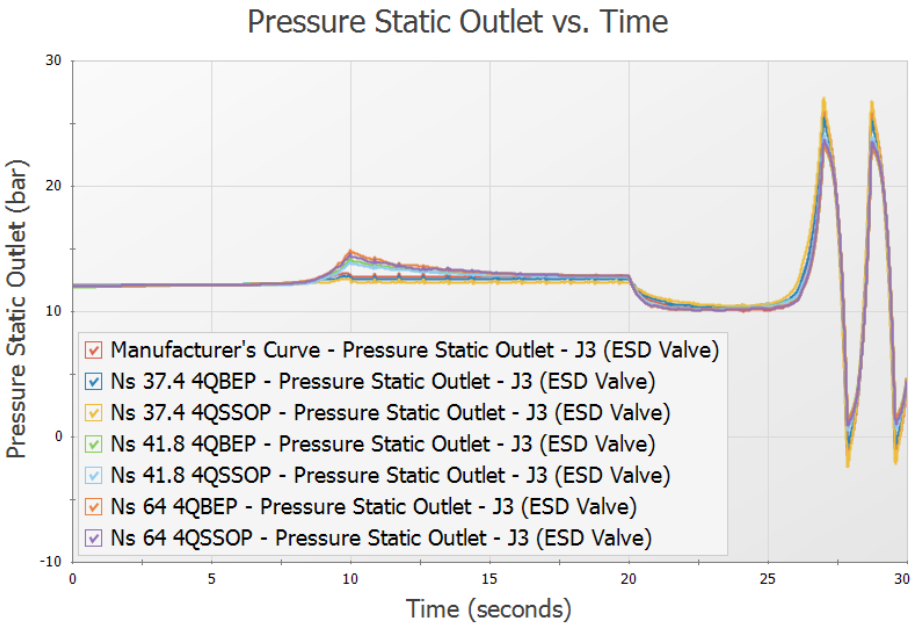


Figure 21: Static pressure at ESD Valve outlet for Case 2 using the manufacturer's curve and each of the four quadrant curve scenarios

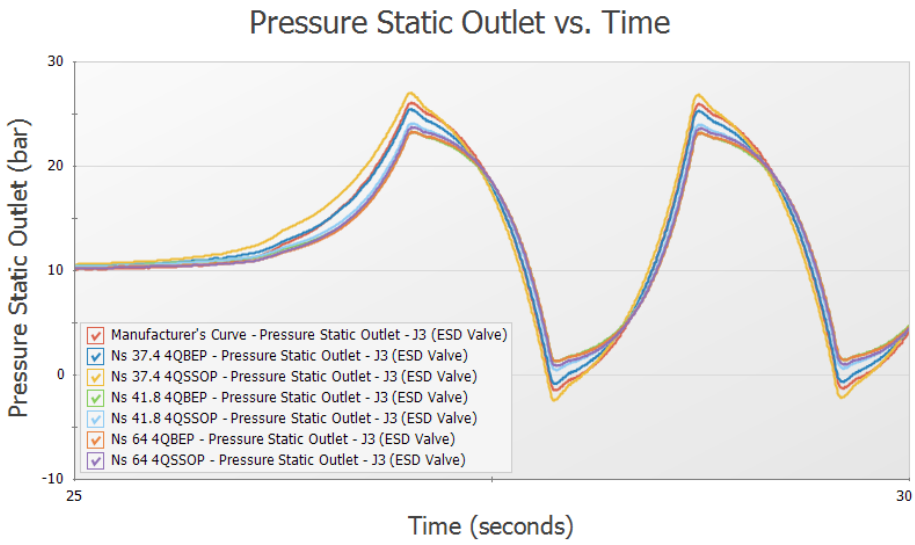


Figure 22: Static pressure at ESD Valve outlet for Case 2 using the manufacturer's curve and each of the four quadrant curve scenarios from 25 to 30 seconds

Step 7. Case 3 - Valve A Fails Open

A. Create Case Scenarios for Case 3

For Case 3, consider the scenario where flow is being transitioned to Tank B, like in Case 1. However, Valve A malfunctions and is stuck fully open so that Valve A and Valve B are fully open. In this case the elevation differences between Tank A and Tank B would cause Tank A to begin draining. To prevent Tank A from fully draining both the ESD valve and Valve B will be closed, and the pump will be intentionally tripped at the same time to bring the system to a safe stop so the Valve A malfunction can be investigated.

Make the following changes in the model:

1. In the Scenario Manager right-click on the Case 2 - Tank B to Tank A scenario and choose Clone With Children. Rename the scenario Case 3 - Valve A Fails.
2. Open the Properties window for Pump J2 and go to the Transient Tab. Under Initiation of Transient change the Time Absolute to 0 seconds so that the pump trip will begin at the beginning of the run.
3. Repeat step 2 for the ESD Valve, junction J3.
4. Open the Valve Properties window for Valve A, junction J5. This valve will need to be set to be open, and remain open throughout the transient.
5. On the Optional Tab, set the Special Condition to None.
6. On the Transient tab set the Transient Special Condition to Ignore Transient Data. This will allow us to keep the transient data entered, but prevent the transient from being used for this scenario.

B. Compare four quadrant curves to manufacturer's curve

Now that the scenario is fully defined, let's examine the results for the model using the manufacturer's curve.

➢**Run the Case 3 - Valve A Fails scenario** from the shortcut on the toolbar, or from the Analysis menu, and go to the Output window.

Once more we will see a warning indicating that the pump did experience reverse flow, which suggests a four quadrant curve should be used. Warnings will also be given for the transient static pressure, as were discussed for Case 2. Lastly, cautions will be given for reverse flow through junctions 5 and 7. These cautions occur since the loss factors through these junctions were entered assuming the losses were for forwards flow through the junctions, and may be inaccurate when reverse flow occurs. We will assume the difference is negligible in this case, though this should be verified for actual analysis.

In the Pump Summary at the top of the window, we can see that the pump operates at 101.2% of BEP.

In the Pipes section of the Output click the Pipes tab to view the steady state results for the pipes. Here we can see that pipes P4 through P6 experience reverse flow at the beginning of the run when both Valve A and Valve B are open, as is expected.

For this scenario the final steady state after the transient is when the pump has tripped and both Valve A and Valve B are closed. This is not meaningful for comparison as all flows will be zero, so the four quadrant pump head curves were simply compared to the initial steady state for the system, as is shown in Figure 23. The power curves can be seen in Figure 24.

As expected the 4QBEP and 4QSSOP curves for each value of specific speed are identical when operating at BEP, so we essentially only need to compare the manufacturer’s curve to 3 four quadrant curves. Since the BEP stays the same in all scenarios, the BEP curves are the same as in the other cases. Thus our conclusions are similar to the conclusions from Case 2. It is clear that the head and power curves for the 37.4 and 41.8 data sets resemble the manufacturer’s curves better than the 64.0 head and power curve. Hence more weight should be given to the results from the 37.4 and 41.8 curves than the 64.0 curve. Between the 37.4 and 41.8 curve, the 37.4 curve seems closer to the manufacturer curve than the 41.8 curve in head, but the 41.8 curve is a better match for the power curve. Since the 37.4 curve is Preferred while the 41.8 curve is Average, the 37.4 curve may be a better choice, even though the qualitative comparison showed a similar match for the curves.

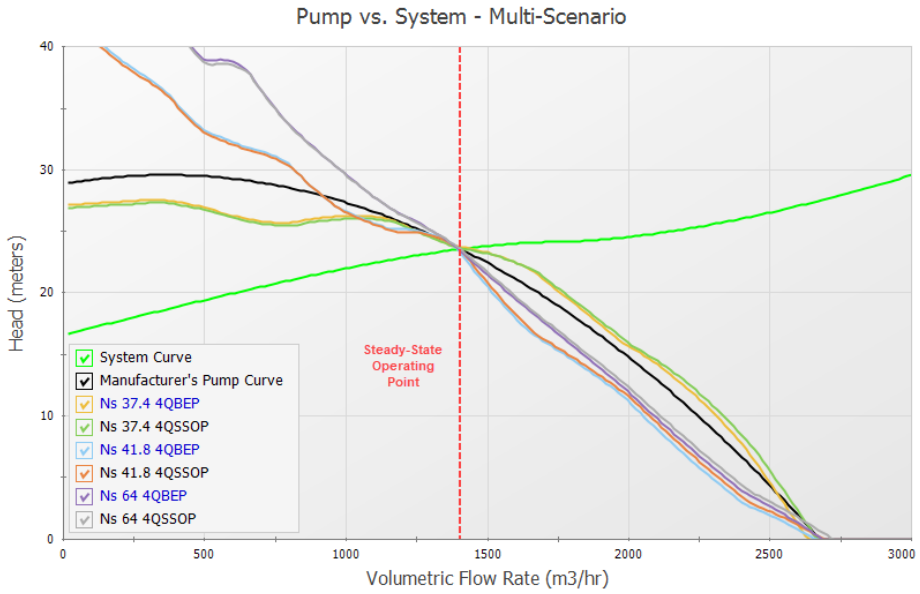


Figure 23: Pump vs. System Curve plot showing the manufacturer provided pump curve compared to pump curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at the initial steady state conditions in Case 3

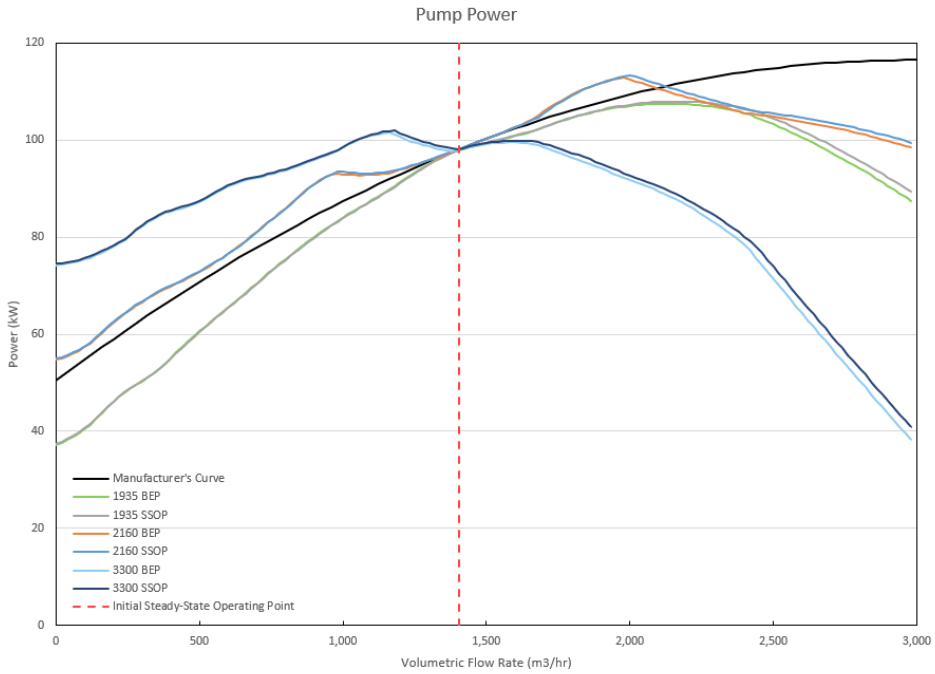


Figure 24: Pump Power Curve plot showing the manufacturer provided power curve compared to power curves from the Ns = 37.4, Ns = 41.8, and Ns = 64 four quadrant data sets at the initial steady state conditions in Case 3

C. Examine transient results for Case 3

From Figure 25 and Figure 26 below it is clear that the four quadrant results diverge from the manufacturer's curve due to backwards flow beginning at the pump at about 3 seconds, and reverse rotation beginning at around 6 - 7 seconds, depending on the four quadrant curve. In Figure 27 we can see a comparison of the pressure results at the outlet of the ESD valve.

While the pump speed results show a significant difference between the scenarios, the velocity results at the pump and the pressure results at the outlet of the ESD valve show relatively minor differences between the four quadrant scenarios. The 37.5 pump curve produces the highest magnitude of reverse velocity out of the four quadrant curves with a minimum velocity of -1.0 meters/second (i.e. a maximum reverse velocity of positive 1.0 meters/second). This larger reverse velocity translates to a slightly larger peak pressure.

When the manufacturer’s curve results are compared to the four quadrant curves, we can see that the difference in reverse velocity is between 0.15 to 0.2 meters/sec, depending on which four quadrant results are being compared to the manufacturer’s curve results. This difference of velocity is reflected in a 1.2 bar difference from the maximum and minimum pressures predicted by the four quadrant curves.

As with Case 2 it should be noted that the ESD valve does not represent the overall maximum/minimum pressures for this model, which instead occur at Valve B. Cavitation is predicted at Valve B, so the engineer would be required to address the low pressures at Valve B based on the requirements for this system.

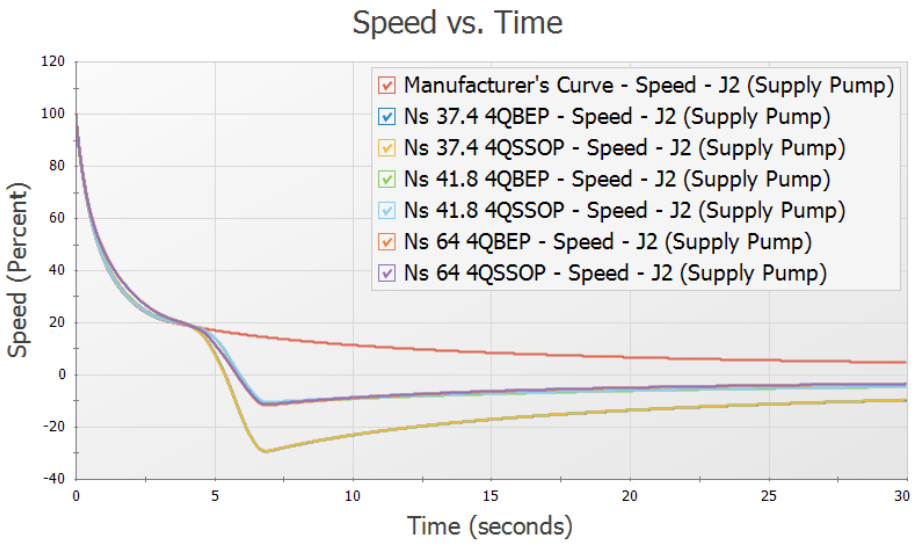


Figure 25: Pump speed vs time for Case 3 using the manufacturer's curve and each of the four quadrant curve scenarios

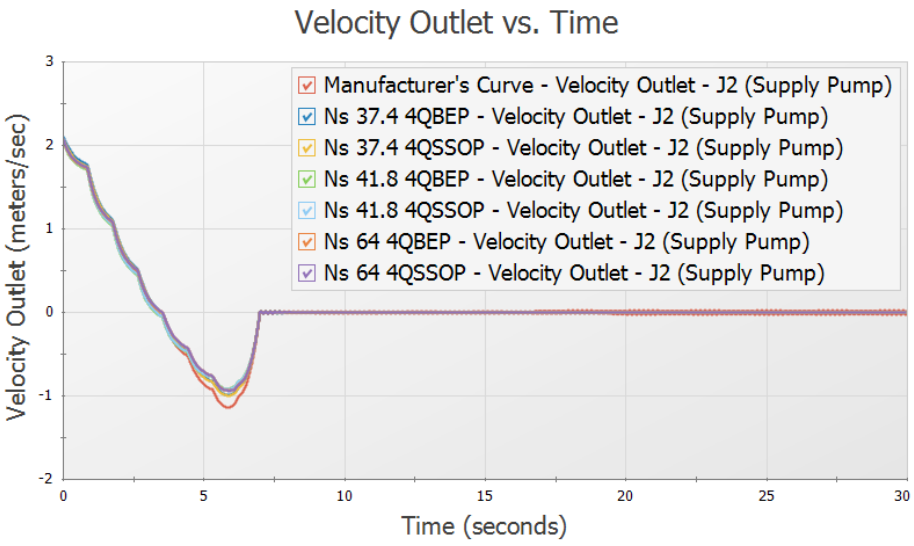


Figure 26: Velocity at pump outlet for Case 3 using the manufacturer's curve and each of the four quadrant curve scenarios

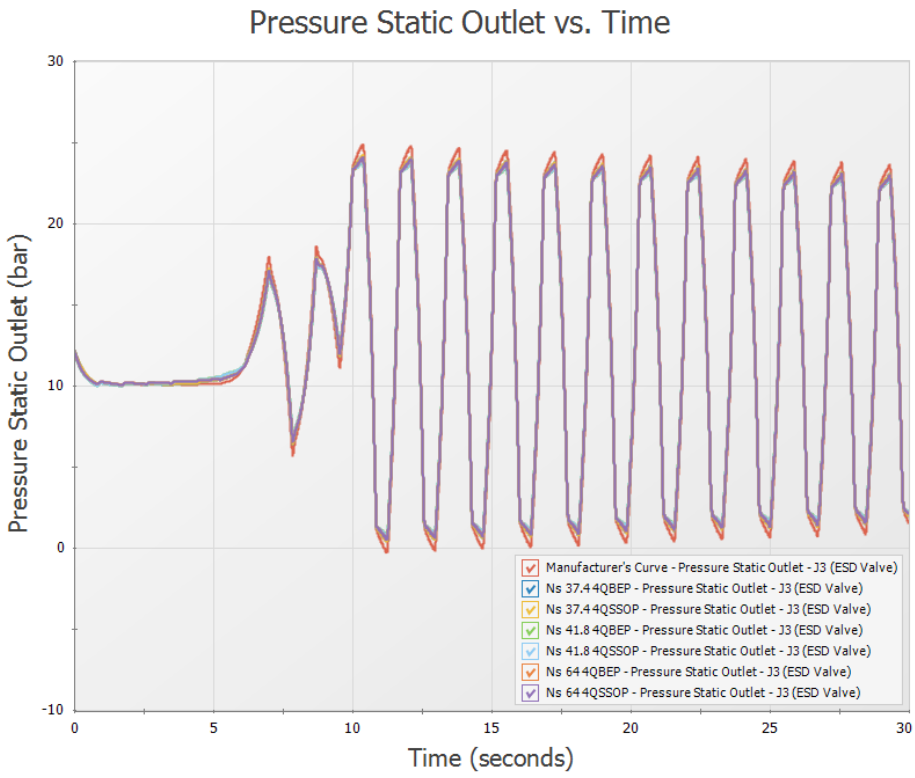


Figure 27: Static pressure at ESD Valve outlet for Case 3 using the manufacturer's curve (shown in black) and each of the four quadrant curve scenarios, which all overlap at the green curve

Conclusion

In this example three different pump trip scenarios were analyzed for a water transfer system in which there was potential for reverse flow and reverse rotation to occur at the pump. Four quadrant data was considered for use to accurately account for this potential reverse flow/rotation.

In situations such as with Case 1 the reverse flow was negligible, but the pump experienced negative head, requiring the engineer to consider four quadrant data. The manufacturer's curve was considered most accurate up until 21 seconds into the run when negative head was calculated at the pump.

In Case 2 and Case 3 where the reverse flow and reverse rotation was significant in duration and magnitude, the four quadrant curves allowed for more accurate results to be found. By comparing the pump head and power curves to the curve data from the manufacturer four quadrant data sets could be selected for the analysis based on the initial steady state, and final steady state where applicable.

From the results above it is clear that the choice of four quadrant data set and dimensional reference point can have a large impact on transient results. Note that while a pump operating closer to BEP will typically be less impacted by the choice of dimensional reference point, this will not always be the case. Performing a sensitivity analysis such as what was described in this example will allow the engineer to determine the impact that using different four quadrant data sets and dimensional reference points will have on the results of the transient simulation.

Pump with Variable Speed Flow Control

Summary

A pump with a variable speed drive for flow control supplies water to two heat exchangers. The flow to one of the exchangers is turned off, and the controlled pump flow rate is reduced by half over five seconds.

Topics Covered

- Using a variable speed controller on a pump
- Modeling a controlled flow rate transient for a pump

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Pump with Variable Speed Flow Control.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard
b. Fluid = Water (liquid)
    i. After selecting, click Add to Model
c. Temperature = 38 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

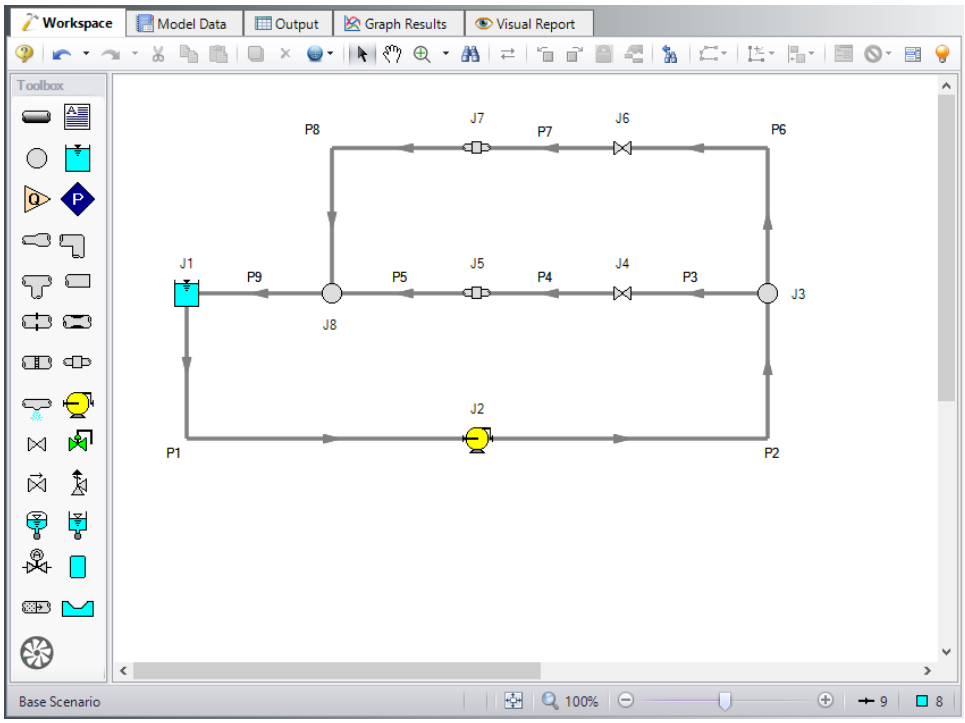


Figure 1: Layout of cooling system with variable speed drive

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Model tab
    a. Name = Use table below
    b. Pipe Material = Steel - ANSI
    c. Size = Use table below

d. Type = STD (schedule 40)
e. Friction Model Data Set = Standard
f. Length = Use table below

Pipe	Name	Size	Length (meters)
1	Pump Suction Pipe	10 inch	9
2	Pump Dis-charge Pipe	10 inch	152
3	Pipe	8 inch	1.5
4	Pipe	8 inch	1.5
5	Pipe	8 inch	3
6	Pipe	8 inch	1.5
7	Pipe	8 inch	1.5
8	Pipe	8 inch	3
9	Pipe	10 inch	152

Junction Properties

1. Reservoir J1
   a. Name = Cooling Pond
   b. Tank Model = Infinite Reservoir
   c. Liquid Surface Elevation = 6 meters
   d. Liquid Surface Pressure = 0 barG (0 kPa(g))
   e. Pipe Depth = 6 meters
2. Pump J2
   a. Name = Main Pump
   b. Inlet Elevation = 0 meters
   c. Pump Model tab
      i. Pump Model = Centrifugal (Rotodynamic)
      ii. Performance Curve Used in Simulation = Standard Pump Curve
      iii. Enter Curve Data

Volumetric	Head
m3/hr	meters
0	18.3
454	16.7

Volumetric	Head
m3/hr	meters
908	12.2
1135	9.1

iv. Curve Fit Order = 2

d. Variable Speed tab
   i. Pump Speed = Controlled Pump (Variable Speed)
   ii. Selection = Volumetric Flow Rate
   iii. Control Setpoint = 454 m3/hr

e. Transient tab
   i. Transient = Setpoint vs. Time
   ii. Transient Special Condition = None
   iii. Initiation of Transient = Time
   iv. Transient Data = Absolute Values

Time (seconds)	Vol. Flow (m3/hr)
0	454
5	227
10	227

3. Branches J3 & J8
   a. Elevation = 0 meters
4. Valve J4
   a. Name = Valve #1
   b. Inlet Elevation = 0 meters
   c. Cv = 500
5. General Components J5 & J7
   a. J5 Name = HEX #1
   b. J7 Name = HEX #2
   c. Inlet Elevation = 0 meters
   d. Loss Model = Resistance Curve
   e. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	0
227	6
454	24

f. Curve Fit Order = 2

6. Valve J6
   a. Name = Valve #2
   b. Inlet Elevation = 0 meters
   c. Loss Model tab
      i. Cv = 500
   d. Transient tab
      i. Transient Special Conditions = None
      ii. Initiation of Transient = Time
      iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	500
0.5	100
1	0
10	0

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 10 seconds for the Stop Time.

Step 6. Run the Model

Click Run Model from the toolbar or from 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.

Step 7. Graph the Results

Figure 2 shows the speed history for the J2 Main Pump while it adjusts from 454 m3/hr to 227 m3/hr.

Figure 3 shows the pressure history for the J2 Main Pump discharge.

Figure 4 shows the head rise across the pump.

Choosing a flow path through the J7 Hex #2, a pressure profile is shown in Figure 5.

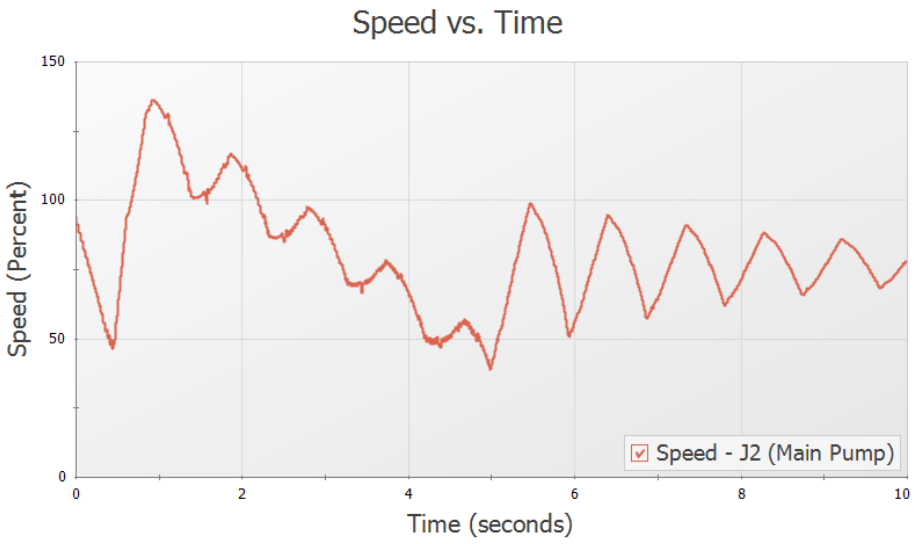


Figure 2: Pump speed transient for J2

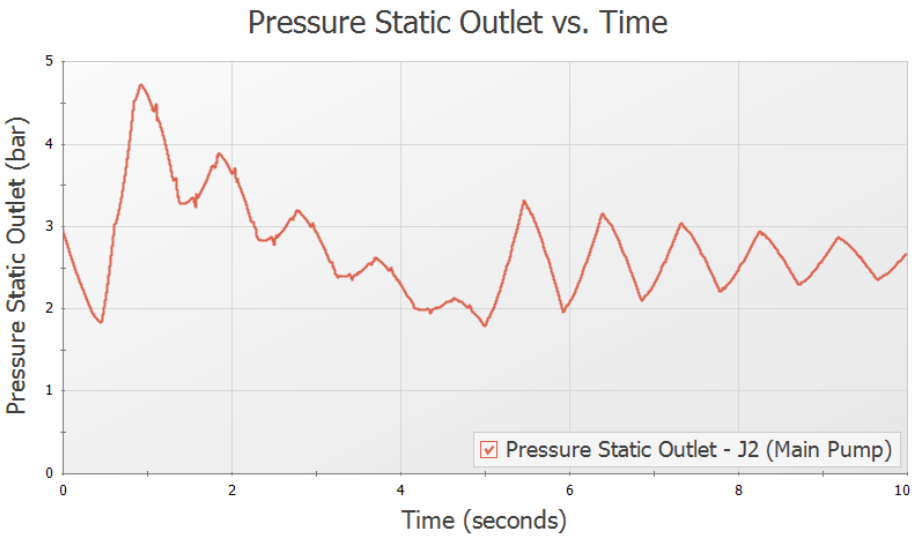


Figure 3: Transient pressures at pump discharge

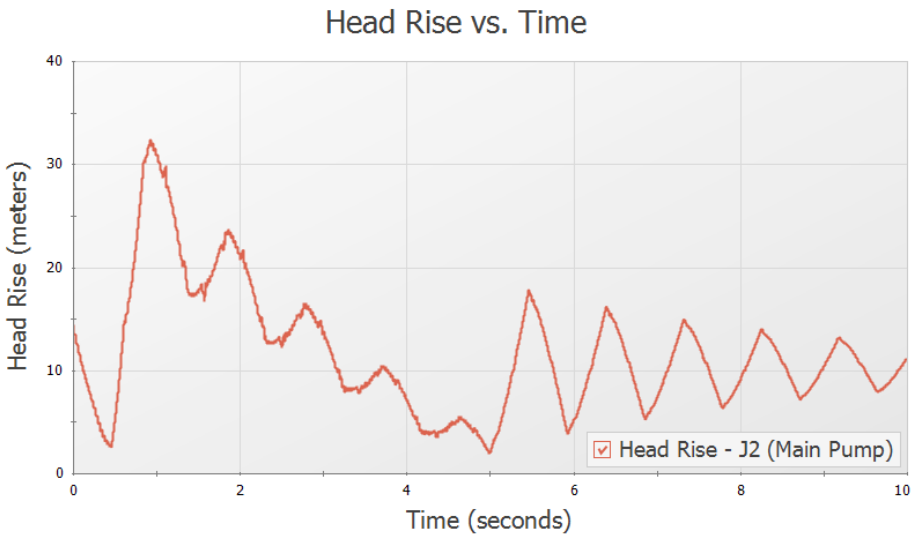


Figure 4: Transient head rise across the pump

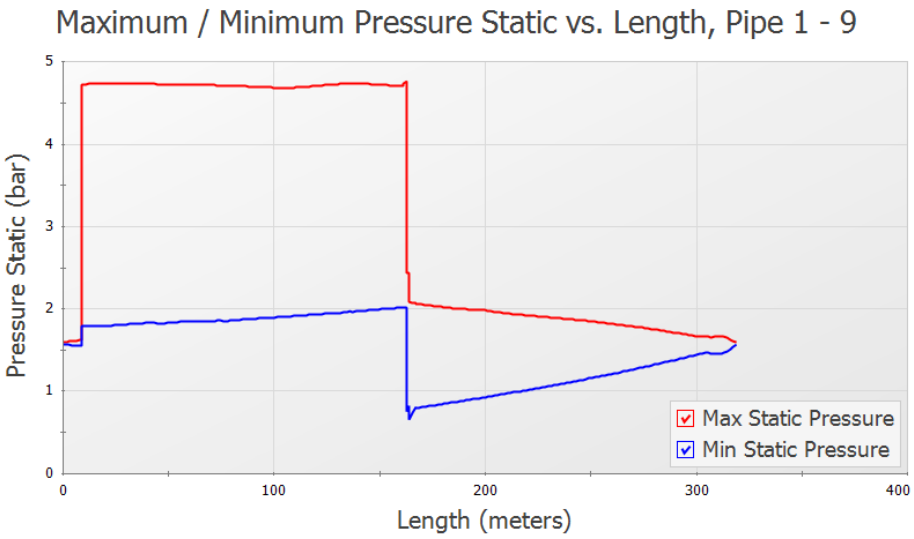


Figure 5: Maximum and minimum pressure profile around loop through J22 Hex #2

Positive Displacement Pump

Summary

This example is designed to illustrate how to model the rapid cycling flow for a positive displacement pump with AFT Impulse. Models of a duplex and triplex positive displacement pump will be analyzed.

Topics Covered

- Using Repeat Transient feature on a pump
- Modeling submerged pumps
- Using the Gas Accumulator junction

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Positive Displacement Pump.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

a. Fluid Library = AFT Standard
b. Fluid = Water (liquid)
    i. After selecting, click Add to Model
c. Temperature = 15.56 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

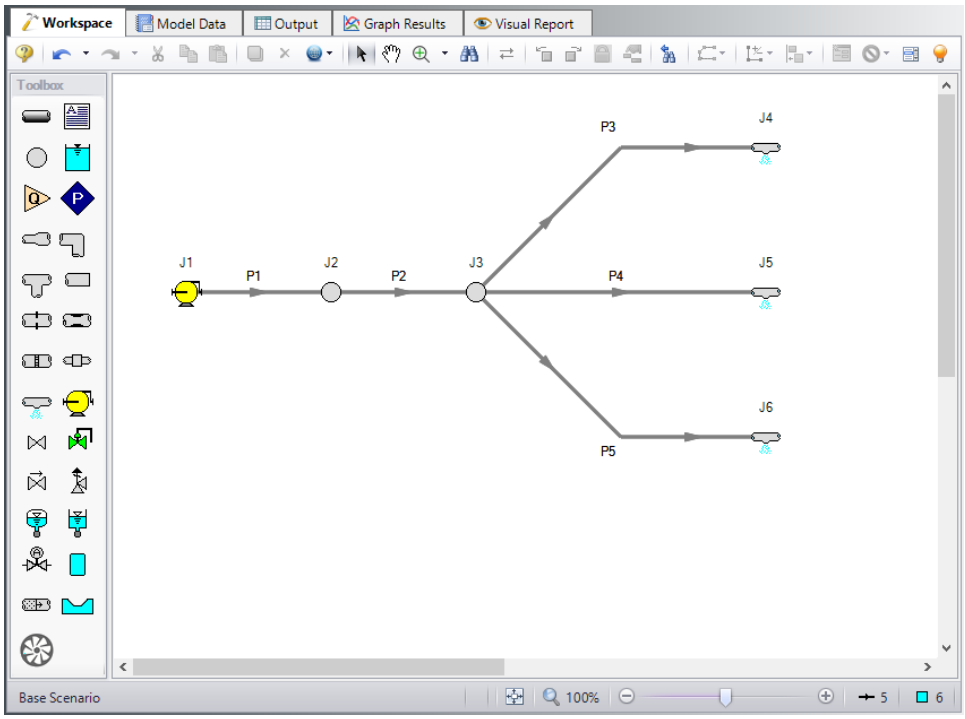


Figure 1: Layout of positive displacement pump model

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipes P1 & P2
    a. Pipe Material = Steel - ANSI
    b. Size = 1-1/2 inch
    c. Type = XS (schedule 80)

   d. Friction Model Data Set = Standard
   e. Length = 1.524 meters
   f. Wavespeed = Calculated Wavespeed
2. Pipes P3 - P5
   a. Pipe Material = (User Specified)
   b. Inner Diameter = 3.81 cm
   c. Wall Thickness = 0.381 cm
   d. Modulus of Elasticity = 20,684.3 MPa
   e. Poisson Ratio = 0.3
   f. Length = 30.48 meters
   g. Wavespeed = Calculated Wavespeed
   h. Friction Model Data Set = User Specified
   i. Selection = Absolute Roughness
   j. Value = 0.00508 cm

Junction Properties

1. Pump J1
   a. Inlet Elevation = 0 meters
   b. Pump Model = Positive Displacement
   c. Parameter = Volumetric Flow Rate
   d. Flow Rate = Steady
   e. Steady Flow rate = 0.00631 m3/sec
   f. Submerged Pump = Checked
   g. Selection = Head (HGL)
   h. Liquid Surface HGL = 0.9144 meters
2. Branches J2 & J3
   a. Elevation = 0 meters
3. Spray Discharges J4 - J6
   a. Elevation = 0 meters
   b. Loss Model = Cd Spray (Discharge Coefficient)
   c. Geometry = Spray Nozzle
   d. Exit Properties = Pressure
   e. Exit Pressure = 0 barG
   f. Cd (Discharge Coefficient) = 0.6
   g. Discharge Flow Area = 0.32258 cm2

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 5 seconds for the Stop Time.

All groups should now be complete and the model is ready to run. If all groups in Analysis Setup have a green checkmark then Click OK and proceed. Otherwise, enter the missing information.

Step 6. Create Child Scenarios

In this model we want to evaluate four displacement pump cases: Duplex and Triplex, each without and with an accumulator.

A. Create the Duplex Scenarios

Using the Scenario Manager on the Quick Access Panel, create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on the Scenario Manager on the Quick Access Panel and then selecting the Create Child icon. Enter the name *Duplex PDP*. A new scenario will appear below the Base Scenario in the list. Select the Duplex scenario and create another child and call it *With Accumulator*. See Figure 2.

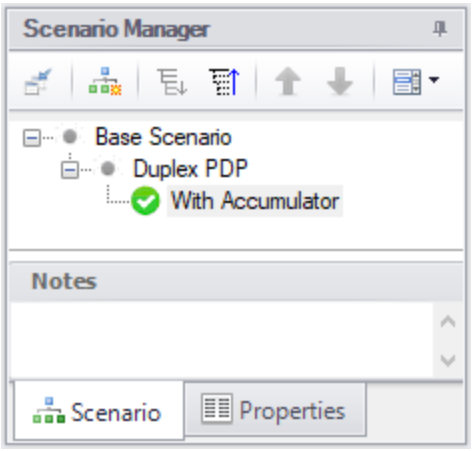


Figure 2: Scenario Manager in the Quick Access Panel with two child scenarios

B. Set up the Duplex PDP Scenario

In the Scenario Manager load the Duplex PDP scenario.

➢**Open the J1 Pump junction** and do the following:

1. Flow Rate = Periodic (simple)
2. Average Flow Rate = 0.00631 m3/sec
3. Number of Heads = 2
4. Operating Speed = 1490 rpm

The periodic flow rate for the duration of one period will be shown. Impulse will continuously repeat this flow rate cycle throughout the transient, using the average flow rate as the initial steady-state flow rate.

The window should appear as in Figure 3. Click OK to exit the window. This scenario is now completed.

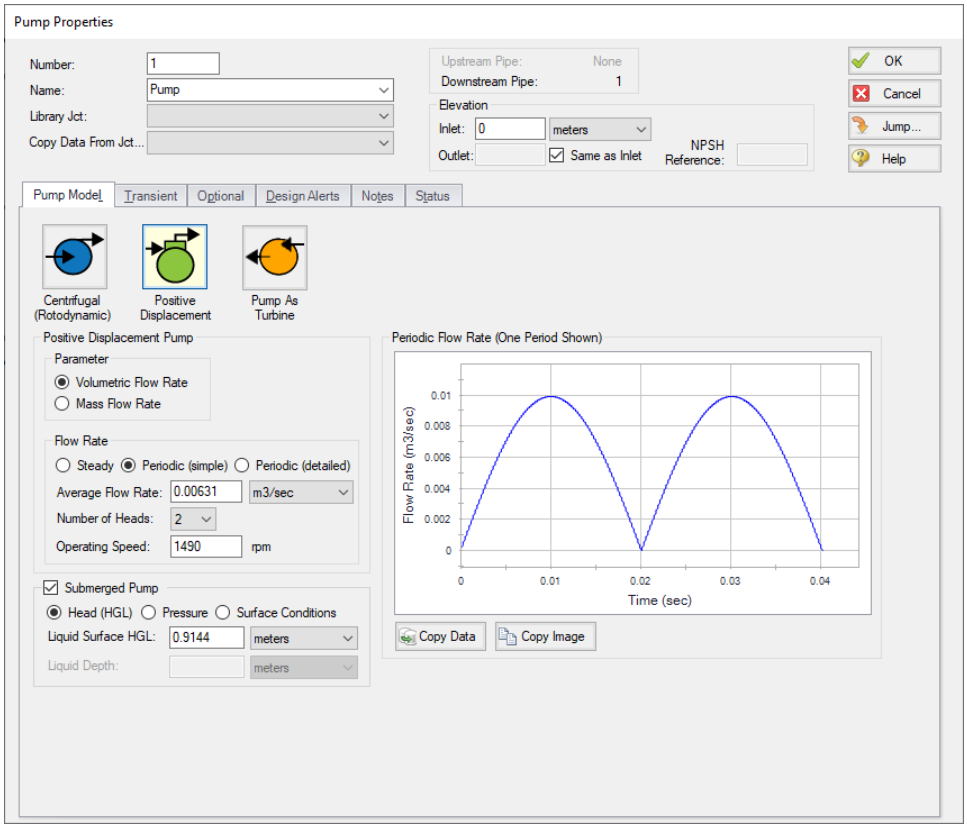


Figure 3: Pump window with duplex flow data entered

C. Commentary on modeling positive displacement pumps

For this example the pump flow vs. time data has been based on an idealized case, where the time varying flow may be represented as half sine waves for each pump cylinder (half is discharge while the other

half of the cycle is suction), one sine wave for each cylinder phased according to the number of cylinders (e.g. 180 degrees for a duplex pump, 120 degrees for a triplex pump). This can be viewed by opening the pump junctions, to see the graph on the Pump Model tab. One will note that the flow for the duplex pump varies from a maximum value to zero flow since each cylinder discharges over 180 degrees of rotation and the two cylinders are 180 degrees out of phase. By comparison, the flow from the individual cylinders of the triplex pump, 120 degrees out of phase, overlap so that minimum flow is greater than zero. Real positive displacement pumps deviate from a sine wave due to the effects of valving and leakage. Accordingly, it is preferable to use manufacturer data when available.

Initial, steady-state flow rate is typically specified based on the nominal flow rate of the pump. Often, one may see a difference between this flow rate and the flow rate as displayed in Graph Results. This difference will typically derive from two sources; 1) differences between a manufacturer's time vs. flow curves and the nominal pump rating and, 2) differences resulting from stepwise integration of the time varying flow. In the case of calculating time varying flow based on an idealized sine wave, the latter will occur because the calculated data points are based on a different stepwise integration basis than performed by AFT Impulse during the analysis. A similar result occurs based on the number of data points taken from a published curve. An example of this difference can be seen in the Duplex PDP with Accumulator scenario by graphing flow in the pipe after the accumulator (P2). Initial flow begins at the specified steady-state value of 22.72 m3/hr, but decreases over time to a steady-state value slightly less than this (~0.2271 m3/hr). It takes a few seconds in this example for the flow to stabilize since the initial accumulator pressure has been calculated based on the initial flow specified for the pump. As the integrated time varying flow is slightly different, system resistance is different than the initial conditions and the accumulator must come to a different pressure to balance this resistance. A change in accumulator pressure is coupled with a net outflow or inflow of liquid in the accumulator, which takes time.

Finally, this example deals only with modeling discharge piping, but what about suction piping? In this example we didn't need to model suction piping to the pump since we selected the optional, 'Submerged Pump' modeling. We could have modeled suction piping supplying the pump and it would be subjected to the same time varying flow variation as the discharge. Modeling the system this way will result in the suction and discharge flows being in phase, whereas in the actual system suction and discharge flows are out of phase. In most situations this difference is not significant since the suction and discharge sides of the system are hydraulically isolated from each other by the pump itself. In other words, one can consider the suction and discharge sides of the piping system as two, distinct systems. In those cases where it is desired to model the out of phase relationship of the suction and discharge flows, the pump can be split in two, so to speak, a suction and discharge half. This example already includes the discharge half so that the suction half could be modeled as an assigned flow junction along with the suction piping. Time varying flow can be specified for an assigned flow junction in the same way as the pump in this example. One would modify the time vs. flow data to reflect the out of phase nature of the suction flow.

D. Set up the With Accumulator Scenario

Now we want to add an accumulator junction to review the effects on pressure and transients. In the Scenario Manager load the With Accumulator scenario

1. Morph the J2 Branch into a Gas Accumulator.
    a. Hold down the CTRL key on the keyboard.
    b. While continuing to hold down the key, drag a gas accumulator junction from the Toolbox and drop it on the J2 Branch.

c. Let go of the CTRL key. The gas accumulator junction should take on the number J7. See Figure 4.

2. Double-click the J7 Gas Accumulator to open the properties window.
3. Polytropic Constant in Transient = 1.2
4. Initial Gas Volume = Known Volume
5. Value = 0.00566 meters3

This scenario is now completed.

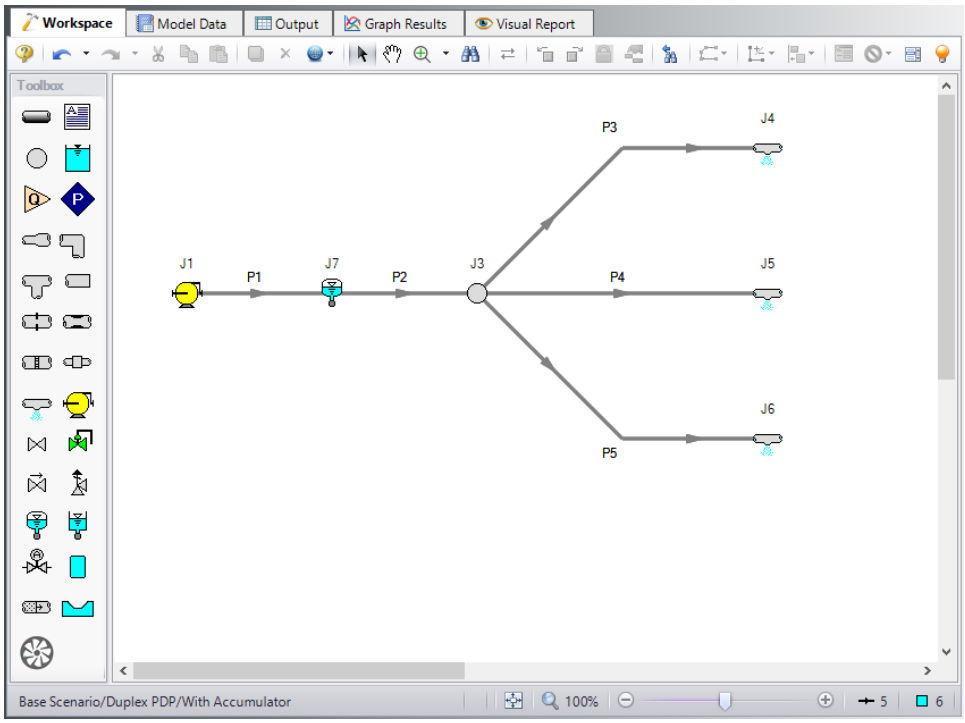


Figure 4: Scenario with duplex pump and gas accumulator junction

E. Set up the Triplex PDP scenario

From this point we can create the Triplex scenario by cloning the Duplex PDP scenario.

1. In the Scenario Manager select the Duplex PDP scenario
2. Right click the scenario name and select the Clone With Children menu item
3. Name the scenario Triplex PDP. This scenario will have a child called With Accumulator. Leave this scenario as is
4. Double-click the Triplex PDP scenario to load it as the current scenario
5. Open the Pump J1 properties window
6. Change the Number of Heads to 3

7. The periodic flow rate for the duration of one period will be shown. Impulse will continuously repeat this flow rate cycle throughout the transient, using the average flow rate as the initial steady state flow rate.
8. Click OK.

This scenario is now completed.

Step 7. Run the Model

Using the Scenario Manager, load each scenario and run it. This can also be accomplished by performing a batch run from the File menu.

Note that once the scenarios are run, a warning will be shown in each of the duplex pump scenarios stating that the Positive Displacement Pump added negative pressure. This is occurring due to the fact that the simple transient used in the positive displacement pump allows the flow in the pump to drop to zero throughout the transient, which requires a negative head rise across the positive displacement pump. Using a more realistic transient would likely resolve this warning.

Step 8. Graph the Results

Graphs of the pressure and flow transients can be created for each scenario. The pump discharge pressures are shown in Figure 5 and Figure 6. The pressures at the J2 Branch & J7 Accumulator are shown in Figure 7 and Figure 8. The flows after the J2 Branch & J7 Accumulator are shown in Figure 9 and Figure 10.

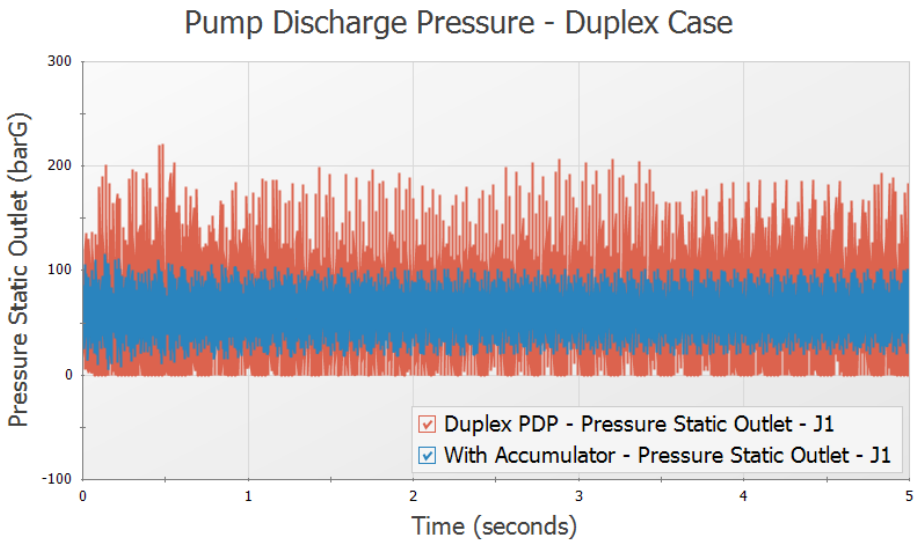


Figure 5: Pump discharge pressure for Duplex case

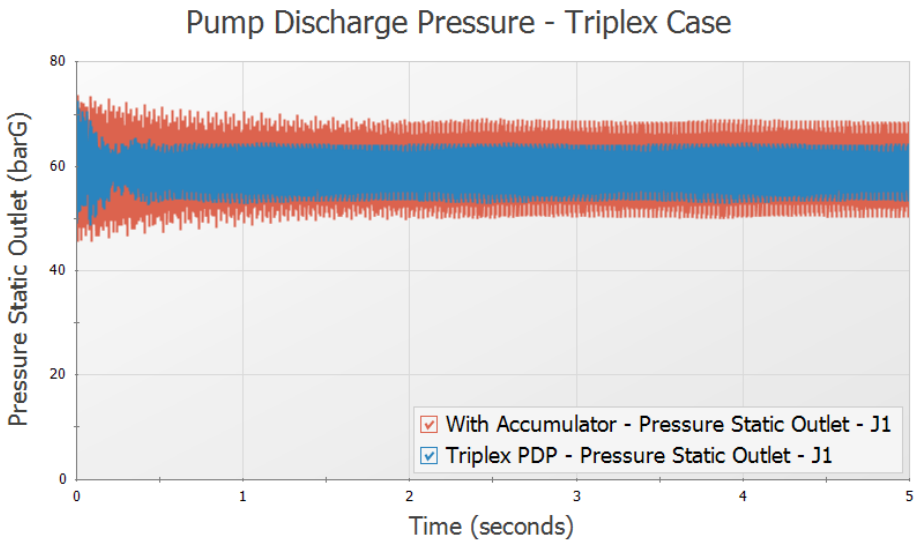


Figure 6: Pump discharge pressure for Triplex case

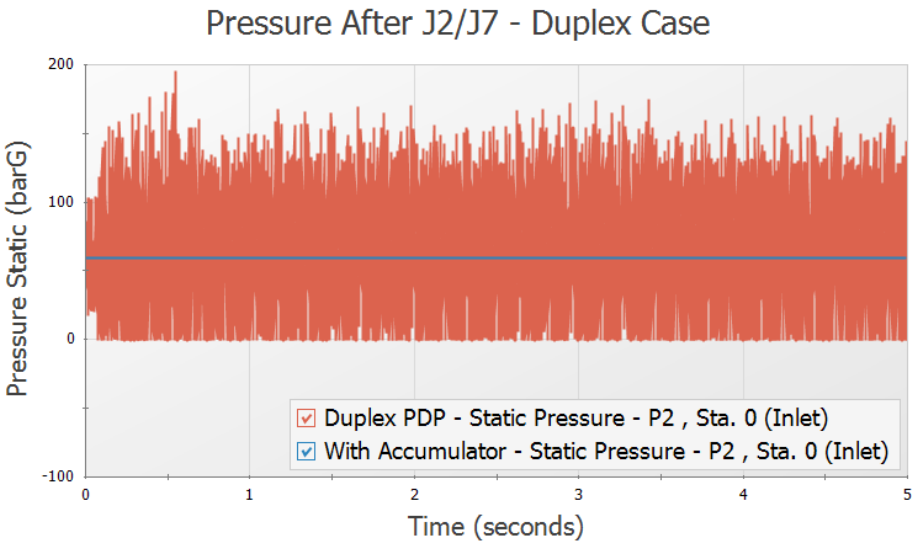


Figure 7: Pressure at J2 Branch/J7 Accumulator for Duplex case

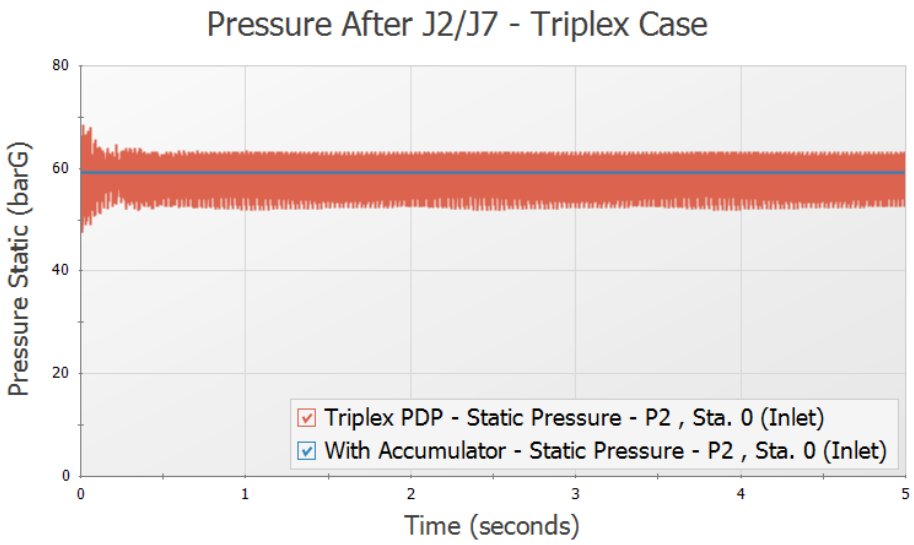


Figure 8: Pressure at J2 Branch for Triplex case

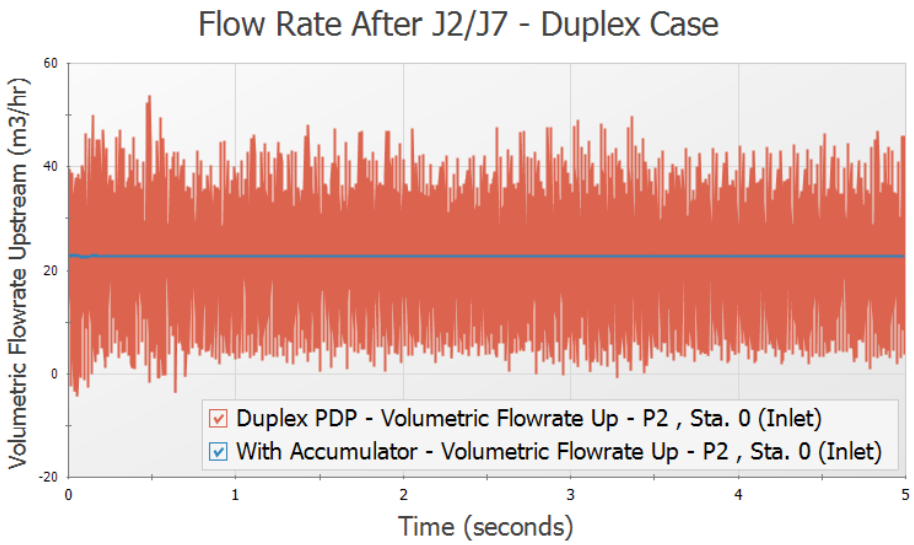


Figure 9: Flow after J2 Branch/J7 Accumulator for Duplex case

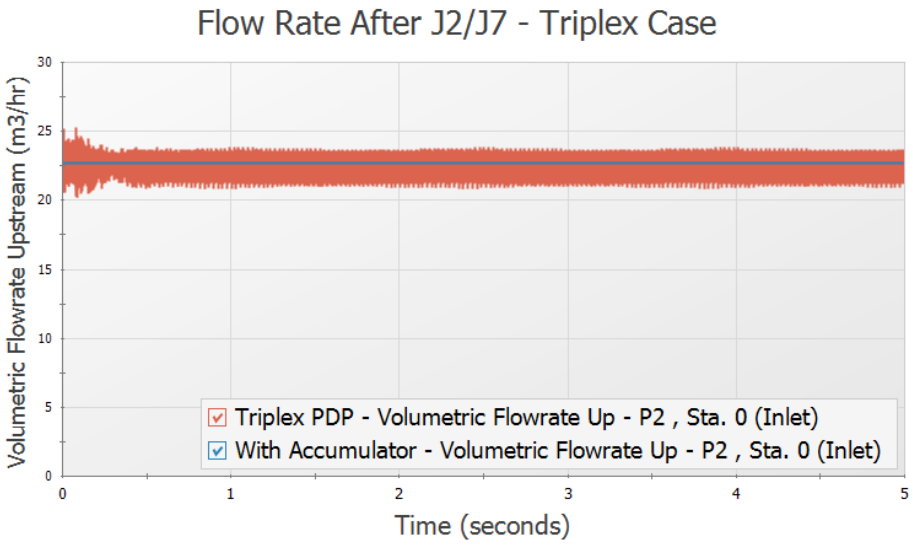


Figure 10: Flow after J2 Branch/J7 Accumulator for Triplex case

Valve Closure with Pipe Forces

Summary

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

Topics Covered

- Defining force sets
- Graphing transient forces
- Evaluating the effect of frictional losses on transient pipe forces
- Using the isometric pipe drawing mode
- Generating a CAESAR II force file

Required Knowledge

This example assumes the user has already worked through the Beginner: Valve Closure example, or has a level of knowledge consistent with that topic. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

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

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
    a. Fluid Library = AFT Standard
    b. Fluid = Water (liquid)
        i. After selecting, click Add to Model
    c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

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 it is more convenient to use the Isometric mode to visually interpret the pipe layout and have a better understanding of the system when interpreting calculated forces.

1. From the Arrange menu, select the option to Show Grid and choose Isometric under Pipe Drawing Mode.
2. Place the junctions as shown below in Figure 1.
3. Double-click the pipe tool and draw the pipes in the model as shown. 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.

Note: You can hold the ALT key while adjusting a pipe by the endpoint to add an additional segment. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

---

4. Align the junctions using the Rotate Icons buttons, or right click on the junctions and select Auto-rotate Icon to automatically align the junctions with the connected pipes. Alternatively, Customize Icon can be selected from the right click menu to manually choose an icon/orientation.
5. The grid can be shown or turned off in the Arrange menu, as shown in Figure 2.

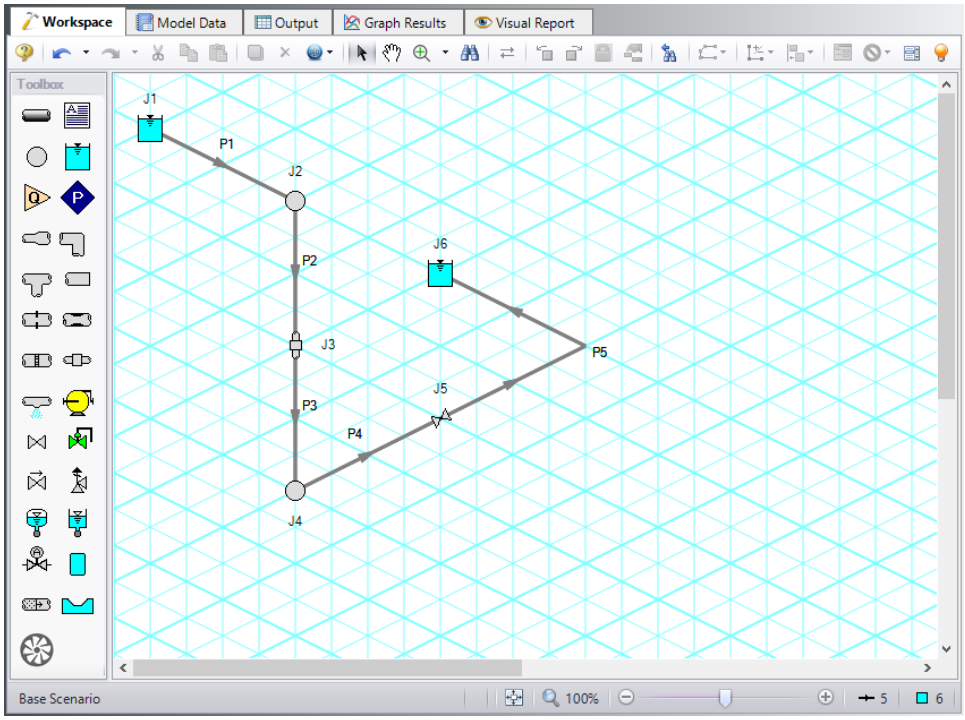


Figure 1: Layout of gravity drain system in Isometric Drawing Mode with grid shown

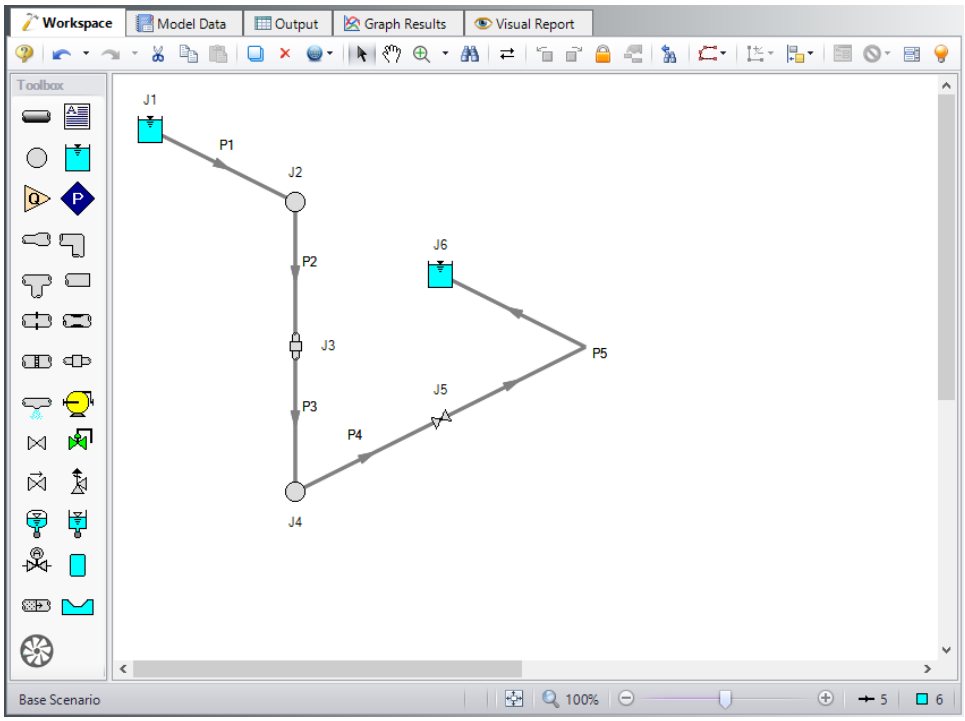


Figure 2: Layout of gravity drain system with Isometric grid turned off

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Size = 20 inch
    c. Type = STD (schedule 20)
    d. Friction Model Data Set = Standard
    e. Length = Use table below
    f. Wavespeed = Calculated Wavespeed

Pipe	Length (meters)
1	150
2	5
3	5
4	125
5	30

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 and junction and 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.

Junction Properties

1. Reservoir J1
    a. Tank Model = Infinite Reservoir
    b. Liquid Surface Elevation = 15 meters
    c. Liquid Surface Pressure = 0 barG (0 kPa(g))
    d. Pipe Elevation = 10 meters
2. Branch J2
    a. Elevation = 10 meter
3. General Component J3
    a. Name = Filter
    b. Inlet Elevation = 5 meters

c. Loss Model = K Factor
d. K = 10

4. Branch J4
   a. Elevation = 0 meters
5. Valve J5
   a. Loss Model tab
      i. Inlet Elevation = 0 meters
      ii. Valve Data Source = User Specified
      iii. Loss Model = Cv
      iv. Loss Source = Fixed Cv
      v. Cv = 13215
   b. Transient tab
      a. Transient Special Condition = None
      b. Initiation of Transient = Time
      c. Transient Data = Relative To Steady-State Values

Time (seconds)	% Cv of Steady State
0	100
2	10
10	10

6. Reservoir J6
   a. Liquid Surface Elevation = 3 meters
   b. Liquid Surface Pressure = 0 barG (0 kPa(g))
   c. Pipe Elevation = 0 meters

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be displayed in the table at the top. Select the row to use one section in the controlling pipe.

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

Step 5. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 10 seconds for the Stop Time.

Step 6. Define the Forces Group

➢**Open the Force Definitions panel** and define a force set by clicking New, entering the name, the start pipe, length to start node, end pipe, and the length to end node. Repeat this process for the second force set as shown in Figure 3.

Note: Force sets can also be defined from the Workspace by right-clicking on the starting pipe(s) to automatically create a difference force set from the start node to the end node of that pipe. The force set can then be edited from the Transient Control window.

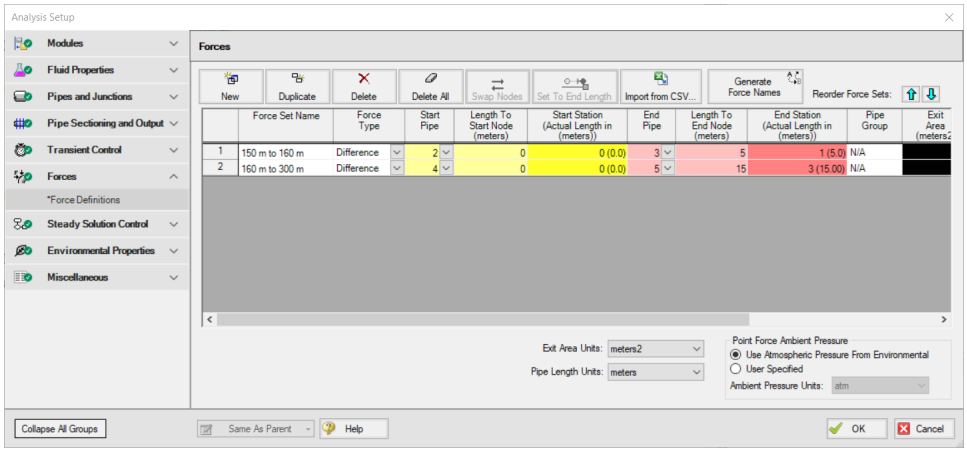


Figure 3: Force sets defined in Transient Control

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 pipe is interrupted. 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 meters to 160 meters force set and then 25 sections and six sections in pipes P4 and P5, respectively for the 160 meters to 300 meters 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 automatically set Pipe Station Output to include only those stations required by the force set nodes up to five. If more than five stations are required, all stations will be specified.

An Impulse model does not contain directional data with regard to the forces, it knows only pipe length and elevation. Since forces are vectors with both magnitude and direction, the user must identify the direction of the calculated forces using data in the pipe arrangement drawing that defines the geometry of the pipe routing. This directional information can then be entered using the Force Unit Vector columns. The force unit vectors will not impact Impulse's calculations, but may be useful if it is desired to export the Impulse force output to a pipe stress software as is discussed in step 8.

➢**Click OK** to save and close Analysis Setup.

Step 7. Run the Model

Click Run Model from the toolbar or from 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.

Step 8. Graph the Results

➢**On the Quick Access Panel, in the Parameters section, select the Forces tab.** The force sets defined on the Force Definitions panel will appear as available forces to graph.

Check the box next to the 150 m to 160 m force set. This corresponds to the elevation change between the branch junctions of 10 meters and 0 meters specified from pipes P2 and P3. Click Generate and the following force versus time graph is displayed (Figure 4).

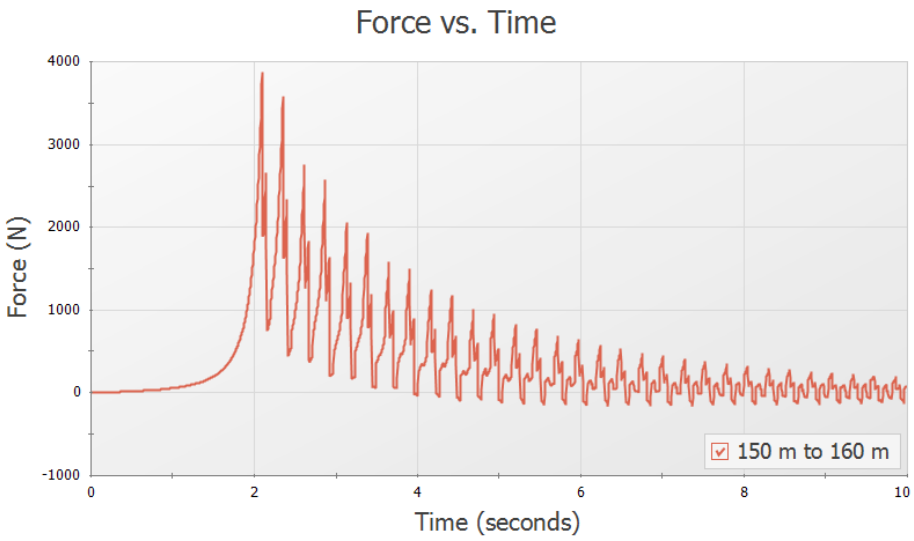


Figure 4: Transient imbalanced forces vs. time for force set 150 m to 160 m

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 4 were created with friction and momentum ignored, the steady state value would be calculated as approximately -11940 N. Due to this effect, Impulse will always include both friction and momentum when calculating and graphing force sets.

Now create the same graph but for the 160 m to 300 m force set. The resulting force vs time is shown in Figure 5.

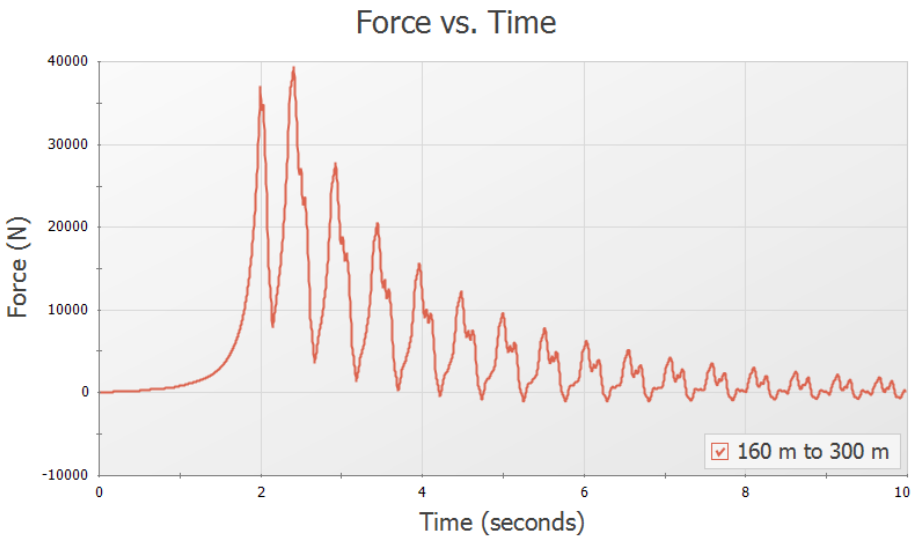


Figure 5: Transient imbalanced forces vs. time for force set 160 m to 300 m

Again we see that with friction included there is no force imbalance under steady state conditions (time 0). If friction were not included in the force calculations as is done here a steady state imbalance of approximately -5100 N would have been observed.

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 fittings weight. A comprehensive analysis of pipe loading must separately include these items.
2. In some cases, ignoring friction and momentum force balance terms 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.

Step 9. Export CAESAR II Force File

➢**Go to the Output window, click on the File menu and select Export Force File and choose CAESAR II Force File** , which will open the export file window shown in Figure 6.

The force sets to include in the force file may be individually selected, and all or only intermediate data points may be included in the force file. For the force sets and options selected the number of data points that will be in the file is displayed. This number should not exceed the maximum number of data points allowed in CAESAR II.

In addition if unit vector information has been entered for the force sets in the Transient Control window, the Export Unit Vector option can be used to include this information in the exported file.

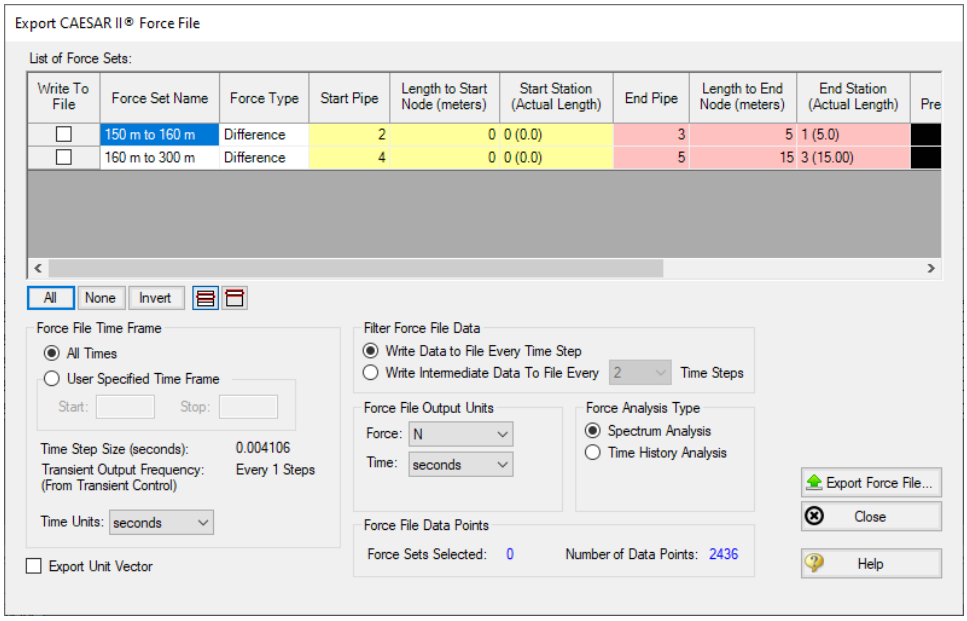


Figure 6: Export CAESAR II Force File window

Final Note

If transient cavitation occurs, the warning shown in Figure 7 will be displayed in the Output window:

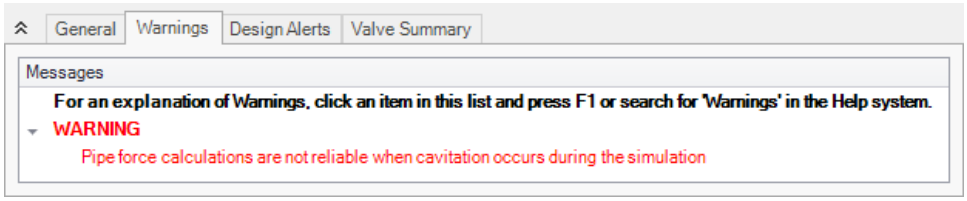


Figure 7: Force calculation cavitation warning

Calculating time varying forces requires a greater level of accuracy in the magnitude and location of pressures, velocity and frictional losses than may be determined with transient cavitation modeling. Accordingly, pipe force calculations will have a usually indeterminable reduction in accuracy when cavitation occurs.

Module Walk-Through Examples

The examples in the AFT Impulse Module Walk-Through are designed to get new users trained to use the AFT Impulse modules correctly in the shortest amount of time.

If this is your first time using AFT Impulse, please start with the Walk-Through Examples to learn how to create and run AFT Impulse models. Start with the Valve Closure - Beginner model, because it provides the most complete, basic description on how to set up and run an AFT Impulse model. After this example, it is recommended that you work through the remaining Walk-Through examples.

Example	Fluid	Pipes	Pumps	Description
Pipe Sizing for Sand Transfer System - SSL	Water Slurry	8	1	Determines if the pipe size and velocity ratio for the system is adequate to prevent sand from settling out of the slurry and plugging the pipeline.
PD Pulsation Study - PFA	User Specified	9	-	Determines the excitation frequencies of the system and creates child scenarios for pump speeds that cause the system to resonate at these frequencies.

Pipe Sizing for Sand Transfer - SSL

Summary

Sand is being transferred from a quarry sand pit and being transferred 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.

Note: This example can only be run if you have a license of the SSL module.

Topics Covered

- Entering solids data
- Reviewing calculated slurry output
- Using Design Alerts

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner: Valve Closure example. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

Model File

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

- ***Metric - Pipe Sizing for Sand Transfer - SSL.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Modules Group

First the SSL add-on modules needs to be activated for use in the model. To do this, open Analysis Setup from the toolbar or Analysis menu. On the Modules panel, check the box next to Activate SSL. After the SSL module is activated, the Settling option should being selected by default.

Step 3. Define the Fluid Properties Group

1. Carrier Fluid panel (Figure 1)
    a. Carrier Fluid = Basic (Water)
    b. Temperature = 21 deg. C

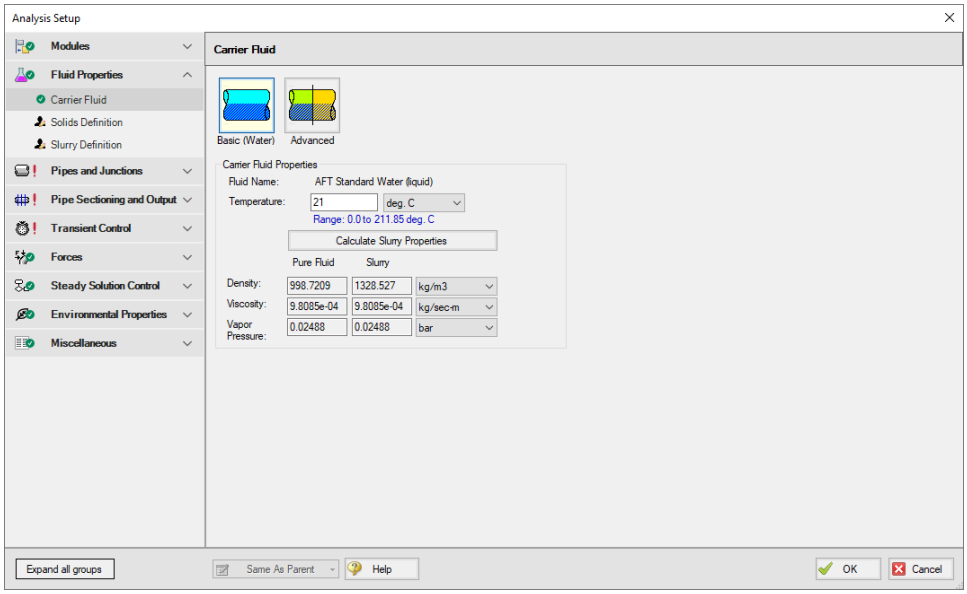


Figure 1: Carrier Fluid panel

2. Solids Definition panel (Figure 2)
    a. Slurry Calculation Method = Detailed
    b. Solids Specifications = User Specified Solids Added
    c. M (Stratification Ratio Exponent) = Calculated
    d. Terminal Velocity Parameter = Vt / Vts (Xi)
    e. Value = 0.55
    f. Density = 2.65 S.G. water
    g. d50 = 0.07 cm
    h. d85 = 0.17 cm
    i. Bulk Modulus = 16000 MPa

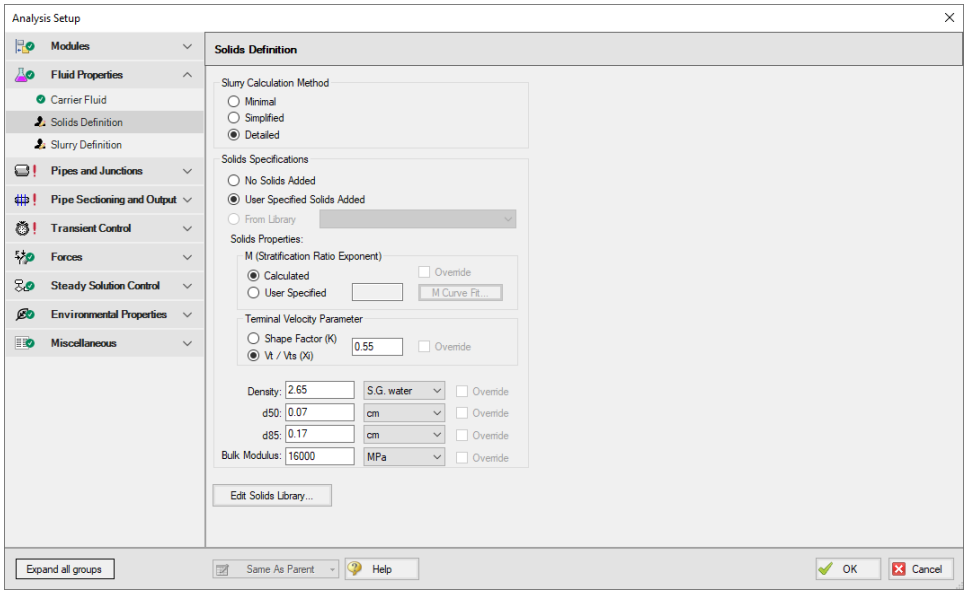


Figure 2: Solids Definition panel

3. Slurry Definition panel (Figure 3)
   a. Concentration Type = Volume Fraction
   b. Amount Solids Added = 20 Percent
   c. Slurry Model = Settling Slurry - Wilson, Addie, Clift

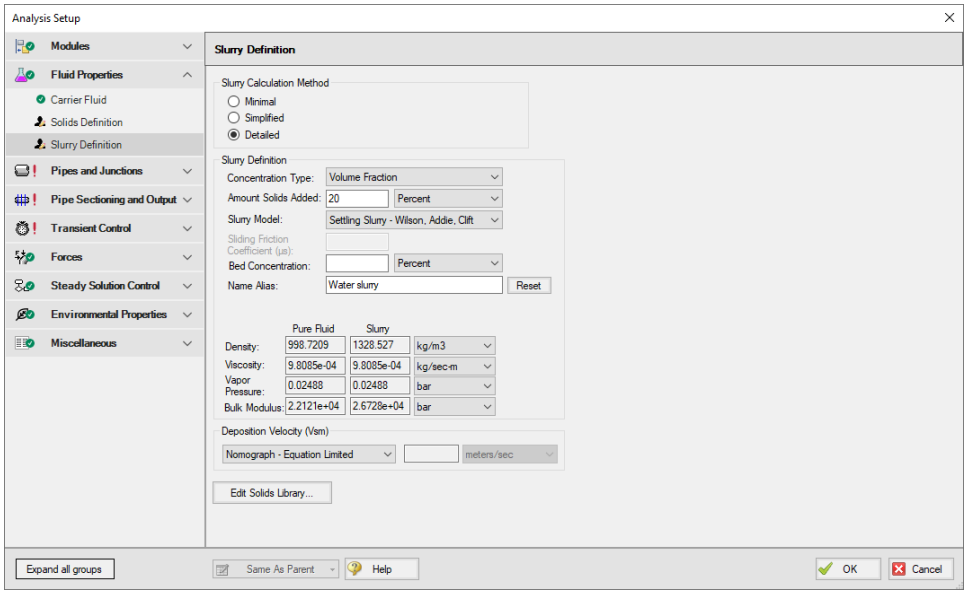


Figure 3: Slurry Definition panel

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

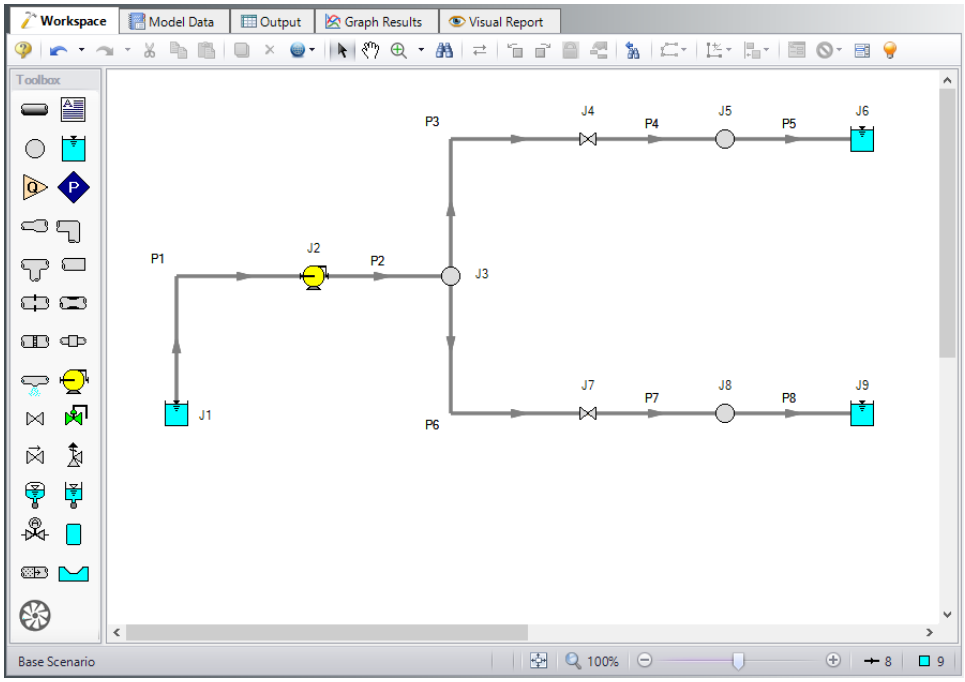


Figure 4: Layout of pipe system for Sand Transfer to Two Locations example

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Model tab
    a. Pipe Material = Steel - ANSI
    b. Size = 10 inch
    c. Type = STD (schedule 40)
    d. Friction Model Data Set = Standard
    e. Length = Use table below
    f. Wavespeed = Calculated Wavespeed

Pipe	Length (meters)
1	3
2	122
3	1.5
4	26
5	122
13	1.5
14	30.5
15	213

Junction Properties

1. Reservoir J1
    a. Name = Sand Pit
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 0 meters
    d. Liquid Surface Pressure = 0 barG (0 kPa(g))
    e. Pipe Depth = 0 meters
2. Pump J2
    a. Name = Main Slurry Pump
    b. Inlet Elevation = 1.2 meters
    c. Pump Model = Centrifugal (Rotodynamic)
    d. Performance Curve Used in Simulation = Standard Pump Curve
    e. Enter Curve Data =

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

    f. Curve Fit Order = 2
3. Branch J3
    a. Elevation = 1.2 meters
4. Valve J4

a. Name = Valve to Deposit #1
b. Inlet Elevation = 1.2 meters
c. Loss Model tab
    i. Valve Data Source = User Specified
    ii. Loss Model = Cv
    iii. Loss Source = Fixed Cv
    iv. Cv = 500
d. Transient tab
    i. Transient Special Condition = None
    ii. Initiation of Transient = Time
    iii. Transient Data = Absolute Values

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

1. Branch J5
    a. Elevation = 6 meters
2. Reservoir J6
    a. Name = Deposit #1
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 6 meters
    d. Liquid Surface pressure = 0 barG (0 kPa(g))
    e. Pipe Depth = 0 meters
9. Valve J7
    a. Name = Valve to Deposit #2
    b. Inlet Elevation = 1.2 meters
    c. Loss Model tab
        i. Loss Model = Cv
        ii. Loss Source = Fixed Cv
        iii. Cv = 400
    d. Transient tab
        i. Transient Special Condition = None
        ii. Initiation of Transient = Time
        iii. Transient Data = Absolute Values

Time (seconds)	Cv
0	400

Time (seconds)	Cv
5	500
10	500

1. Branch J8
    a. Elevation = 9 meters
2. Reservoir J9
    a. Name = Deposit #2
    b. Tank Model = Infinite Reservoir
    c. Liquid Surface Elevation = 9 meters
    d. Liquid Surface pressure = 0 barG (0 kPa(g))
    e. Pipe Depth = 0 meters

Design Alerts

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

1. Fro m the Tools menu, select Design Alerts to open the Design Alert Manager.
2. Click Create New Design Alert.
3. Name = Min Velocity Ratio (Vm/Vsm)
4. Object Type = Pipes
5. Parameters = Velocity Ratio (Vm/Vsm)
6. Selection = Minimum
7. Value = 1.2
8. Click Add to List
9. Click Update/Add Design Alert
10. Select the new design alert in the Existing Design Alerts list, then click All below the Pipes in Design Alert area to apply the Design Alert to all pipes in the model.
11. The Design Alert Manager window should now appear as shown in Figure 5.
12. Click OK to return to the Workspace.

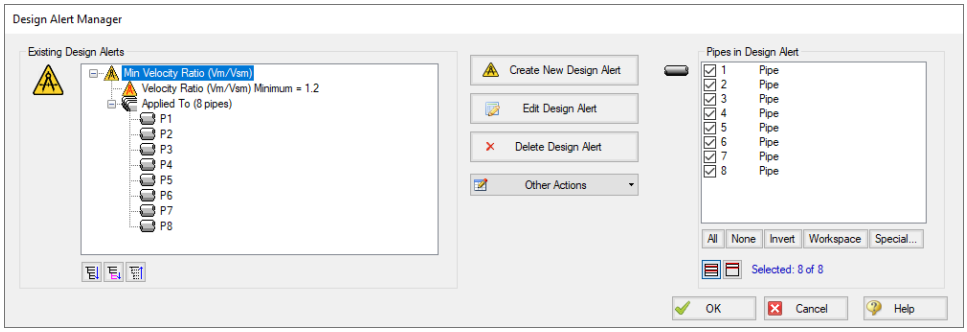


Figure 5: Creating a minimum velocity ratio Design Alert of 1.2 in the Design Alert Manager

Open the Pipe Properties window of any pipe and click the Design Alerts tab. Check to confirm that the Design Alert that you created has been applied (Figure 6).

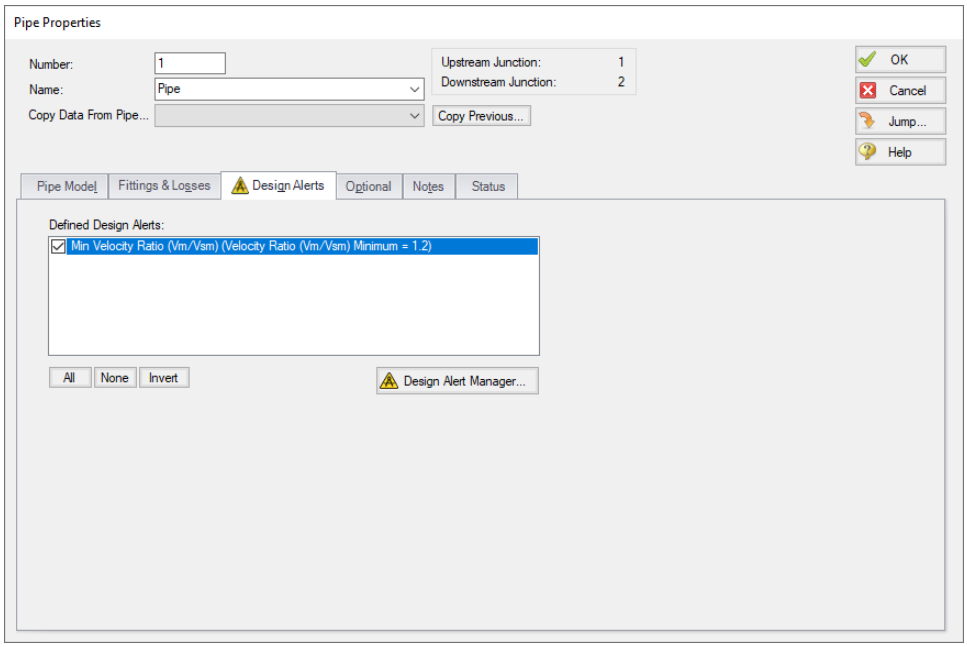


Figure 6: Design Alerts tab showing a defined design alert

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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 5. Define the Pipe Sectioning and Output Group

➢**Open Analysis Setup and open the Sectioning panel.** When the panel is first opened it will automatically search for the best option for one to five sections in the controlling pipe. The results will be

displayed in the table at the top. Select the row to use one section in the controlling pipe.

Step 6. Define the Transient Control Group

➢**Open the Simulation Mode/Duration panel in the Transient Control group.** Enter 10 seconds for the Stop Time.

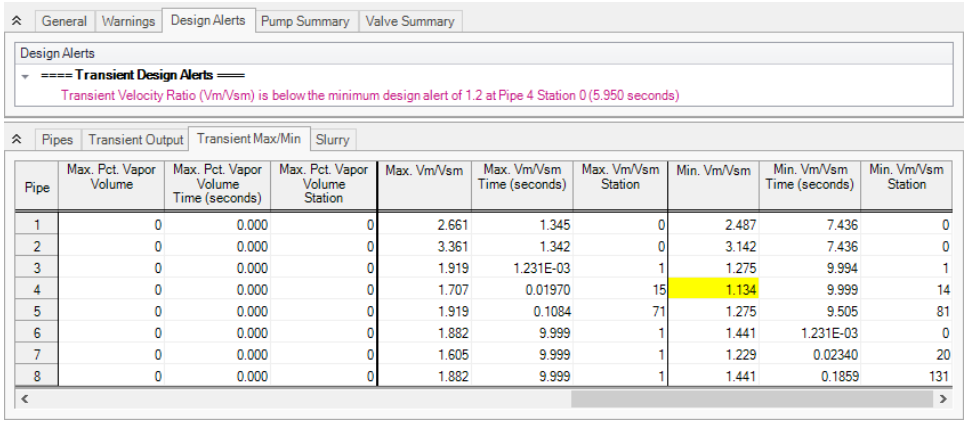
All groups should now be complete and the model is ready to run. If all groups in Analysis Setup have a green checkmark then Click OK and proceed. Otherwise, enter the missing information.

Step 7. Run the Model

Click Run Model from the toolbar or from 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 Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

In the Output window, there is a warning that the Design Alert was violated for Pipe P4. The minimum value for the run was 1.134 in station 14 at 9.999 seconds as shown in the Transient Max/Min table.



Pipe	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	0.000	0	2.661	1.345	0	2.487	7.436	0
2	0	0.000	0	3.361	1.342	0	3.142	7.436	0
3	0	0.000	0	1.919	1.231E-03	1	1.275	9.994	1
4	0	0.000	0	1.707	0.01970	15	1.134	9.999	14
5	0	0.000	0	1.919	0.1084	71	1.275	9.505	81
6	0	0.000	0	1.882	9.999	1	1.441	1.231E-03	0
7	0	0.000	0	1.605	9.999	1	1.229	0.02340	20
8	0	0.000	0	1.882	9.999	1	1.441	0.1859	131

Figure 7: Design Alert violation in pipe P4 station 17

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, the model has the pipes all 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.

Step 9. Increase the Pipe Diameter and Rerun the Model

Return to the Workspace and change pipes P1 and P2 to 14-inch diameter. Since the size of the pipes were changed they need to be re-sectioned by opening the Pipe Sectioning and Output group in Analysis Setup. Click OK to accept the results.

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.

Step 10. Graph the Results

1. Go to the Graph Results window from the Window menu or by clicking on the Graph Results primary window tab.
2. On the Profile tab in the Quick Access Panel, select pipes P1 through P5 (Figure 8). These represent the flow path to Deposit #1.
3. In the Parameter section, choose Velocity Ratio (Vm/Vsm).
4. Check the boxes for Mx, Mn, and DA in order to display the maximum, minimum, and Design Alert.
5. Click Generate.

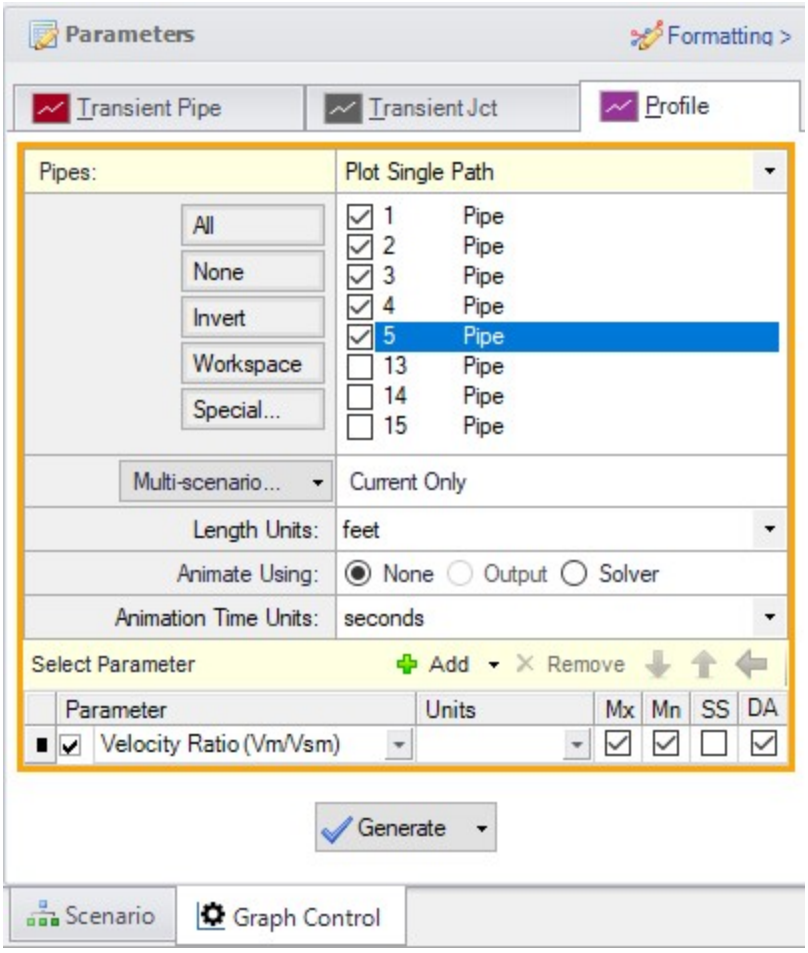


Figure 8: Data selection to view the velocity ratio along the flow path to Deposit #1

The resulting graph is shown in Figure 9. This represents the flow path that had the velocity ratio Design Alert warning when 10-inch pipe was used. Now with 14-inch pipe the minimum is above the 1.2 limit along the entire path.

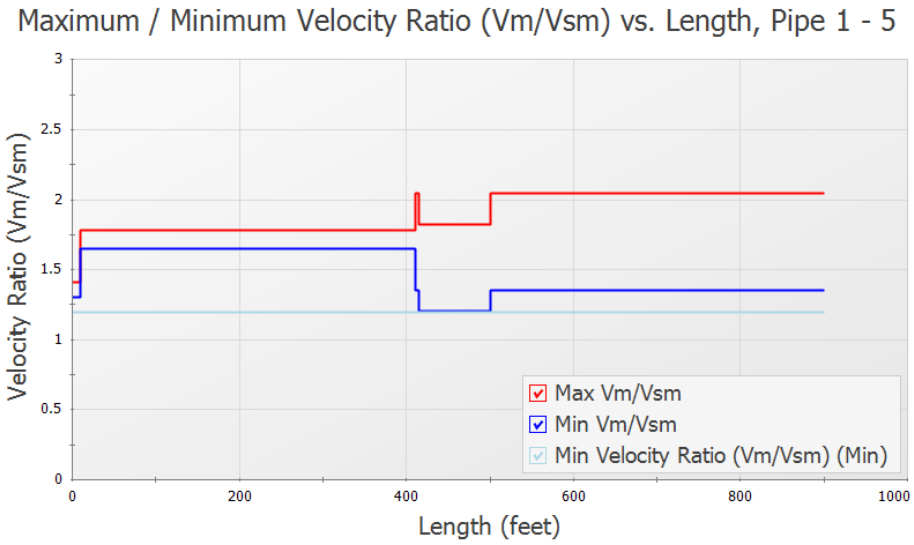


Figure 9: Velocity ratio in the flow path to Deposit #1 is above the minimum Design Alert of 1.2 at all locations

In the Graph Parameters section on the Quick Access Panel change the parameter to Pressure Static and the units to barG. Click show to view the new graph (Figure 10).

You can also check the same parameters for the flow path leading to Deposit #2.

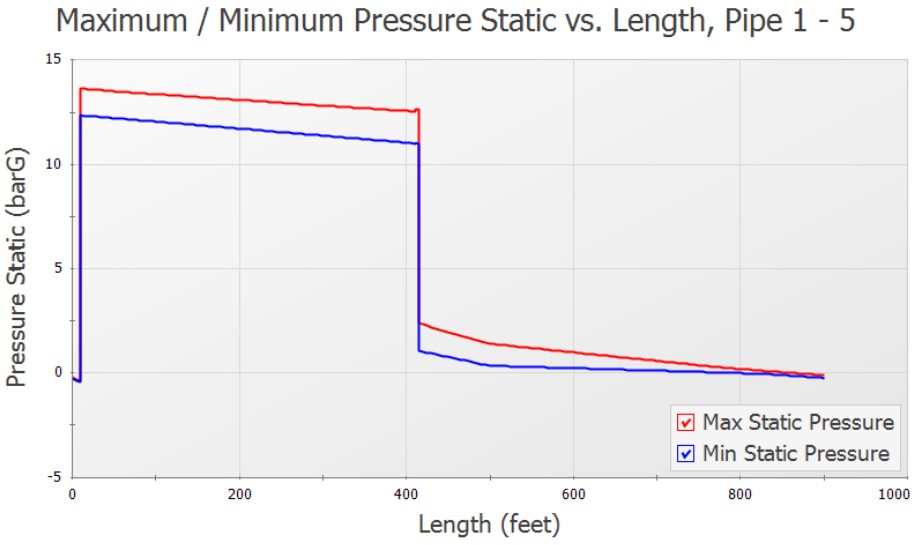


Figure 10: Static pressure along the flow path to Deposit #1

A time comparison of the velocity ratio in the critical pipes along the two flow paths is important to consider.

1. On the Transient Pipe tab, select pipes P4 Inlet and P7 Inlet and add them to the pipe stations to graph (Figure 11). These are the two sloped pipes which have the lowest velocity ratio.
2. In the Graph Parameters list choose Velocity Ratio (Vm/Vsm).
3. Check the DA box to display Design Alerts on the graph.
4. Click Generate.

The resulting graph is shown in Figure 12. This shows that as the valve J4 closes and valve J7 opens the velocity ratios in the pipe immediately downstream change but stay in the acceptable range.

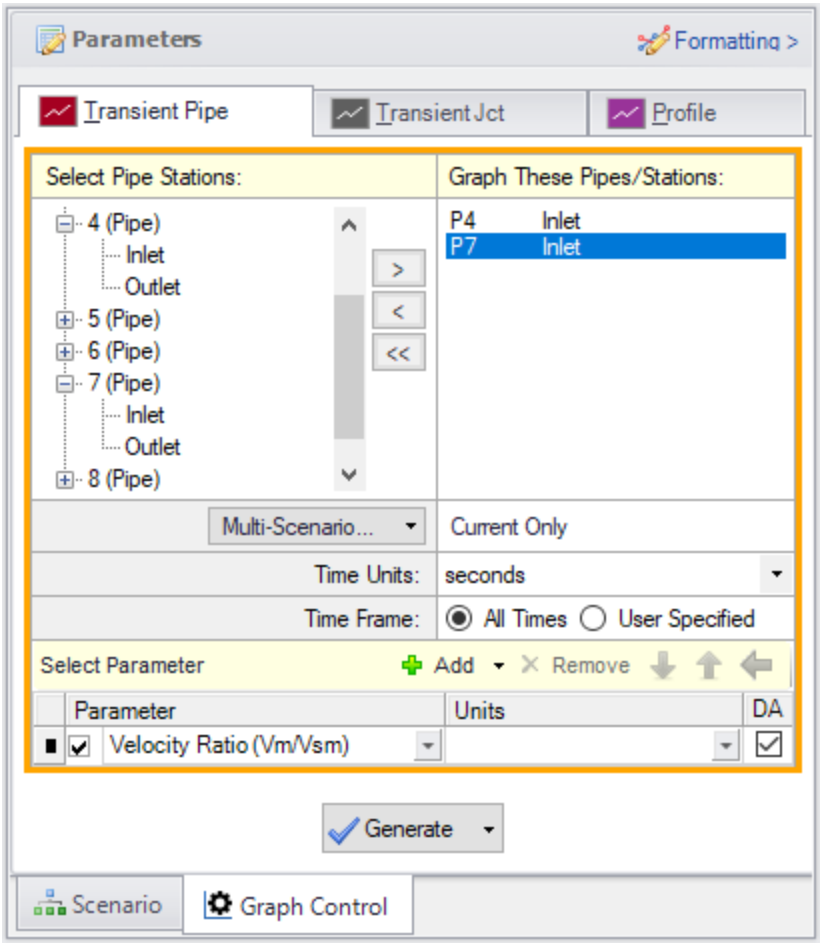


Figure 11: Data selection to view the velocity ratio over time

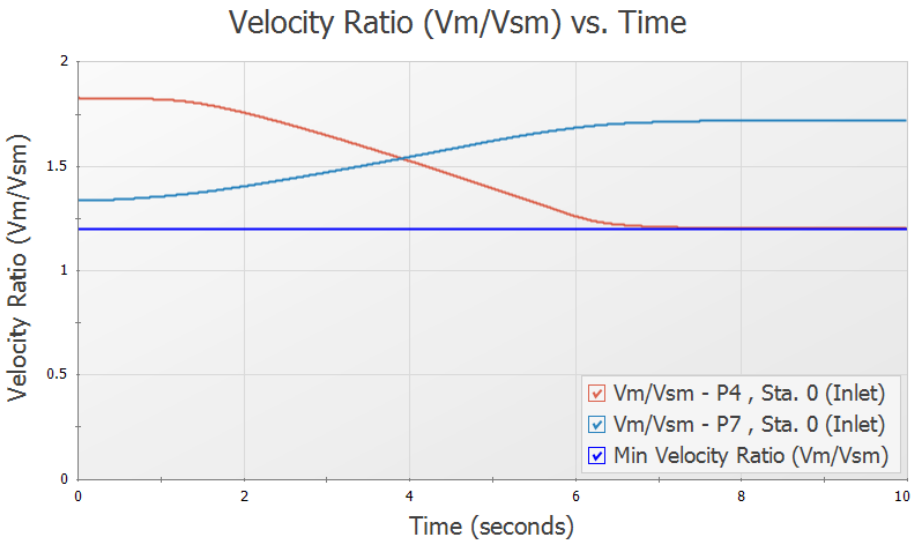


Figure 12: The velocity ratio in the sloped pipes as the valves change position

SSL Analysis Summary

A slurry of sand and water flowing to two deposits was modeled. The velocity ratio was examined to insure a minimum valve was maintained throughout the transient to avoid the danger of the sand settling out of the slurry 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.

PD Pulsation Study - PFA

Summary

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 matching up with certain PD pump frequencies, resulting in severe pressure oscillations at the closed valve. Use AFT Impulse and the PFA (Pulsation Frequency Analysis) module to discover the problematic pump frequencies and what the pressure response at the closed valve is at these specific pump frequencies.

Note: This example can only be run if you have a license of the PFA module

Topics Covered

- Ringing systems to find natural acoustic frequencies
- Finding system excitation frequencies
- Determining 'worst case' PD pump speeds at excitation frequencies
- Calculating pressure response of system at above PD speeds

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the Beginner: Valve Closure example. You can also watch the AFT Impulse Quick Start Video (Metric Units) on the AFT website, as it covers the majority of the topics discussed in the Valve Closure example.

This example is based on the paper by James M. Blanding and Trey Walters.

Model File

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

- ***Metric - PD Pulsation Study - PFA.imp***

Step 1. Start AFT Impulse

From the Start Menu choose the AFT Impulse 9 folder and select AFT Impulse 9.

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

Step 2. Define the Modules Panel

First the PFA add-on modules needs to be activated for use in the model. To do this, open Analysis Setup from the toolbar or Analysis menu. On the Modules panel, check the box next to Activate PFA. After the PFA module is activated, the Use option should being selected by default.

Step 3. Define the Fluid Properties Group

Open the Fluid panel then define the fluid:

1. Fluid Library = User Specified Fluid
2. Name = PFA Fluid
3. Density = 0.92 gram/cm3
4. Dynamic Viscosity = 0.95 centipoise
5. Bulk Modulus = 9653 bar

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model as shown in the figure below.

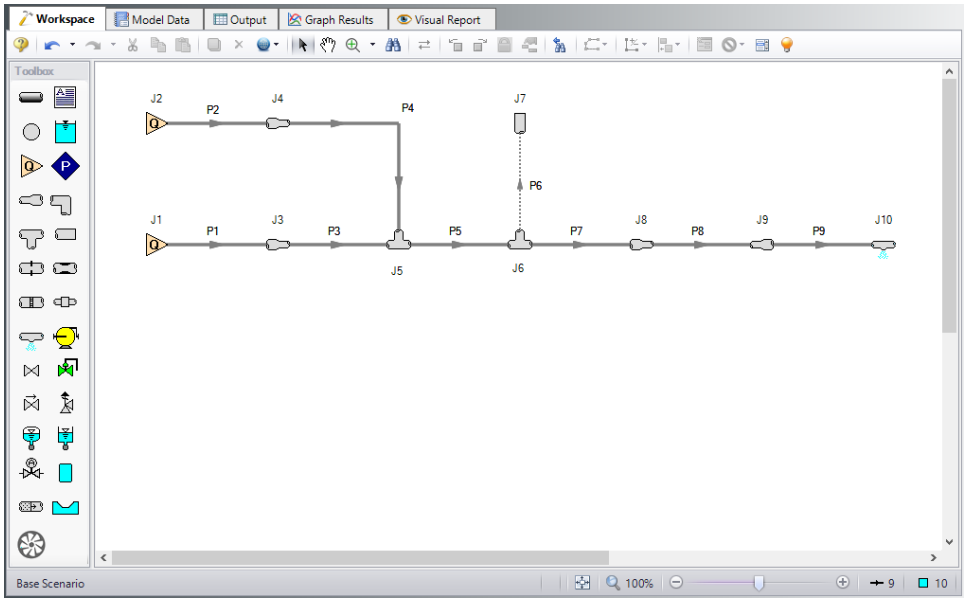


Figure 1: Layout of PD Plunger Pulsation Study example

The system is in place but now we need to enter the input data for the pipes and junctions. Double-click each pipe and junction and enter the following data in the properties window.

Pipe Properties

1. Pipe Material = (User Specified)
2. Inner Diameter = Use table below
3. Length = Use table below
4. Wavespeed = User Specified Wavespeed
    a. Wavespeed = 915 meters/sec
5. Friction Model Data Set = User Specified
6. Selection = Absolute Roughness
7. Value = 0.00152 cm

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

Junction Properties

Note: For the sake of simplicity, assigned flow junctions J1 and J2 are being used to represent the PD pumps. 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. Assigned Flows J1 & J2
    a. Elevation = 0 meters
    b. Volumetric Flow Rate = 8 m3/hr
2. Area Changes J3 & J4
    a. Inlet Elevation = 0 meters
    b. Type = Abrupt Transition (Cylindrical)

3. Tee/Wye J5 & J6
    a. Elevation = 0 meters
    b. Loss Model = Simple (no loss)
4. Dead End J7
    a. Elevation = 0 meters
5. Area Changes J8 & J9
    a. Inlet Elevation = 0 meters
    b. Type = Conical Transition
    c. Angle (μ) = 90 degrees
6. Spray Nozzle J10
    a. Elevation = 0 meters
    b. Loss Model = Cd Spray (Discharge Coefficient)
    c. Geometry = Spray Nozzle
    d. Exit Properties = Pressure
    e. Exit Pressure = 0 barG
    f. Cd (Discharge Coefficient) = 1
    g. Discharge Flow Area = 0.148 cm2

➢**Turn on the Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the 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 5. Define the Pulsation Setup Group

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

Open Analysis Setup to see the Pulsation Setup group below the Transient Control group. Use the Pulsation Setup group to specify input regarding the initial pulse and other information about the pulsation source. There are two panels in the Pulsation Setup group: the Pulse Setup and PD Pump Settings panels. The PD Pump Settings panel 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. Enter the data below to define the Pulse Setup panel.

1. Applied At = J2 (Assigned Flow)
2. Start Time = 0 seconds
3. Magnitude = Automatic (Twice Steady-State Flow)
4. Peak = 16.00 m3/hr
5. Cutoff Frequency = PD Pump Characteristics
6. Filter Type = Chebyshev
7. Order = 10

8. Ripples Allowed = 1 dB
9. Pulsation Source = Positive Displacement (PD) Pump
10. Number of Heads = 3
11. Pump Speed = 250 RPM
12. Speed Buffer = 15 Percent (%)

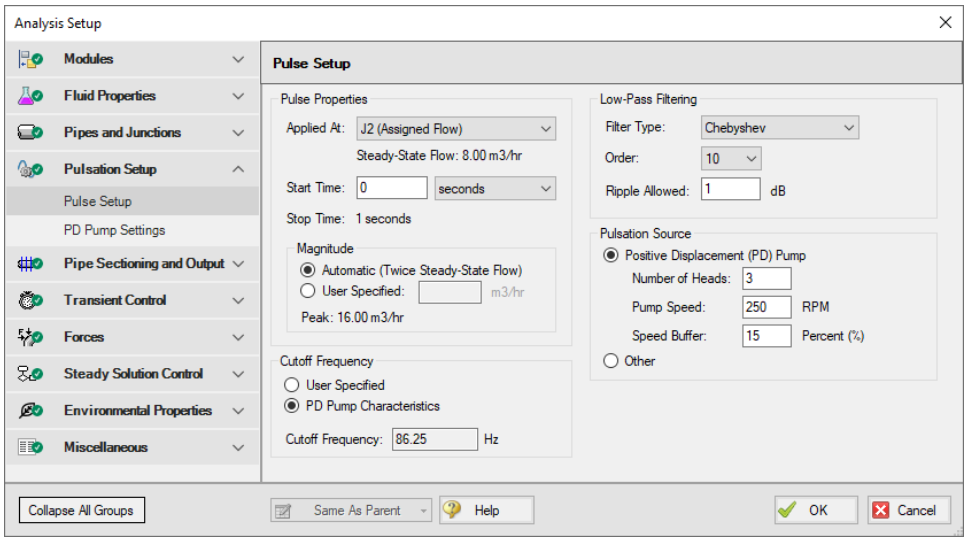


Figure 2: Pulsation Setup window with Input

Now that all required input has been entered in the Pulse Setup panel, enter the following information on the PD Pump Settings panel (see Figure 3).

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

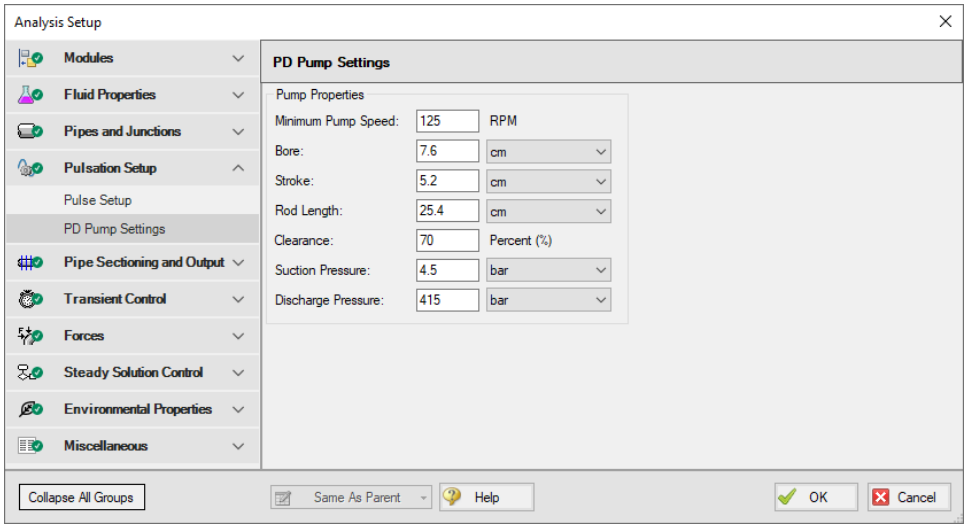


Figure 3: PD Pump Settings tab with Input

Step 6. Define the Pipe Sectioning and Output Group

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

1. Select the significant pipes that should consider wavespeed variance. In this example we will consider the variance in all modeled pipes, except for Pipe P8. You can see that this pipe has been excluded in Figure 4. Pipe P8 is excluded because it represents a flow meter and not an actual pipe. Units such as flow meters and dampeners may be better modeled with a pipe than with a junction if they have significant length and associated acoustic interaction aspects in the physical system. Ensure that the box next to each pipe is selected under Consider Variances, except for Pipe P8.
2. Click Update Results and select one section in the controlling pipe by clicking on one section in the table under Search Results. This window is displayed in Figure 4.
3. Select Show Pulsation Graphs located at the bottom left of the Sectioning panel. A graph of the pulse will appear (see Figure 5). 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. The LPF of Pulse is shown in Figure 6. After you are done viewing these graphs, click Close to return to the Section Pipes window.
4. Click OK to save and close Analysis Setup.
5. 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.
6. Open J2 to verify that the numerical ring has been applied to the transient (see Figure 7 for the transient input). This transient is a brief jump in flow rate at junction J2.

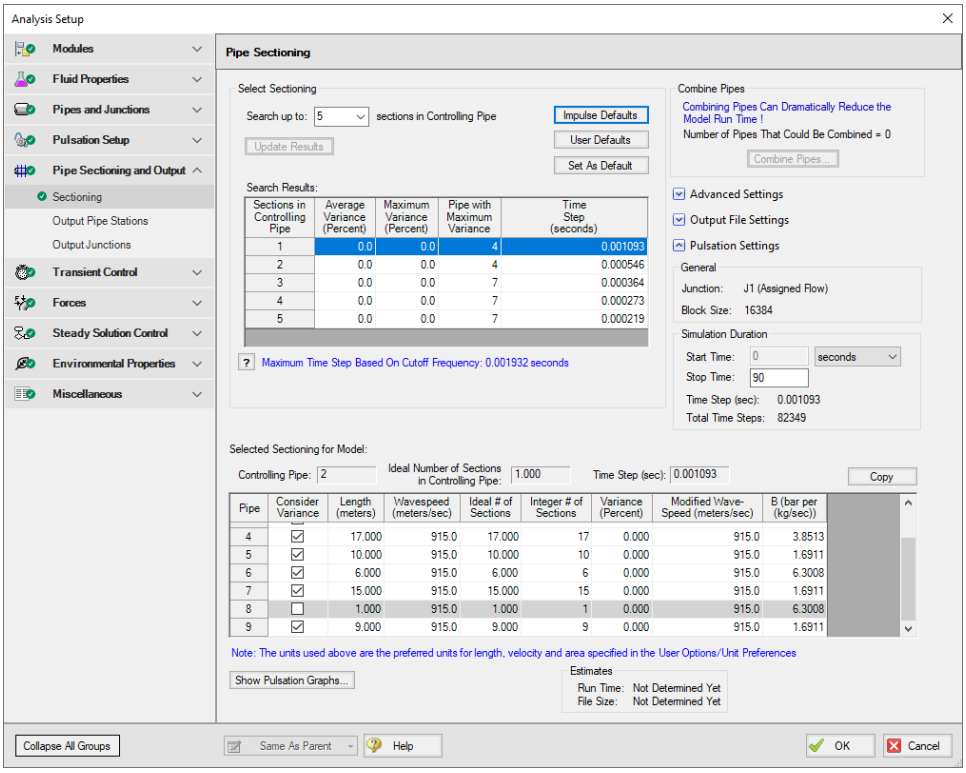


Figure 4: Section Pipes window

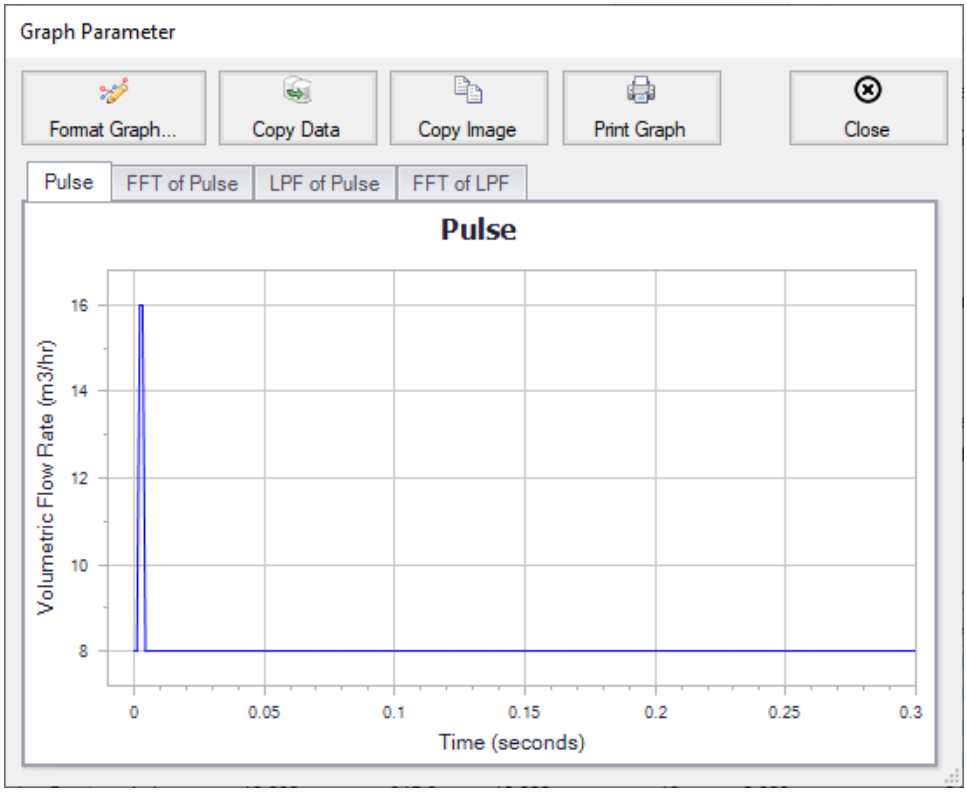


Figure 5: Graph of Pulse in this example

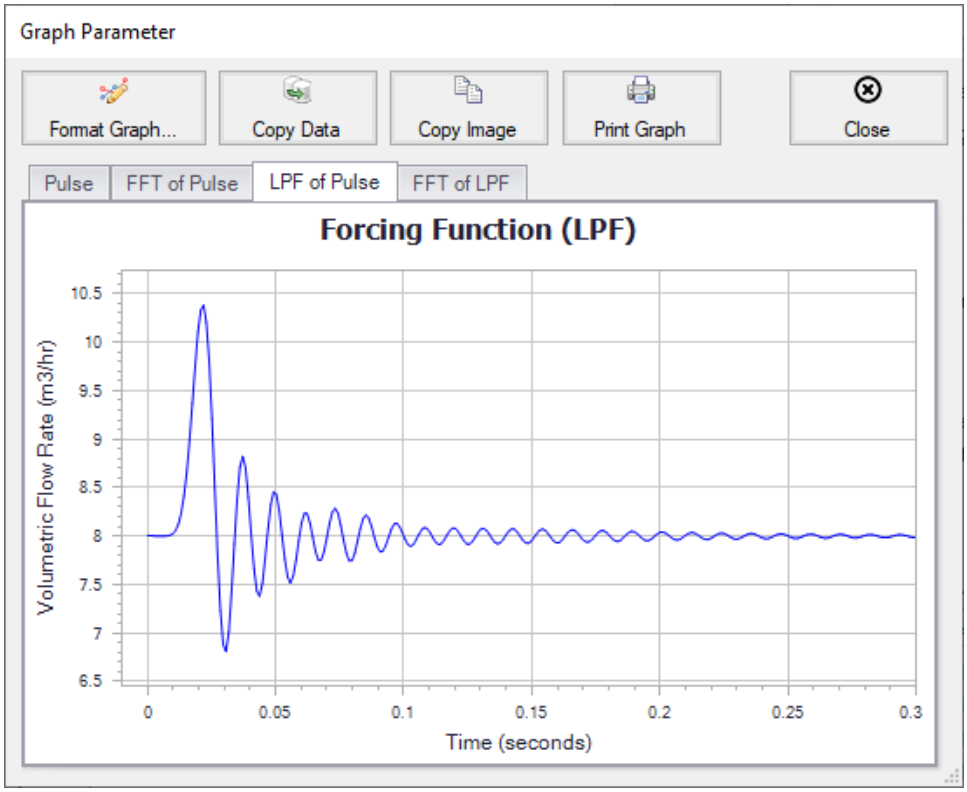


Figure 6: Graph of LPF of Pulse

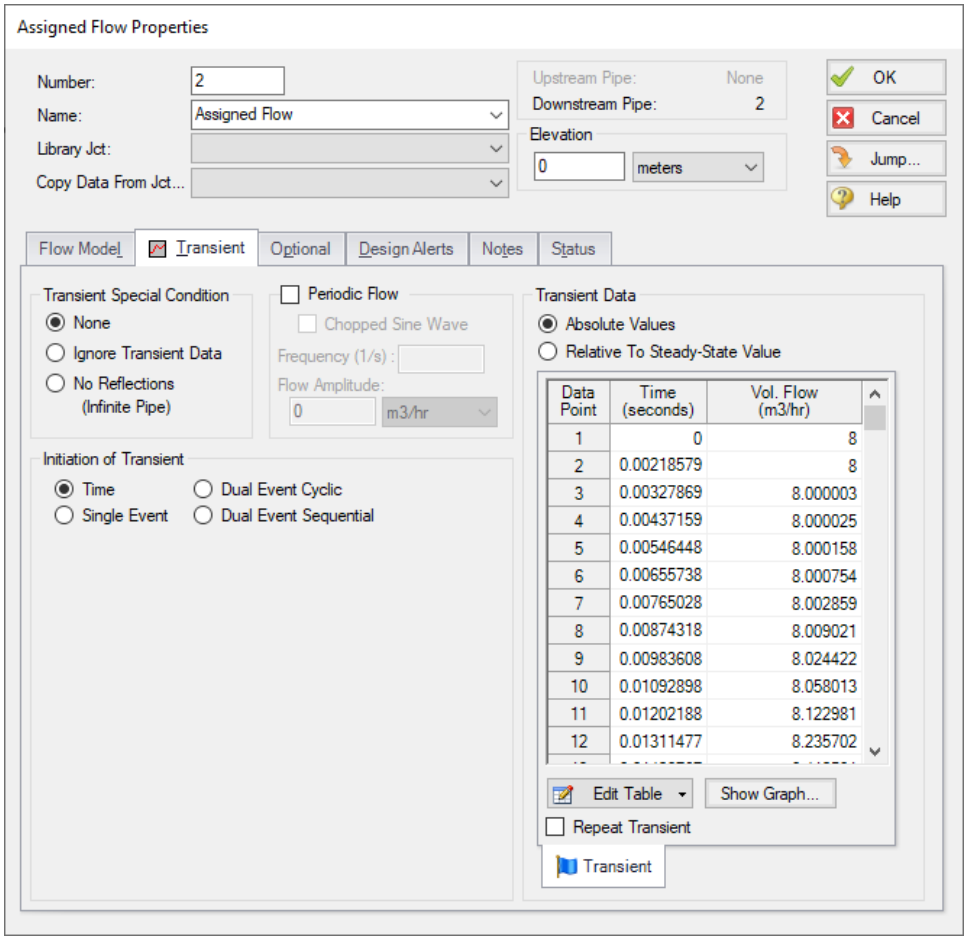


Figure 7: Assigned Flow Properties with Transient from Pulsation Setup. The Show Graph button under the Transient Data table can be selected to view the graph of this data

Step 7. Run the Model

Click Run Model from the toolbar or from 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.

Step 8. Examine the Results

1. **Find excitation frequencies to study**
   a. On the Graph Control tab in the Quick Access Panel, click the Frequency tab to begin generating an Excitation Frequency Analysis graph
   b. 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.

c. 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 (see Figure 8).

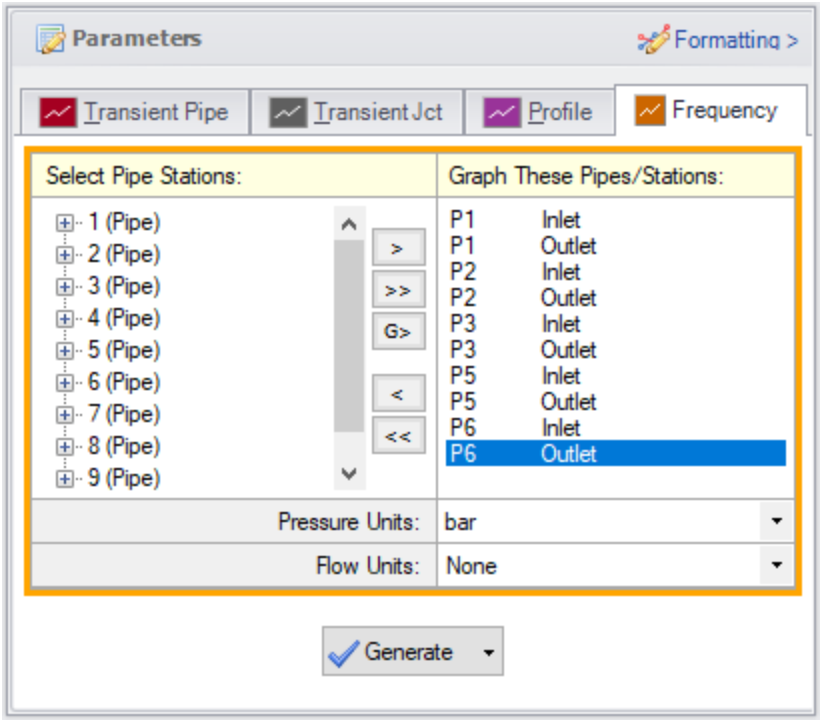


Figure 8: Graph Parameters selected for Excitation Frequency Analysis graph

d. Click Generate. The resulting graph (shown in Figure 9) represents the frequencies that are excited by the pulse defined in the Pulsation Setup window.

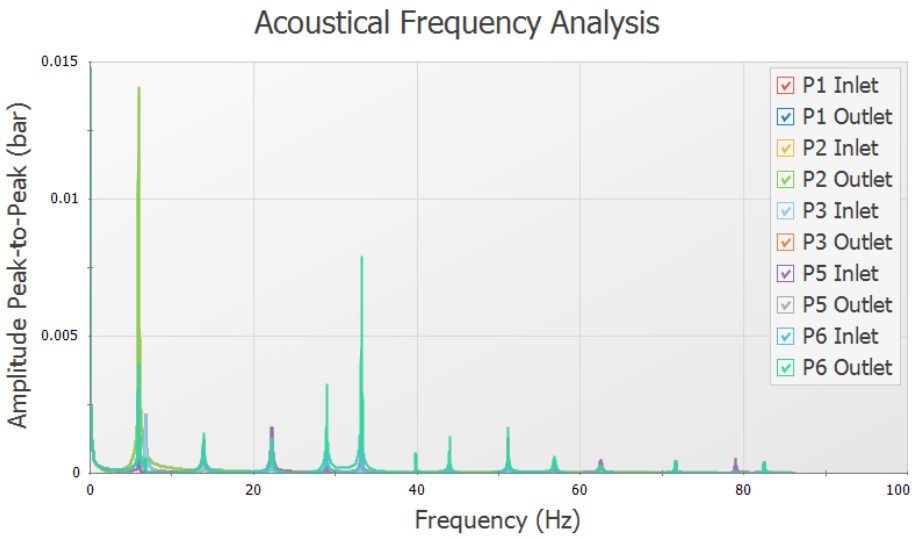


Figure 9: Excitation Frequency Analysis graph

e. From the graph in Figure 9, we see that there are several frequencies that produce a large pressure magnitude. Points on the plot with a large response magnitude are easily identified by finding local maxima on the graph. These maxima are the acoustic natural frequencies that are potentially the most damaging when excited by the identified pump speeds.

f. Note that the graph in Figure 9 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. See Figure 10 for an image of the graph in Figure 9 with the amplitude changed to be displayed on a logarithmic scale.

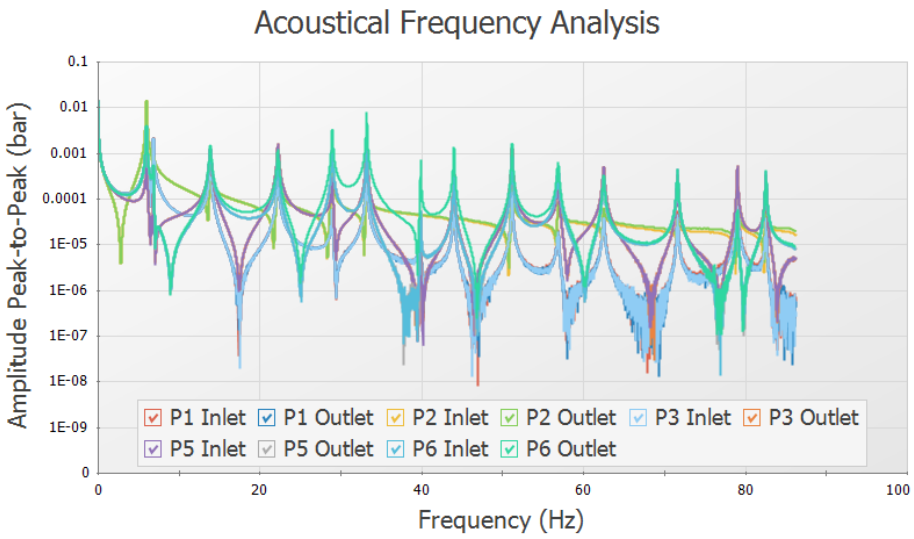


Figure 10: Excitation Frequency Analysis Graph with Magnitude on Logarithmic Scale

2. **Evaluate excitation frequencies**
   a. The identified frequencies can now be evaluated to determine pump speeds that would cause them to be excited, making them potentially problematic. Follow the steps below to select and evaluate the excitation frequencies.
   b. Left-click near the frequency of about 33.24 Hz and drag the mouse over the peak on the graph near this frequency. This will highlight and flag the local maximum. Figure 11 shows the frequency graph after the dialog flag appears.

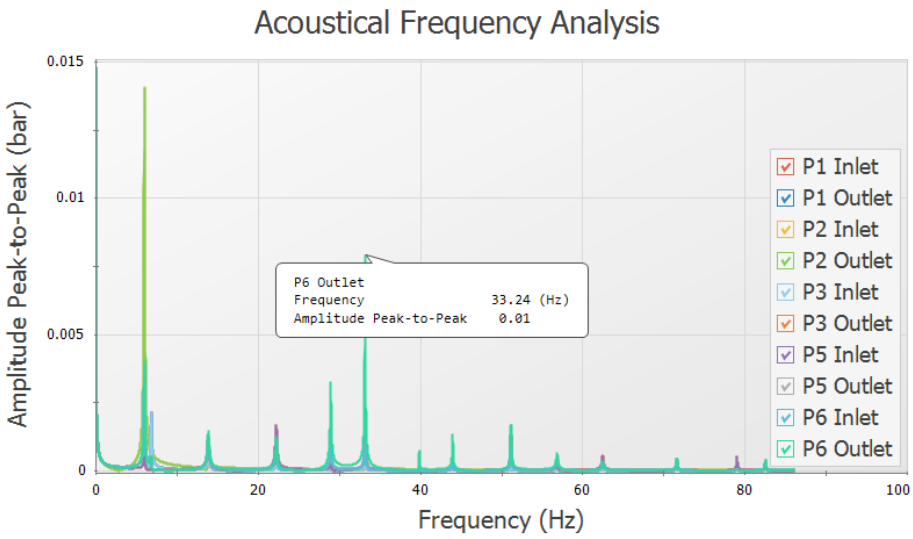


Figure 11: Excitation Frequency Analysis with Dialog Flag

c. Right-click on the dialog flag that appears on the peak of this frequency as shown in Figure 12.

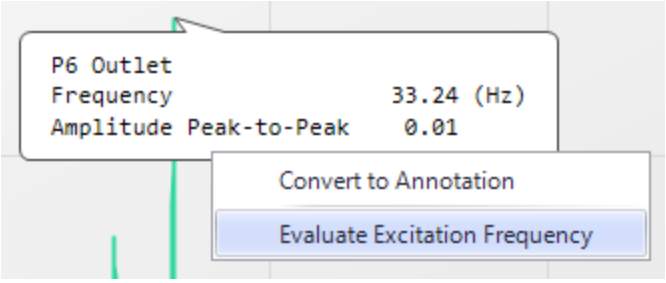


Figure 12: Right Click on the Dialog flag to evaluate the frequency

d. Select Evaluate Excitation Frequency to add this frequency to the Pump RPM Evaluation panel, which is labeled with Excitation Frequencies and Pump Speeds (RPM) (see Figure 13)

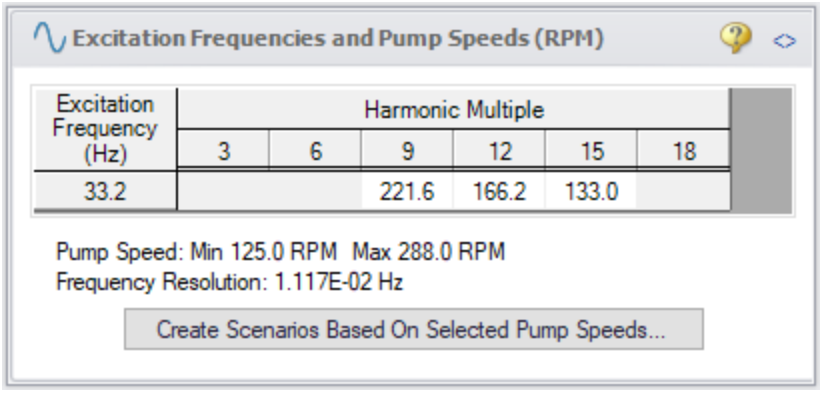


Figure 13: Pump RPM Evaluation panel

e. Repeat steps a through c for the local maxima at the frequencies of 28.98 Hz and 6.89 Hz. Figure 14 shows the Pump RPM Evaluation Panel with the excitation pump speeds displayed.

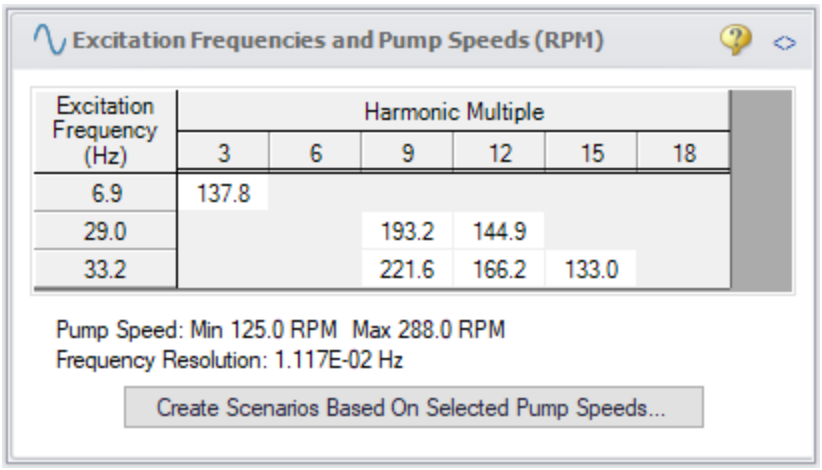


Figure 14: Pump RPM Evaluation Panel with Excitation Frequencies and their Harmonic Multiples Displayed

f. 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 15 appears.

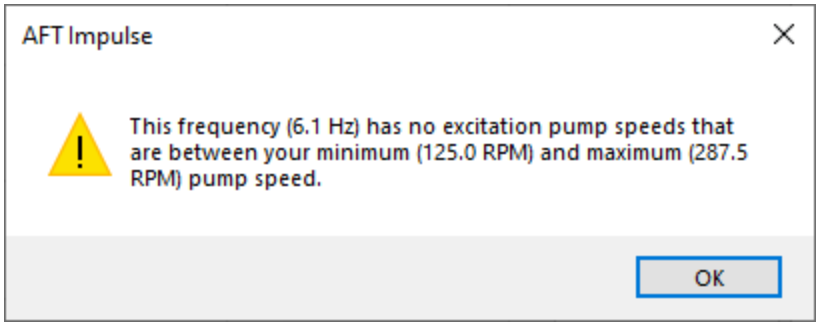


Figure 15: Message Indicating No Excitation Pump Speeds Between the Minimum and Maximum Pump Speeds Exist

3. **Determine the pressure response of the system at excited frequencies**
   a. 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. Scenarios can then be created that will use the information from the PD Pump Settings to create a child Pump Speed scenario that will have the Flow vs. Time profile at the selected speed (RPM) that will excite the evaluated frequencies.
   b. We will create child scenarios for all six pump speeds in the Pump RPM Evaluation panel. To do this, ensure that all pump speeds are selected (not italicized).
   c. Click Create Scenarios Based On Selected Pump Speeds…
   d. A dialog box will appear stating that six scenarios will be created based on the selected speeds in the Pump RPM Evaluation panel. Click Yes to continue creating the child scenarios.
   e. Load the 133 RPM (33.2 Hz) scenario.
   f. The scenario will now have different transient information in J2, modeling the pump performance at 133 RPM. Go to the Workspace and open the Properties window for J2. On the Transient tab click Show Graph under the Transient Data table to verify the Pump Flow Forcing Function has been applied, as shown in Figure 16.

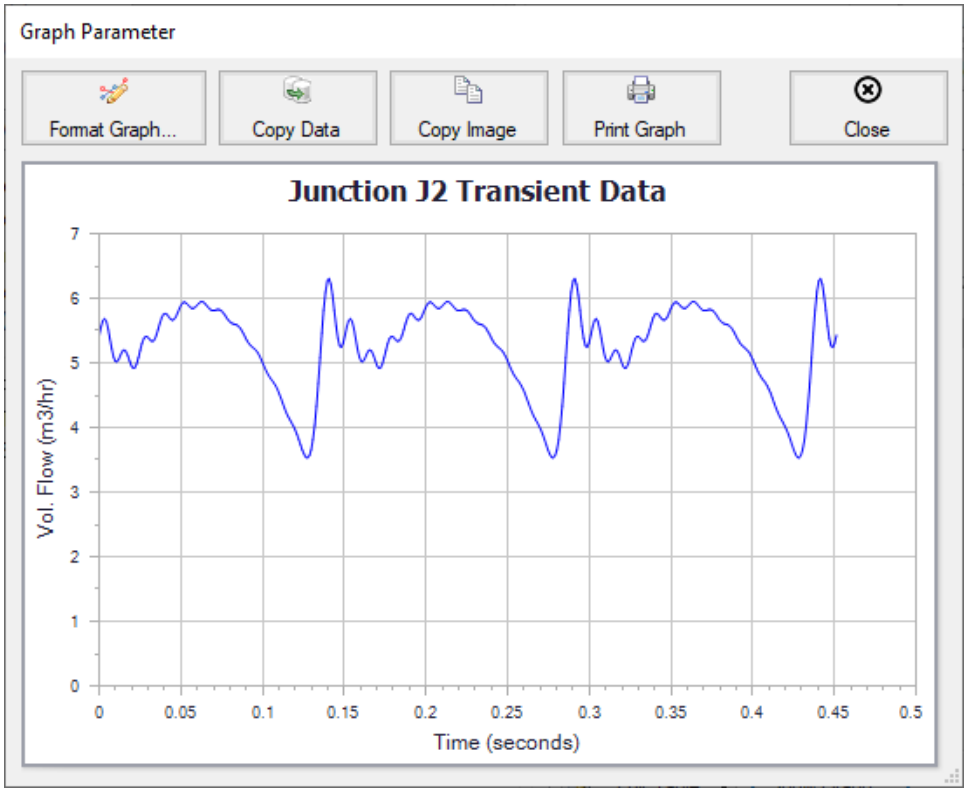


Figure 16: New Assigned Flow transient representing the PD Pump operating at 133 RPM

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

1. To create a pressure profile graph in pipe P6, go to the Graph Results tab and 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

Figure 17 shows the profile plot for Pipe P6 at a pump speed of 133 RPM .

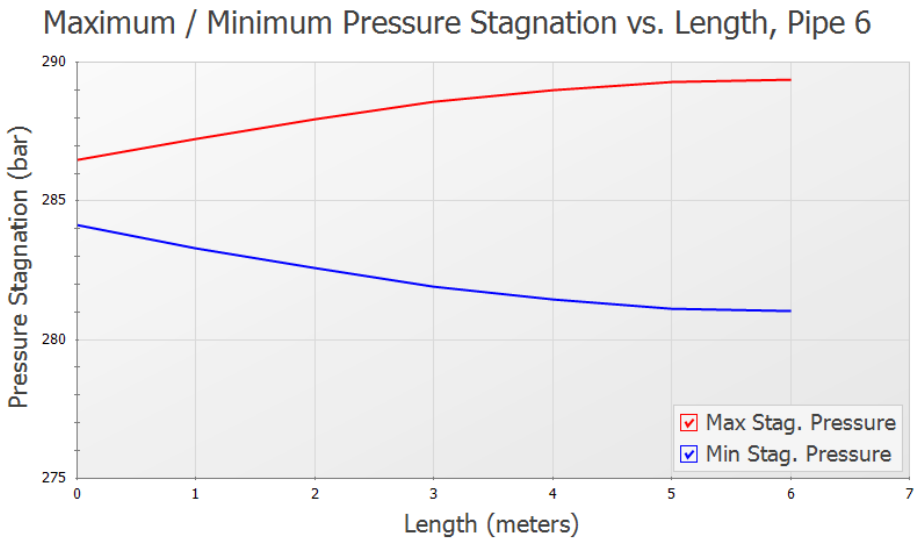


Figure 17: Max/Min Pressure Profile of Pipe P6 for 133 RPM

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

1. In the Quick Access Panel, select Transient Pipe for the graph type
2. Expand the pipe station menu for Pipe P6
3. Add the outlet and inlet of Pipe P6 to the Graph These Pipes/Stations list
4. Select Pressure Stagnation for the Parameter
5. Use bar for the units
6. Set the Time Frame to User Specified with a Start Time of 0 and a Stop Time of 2 seconds.
7. Click Generate (see Figure 18).

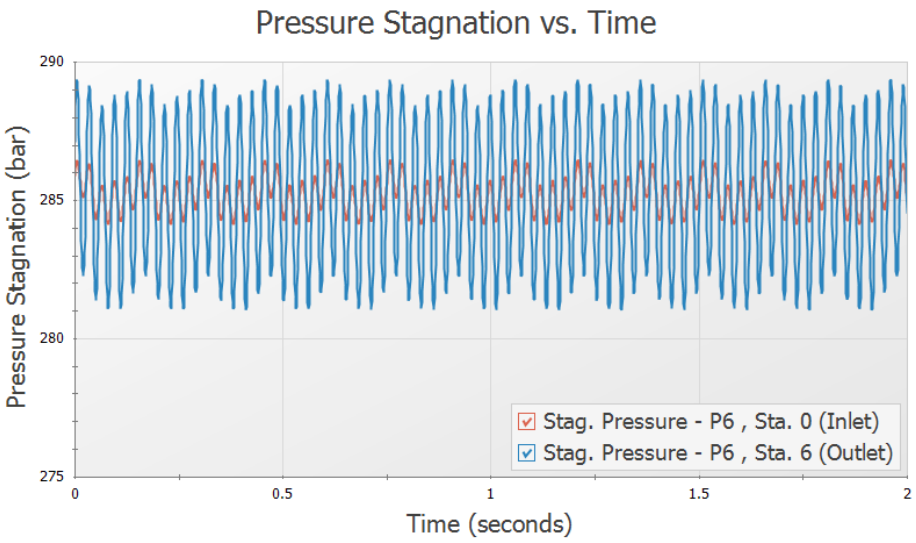


Figure 18: Pressure Oscillations at Inlet and Outlet of Pipe P6 for 133 RPM

After looking at these graphs, go to the Output tab, which should open to the Warnings tab. 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. In this case, the the Peak-to-Peak pressures exceed the allowable limits. See Figure 19. 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. For a full summary, click on the Pulsation Summary tab, as is shown in Figure 20.

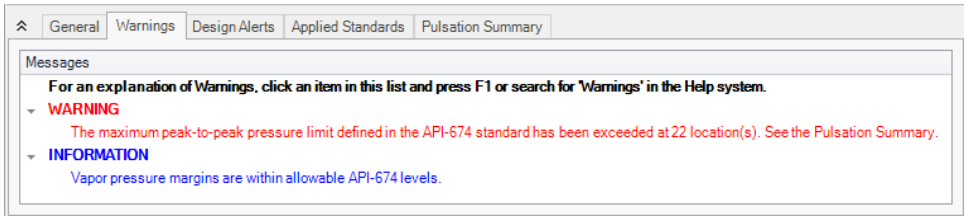


Figure 19: Warnings section the Output window with information about API-674 being exceeded

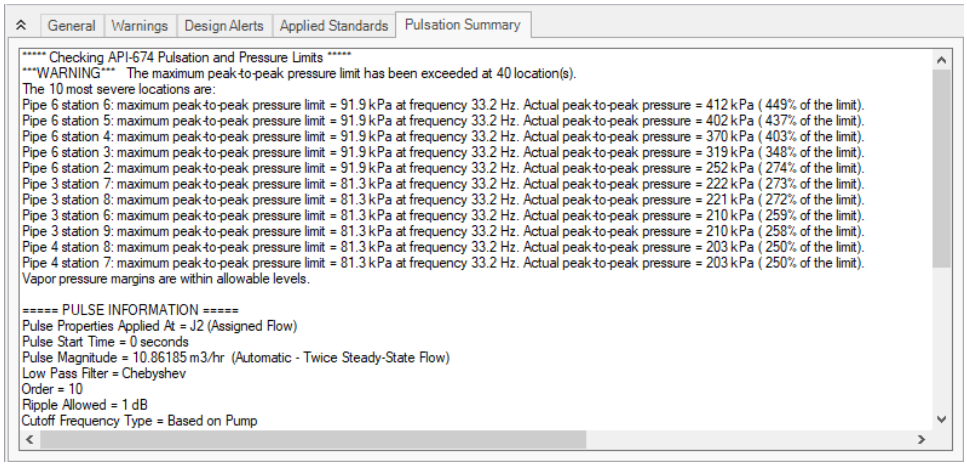


Figure 20: Pulsation Summary in the Output window

For the purposes of this example, we will repeat these steps for the pump speed of 221.6 RPM because this pump speed causes the largest pressure oscillations. We will show these graphs here. Though not shown, multi-scenario graphing was used to determine the pump speed that caused the maximum oscillations. When performing an analysis on your system, all pump speeds that excite the system should be evaluated.

Figure 21 shows the Max/Min Pressure Profile for P6 for the pump speed of 221.6 RPM .

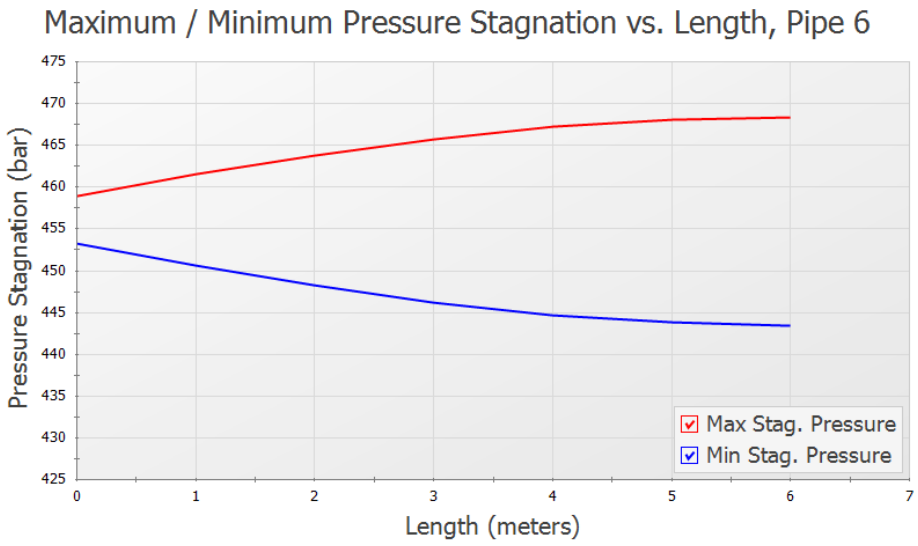


Figure 21: Max/Min Pressure Profile of Pipe P6 for 221.6 RPM

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

Figure 22 shows the Max/Min Pressure Profile for P6 for the pump speed of 221.6 RPM.

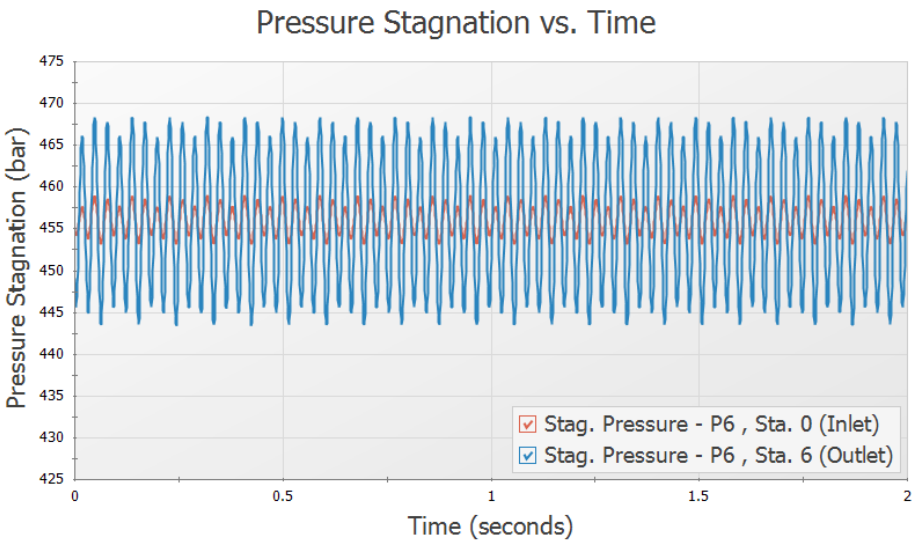


Figure 22: Pressure Oscillations at Inlet and Outlet of Pipe P6 for 221.6 RPM

The large pressure oscillation in Pipe P6 can be seen in Figure 22. 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.

Return to the Output tab of the Graph results window for the scenario with the pump speed of 221.6 RPM, and make note of the message in red at the bottom right of the Graph Results screen that states that Warnings Exist. This scenario also violates API-674 pulsation and pressure limits similar to the 133 RPM scenario.

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.