

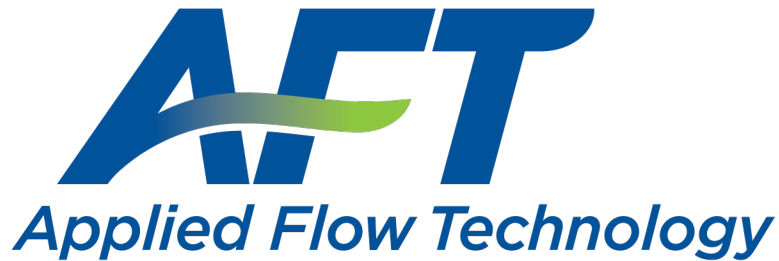
# ***AFT xStream***<sup>™</sup>

## **Examples - Metric Units**

**AFT xStream Version 3**

**Gas Transient Modeling in Piping Systems**

**Published: November 01, 2023**



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

# Contents

---

<b>Walk-Through Examples</b> .....	<b>3</b>
<b>Beginner - Tank Blowdown Example (Metric Units)</b> .....	<b>4</b>
<b>Gas Turbine Fuel System Example</b> .....	<b>46</b>
<b>Fluid Library Comparison - Steam Turbine Shutdown Example</b> .....	<b>62</b>
<b>Pipe Forces from Turbine Shutdown Example</b> .....	<b>79</b>
<b>Frequency Analysis - PFA Example</b> .....	<b>94</b>
<b>Index</b> .....	<b>108</b>

## Walk-Through Examples

The AFT xStream Walk-Through Examples are designed to get new users trained to use AFT xStream correctly in the shortest amount of time. If you are using AFT xStream for the first time, start with the [Beginner - Tank Blowdown](#) example because it offers an in-depth explanation of how to use AFT xStream. Otherwise, choose an example from the table below that you wish to learn more about.

Completed versions of the example model files are included with the default installation: C:\AFT Product-s\AFT xStream 3\Examples

Example	Fluid	Pipes	Description
<a href="#">Beginner - Tank Blowdown</a>	Nitrogen	2	An in depth discussion of model building in AFT xStream and a basic overview of the software interface and Primary Windows. Models a simple valve opening during a tank blow-down.
<a href="#">Gas Turbine Fuel System</a>	Methane	11	A compressor is passing natural gas through a heat exchanger before being used in the combustion baskets of a turbine system. This example evaluates the effect the sudden closure of one of the 3 control valves has on the temperature of the system. The effect of the Minimum Number of Sections per Pipe parameter is also shown.
<a href="#">Fluid Library Comparison - Steam Turbine Shutdown</a>	Steam	12	Two turbines in a high pressure steam system are simultaneously shut off as a bypass line valve is opened. This model demonstrates factors to know to evaluate which fluid library to use in your system, as well as demonstrating how to construct a model in the isometric drawing mode.
<a href="#">Pipe Forces from Turbine Shutdown</a>	Steam	14	Two turbines in a high pressure steam system are simultaneously shut off. This model demonstrates the transient forces of the steam within the system as well as the effect that the transient duration has on these forces. Finally, this example shows how to export the force file once the forces are obtained.
<a href="#">PFA Add-on Module - Frequency Analysis</a>	Methane	5	A frequency analysis is performed on a system with a positive displacement compressor to determine the acoustical frequencies for the existing piping configuration using the Pulsation Frequency Analysis (PFA) Module.

## Beginner - Tank Blowdown Example (Metric Units)

### Summary

---

This example will give you the big picture of AFT xStream's layout and structure. To accomplish this purpose, a simple model will be built to evaluate the initial design for a tank that undergoes a blowdown. Nitrogen flows out of a supply tank into a 2 inch line as a part of the blowdown. The flow is controlled by opening and closing a valve in the 2 inch line. This model will simulate opening the valve in 0.1 seconds, after which the valve will remain open for the remainder of the simulation.

### Topics Covered

- Model building basics
- Entering Pipe and Junction data
- Entering transient data
- Using the Primary Windows: Workspace, Model Data, Output, & Graph Results
- Analysis Setup: fluid libraries, sectioning pipes, etc
- Graph types
- Workspace Layers

### Required Knowledge

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

### Model File

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

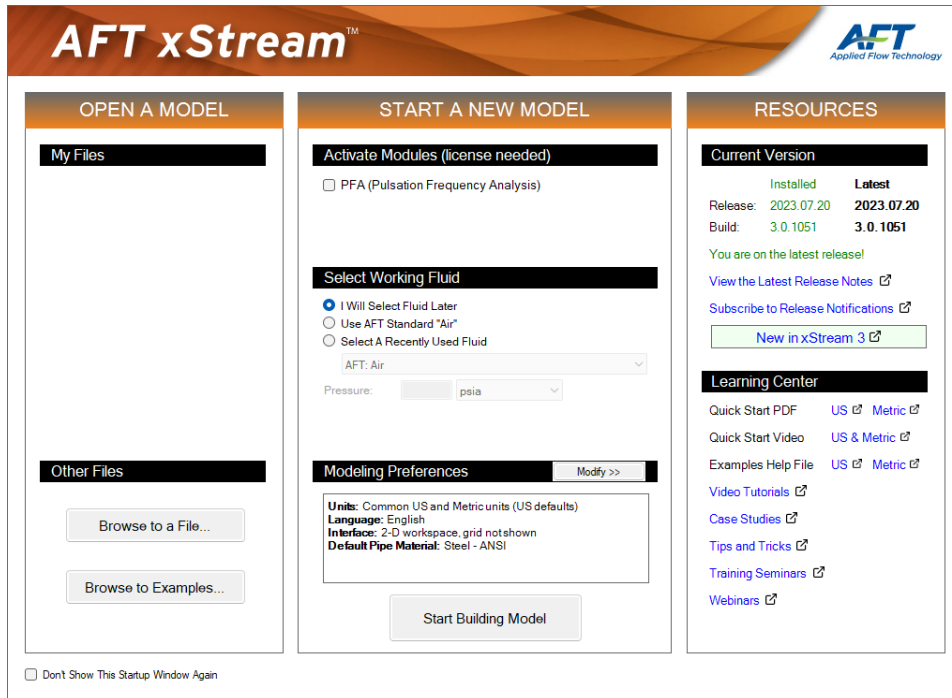
C:\AFT Products\AFT xStream 3\Examples

- ***Metric - Beginner - Tank Blowdown.xtr***

## Step 1. Start AFT xStream

➤ **To start AFT xStream**, open the all applications list in the Windows Start Menu, expand the AFT xStream 3 folder, and choose AFT xStream 3. This refers to the standard menu items created by a standard installation; you may have chosen to specify a different installation location.

As AFT xStream starts, the Startup Window appears, as shown in Figure 1. This window provides you with several options before you start building a model.



**Figure 1: Startup Window with Modeling Preferences collapsed**

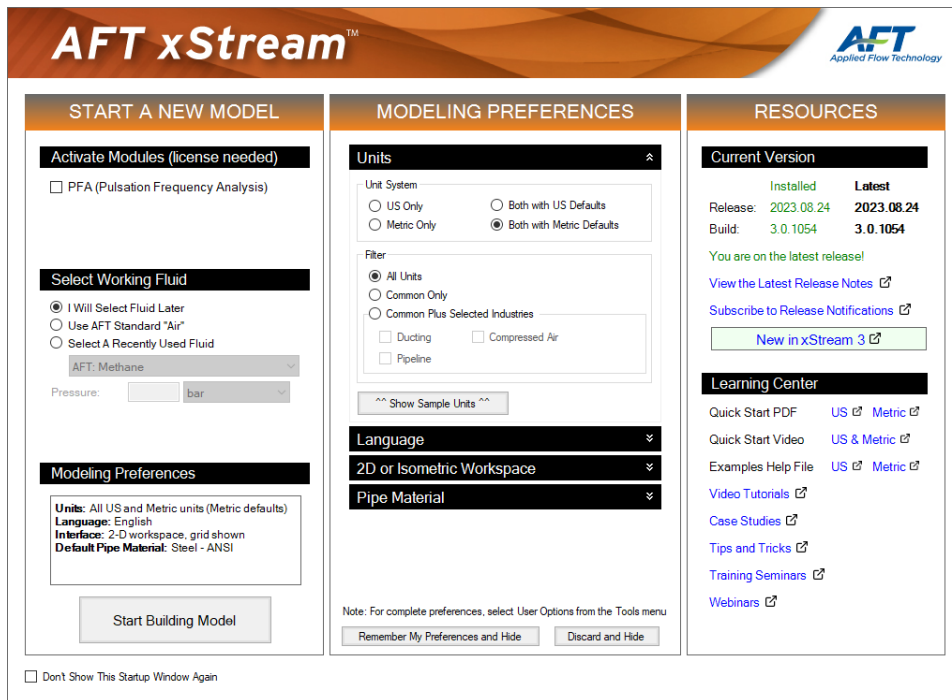
Some of the actions available from the Startup window are:

- Open a previous model, browse to a model, or browse to an example
- Activate an Add-on Module
- Select AFT Standard “Air” or a recently used fluid to be the Working Fluid
- Review or modify Modeling Preferences: select a Unit System, filter units, choose a Grid Style, select a Default Pipe Material, and more
- Access other Resources such as Help Files and Video Tutorials

If this is the first time that you have started AFT xStream, Modeling Preferences will be expanded in the middle section of the Startup Window, as shown by Figure 2. If this is not the first time that you have started AFT xStream, the Startup Window will appear with Modeling Preferences Collapsed, as shown in Figure 1.

## Beginner - Tank Blowdown Example

When collapsed, you can view your current Modeling Preferences at the bottom of the Start a New Model section. To further review or adjust your preferences, click the Modify button (Figure 1).



**Figure 2: Startup Window with Modeling Preferences expanded**

➤ **With Modeling Preference expanded, as in Figure 2, select Both with Metric Defaults under Unit System.** The Filter section can be used to limit the number of units that are shown in the interface.

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

➤ **Click Start Building Model.**

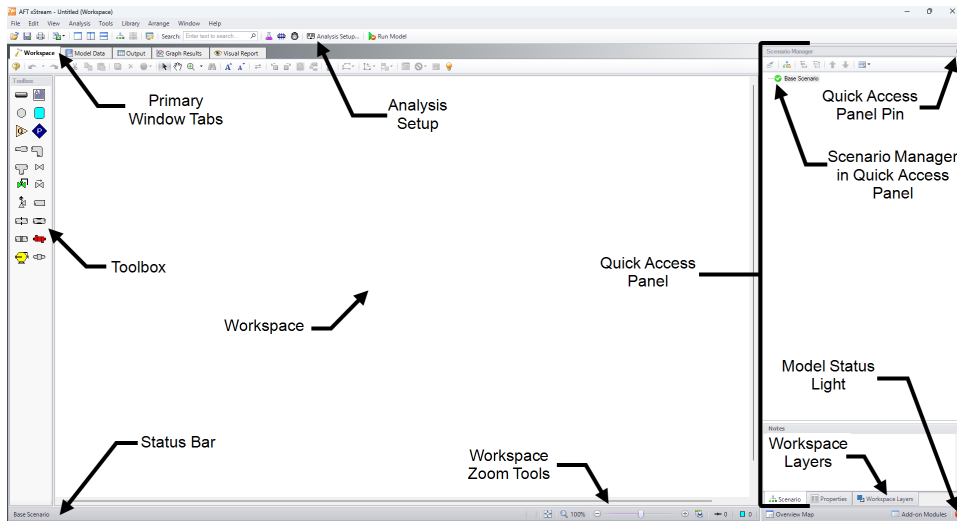
The Workspace window is shown initially, as seen in . The tabs in the AFT xStream window represent the Primary Windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

AFT xStream supports multiple monitor usage. You can click and drag any of the Primary Window tabs off of the main xStream window, where you can then 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 xStream 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 xStream settings, unless you are specifically instructed to do otherwise.

### ***The Workspace window***

The Workspace window is the primary interface for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox is the bundle of tools on the far left. The Quick Access Panel is displayed on the far right. It is possible to collapse 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.



**Figure 3: The Workspace Primary Window**

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

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

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

## **Step 2. Build the model**

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

### **I. Place the Tank junction**

➤ **To start, click and drag a Tank junction from the Toolbox and drop it on the Workspace.** Figure 4 shows the Workspace with one Tank.

### Objects and ID numbers

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

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

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

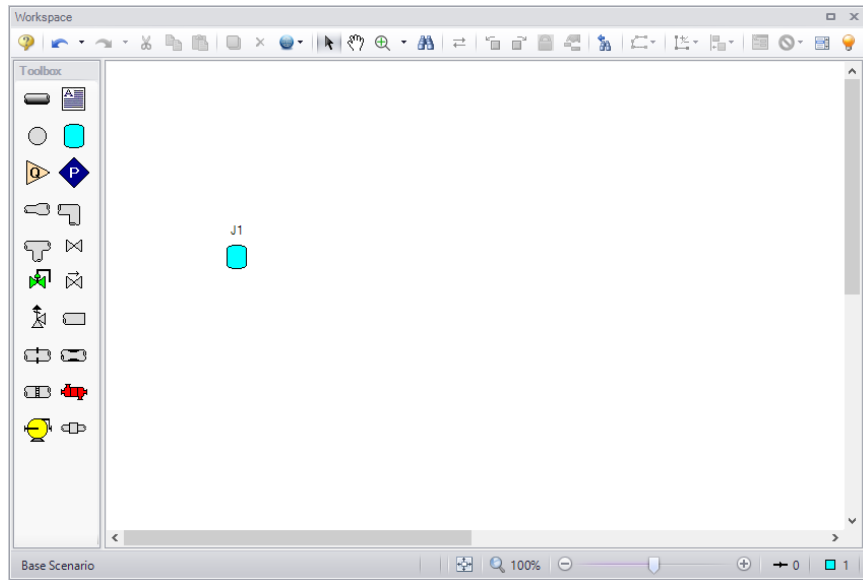
### Editing on the Workspace

You can move junctions on the Workspace to new locations. You can also cut, copy, paste, and duplicate junctions from the Edit menu or using keyboard shortcuts. Undo is available for most arrangement actions on the Workspace.

---

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

---

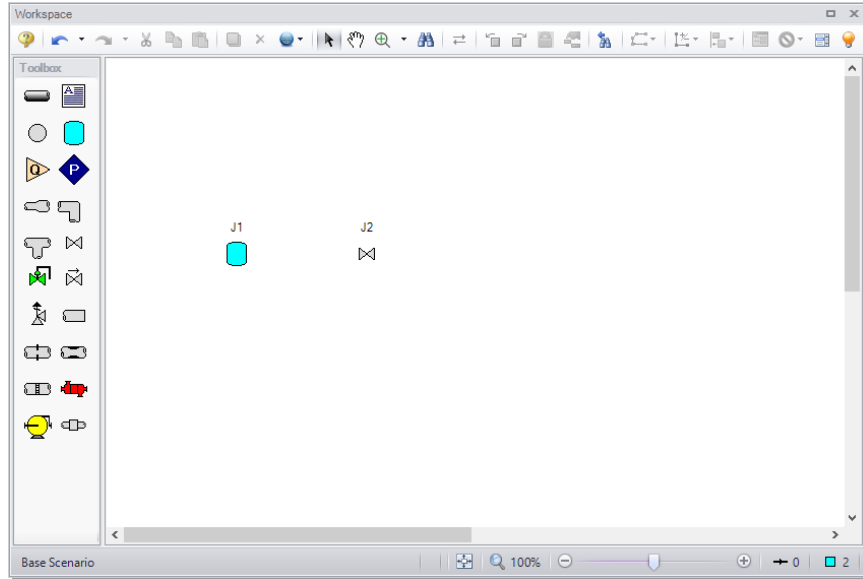


**Figure 4: Tank junction placed**

## II. Place the Valve junction

➤ **To add a Valve junction**, click and drag a valve from the Toolbox and place it on the Workspace as shown in Figure 5. The Valve will be assigned the default number "J2".

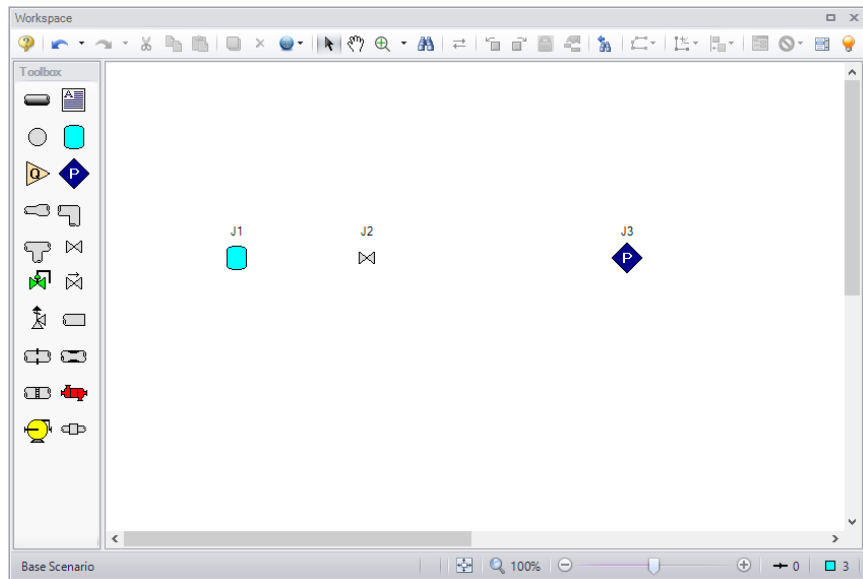




**Figure 5: Tank Blowdown model with Valve placed**

### III. Place an Assigned Pressure junction

➤ **To add an Assigned Pressure junction**, click and drag an Assigned Pressure junction from the Toolbox and place it on the Workspace as shown in Figure 6. The Assigned Pressure junction will be assigned the default number "J3".



**Figure 6: All junctions placed**

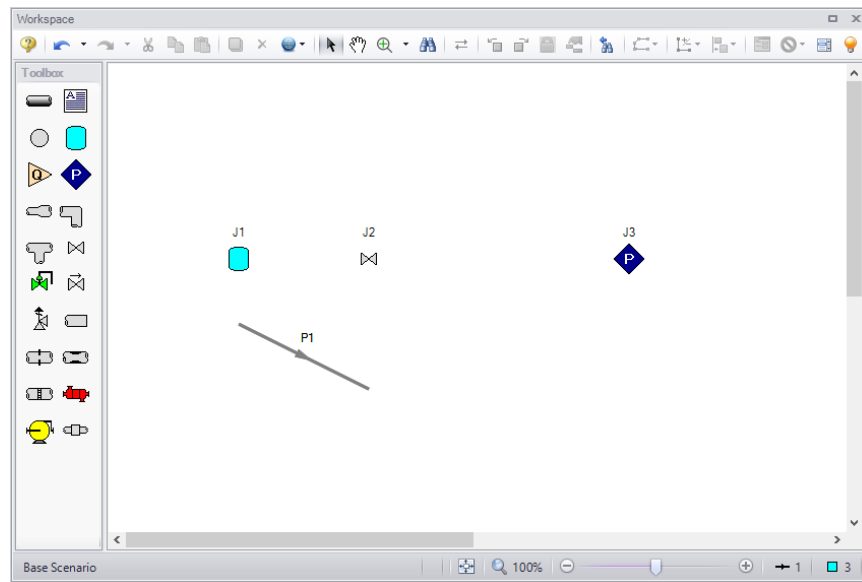
➤ **Before continuing, save the work you have done so far** by either choosing Save As... from the File menu, clicking the Save button on the Common Toolbar, or using the CTRL+S keyboard shortcut. Enter a file name (Tank Blowdown, perhaps) and AFT xStream will append the \*.xtr extension to the file name.

### IV. Draw a pipe between J1 and J2

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

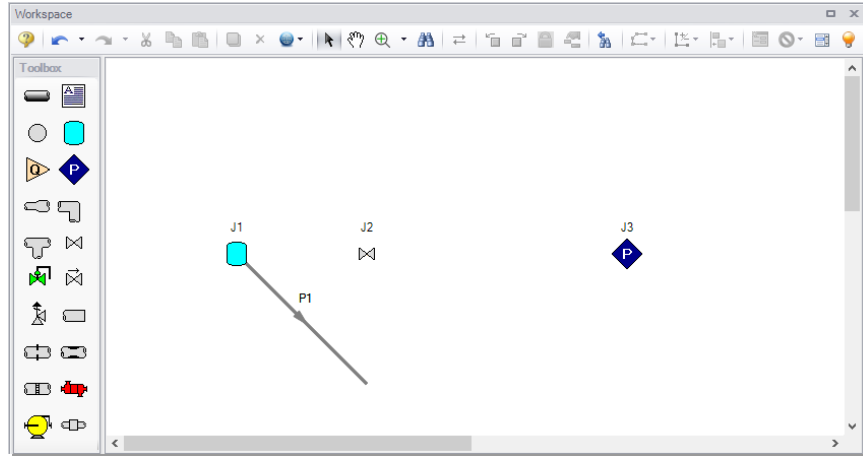
➤ **To create a pipe, click the Pipe Drawing Tool icon on the Toolbox.** The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe near the junctions by clicking and dragging the mouse, similar to that shown in Figure 7.

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

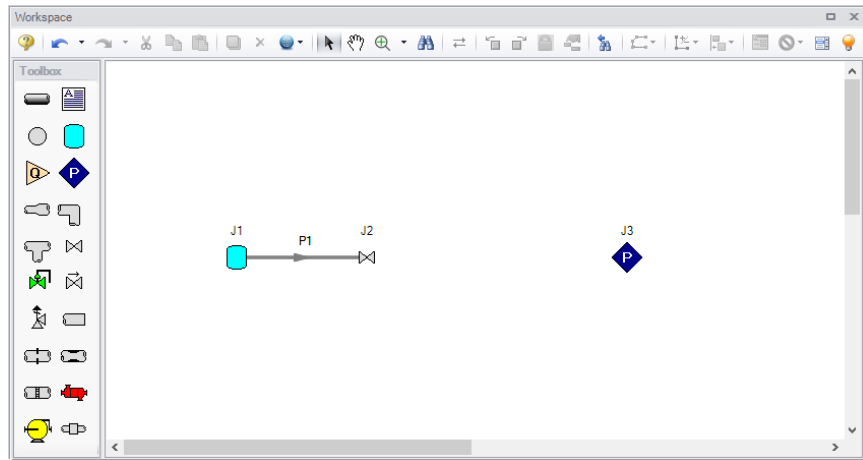


**Figure 7: First pipe drawn**

➤ **To place the pipe between J1 and J2,** use the mouse to grab the pipe in the center and drag it so that the left endpoint falls within the J1 Tank icon, then drop it there (see Figure 8). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J2 Valve icon (see Figure 9).



**Figure 8: First pipe partially connected**



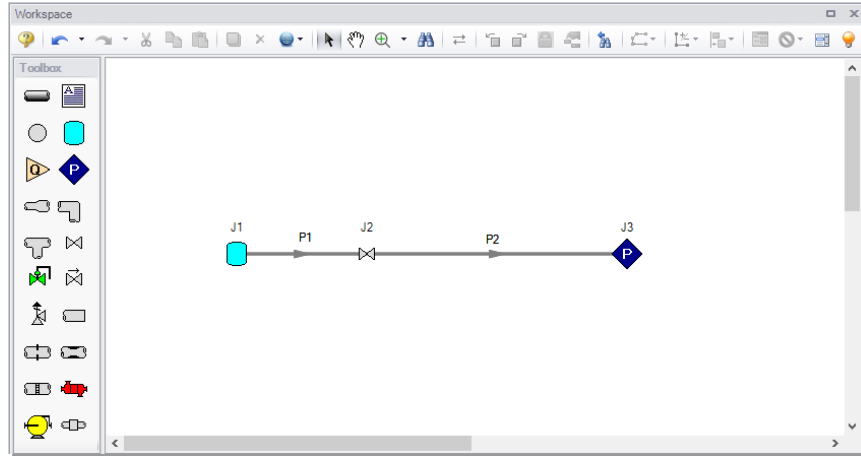
**Figure 9: First pipe fully connected**

### V. Add the remaining pipe

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 Valve, then press and hold the left mouse button. Stretch the pipe across to the J3 Assigned Pressure junction, then release the mouse button. All objects in the model should now be graphically connected as they are in Figure 10.

Save the model by selecting "Save" in the File menu, by clicking the "Save" icon on the Toolbar, or by entering CTRL+S.



**Figure 10: All pipes and junctions placed**

### **Reference positive flow direction**

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

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

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

---

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

---

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

---

## **A. Define the model components**

The model must have all pipes and junctions defined in order for it to run. This step encompasses the proper input data and connectivity for all pipes and junctions.

### **Object status**


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

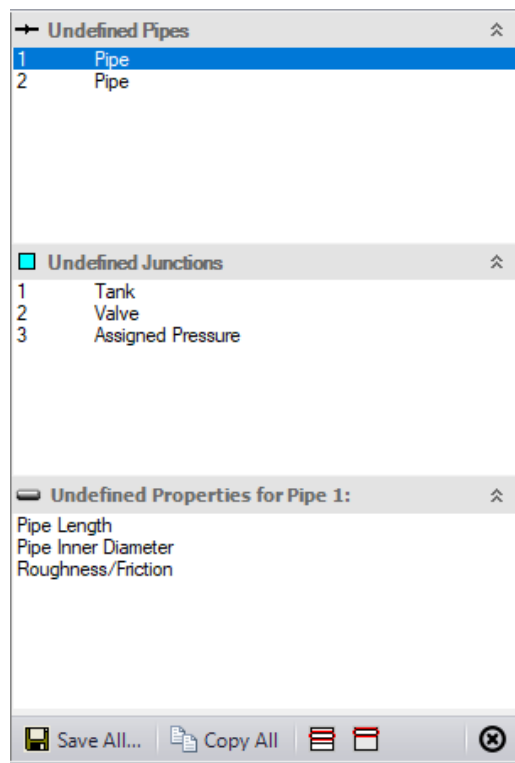
When Show Object Status is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. These colors are configurable through User Options from the Tools menu.

Because you have not yet defined the pipes and junctions in this model, all the object ID numbers will change to red when you turn on Show Object Status.

### **Showing undefined objects**

Another useful feature is the List Undefined Objects window (Figure 11). This can be opened from the View menu by clicking on List Undefined Objects, or by clicking the List Undefined Objects icon on the

Workspace toolbar . All objects with incomplete information are listed. Click on an undefined pipe or junction to display the property data that is missing. Click the close button on the bottom-right to stop showing this window.



**Figure 11: The List Undefined Objects window lets you see the undefined properties for each object**

## I. Enter data for the J1 Tank

➤ **To define the first tank**, open the J1 Tank Properties window (Figure 12) by double-clicking on the J1 junction.

---

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

---

➤ **Enter the following properties into the J1 Tank Properties window:**

1. Name = Supply Tank
2. Elevation = 0 meters
3. Tank Model = Finite
4. Pressure = 45 bar
5. Temperature = 150 deg. C
6. Fixed Volume = 8.5 meters<sup>3</sup>
7. Initial Conditions = Known
8. Thermodynamic Process = Adiabatic

---

**Note:** You can specify default units for many parameters (such as meters for length) in the User Options window, accessed from the Tools menu under Preferred Units.

---

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

Most junction types can be entered into a custom Junction 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 List in the junction's Properties window. The current junction will get the properties from the library component.

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

Tank Properties

Number: 1

Name: Supply Tank

Library Jct:

Copy Data From Jct...:

Elevation: 0 meters

OK Cancel Jump... Help

Tank Model Optional Design Alerts Notes Status

Tank Model

Infinite

Finite

Initial Conditions

Pressure: 45 bar

Temperature: 150 deg. C

Enthalpy:

Finite Tank Parameters

Fixed Volume: 8.5 meters3

Initial Conditions

Known

Calculate From Steady State

Thermodynamic Process

Adiabatic

Isothermal

Known Heat and Volume vs. Time

**Figure 12: Properties window for Tank J1**

### **Using the tabs in the Properties windows**

The information in the Properties windows is grouped into several categories and placed on separate tabs. Click the tab to bring its information forward. Figure 12 is an example of a Tank's Properties window.

The Tank Model tab is the tab that appears first when the Tank junction is opened and contains all of the inputs required to define the Tank in steady-state. While the Tank Model tab is specific to the Tank junction, the first tab opened for any junction will contain all inputs required for that junction to be defined.

The Optional tab allows you to set Special Conditions (like closing a valve or turning off a compressor), and change the junction icon.

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

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

The Status tab will state whether all required input data is present for a given junction. If not, it will list all inputs that are incomplete.

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

### The Inspection feature

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspection feature. Position the cursor on the Tank J1 junction and hold down the right mouse button. An information box appears, as shown in Figure 13.

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

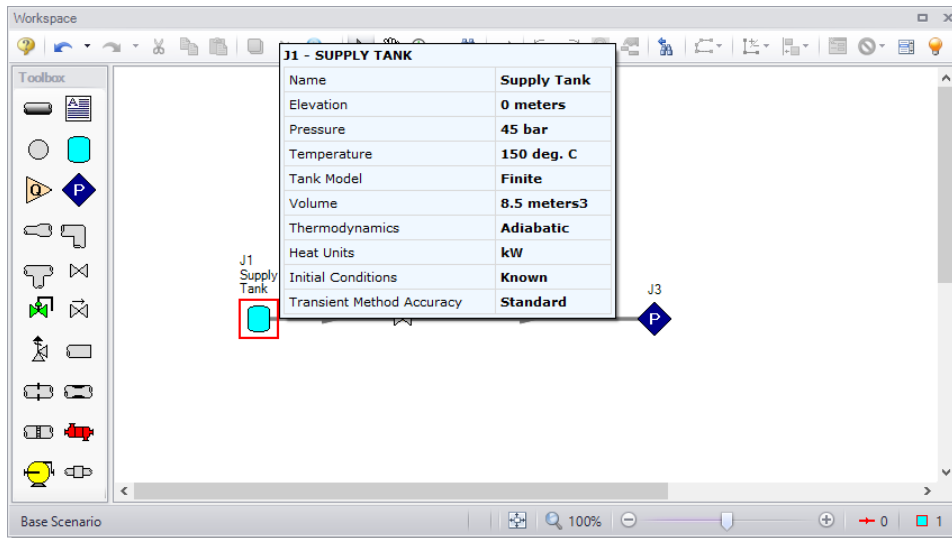


Figure 13: Inspect junction properties from the Workspace by clicking and holding the right mouse button

## II. Enter data for the J2 Valve

The Valve is the element which will cause the transient in this model. The purpose of the model will be to understand how the conditions within the supply tank and discharge pipe change after the valve opens.

➤ **Double-click the J2 junction icon** to open the Valve Properties window and enter the following data on the Loss Model tab (Figure 14).

1. Elevation = 0 meters (by default, the elevation will be set to the value specified in the last junction, and this behavior can be changed in the User Options > Parameters > Pipes and Junctions section)
2. Loss Source = User Specified
3. Loss Model = Kv
4. Kv = 52
5.  $xT = 0.7$

➤ **Click the Optional tab and set the Special Condition to Closed.** This will set the initial steady-state Kv equal to 0. The Kv value defined above will represent the final steady-state value after the transient



valve movement has completed.

**Figure 14: Input data for the J2 Valve**

➤ **Click the Transient tab** and enter the following data (Figure 15).

1. Initiation of Transient = Time (indicates that the valve closure will be initiated at the time you specify)
2. Enter the following Kv vs. Time data:

<b>Time (seconds)</b>	<b><math>K_v</math></b>	<b><math>xT</math></b>
0	0	0.7
0.1	52	0.7
240	52	0.7

The first data point of the table must match the steady-state value of the junction; therefore, the values are grayed out. In this case, the valve is closed in the initial steady-state due to the Special Condition specified in the previous step. Therefore, the initial transient data point automatically populates the valve as being closed ( $K_v$  of zero means the valve is closed). The valve then fully opens over a period of 0.1 seconds. It remains in this position for the rest of the simulation.

Valve Properties

Number: 2 Upstream Pipe: 1  
Name: Valve Downstream Pipe: 2  
Library Jct.: Elevation: 0 meters  
Copy Data From Jct.:

Loss Model  Transient  Optional Design Alerts Notes Status

Transient Special Condition  
 None  
 Ignore Transient Data

Initiation of Transient  
 Time  Dual Event Cyclic  
 Single Event  Dual Event Sequential

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

Data Point	Time (seconds)	Kv	Xt
1	0	0	0.7
2	0.1	52	0.7
3	240	52	0.7
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			

Edit Table Show Graph...  
 Repeat Transient  
Transient

OK Cancel Jump... Help

Figure 15: Transient input data for the J2 Valve

➤ Click the **OK** button to save and close the **Valve Properties** window. There should now be a “T” symbol next to the Valve junction on the Workspace, indicating that transient data is entered for the junction. The Valve junction on the Workspace will additionally have a red “X” next to it indicating that the junction contains a Special Condition.

### III. Enter data for the J3 Assigned Pressure

➤ **Double-click the J3 junction icon** to open the Assigned Pressure Properties window and enter the following data (Figure 16).

1. Elevation = 0 meters
2. Boundary Specification = Stagnation
3. Stagnation Pressure = 1 bar
4. Temperature = 21.1 deg. C
5. Click the OK button to save and close the Assigned Pressure Properties window

Assigned Pressure Properties

Number:

Name:

Library Jct:

Copy Data From Jct...:

Elevation:  meters

OK Cancel Jump... Help

Pressure Model Loss Coefficient Transient Optional Design Alerts Notes Status

Boundary Specification

Stagnation  Static

Stagnation Pressure:  bar

Stagnation  Temperature  Enthalpy  deg. C

**Figure 16: Input data for the J3 Assigned Pressure**

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

#### IV. Enter data for Pipe P1

Data for pipes and junctions can be entered in any order. In this example, the junctions were done first. The next step is to define all of the pipes.

➤ **Open the Pipe Properties window for Pipe P1.** The Pipe Model tab opens by default (Figure 17).

1. Pipe Material = Steel - ANSI
2. Size = 2 inch
3. Type = STD (schedule 40)
4. Length = 6.1 meters
5. Friction Model = Standard

The screenshot shows the 'Pipe Properties' dialog box for 'Pipe P1'. The 'Pipe Model' tab is selected. The 'Number' field is set to 1, and the 'Name' is 'Pipe'. The 'Upstream Junction' is 1 and the 'Downstream Junction' is 2. The 'Pipe Material' is 'Steel - ANSI', 'Size' is '2 inch', 'Type' is 'STD (schedule 40)', and 'Inner Diameter' is '5.250179 cm'. The 'Length' is '6.1 meters'. The 'Friction Model' is set to 'Standard' with an 'Absolute Roughness' of '0.004572 cm'. Buttons for 'OK', 'Cancel', 'Jump...', and 'Help' are visible in the top right corner.

**Figure 17: Properties window for Pipe P1**

### **The Pipe Properties window**

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

To Inspect a junction connected to a given pipe, position the mouse pointer on the connected junction's ID number in that pipe's Properties window (located at the top right of the Pipe Properties window) and hold down the right mouse button. This process can be repeated for any junction that states the upstream and downstream pipe in the junction's Properties window. You can jump to that junction's Properties window by double-clicking the connected junction's ID number, or use the Jump button to jump to any other part of the model.

➤ **Click OK to save and close the Pipe Properties window for Pipe P1.**

## **V. Enter data for Pipe P2**

➤ **Open the Pipe Properties window for Pipe P2 and enter the following data:**

1. Pipe Material = Steel - ANSI
2. Size = 2 inch
3. Type = STD (schedule 40)
4. Length = 457 meters
5. Friction Model = Standard
6. Click OK to save and close the Pipe P2 Properties window.

➤ **Check if all the pipes and junctions are defined** by clicking on the List Undefined Objects button. If all data is entered, no items should appear in the Undefined Objects window. If items are present, open the corresponding Properties windows for the incomplete items from the Workspace. The Status tab on each Properties window will indicate what information is missing.

## B. Review model data

➤ **Save the model.** 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 18. To change to this window, you can click on the Model Data tab, select it from the Window menu, or use the CTRL+M keyboard shortcut. The Model Data window gives you a table-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to a PDF, or printed out for review. Figure 19 shows an expanded view of the Transient Data tab from Figure 18. Here, all transient input data for the model is shown.

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

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

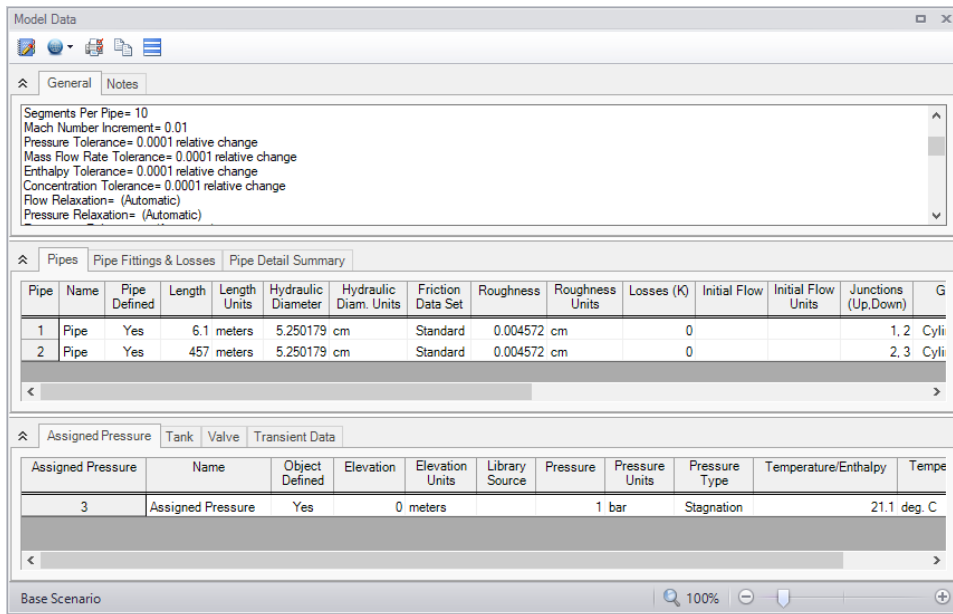
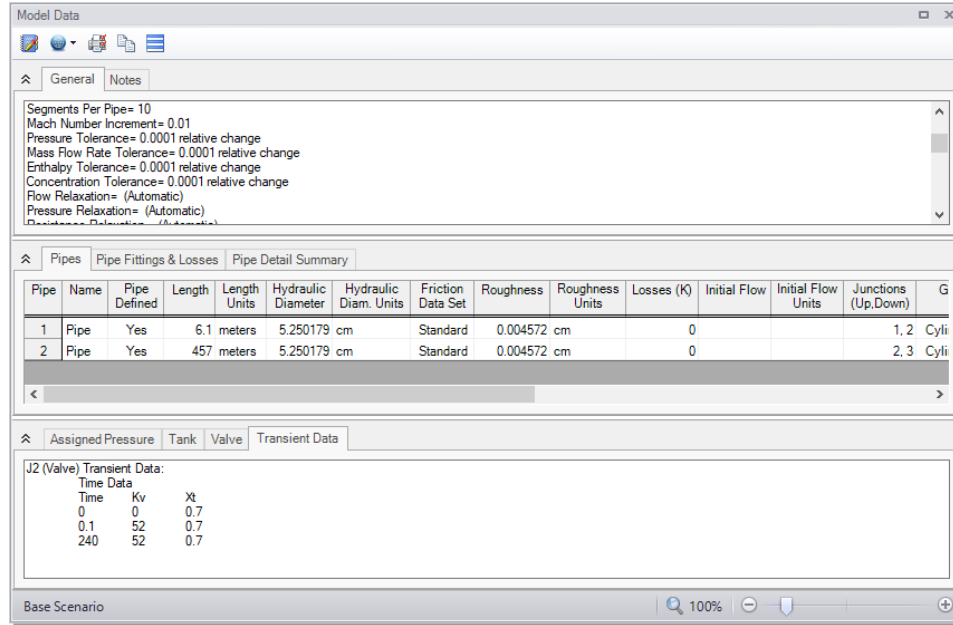


Figure 18: The Model Data window displays all input in table form



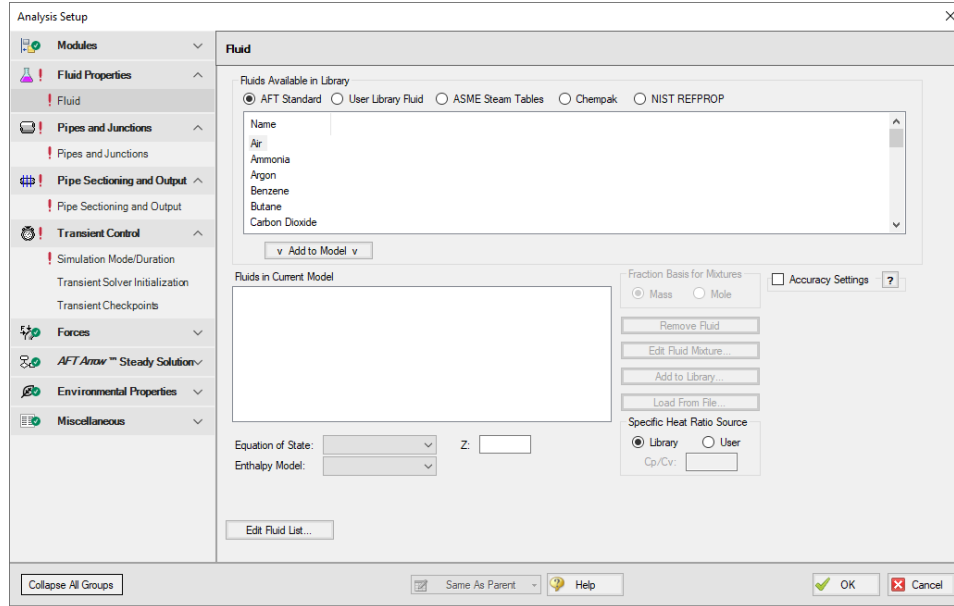
**Figure 19: The Transient data tab in the Model Data window displays transient input data**

### Step 3. Specify Analysis Setup

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

➤ **Click Analysis Setup on the Main Toolbar** to open the Analysis Setup window (see Figure 20). The Analysis Setup window contains various *groups* depending on which modules you may be using. Each group has one or more *items*. A fully defined group has every item within it defined with all required inputs. Complete every group to run the Solver. A group or item that needs more information will have a red exclamation mark to the left of the group or item name. After completing a group, the group icon shows a green circle with a checkmark. After defining all groups, thus completing the Analysis Setup, the Model Status light in the lower-right corner of the Workspace turns from red to green.

The Analysis Setup window can also be opened by clicking on the Model Status Light on the Status Bar at the bottom-right corner of the AFT xStream window (Figure 3).



**Figure 20:** The Analysis Setup window tracks the defined and undefined model inputs

### A. Modules

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 20. Here you can enable and disable the Pulsation Frequency Analysis (PFA) Add-on Module if it is available on the license. By default, the model will have all modules disabled. Make no changes to this panel for this example.

### B. Fluid Properties

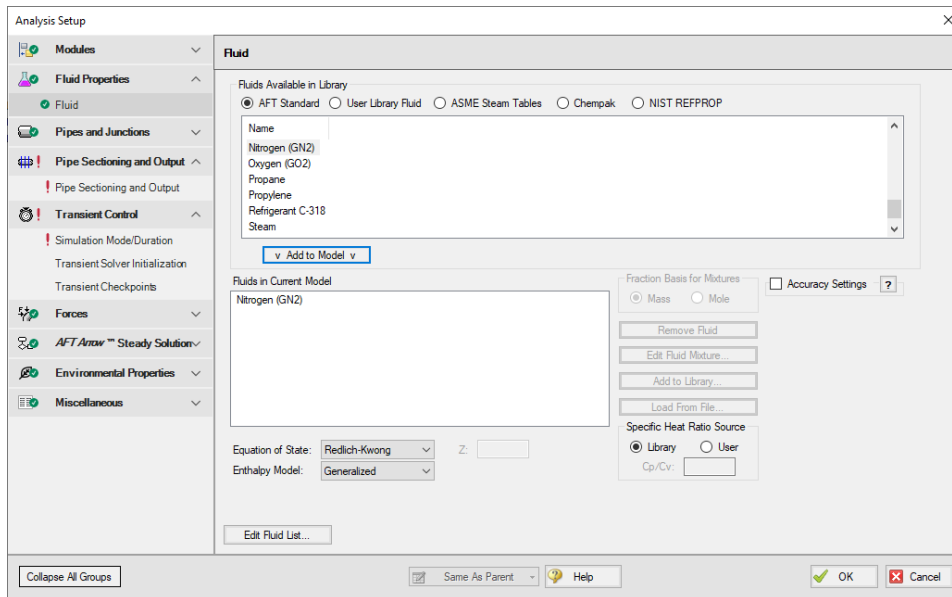
➤ **Select the Fluid item** to open the Fluid panel (Figure 21).

This panel allows you to specify the fluid used in the model. There are several fluid libraries that are available:

- **AFT Standard** - Common fluids from various industry standard sources have been curve fit to optimize calculation speed.
- **User Library Fluid** - If the desired working fluid is not available via the other sources, custom fluid properties can be entered and saved to the User Library for repeated use in the future. Custom fluids are created by opening the Library Manager from the Library menu or by clicking the Edit Fluid List button in the Fluid panel and clicking Add New Fluid.
- **ASME Steam Tables** - Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation (1997).
- **Chempak** - Optional add-on for purchase which contains data for several hundred fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.

- **NIST REFPROP** - The Reference Fluid Thermodynamic and Transport Properties (REFPROP) Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via Equations of State. REFPROP supports user-specified fluid mixtures.

➤ **Select the AFT Standard fluid option**, then choose Nitrogen (GN2) from the list and click the Add to Model button. Leave the Equation of State as Redlich-Kwong and the Enthalpy Model as Generalized.



**Figure 21: Fluid panel of Analysis Setup**

### C. Pipes and Junctions

The Pipes and Junctions panel is used to show the status of the pipes and junctions on the Workspace. This item is incomplete if there are any undefined pipes/junctions on the Workspace, or if there are no pipes/junctions on the Workspace. In this example the objects in the Workspace are fully defined, so this item is marked as complete.

#### **Running models in steady-state**

After fully specifying the Fluid panel and the pipes and junctions, sufficient information exists to run the model in steady-state. You can run the model in steady-state by selecting the Simulation Mode/Duration panel from the Transient Control group and then choosing AFT Arrow™ Steady for the Time Simulation. If you do this, the other inputs on the Simulation Mode/Duration panel and the Pipe Sectioning and Output panel will no longer be required, and the model can be run.

In general, it is a good idea to always run your model in steady-state before running the full transient analysis to make sure the model is giving reasonable results. However, in this case the model has no flow in the steady-state due to the closed valve, and a steady-state run is trivial. Keep the Transient option selected.



## D. Pipe Sectioning and Output

The next panel is the Pipe Sectioning and Output panel in the corresponding group. The Pipe Sectioning and Output panel divides pipes into computation sections in a manner which is consistent with the Method of Characteristics (MOC).

To satisfy the MOC, the following equation must be valid:

$$\Delta t_{min} = \frac{\Delta x}{2 c_{T,max}}$$

where  $\Delta t$  is the time step,  $\Delta x$  is the section length, and  $c$  is the sonic velocity.

### ➤ Enter 1 for the Minimum Number of Sections Per Pipe.

The Minimum Number of Sections Per Pipe determines the section length by taking the quotient of the shortest pipe's length and the Minimum Number of Sections. Increasing the number of sections in a model will always have a positive correlation to the model accuracy as the error inherent to the Method of Characteristics decreases when the step size is made smaller. However, the run time of the model increases quadratically as well, since each added section to the shortest pipe increases the number of calculations for each time step. This increase in run time is exacerbated when there is a large disparity between the minimum pipe length and the length of the longest pipe, such as the case in this model.

The Minimum Number of Sections Per Pipe for this example will be set to 1 for expediency, but note that it would be wise to use more computational sections if you are running an AFT xStream model for real projects. Typically, less sections and faster run time is useful during the initial model development and troubleshooting, and more sections can be used for final runs.

### ➤ Define the Estimated Maximum Pipe Temperature During Transient as 930 deg. C.

The Estimated Maximum Pipe Temperature During Transient is used to calculate the maximum sonic velocity. Since all pipes in the network must be solved together, the time step must be the same for each pipe. However, the sonic velocity will not be equivalent for all of the pipes. Therefore, the sonic velocity at the specified maximum temperature is used to find a uniform time step for the system. Without a uniform time step, the MOC could fail. Choosing a maximum temperature that is much higher or much lower than the maximum temperature calculated during the run can result in additional uncertainty in the MOC solution, as is discussed in the [Pipe Sectioning - Introduction to Method of Characteristics](#) help topic. It is generally recommended to choose a conservatively high temperature for the model initially. The results of the initial run can then be used to refine the Estimated Maximum Temperature During Transient for subsequent runs.

The Estimated Maximum Pipe Mach Number During Transient is an additional parameter that affects result accuracy. The value of this parameter is set to 1 by default so that the Method of Characteristics grid is able to bound the sonic velocity obtained at the Estimated Maximum Pipe Temperature During Transient. Setting the Mach number to 1 ensures that the model will not fail assuming the input values for that variable are correct.

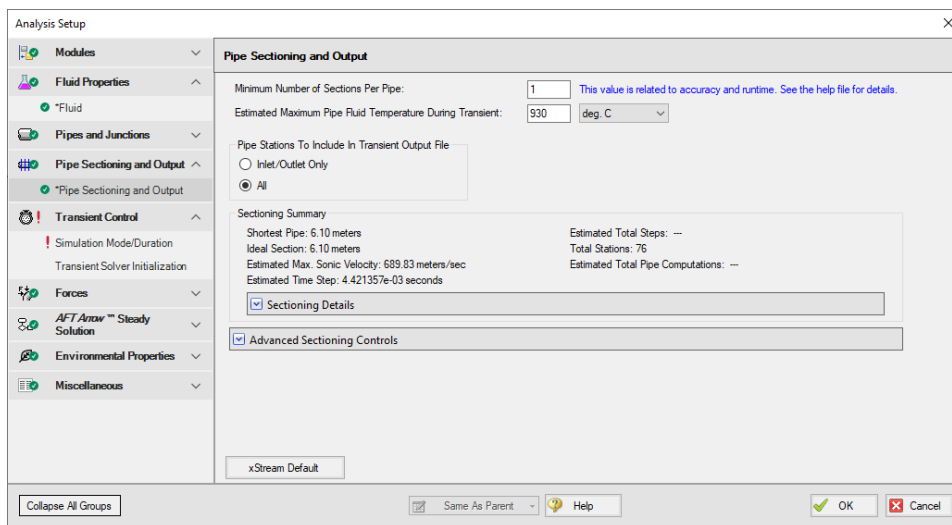
However, because of the computational methodology of the MOC, using the default Mach number can introduce error due to the difference between the Estimated Maximum Pipe Mach Number During Transient and the actual maximum. This error is particularly noticeable when the Mach number during the

simulation is near zero. If the maximum Mach number reached during the simulation is near zero, a more accurate model can be found by lowering the estimated maximum based on the actual Maximum Mach number of the system.

To change the Estimated Maximum Pipe Mach Number During Transient, you must expand the Advanced Sectioning Controls dropdown in the Pipe Sectioning and Output panel of Analysis Setup. It is highly recommended that you first run the model with the default Mach number in order to ensure that choking does not occur in the pipes before altering this number. Additionally, be cognizant of the fact that altering sectioning, transient components, or maximum estimated temperatures between runs could have an effect on the maximum Mach number reached during the simulation. As such, it is generally recommended to only change this number once your model has reached a more finalized state.

Sonic choking will be seen in this model, so no adjustment to Maximum Mach Number will be made.

The Sectioning Summary underneath the inputs will be partially filled in. Certain properties such as the Estimated Total Steps and Total Pipe Computations will need the Simulation Mode/Duration panel to be completed before values can be displayed.

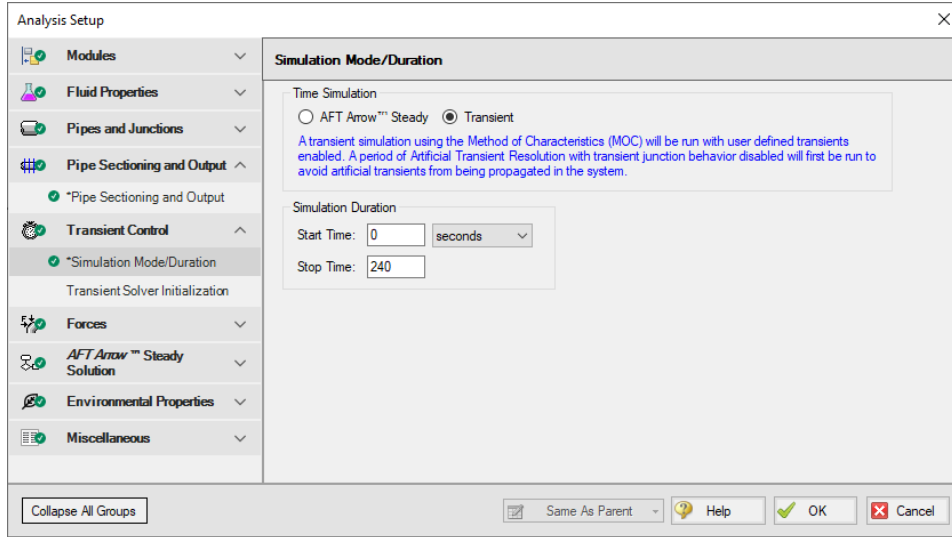


**Figure 22:** The Sectioning panel in the Analysis Setup window is used to calculate the time step of the model

## E. Transient Control

➤ **Select the Simulation Mode/Duration panel.** Enter 0 seconds for the Start Time and 240 seconds for the Stop Time (Figure 23).

➤ **Click OK to accept the changes in Analysis Setup.**



**Figure 23:** The Simulation Mode/Duration panel is used to specify the simulation type and the time span for the transient

➤ Save the model.

### Step 4. Run the model

➤ Choose **Run Model...** from the **Analysis** menu or click the Run Model button on the Common Toolbar. During execution, the Solution Progress window is displayed and can be used to pause or cancel the Solver's activity (Figure 24).

This simple model will take several minutes to run. A typical xStream model may take hours to solve depending on the simulation duration, sectioning, and which fluid library is used.

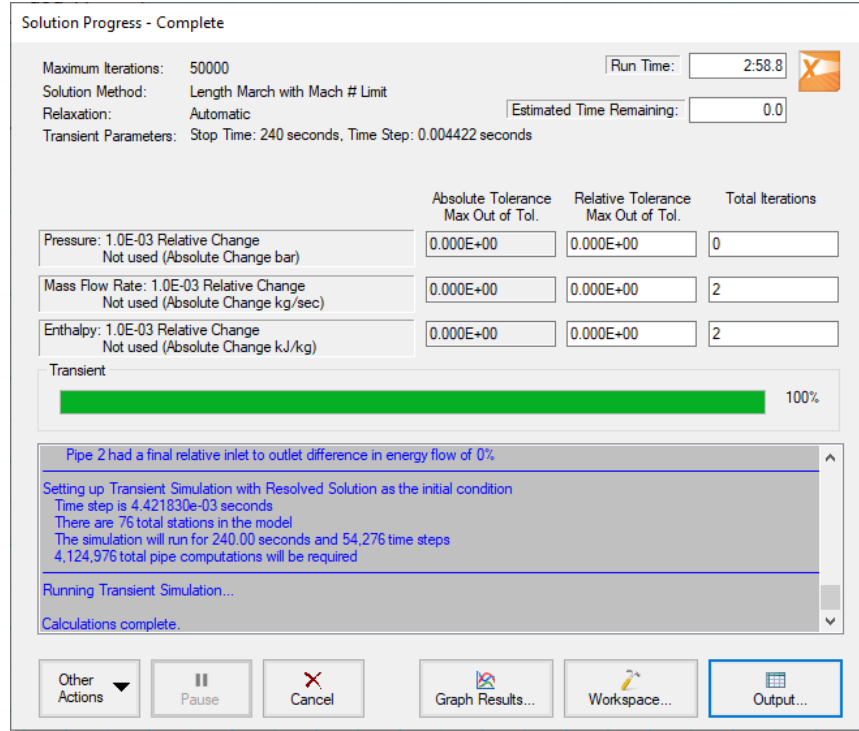


Figure 24: The Solution Progress window displays the state of the simulation

**The two Solvers**

AFT xStream has two Solvers. The first is called the AFT Arrow™ Steady Solver, which as its name suggests, obtains a steady-state solution to the pipe network. The second Solver is called the Transient Solver and solves the gas transient equations via the Method of Characteristics (MOC).

The AFT Arrow™ Steady Solver and Transient Solver use different sectioning methods which causes small discrepancies between the converged AFT Arrow™ Steady solution and the first time step of the transient solution. AFT xStream runs the Transient Solver for a period of time before the transient simulation begins in order to allow these artificial waves caused by these discrepancies to dampen and die out. Once the artificial transients have settled, AFT xStream checks the solution with resolved artificial transients against the AFT Arrow™ Steady solution. If the mismatch between the beginning and end of the artificial transient resolution process is sufficiently small AFT xStream proceeds with the transient simulation. See the chart in Figure 25 as a visual representation of the process described above.

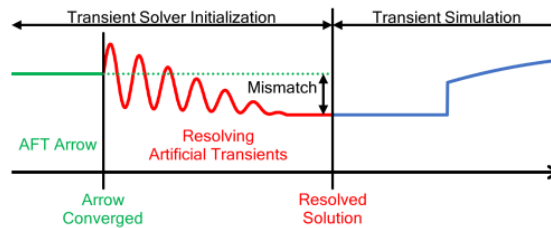


Figure 25: Illustration of the Transient Solver Initialization process with artificial transient resolution

➤ **When the solution is obtained, click the Output button** to display the table-based Output window. The information in the Output window can be reviewed visually on the screen, saved to a file, exported to a spreadsheet-ready format, copied to the clipboard, or printed.

### ***The output file***

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

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

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

## **Step 5. Review the Output window**

---

The Output window (Figure 26) 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. The items displayed in the tabs can be customized with the Output Control window, which can be opened by clicking the spreadsheet with a pencil icon from the Common Toolbar or from the Tools dropdown menu.

The General section will open by default to the Warnings tab if any Warnings or Cautions were encountered during the simulation. This model should generate a Caution that the Maximum Transient Pipe Mach Number is at least 0.2 higher than the steady pipe Mach number, which occurs since there is no flow in the system at steady-state. When this Caution is present, the effect of pipe sectioning on results can be significant. The impact of pipe sectioning will be addressed in greater detail in a later step.

The Output window allows you to review both the steady-state and transient results. The Pipes tab, All Junctions tab, and any specific junction tabs in the Junction section (such as Tank, Valve, etc.) show the steady-state results. A summary of the maximum and minimum transient results for each pipe is given on the Transient Max/Min tab in the Pipe section and is the active tab displayed in Figure 26. You can review the solutions for each time step (i.e., a time history) on the Transient Output tab shown in Figure 27. Note that in order to display all pipe stations the model will need to be set to save data for all stations. By default, xStream saves all data points. This can be changed in the Pipe Sectioning and Output panel.

## Beginner - Tank Blowdown Example

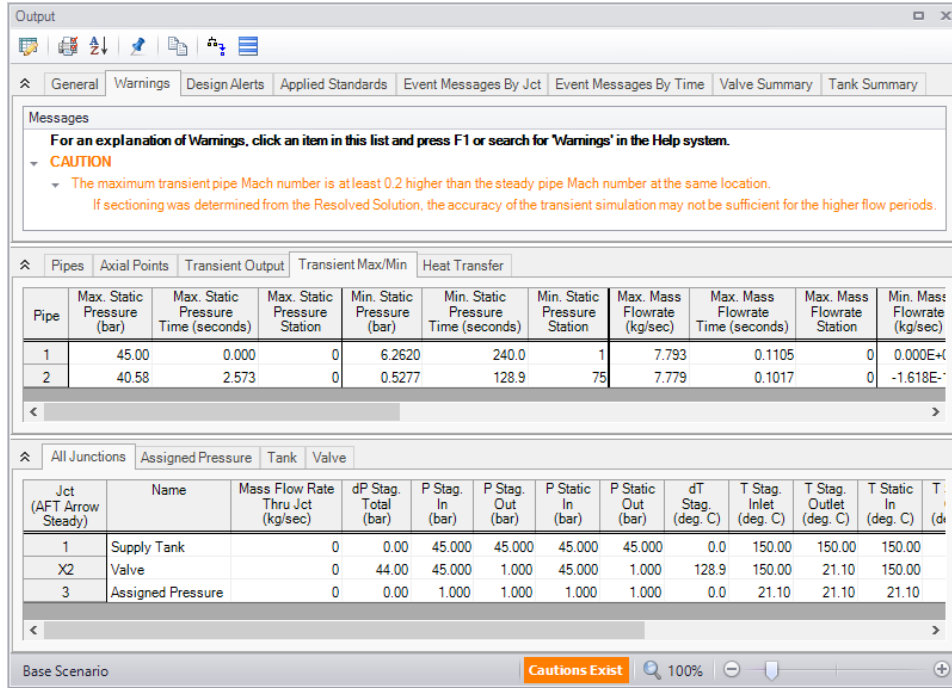


Figure 26: The Output window displays steady-state and transient output in tabular form

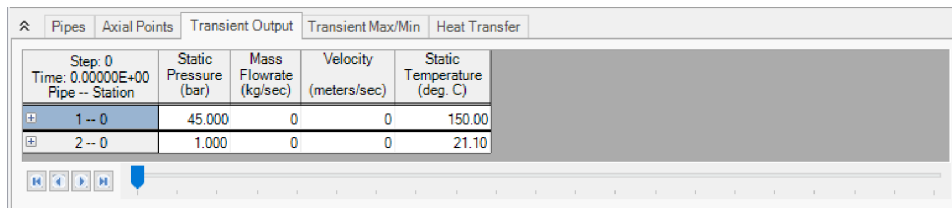


Figure 27: The Output window displays transient data from each time step

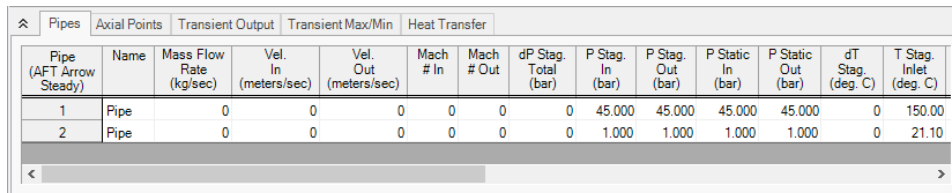
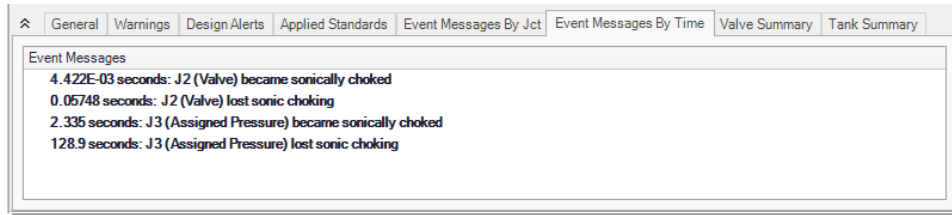


Figure 28: The Output window displays the values of steady-state parameters in the Pipes tab

➤ **Click on the Event Messages By Time tab.** This tab allows you to review changes of states at junctions, such as the onset of choking or the initiation of transient events (see Figure 29). There should be messages indicating that the J2 Valve experienced a near instantaneous choking that dissipated after about 0.06 seconds while the J3 Assigned Pressure experienced choking after 2.3 seconds that lasted until approximately 128.9 seconds. Choking is an important phenomenon to pay attention to as it indic-

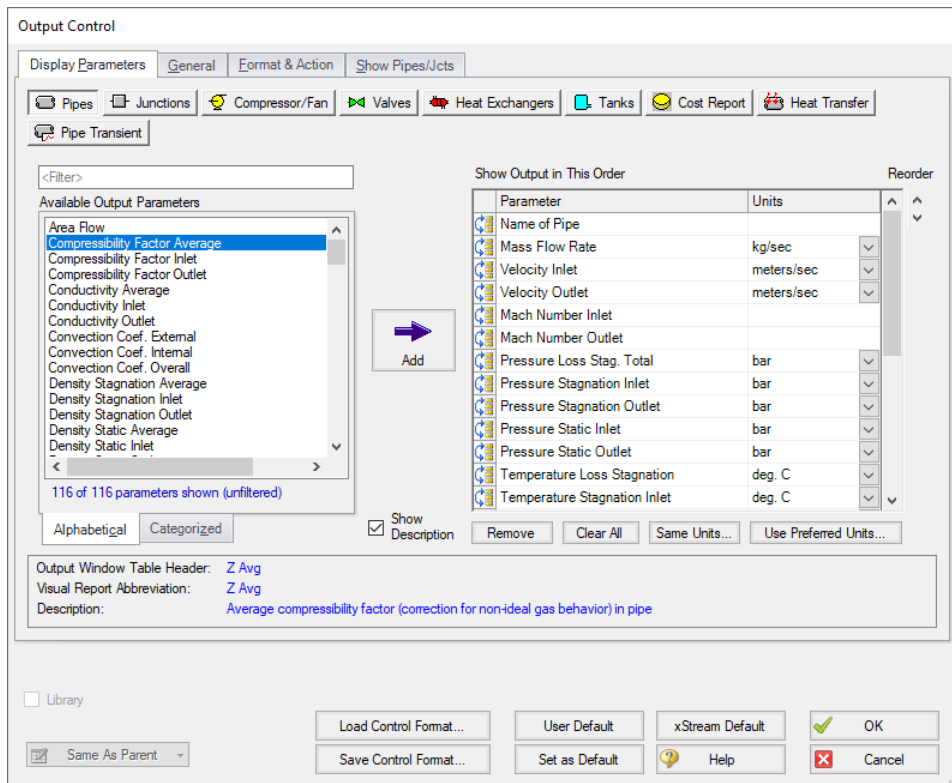
## Beginner - Tank Blowdown Example

ates that the fluid has reached sonic velocity. This has significant implications for the MOC calculation method as will be discussed later.



**Figure 29: Event Messages by Time tab displays important system behavior**

The Output Control window (Figure 30) can be accessed from either the Tools menu or from the Output window Toolbar, and it allows you to select the specific output parameters you want to display in the Output window. You can choose the units and reorder the parameters as desired. If you do not change any of the Output Control settings, the default Output Control parameters are assigned.



**Figure 30: The Output Control window lets you customize the output**

Units for each column in the Output window can also be changed by double-clicking the column header. This will open a window in which you can select the units you prefer (Figure 31). These changes are extended to the Output Control parameter data that is set.

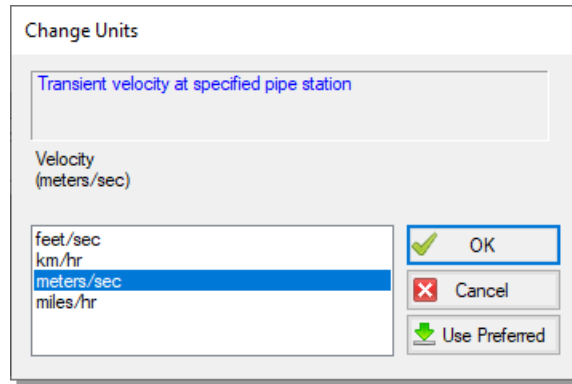


Figure 31: The Change Units window is opened from the Output window tabs by double-clicking a column header

### Step 6. View the Graph Results

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

➤ **Open the Graph Results window** by clicking the Graph Results tab, choosing it from the Windows menu, or by pressing CTRL+G.

Figure 32 shows the Graph Results window with the Graph Control tab selected on the Quick Access Panel.

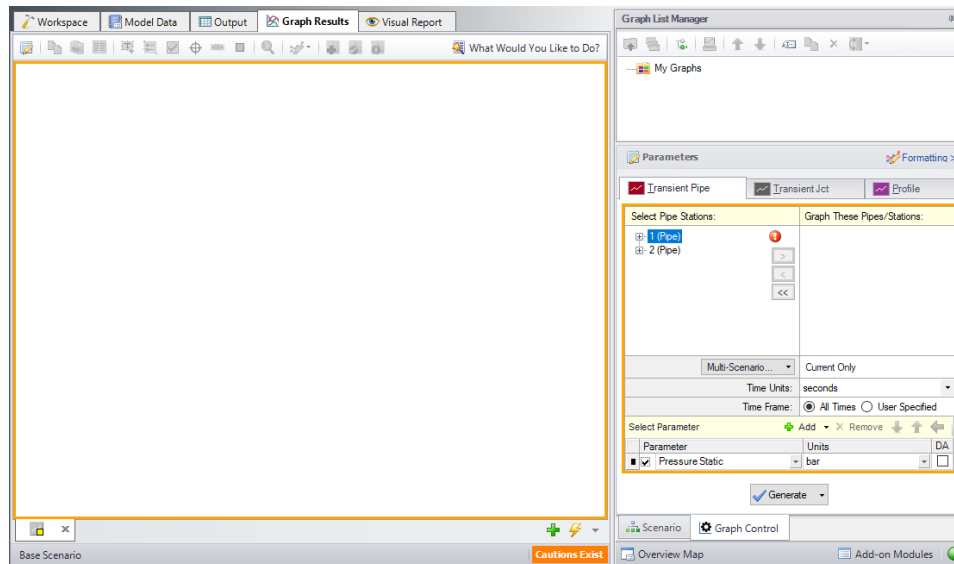


Figure 32: The Graph Results window and Quick Access Panel Graph Control tab is where various system parameters (both steady-state and transient) can be graphed



## A. Create a Transient Pipe graph

AFT xStream gives you the ability to create *stacked graphs*. These are graphs that are vertically aligned such that they use the same x-axis but have different parameters on the y-axis of each graph. This feature is helpful when you want to look at how different parameters vary with time at a given position without having to create separate graphs.

➤ **Access the graphing parameters by opening the Graph Control tab on the Quick Access Panel (this tab is automatically selected when the Graph Results window is opened).** Because we are interested in seeing how the pressures and flows in specific pipe sections respond over time, ensure that the Transient Pipe tab is selected in the Parameters/Formatting area on the Quick Access Panel. Alternatively, you can open the Select Graph Parameters window by clicking on the corresponding icon on the Graph Results Toolbar (Figure 32).

➤ **Add the Pipe P2 Outlet to the Graph These Pipes/Stations list** by expanding the tree for Pipe P2 under the Select Pipe Stations list and double-clicking the Outlet station. Pipe P2 Outlet represents the pipe computing station at the discharge of the blowdown pipe. Alternatively, you can click on the right arrow button after selecting the pipe station you want to graph to add it to the Graph These Pipes/Stations list.

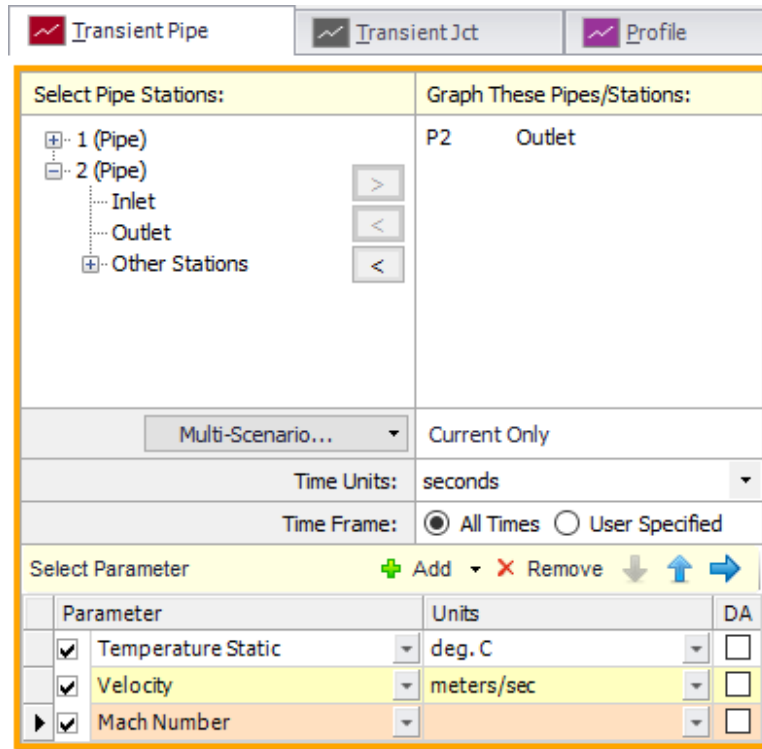
➤ **Verify the Time Units are set to seconds and the Time Frame is set to All Times.**

➤ **Click the Add button twice**, which is the green “+” icon next to Select Parameter. Two new rows will appear under the Parameter definition area which will create stacked graphs (Figure 33).

➤ **Select the following parameters from the dropdown lists on each row:**

1. Temperature Static with units of deg. C
2. Velocity with units of meters/s
3. Mach Number

Figure 33 shows the input in the Parameters/Formatting area on the Quick Access Panel.



**Figure 33: The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area**

- **Click the Generate button at the bottom of the Quick Access Panel** to create the static temperature, velocity, and Mach number graphs at the pipe discharge over the duration of the simulation.
- **Format the legend font size to 20 pt** by right-clicking on any legend and using the scroll bar. Drag the legend to the top-right corner of the graph.
- **Format the y-axis and x-axis font size to 20 pt** by right-clicking on each axis parameter and using the scroll bar.

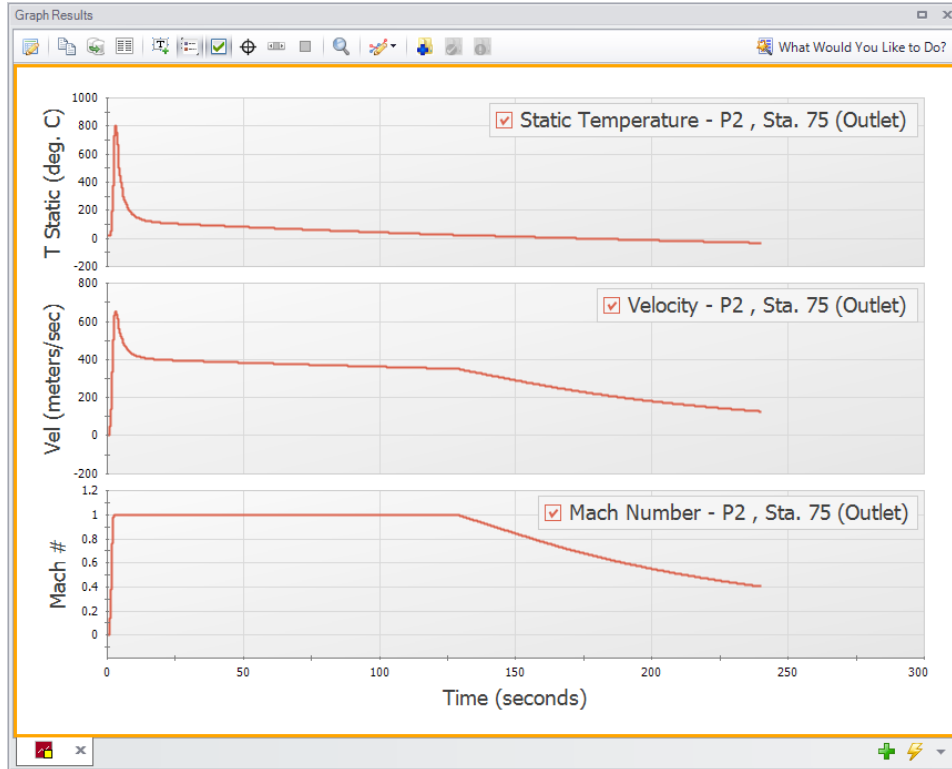
Figure 34 shows the stacked graphs detailing the static temperature, velocity, and the Mach number at Pipe P2 outlet over the course of 240 seconds. Here you can see that the maximum temperature at the discharge is about 800 deg. C. Also note that the maximum temperature in Figure 34 does not approach the value indicated in the Transient Max/Min tab. This is because the Transient Pipe graph shows only the outlet station (station 75), while the maximum temperature in Pipe P2 is reached at station 66.

---

**Note:** The axis titles in the stacked graphs in Figure 34 have been modified for clarity.

---

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



**Figure 34: Static temperature, velocity, and Mach number of Pipe P2 outlet**

Further review of the graph results in Figure 34 shows that sonic choking occurs after several seconds, as was seen on the Event Messages tab in the Output window, and continues for the next 130 seconds. Also note that the temperature, velocity, and Mach number all rise sharply shortly after the valve opens, with the Mach number rapidly reaching one. After the initial rise, the temperature and velocity both decay while the Mach number remains at one. The velocity drops because sonic velocity has a positive correlation with temperature, so as the temperature decays, the velocity required for choked flow also decreases. This relationship is important to understand when looking at systems that encounter choking.

### B. Create a Transient Junction graph

For the purpose of this analysis, it is also helpful to look at the temperature and pressure of the tank over time.

- **Create a new graph tab** by clicking the New Tab button which is the green plus icon located on the bottom-right, immediately below the graph area (Figure 32).
- **Click the Transient Jct tab** on the Graph Control tab of the Quick Access Panel (Figure 35). Add the J1 (Supply Tank) junction.
- **Verify the Time Units are set to seconds and the Time Frame is set to All Times.**
- **Click the Add button** to add a new row under the Parameter definition area.
- **Select the following parameters from the dropdown lists on each row:**

## Beginner - Tank Blowdown Example

1. Pressure Stagnation with units of bar
2. Temperature Stagnation with units of deg. C

➤ **Click the Generate button.** These graphs show the tank pressure and temperature initially decrease quickly, then decrease more slowly as time progresses. This change in slope is due to a decreasing pressure gradient between the tank and the atmosphere as the tank is emptied, which slows the mass flow rate out of the tank (Figure 36).

**Note:** AFT xStream assumes all pipes are adiabatic (perfectly insulated). Over a 240 second run time this assumption may not be accurate. Consider this when evaluating simulation results over longer times.

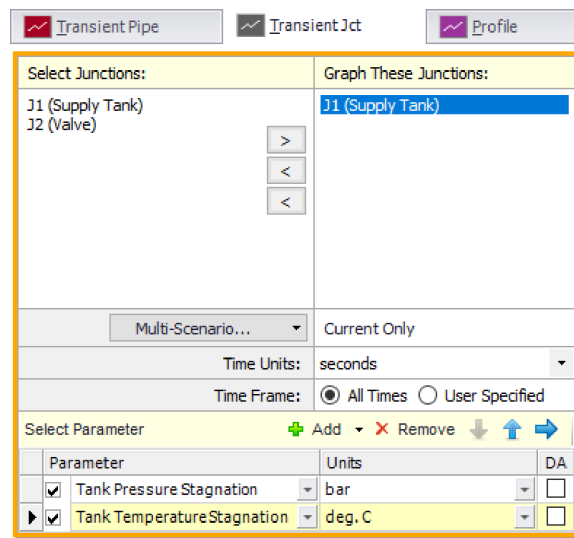
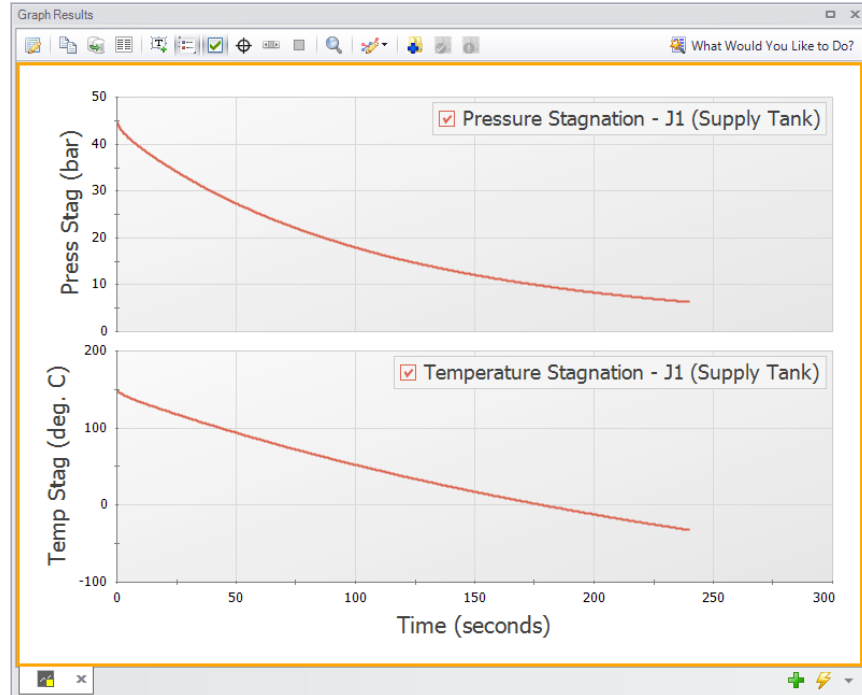


Figure 35: Graph Parameters window for the Transient Junction graph



**Figure 36: The Transient Junction graph shows how variables within a junction change over time, such as with the temperature and pressure of Tank J1**

This junction graph could be used to determine when certain conditions within the tank were met. For instance, suppose there was a need to know when the tank pressure dropped below 25 bar. The junction graph would indicate the tank pressure would reach such a point around 60 seconds after beginning the tank blowdown.

### C. Create an Animated Profile graph

Lastly, we will animate the profile of the system to show the wave caused by opening the valve.

➤ **Create a new graph tab and select the Profile tab.** Under Plot Single Path, select Pipe P1 and P2. After ensuring your Length Units are meters, select the option to Animate Using Output and ensure the Animation Time Units are set to seconds.

➤ **Add new parameters until you have 4 in total, and set them to the following (see Figure 37):**

1. Pressure Static with units of bar
2. Temperature Static with units of deg. C
3. Velocity with units of meters/sec
4. Density Static with units of kg/m<sup>3</sup>

➤ **Click the Generate button.** This will show the max and min envelopes of the parameters throughout the selected time frame. The animation can be started by clicking Play in the top-left playback controls (Figure 38).

## Beginner - Tank Blowdown Example

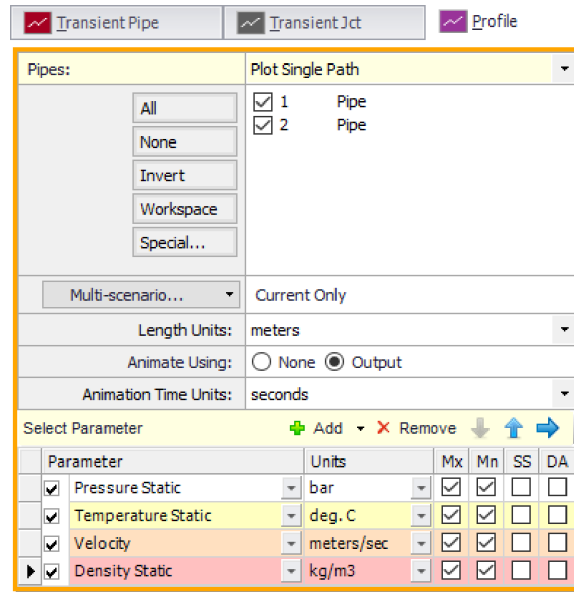


Figure 37: Profile Graph Parameters window

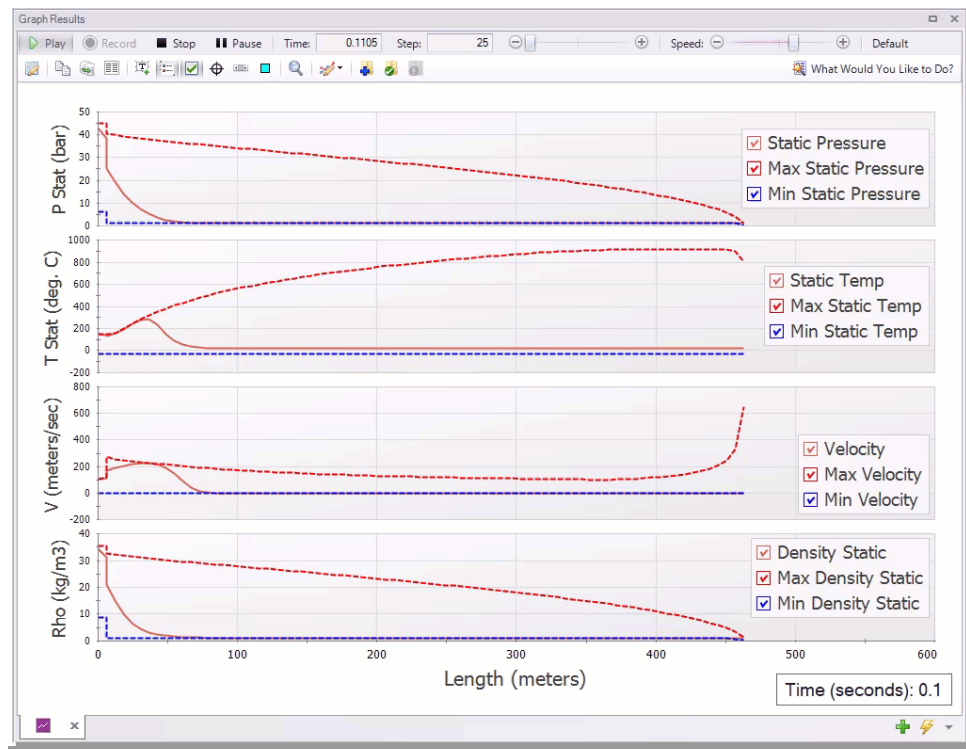


Figure 38: The Profile Graph show the Maximum and Minimum values of all parameters shown in the Graph over the specified time Frame, and can also animate the values

Take note of the pressure wave that is created at the valve just after animation is started. This wave travels to the Assigned Pressure junction before reflecting back towards the valve. The amount of time it

takes this wave to travel to the reflection point and back is called the *communication time*. It should be noted when creating your transient that any event which occurs over a shorter length of time than the communication time of the system is considered to happen instantaneously.

### Step 7. View the Workspace Layers

---

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

The Workspace Layers can also animate the transient pipe results in a color animation overlaid on the model.

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

Within the Layers section, there are two expandable or collapsible groups which contain the two *layer types* named Color Maps and Standard layers. We will integrate the model results with the graphical layout of the pipe network by using the All Objects Layer located under the Standard group. The All Objects Layer is the bottommost layer for all models and shows all pipes, junctions, and annotations when the layer is visible. Layer visibility can be toggled on/off using the eye icon next to the layer. The combination of layers and toggling visibility allows for a wide range of visual customization, and the All Objects Layer will always provide a safety net to keep track of all objects in the model.

➤ **Open the Layer Settings window for the All Objects Layer** by double-clicking the All Objects Layer, clicking the gear icon to the right of the layer name, or selecting the layer and clicking the single gear icon on the Layers Toolbar. The Layer Settings window is shown in Figure 39.

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

## Beginner - Tank Blowdown Example

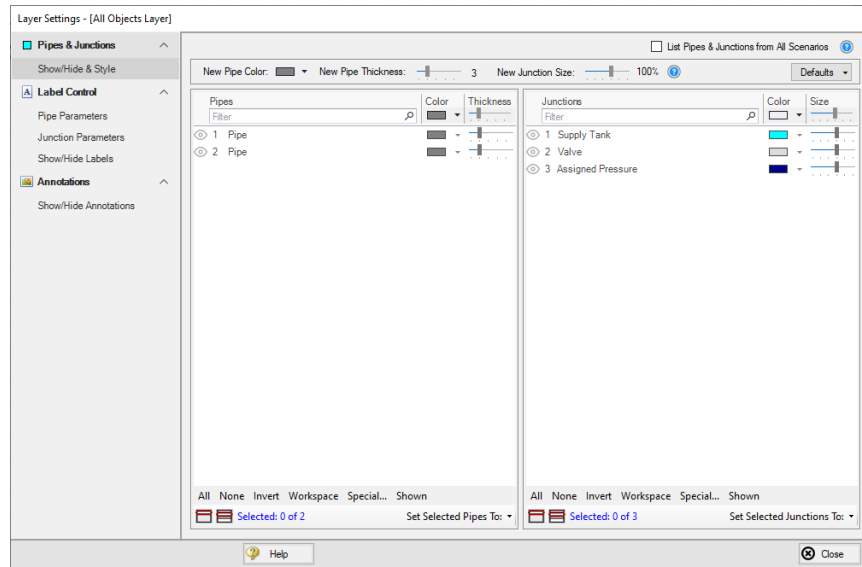


Figure 39: The Layer Settings window specifies content to be superimposed on the Workspace

- **Select the Pipe Parameters item under the Label Control group.** The left pane lists input and output parameters available for display, and the right pane lists the parameters currently being displayed by the layer on the Workspace.
- **In the left pane, expand the Pipe Max/Min Parameters list.** Then, select Max Temperature Static in the list with units of deg. C. Add it to the right pane by either double-clicking the parameter, or selecting the parameter and clicking the arrow between the panes. Repeat the process to add the Min Temperature Static parameter with the same units.
- **Add additional clarity by repositioning the labels and adding a legend.** It is common for the text on the Workspace Layers to overlap with objects when first generated. Labels can be dragged to a new area or a smaller font can be selected using the Font Decrease icon on the Workspace Toolbar or from within the Global Layer Settings. Parameter units can be consolidated in a legend which can be added via the Global Layer Settings. In the Layers Toolbar, click the double gear icon to open the Global Layer Settings. Then, enable the checkbox for Show Units in Legend. The result of these adjustments is shown in Figure 40.

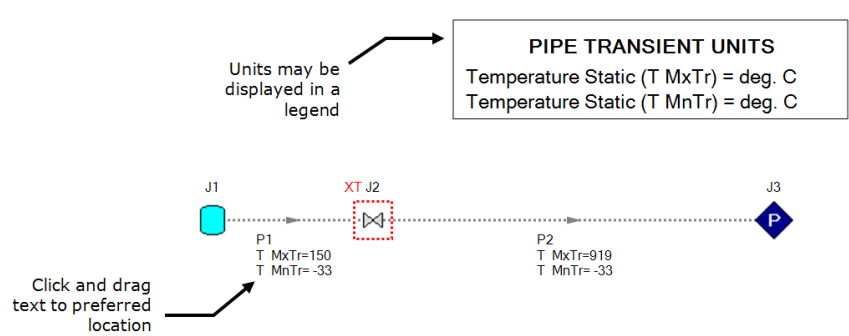


Figure 40: Results are superimposed on the model layout via Workspace Layers



## Step 8. Examine the effect of heat transfer

By default AFT xStream models all pipes as adiabatic at the internal surface, meaning no heat is transferred from the fluid to the pipe walls or surroundings. Modeling the pipes as adiabatic is reasonable when the pipes are well insulated and the temperature difference between the fluid and the atmosphere is relatively low. However, for cases such as the tank blowdown modeled in this example, heat transfer from the fluid to the pipe walls and environment has a large impact on results.

➤ **Create a child scenario named Convective Heat Transfer.**

➤ **Open the Pipe Properties window for each pipe and enter the following on the Heat Transfer tab:**

1. Heat Transfer Model = Convective Heat Transfer
2. External Convection Coefficient Correlation = Free-Horizontal (Churchill-Chu)
3. Ambient Temperature = 21.1 deg. C

After updating each of the pipes open the Analysis Setup window and go to the Pipe Sectioning and Output panel. Change the Estimated Maximum Pipe Fluid Temperature During Transient to 175 deg. C and click OK to accept the changes.

➤ **Run the model** and go to the Output window.

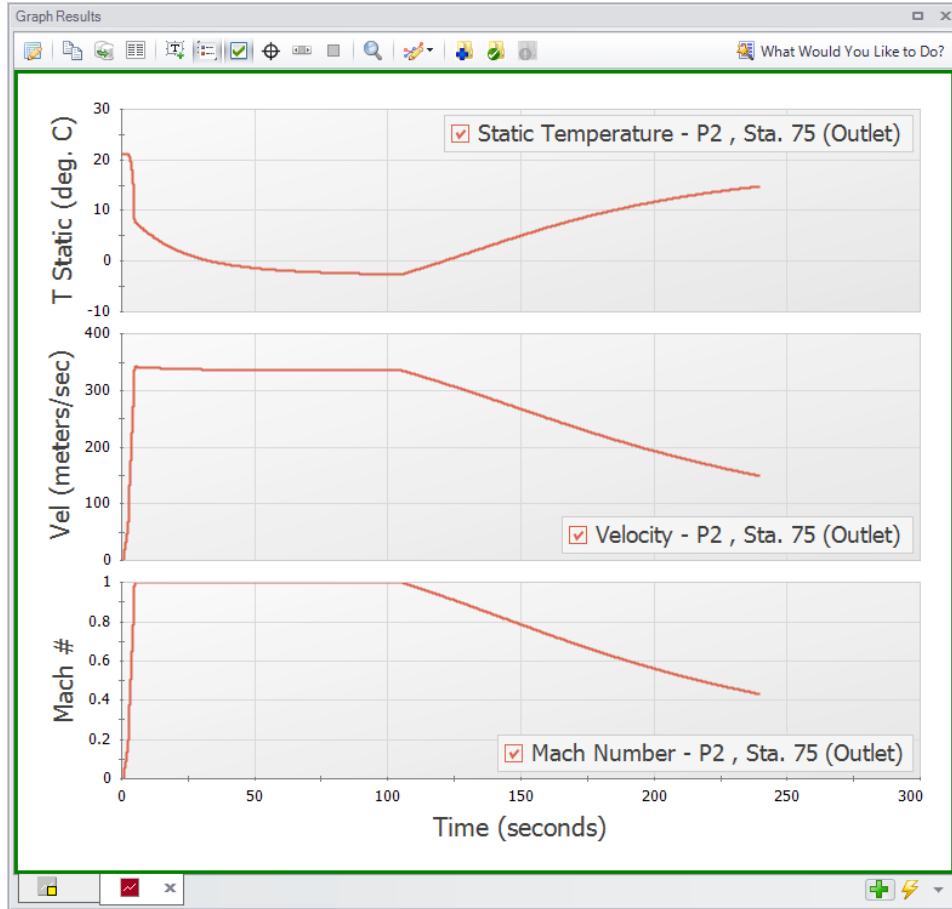
Note that a new caution is shown for heat transfer results in the steady-state. AFT xStream can't solve for convective heat transfer when there is no flow in a pipe in the steady-state, so xStream treats all stagnant pipes as isothermal at the temperature of the connected boundary junctions. The Temperature Override feature on the Optional tab of the Pipe Properties window can be used to specify a different temperature for the pipe if needed.

In the Transient Max/Min tab shown in Figure 41 it can be seen that the maximum temperature in Pipe P2 is only 112 deg. C as opposed to 919 deg. C which was calculated with adiabatic heat transfer.

Go to the Graph Results window and recreate the Pipe P2 outlet graph from Figure 34 as is shown below in Figure 42.

Pipe	Max. Static Temperature (deg. C)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. C)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	Max. Static Pressure (bar)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station
1	150.0	0.000	0	-29.83	240.0	0	45.00	0.000	1
2	111.5	16.34	0	-18.36	0.02114	0	38.51	2.988	0

**Figure 41: Transient Max/Min temperature output with convective heat transfer modeled**



**Figure 42: Temperature, velocity, and Mach number for the discharge of Pipe P2 with convective heat transfer**

### Step 9. Examine the effect of sectioning

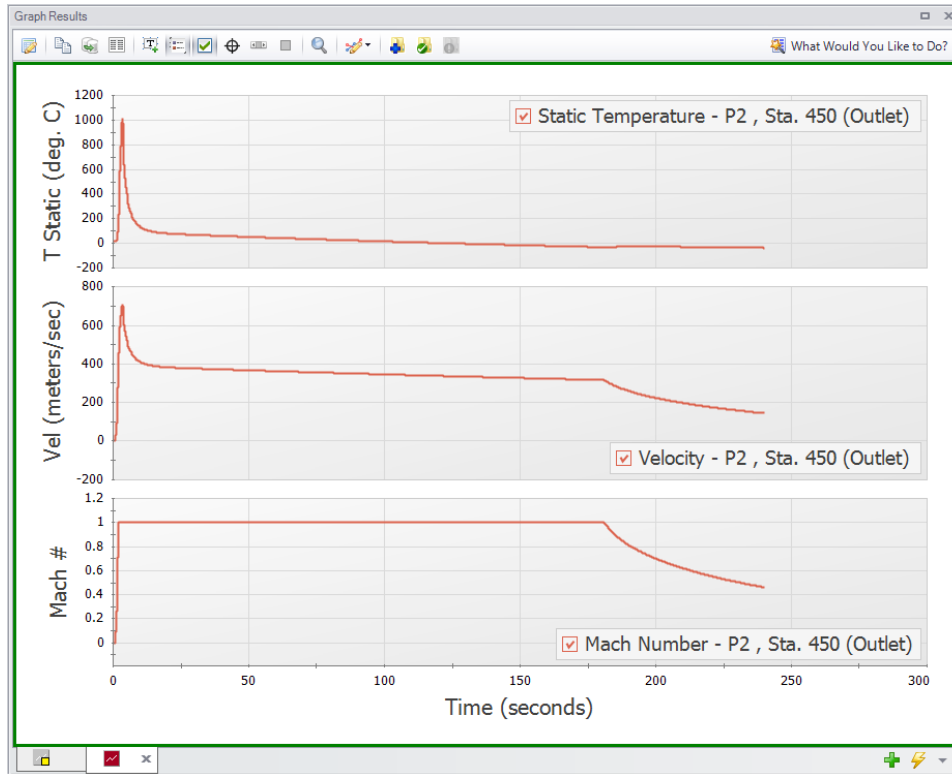
In AFT xStream, sectioning plays an important role in model accuracy. Due to how the MOC operates, accuracy increases with the number of sections you have in the model. To illustrate this, the model was re-run with a Minimum Number of Sections Per Pipe of 6 instead of 1. Comparing Figure 36 to Figure 45, values such as the temperature and pressure for Tank J1 remain similar with the increase in the number of sections. However, values such as the maximum temperature reached in Pipe P2 (Figure 26 and Figure 43), changed substantially. This value increased from 919 deg. C to 1203 deg. C. This change in the maximum temperature indicates that the difference in sectioning has significant implications if your interest is in the conditions in the blowdown pipe. It should be noted that changing the Minimum Number of Sections Per Pipe from 1 to 6 caused the computation time of the model to increase from a couple minutes to about 1.75 hours. Therefore, it is of critical importance to balance accuracy and computation time when using AFT xStream. This idea is discussed in more detail on the [Pipe Sectioning - Introduction to the Method of Characteristics](#) help topic.

Due to the long run time, it was not sensible to ask you to run the 6 section scenario as a part of this example. If you are interested, you may want to run this scenario overnight.

## Beginner - Tank Blowdown Example

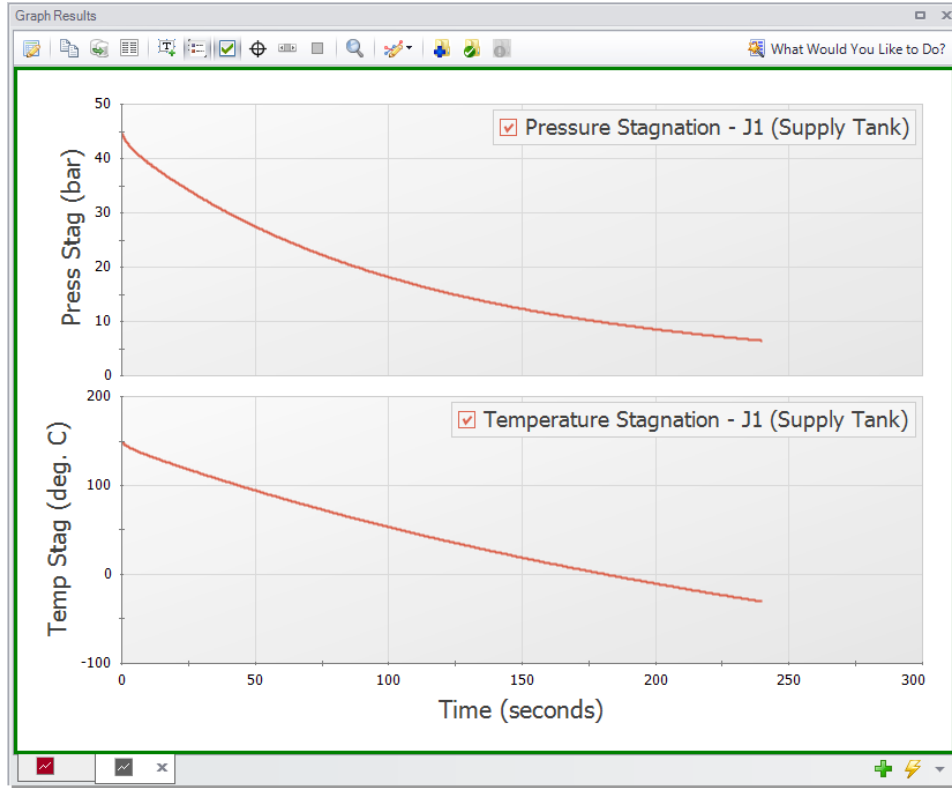
Pipe	Max. Static Temperature (deg. C)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. C)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	Max. Static Pressure (bar)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station
1	150.0	3.986E-03	0	-30.73	240.0	0	45.00	1.993E-03	0
2	1,202.8	2.974	367	-37.66	240.0	450	40.52	2.614	0

**Figure 43: Transient Min/Max tab of the Output for a modified sectioning scenario**

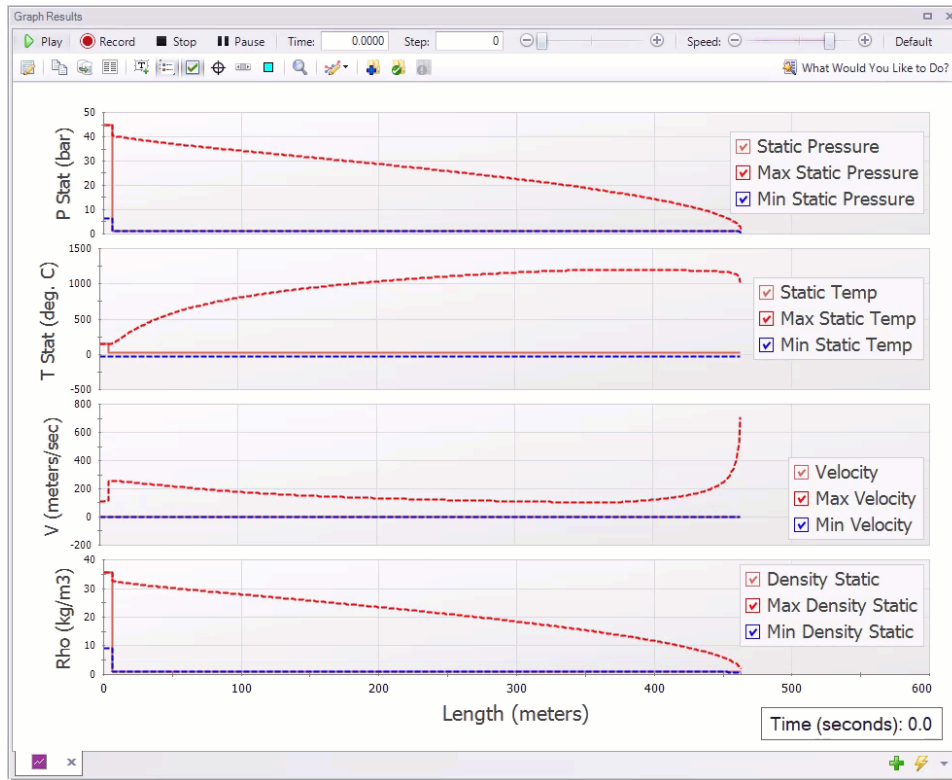


**Figure 44: Temperature, velocity, and Mach number for the discharge of Pipe P2 with 6 sections minimum per pipe**

## Beginner - Tank Blowdown Example



**Figure 45: Temperature and pressure stagnation for Tank J1 with 6 sections minimum per pipe**



**Figure 46: Profile Graph for Tank Blowdown system with 6 Sections Minimum Per Pipe**

## Conclusion

You have now used the Primary Windows in AFT xStream to build and analyze a simple gas transient model. Pipes and junctions were placed on the Workspace and defined by entering data into their Properties windows. Analysis Setup was used to select a fluid, section the pipes, and enter a duration for the transient simulation. The model was run and results were viewed in the Output window. Different types of graphs were created to analyze the system behavior, and the effect of sectioning was discussed.

# Gas Turbine Fuel System Example

## Summary

---

The combustion baskets in a natural gas fuel system are fed by a positive displacement compressor. The natural gas is preheated by a heat exchanger before passing through the three control valves, with each feeding a separate combustion basket. One of the control valves experiences a nearly instantaneous closure. The objective is to determine the maximum fluid temperature reached during the transient, both in the entire system, and at the inlets of the combustion baskets. The effect of this transient event on the temperature, pressure, and velocity of the system should be observed.

As the design engineer, you are also responsible for evaluating the sensitivity of the results to the Minimum Number of Sections Per Pipe parameter.

## Topics Covered

- Modeling a valve closure as a transient event
- Modeling a heat exchanger
- Using the Scenario Manager
- Importance of the Estimated Maximum Temperature During Transient setting
- Finding the max and min values for transient parameters
- Creating animated profile graphs

## Required Knowledge

This example assumes that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the [Beginner - Tank Blowdown](#) example for more information on these topics.

## Model File

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

C:\AFT Products\AFT xStream 3\Examples

- ***Metric - Gas Turbine Fuel System.xtr***

## Step 1. Build the model

### A. Place the pipes and junctions

➤ In the Workspace window, assemble the model as shown in Figure 1.

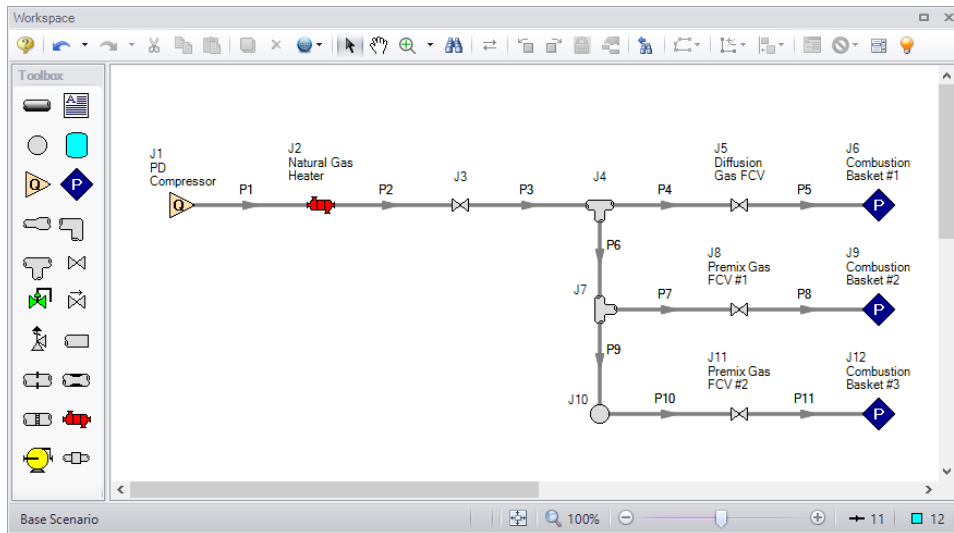


Figure 1: Layout of the system on the Workspace

### B. Enter the pipe and junction data

The system is in place but now the input data for the pipes and junctions needs to be entered. Enter the following data in each Pipe and Junction Properties window.

All Pipes are Stainless Steel - ANSI, Schedule 10S, and have standard roughness. The rest of the pipe information is as follows:

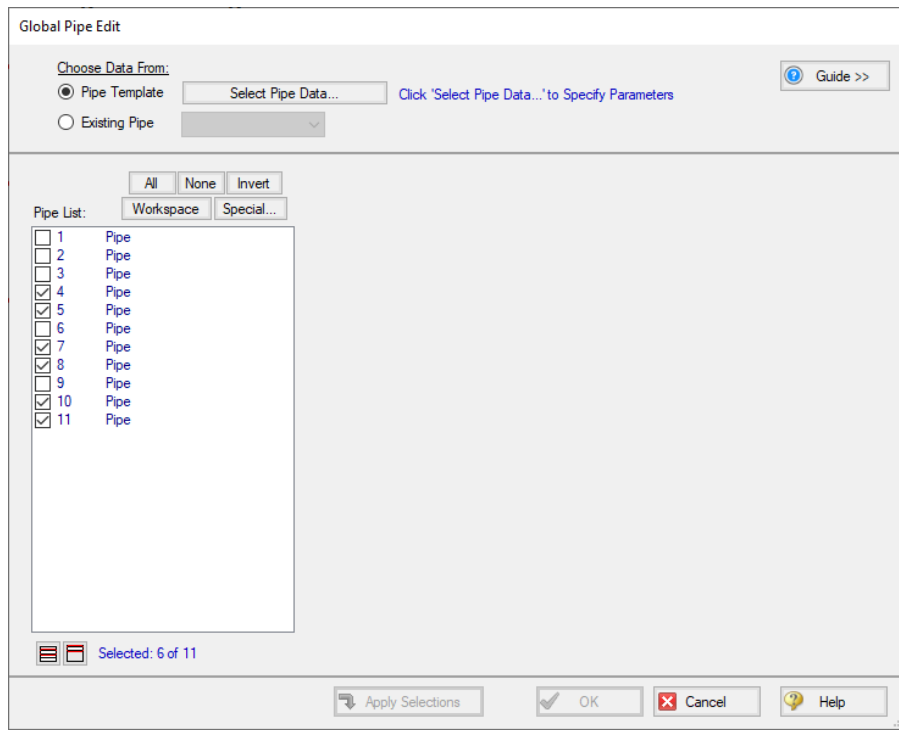
Pipe #	Size (inches)	Length (meters)
1	8	23
2	8	7.5
3	8	1.5
4	5	1.5
5	5	1.5
6	8	1.5

<b>Pipe #</b>	<b>Size (inches)</b>	<b>Length (meters)</b>
7	5	1.5
8	5	1.5
9	8	1.5
10	5	1.5
11	5	1.5

Because several of the pipes are the same, you can use the Global Pipe Edit tool to speed up the process of entering the data. The Global Pipe Edit tool allows you to apply changes to multiple pipes at the same time.

➤ **Select Global Pipe Edit** either from the Edit menu, or by selecting the Global Edit dropdown on the toolbar (blue globe symbol) and selecting the Global Pipe Edit option. This will open the Global Pipe Edit window.

➤ **Select Pipes P4, P5, P7, P8, P10, and P11 from the list of pipes** as shown in Figure 2.



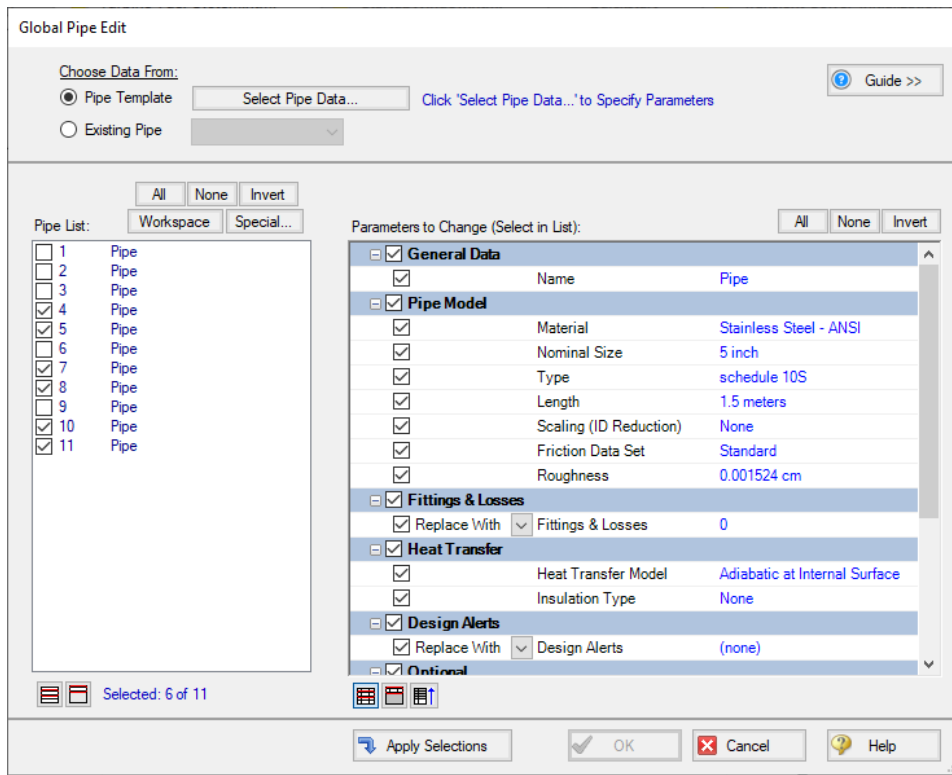
**Figure 2: Global Pipe Edit window**

➤ **Click the Select Pipe Data button.** This will open a template Pipe Properties window for entering the data to be applied to all of the selected pipes.



➤ **Fill out the data for the selected pipes.** When you have entered all of the data for the pipes, close the Pipe Properties window by selecting OK. The Global Pipe Edit window will now display a list of all of the parameters that may be applied to the selected pipes. The parameters are categorized as they are displayed on the tabs in the Pipe Properties window.

➤ **Select the All button above the Parameters to Change list,** since for this example, you want all of the parameters for all of the selected pipes to be updated to the specified values. The Global Pipe Edit window should now appear as shown in Figure 3.



**Figure 3: Global Pipe Edit window with all pipe parameters selected for updating**

➤ **Apply the changes to the selected pipes by clicking the Apply Selections button.** AFT xStream will notify you that the changes have been completed with a message in the bottom-left of the Global Pipe Edit window. At this point, all of the changes have been applied but not saved. If you wish to cancel all of the changes you just implemented, you may do so by clicking Cancel on the Global Pipe Edit window.

➤ **Save the applied changes by clicking OK.**

---

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

---

1. Assigned Flow J1
  - a. Name = PD Compressor
  - b. Elevation = 0 meters
  - c. Mass Flow Rate = 41400 kg/hr

## Gas Turbine Fuel System Example

---

**Note:** Be advised that the unit for mass flow is not the default, it is kg/hr as opposed to kg/s.

---

- d. Temperature = 21 deg. C
- 2. Heat Exchanger J2
  - a. Name = Natural Gas Heater
  - b. Elevation = 0 meters
  - c. K = 6.6
  - d. Steady Heat Rate = 1035 kW
- 3. Valve J3
  - a. Elevation = 0 meters
  - b. Kv = 8650
  - c. xT = 0.3
- 4. Valve J5
  - a. Name = Diffusion Gas FCV
  - b. Elevation = 0 meters
  - c. Kv = 80
  - d. xT = 0.7
- 5. Valve J8
  - a. Name = Premix Gas FCV #1
  - b. Elevation = 0 meters
  - c. Kv = 170
  - d. xT = 0.7
- 6. Valve J11
  - a. Name = Premix Gas FCV #2
  - b. Elevation = 0 meters
  - c. Kv = 170
  - d. xT = 0.7
  - e. Click on the Transient Tab and enter the following data:

<b>Time (seconds)</b>	<b>Kv</b>	<b>xT</b>
0	170	0.7
0.01	0	0.7
2	0	0.7

- 7. Assigned Pressures J6, J9, J12
  - a. J6 Name = Combustion Basket #1
  - b. J9 Name = Combustion Basket #2
  - c. J12 Name = Combustion Basket #3
  - d. Elevation = 0 meters
  - e. Stagnation Pressure = 15.7 bar
  - f. Temperature = 60 deg. C.

8. Junctions J4, J7, J10
  - a. Elevation = 0 meters

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

➤ **Open the List Undefined Objects window** from the View menu or the Workspace Toolbar to verify if all pipes and junctions are specified. If not, the incomplete pipes or junctions will be listed. If undefined objects are present, go back to the incomplete pipes or junctions and enter the missing data.

## Step 2. Specify Analysis Setup

---

### A. Specify the Fluid panel

- **Open the Analysis Setup window** by clicking Analysis Setup on the Common Toolbar.
- **Select Methane from the AFT Standard fluid library, and click the Add to Model button.**

---

**Note:** Methane is used to approximate natural gas in order to minimize the run time. This is typically a fair assumption for preliminary runs. If you need to model a real natural gas mixture, such as for a final run, then the NIST REFPROP library or the optional Chempak add-on can be used. Be aware that mixtures with multiple components may drastically increase the runtime. Limiting the mixture to 3 or 4 components is advised.

---

### B. Specify the Pipe Sectioning and Output panel

- **Open the Pipe Sectioning and Output panel and specify the following:**
  1. Minimum Number of Sections Per Pipe = 8
  2. Estimated Maximum Pipe Temperature During Transient = 60 deg. C

---

**Note:** If following along in the completed model file, the Estimated Maximum Pipe Temperature During Transient will have a different value. This will be discussed in later steps.

---

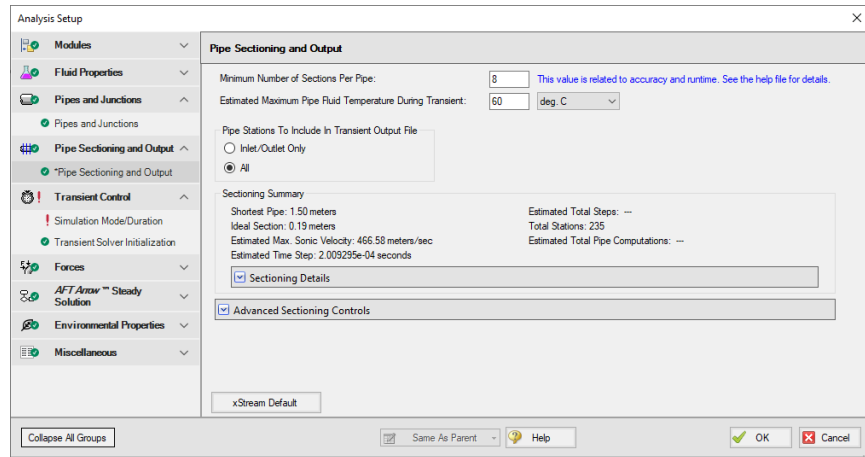


Figure 4: Sectioning panel

## C. Specify the Simulation Mode/Duration panel

➤ Enter a Stop Time of 2 seconds on the Simulation Mode/Duration panel from the Transient Control group. Verify the Time Simulation is set to Transient. Click the OK button.

## Step 3. Create scenarios to model different sectioning setups


### A. Create scenarios

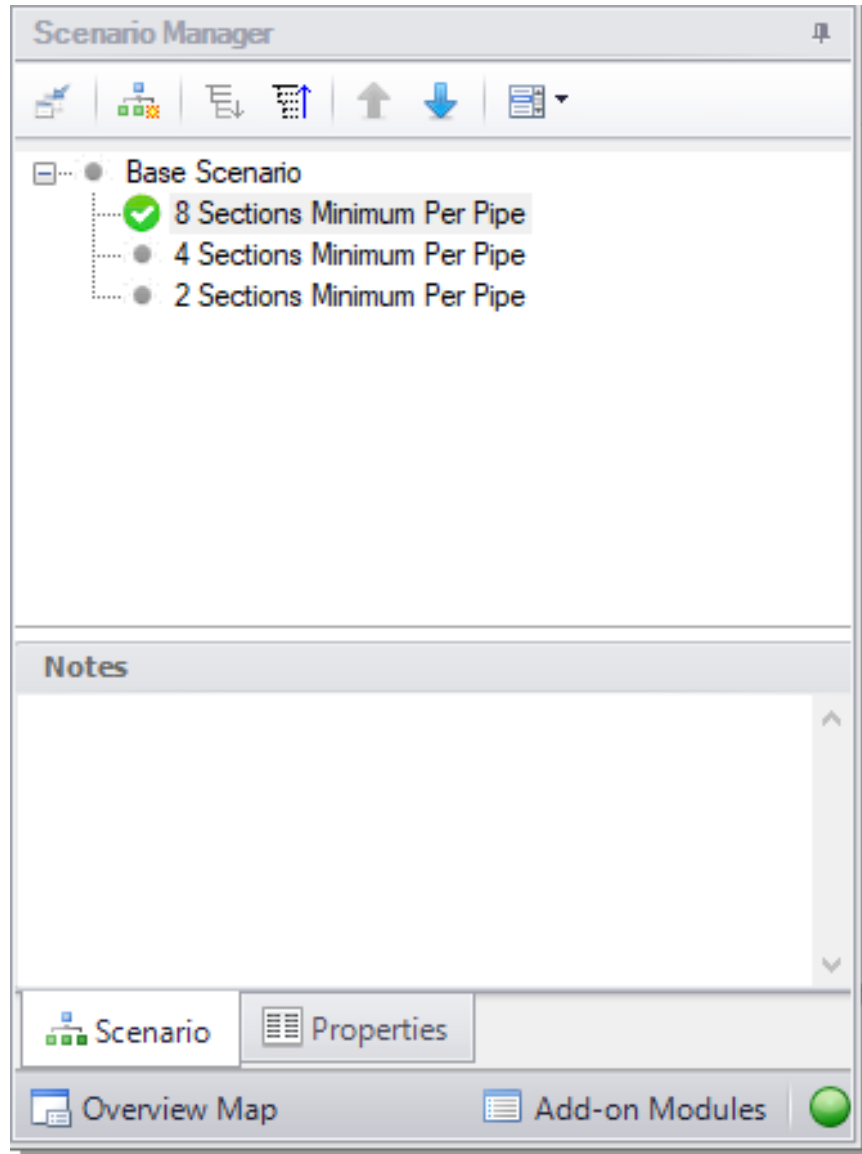
In this model, the following three types of Sectioning cases for the system are to be evaluated:

1. 8 Sections Minimum per Pipe
2. 4 Sections Minimum per Pipe
3. 2 Sections Minimum per Pipe

To evaluate the three cases, we will utilize the Scenario Manager to create three scenarios for the three cases.

Open 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 8 Sections Minimum Per Pipe in the Create Child Scenario window, and select the OK button. The new 8 Sections Minimum Per Pipe 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 named 4 Sections Minimum Per Pipe. Finally, create one more child and call it 2 Sections Minimum Per Pipe. The Child Scenarios should now be displayed as shown in Figure 5.



*Figure 5: Create model variants and keep them organized using the Scenario Manager on the Quick Access Panel*

## B. Set up scenarios

Child scenarios inherit data from their ancestor scenarios. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent. Scenarios can be loaded onto the Workspace by double-clicking their name in the Scenario Manager. The currently loaded scenario is indicated in the scenario tree with a green circle with a checkmark (Figure 5). Since the Base Scenario already has been set up with 8 Minimum Sections Per Pipe, the 8 Sections Minimum Per Pipe child does not need to be modified.

- **Load the 4 Sections Minimum Per Pipe scenario** by double-clicking the name in the Scenario Manager. Open the Analysis Setup window from the Common Toolbar. Open the Pipe Sectioning and Output panel and then click OK.
- **Change the Minimum Number of Sections Per Pipe to 4** from the Pipe Sectioning and Output panel in Analysis Setup, then click OK. The required input for this scenario is now complete.
- **Repeat the steps above for the 2 Sections Minimum Per Pipe scenario, and change the value to 2.**

### Step 4. Run the first scenario

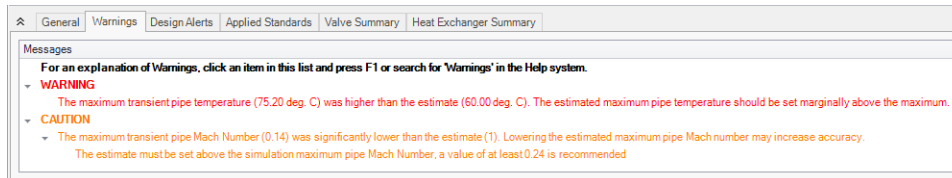
- **Double-click the 8 Sections Minimum Per Pipe scenario** in the Scenario Manager.
- **Click the Run Model button on the Common Toolbar** or select the option from the Analysis menu. AFT xStream will prompt you to save the model if you have not done so already, then will open the Solution Progress window.

### Step 5. Review the results

- **Click the Output button.** This will take you to the Output window, which will display any warnings if they exist. There should be a warning indicating that the maximum transient pipe temperature was higher than the estimate as well as a caution that the maximum transient pipe mach number was significantly lower than the estimate (see Figure 6). The Estimated Maximum Pipe Temperature During Transient is critical to the calculation of transient conditions due to its necessity in the Method of Characteristics. A review on how the MOC handles sectioning and tabulation can be found [here](#).

The Estimated Maximum Pipe Temperature During Transient is used to calculate the sonic velocity of the fluid, which is then used to create the upper boundary of the characteristic grid. If the maximum temperature during the transient exceeds the specified Estimated Maximum Pipe Temperature During Transient and the flow is sonic, the Method of Characteristics may have to extrapolate data, which can lead to instability and will stop the transient solver. In this model, sonic velocity was not reached when the temperature was higher than the estimated maximum. However, to ensure that this model can handle sonic flow, we will re-run the model with a higher Estimated Maximum Pipe Temperature During Transient.

The Maximum Transient Pipe Mach Number warning indicates that error could be introduced through having a Mach Number substantially lower than the Maximum Transient Pipe Mach Number used for the calculation (in this case, the xStream default value of one). This caution will be ignored in this example for simplicity.



**Figure 6: The maximum transient pipe temperature exceeds estimated maximum temperature warning indicates that the model is at risk of instability if changes are made**

## Step 6. Adjust the Pipe Sectioning panel

- **Double-click the Base Scenario** in the Scenario Manager. Open the Analysis Setup window from the Common Toolbar and select Change the Click OK.
- **Change the Estimated Maximum Pipe Temperature During Transient** to 80 deg. C from the Pipe Sectioning and Output panel.

**Important:** Making this change in the Base Scenario will save time since it will propagate the change to all scenarios automatically.

## Step 7. Re-run the model and review results

### A. Review Output window

- **Re-run the 8 Sections Minimum Per Pipe scenario**, then proceed to the Output window. There should be a caution indicating that the maximum transient pipe mach number was significantly lower than the estimate. Disregard this caution as discussed previously.
- **Select the Transient Max/Min tab** as shown in Figure 7. Here we can see the maximum static temperatures in the pipes (scroll to the right in the Pipes section of the Output). Note that the values here are accounting for the entire length of the pipe, and the corresponding Station at which the max/min values occurred is reported.

**The max static temperature for the entire system occurs at the inlet of Pipe P2 (Station 0), with a value of about 77.6 deg. C.** This is intuitive because the Pipe P2 inlet is equivalent to the J2 Heat Exchanger outlet, which adds heat to the system. Furthermore, if we observe the pipes that feed the combustion baskets (Pipes P5, P8, and P11) we can see that max static temperatures occur at the inlets (Station 0). Therefore, to find the max temperatures for the pipe outlets, which correspond to the combustion basket inlets, we will need to create transient graphs.

Output Window									
Pipes									
Transient Max/Min									
Pipe	Velocity (m/s)	Min. Velocity Station	Max. Static Temperature (deg. C)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. C)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	
1	0.04304	123	26.95	0.8109	123	20.94	0.2391	2	
2	0.02524	40	77.63	0.04304	0	58.91	0.000	2	
3	0.02172	8	74.06	0.3676	0	58.92	0.000	3	
4	0.02700	0	73.84	0.4202	0	59.11	5.478E-03	8	
5	9.99E-03	5	72.64	0.4906	0	58.44	0.02895	4	
6	0.02015	2	73.87	0.4202	0	59.00	1.956E-03	8	
7	0.000	0	72.96	0.5079	2	58.92	2.543E-03	8	
8	2.60E-03	0	71.96	0.5298	0	58.08	0.02504	5	
9	0.01722	0	73.46	0.5020	0	59.12	0.000	8	
10	0.01389	0	69.72	2.000	0	58.92	0.000	8	
11	0.01330	8	69.99	0.01643	0	56.35	0.01056	3	

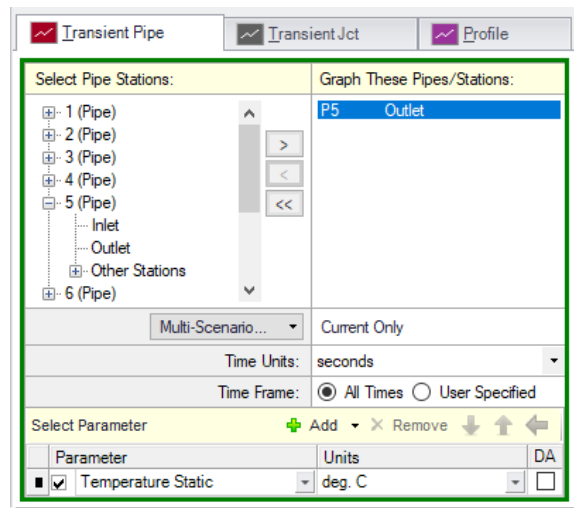
Figure 7: Transient Max/Min tab in the Output window

## B. Graph the results

Go to the Graph Results window by clicking on the Graph Results tab or pressing CTRL+G on the keyboard.

➤ **Create a graph that shows the temperature over time at the entrance to Combustion Basket #1** using the following steps (Figure 8):

1. From the Graph Control tab on the Quick Access Panel, select the Transient Pipe tab
2. Select Pipe P5 and add the outlet station
3. Verify that the time units are set to seconds
4. In the Parameters definition area, select Temperature Static with units of deg. C
5. Click the Generate button



**Figure 8: Graph Parameters for Combustion Basket #1 Inlet**

➤ **Right-click on the resulting graph and select Show Cross Hair**, or enable it via the icon on the Graph Toolbar. Moving your mouse cursor over the graph will display the numerical values of the graph data points. This can be used to find the max value, or you can right-click on the graph and select Copy X-Y Data, which can be pasted into Excel or other software utilities for further inspection.

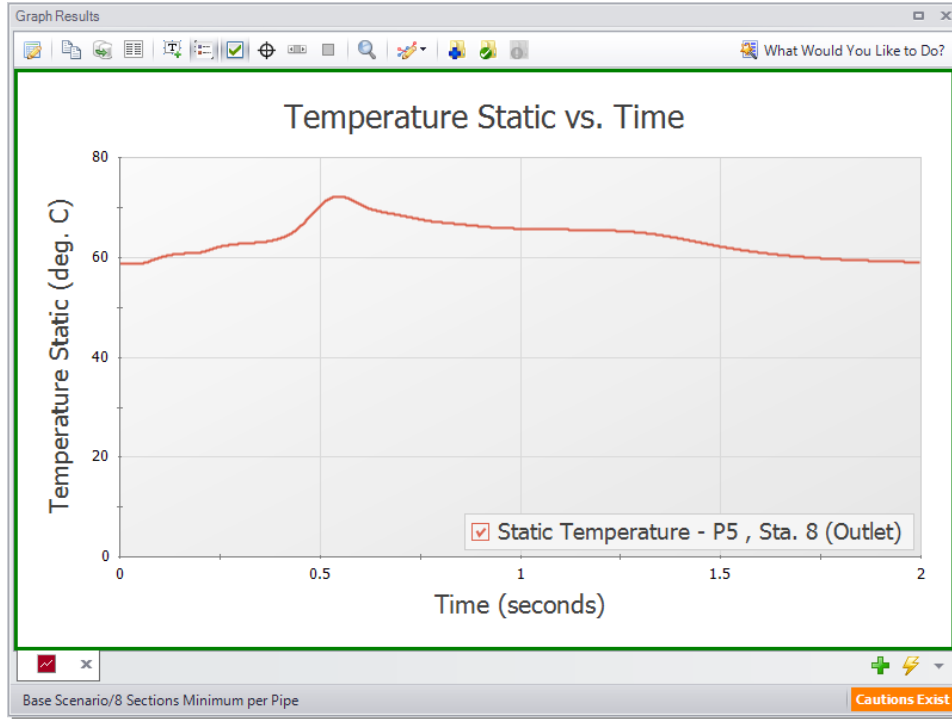
The max temperature at the inlet of Combustion Basket #1 is about 72.3 deg. C. Note that this temperature is lower than the temperature shown on the Transient Max/Min tab because this graph is showing the outlet of Pipe P5, while the maximum temperature in Pipe P5 occurs at the inlet of the pipe. The maximum temperature for the outlet of Pipe P5 occurs at about 0.55 seconds before trending back towards the previous steady-state temperature of 60 deg. C as the flow normalizes.

---

**Note:** The Assigned Pressure junctions in this model were defined with Stagnation Temperature. Therefore, the Static Temperature will be lower than the Stagnation value, as is seen in Figure 9. In the case of Pipe P11 where the flow is zero after the valve closure, the Static and Stagnation Temperatures will be equal.

---






**Figure 9: Temperature Static vs Time for the inlet to Combustion Basket #1**

To easily recreate this graph in the other scenarios or in the future, it can be added into the Graph List Manager on the Quick Access Panel.

➤ **Right-click the graph and select Add Graph to List** from the context menu, or use the icon from the

Graph Results Toolbar . Give the graph a name, such as “Combustion Basket #1 Temperature vs Time”, then click OK. The graph will be added to the currently selected folder in the Graph List Manager. Note that graphs added to the Graph List Manager are not permanently saved until the model is saved again. Also be aware that only the graph settings are saved as opposed to the graph data.

➤ **Create the same graph above for Combustion Basket #2 and #3, and save them as Graph List Items.** Combustion Basket #2 corresponds to the Pipe P8 Outlet station, and Combustion Basket #3 corresponds to the Pipe P11 Outlet station. Combustion Basket #2 will have a similar graph with a slightly lower maximum. Combustion Basket #3 will have a flat graph due to the valve closure, and the temperature settles out to the boundary condition of the J12 Assigned Pressure since Pipe P11 is adiabatic with no heat transfer.

**Table 1: Summary of the Combustion Basket max static temperatures for the 8 Sections scenario**

<b>Combustion Basket</b>	<b>Max Transient Temperature (deg. C)</b>
#1 (Pipe P5 Outlet)	72.3
#2 (Pipe P8 Outlet)	71.9
#3 (Pipe P11 Outlet)	60.0

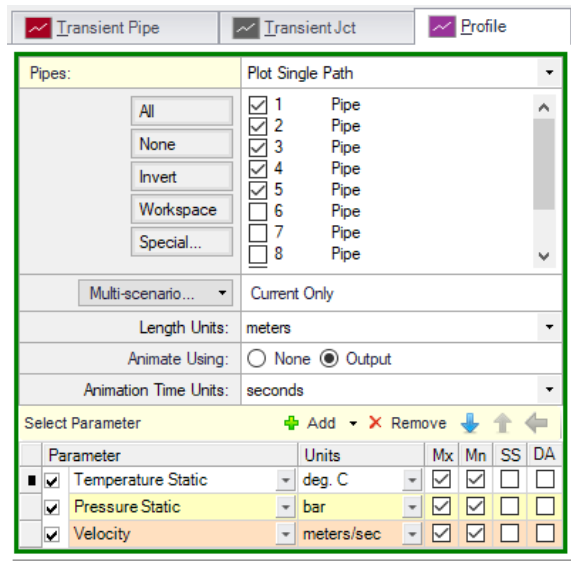
### C. Animate the results

AFT xStream can animate output along pipeline paths in your model in the Graph Results window. This feature can help you understand how your system will respond when a transient event occurs. You can also animate multiple graphs at the same time.

Here, you will animate the temperature static, pressure static, and velocity along the flow path between the positive displacement compressor and Combustion Basket #1 in order to gain an understanding of how these parameters respond as the control valve closes.

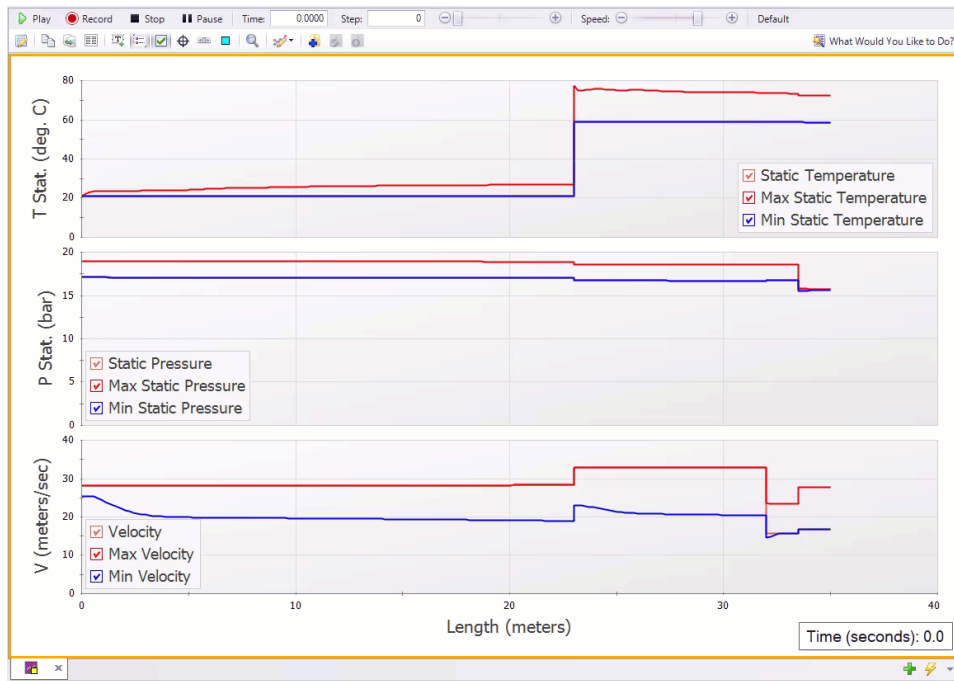
➤ **Specify the following graph settings** (Figure 10):

1. Create a new graph tab by clicking the New Tab button located in the bottom-right (green “+” icon).
2. Select the Profile tab under the Graph Control tab in the Quick Access Panel.
3. Select Pipes P1 through P5.
4. Ensure the Length Units are meters for the Profile graph.
5. Select the option to Animate Using Output. To use this feature, data from all pipe stations must be saved (which is the default on the Pipe Sectioning and Output panel), otherwise the option will be grayed out.
6. Ensure that the Animation Time Units are set to seconds.
7. Add two additional parameters using the Add button, and specify the three parameters:
  - a. Temperature Static with units of deg. C
  - b. Pressure Static with units of bar
  - c. Velocity with units of meters/sec
8. Check the Mx and Mn boxes next to the graph parameters to plot the maximum and minimum values on the graph.
9. Click the Generate button.



**Figure 10: Graph parameters for Combustion Basket #1 animated profile**

**Note:** Figure 11 have been modified to increase clarity.



**Figure 11: Flow to Combustion Basket #1 profile animation**

Additional animation control features appear at the top of the Graph Results window (Figure 11). Press the Play button and watch the temperature, pressure, and velocity responses move between the maximum and minimum values. This animation can also be recorded to a file.

➤ **Add the graph to the Graph list manager** by either right clicking on the graph or using the Add Graph to List icon. Give it a name, such as "Combustion Basket #1 Animated Profile".

Playing the animation reveals that the highest temperature in the system is observed at the J2 heat exchanger. It should also be noted that the rise to this maximum temperature occurs when the pressure wave from the valve closing passes through this junction. However, this heightened temperature is not immediately seen by the combustion baskets downstream because the gas velocity throughout the system drops immediately following the valve closure. Instead, the heightened temperature moves slowly downstream until the gas wave is reflected at the positive displacement compressor and passes through the heat exchanger going the opposite direction. This causes the local gas velocity to rise and causes the temperature wave to move downstream towards the combustion baskets more quickly. This wave eventually reaches Combustion Basket #1 at around Time = 0.55 seconds, at which point the inlet to the combustion basket reaches its maximum temperature.

## Step 8. Run the other scenarios and graph the results

- Load and run the other two scenarios using the Scenario Manager.
- Observe the Transient Max/Min tab for each scenario, and note the overall system max static temperature. The max temperature and pressure during both the steady-state and the transient simulation can also be found at the bottom of the General tab in the Output. A summary of the system max temperature for each scenario is shown below in Table 2.

**Table 2: Overall system max static temperatures for the three sectioning counts**

Scenario	System Max Transient Temperature (deg. C)
8 Sections	77.6
4 Sections	76.0
2 Sections	75.2

- Create the same graphs from the previous steps for each of the scenarios by double-clicking on the saved Graph List Items in the Graph List Manager.

**Table 3: Combustion Basket max static temperatures for the three sectioning counts**

Scenario	Combustion Basket #1 Max Transient Temperature (deg. C)	Combustion Basket #2 Max Transient Temperature (deg. C)	Combustion Basket #3 Max Transient Temperature (deg. C)
8 Sections	72.3	71.9	60.0
4 Sections	70.7	70.3	60.0
2 Sections	69.6	69.2	60.0

## Conclusion

Use of transient conditions within a valve junction allowed us to model the effects of an unexpected valve closure in a system. The Scenario Manager was used to easily create variations of the model that analyzed the effect of different pipe sectioning. While this example was focused on temperature, it also showed that values such as pressure and velocity can be tracked by AFT xStream. The overall maximum temperature for the system was found in the Transient Max/Min tab of the Output, and also at the bottom of the General tab for each scenario (Table 2). Different pipe sectioning resulted in different max temperatures (Table 3).

Additionally, this example identified the Estimated Maximum Temperature During Transient as a critical variable in the tabulation of the transient results. This value sets the boundary for the characteristic grid used by the Method of Characteristics. If sonic choking occurs at a temperature higher than the

## Gas Turbine Fuel System Example

---

estimated maximum, the model may not converge. As such, this value should be conservatively high on early runs of a model and then refined once the temperature that the transient approaches is known.

## Fluid Library Comparison - Steam Turbine Shutdown Example

### Summary

---

Two turbines in a high pressure steam system are simultaneously shut down while a bypass line is opened. Evaluate the maximum pressure at the turbine inlets. Additionally, quantify the mass flow rate through the bypass valve throughout the transient and perform an analysis on whether condensation or two-phase flow occurs in the bypass line. Lastly, conduct a sensitivity analysis based on the selection of different fluid library data.

### Topics Covered

- Using the isometric grid to create a model
- Modeling a transient turbine shutdown
- Creating transient graphs
- Animating profile graphs
- Evaluating the selection of a fluid library

### Required Knowledge

This example assumes that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the [Beginner - Tank Blowdown](#) example for more information on these topics.

### Model File

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

C:\AFT Products\AFT xStream 3\Examples

- ***Metric - Fluid Library Comparison - Steam Turbine Shutdown.xtr***

## Step 1. Build the model

---

### A. Place the pipes and junctions

#### *Isometric Drawing Mode*

The previous example models were drawn using the default Pipe Drawing Mode, 2D Freeform. AFT xStream has two additional available drawing modes, 2D Orthogonal and Isometric. In this model, the isometric mode is used to visually interpret the pipe layout and provide a better understanding of the system.

1. From the Arrange menu, select the option to Show Grid.
2. Open the Arrange menu again, go to Pipe Drawing Mode, and select Isometric.
3. Place the junctions as shown in Figure 1.
4. Click the Pipe Drawing 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.
5. Use the Rotate Icons buttons on the Workspace Toolbar, or right-click on the junction and select the Auto-rotate option to automatically align the junction with the connected pipes. Alternatively, Customize Icon can be selected from the right-click menu to manually choose an icon/orientation.
6. The grid can be shown or turned off in the Arrange menu, as shown in Figure 2.

---

**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.

---

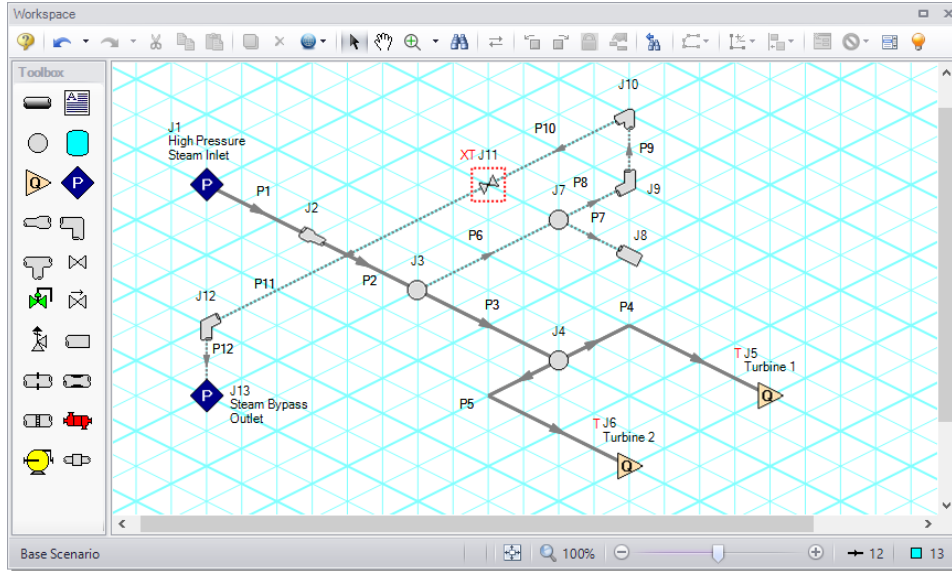


Figure 1: Isometric layout with the grid shown

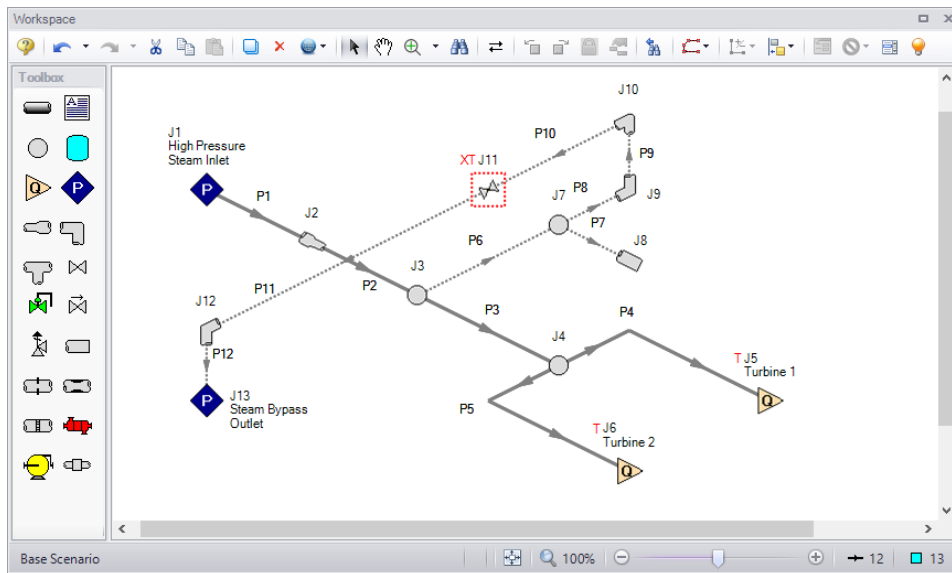


Figure 2: Isometric layout with the grid turned off

## B. Enter the pipe and junction data

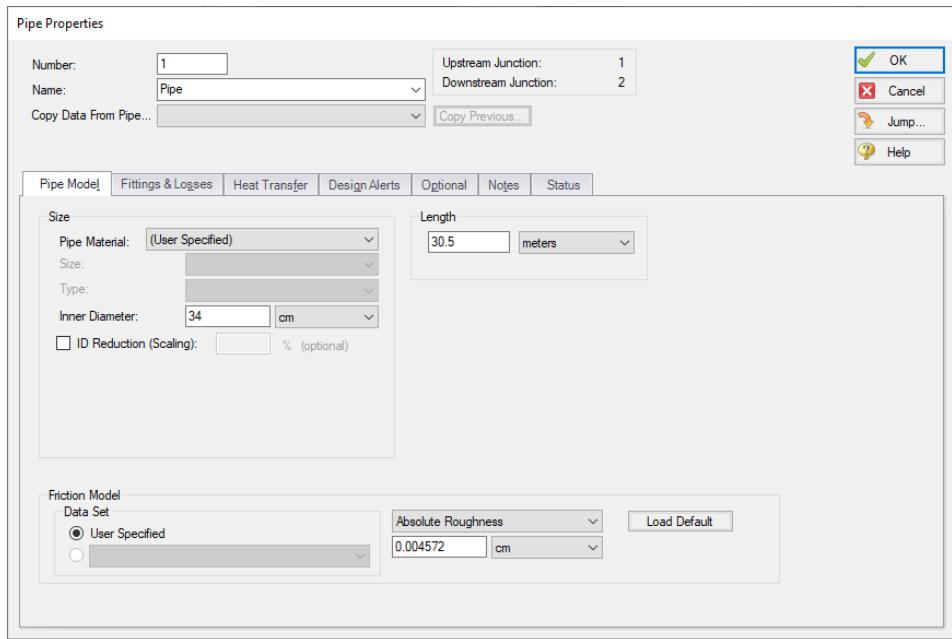
The system is in place but now the input data for the pipes and junctions needs to be entered. Double-click each pipe or junction and enter the following data in the Pipe or Junction Properties window.

All pipe materials are User Specified in this model (see Figure 3). Use the Absolute Roughness of Steel (0.004572 cm) as the Friction Model, along with the following data:



**Note:** The pipes below are user defined as they utilize non-standard pipe sizes.

<b>Pipe #</b>	<b>Inner Diameter (cm)</b>	<b>Length (meters)</b>
1	34	30.5
2	30.5	20
3	30.5	15
4	30.5	4.5
5	30.5	4.5
6	15.5	2
7	10	2
8	15.5	3
9	15.5	1.5
10	15.5	6
11	49	6
12	49	1.5



**Figure 3: User Specified Pipe Properties window**

1. Assigned Pressure J1
  - a. Name = High Pressure Steam Inlet
  - b. Elevation = 0 meters
  - c. Static Pressure = 170 bar
  - d. Temperature = 570 deg. C
2. Area Change J2
  - a. Type = Conical Transition
  - b. Angle = 45 degrees
3. Branches J3, J4, and J7
  - a. Elevation = 0 meters
4. Assigned Flows J5 and J6
  - a. J5 Name = Turbine 1
  - b. J6 Name = Turbine 2
  - c. Elevation = 0 meters
  - d. Mass Flowrate = 228600 kg/hr

---

**Note:** Be advised that the unit for mass flow is not the default, it is **kg/hr** as opposed to **kg/s**.

---

- e. Temperature = 570 deg. C
5. Dead End J8
  - a. Elevation = 0 meters
6. Bend J9
  - a. Elevation = 0 meters
  - b. Type = Standard Elbow (knee, threaded)
7. Bend J10, J12
  - a. Elevation = 1.5 meters
  - b. Type = Standard Elbow (knee, threaded)
8. Valve J11
  - a. Kv = 2600
  - b. xT = 0.7
  - c. Elevation = 1.5 meters
  - d. On the Optional tab, set the Special Condition to Closed
  - e. Click on the Transient Tab and enter the following data:

<b>Time (seconds)</b>	<b>Kv</b>	<b>xT</b>
0	0	0.7
2	2600	0.7
5	2600	0.7

9. Assigned Pressure J13
  - a. Name = Steam Bypass Outlet
  - b. Elevation = 0 meters
  - c. Static Pressure = 30 bar
  - d. Static Temperature = 350 deg. C

## C. Check if the pipes and junctions are defined

➤ **Click the List Undefined Objects button** to see if the model is fully defined. If it is not, the undefined pipes and junctions window will show a list of incomplete items. If undefined objects are present, go back to the incomplete pipes or junctions and enter the missing data.

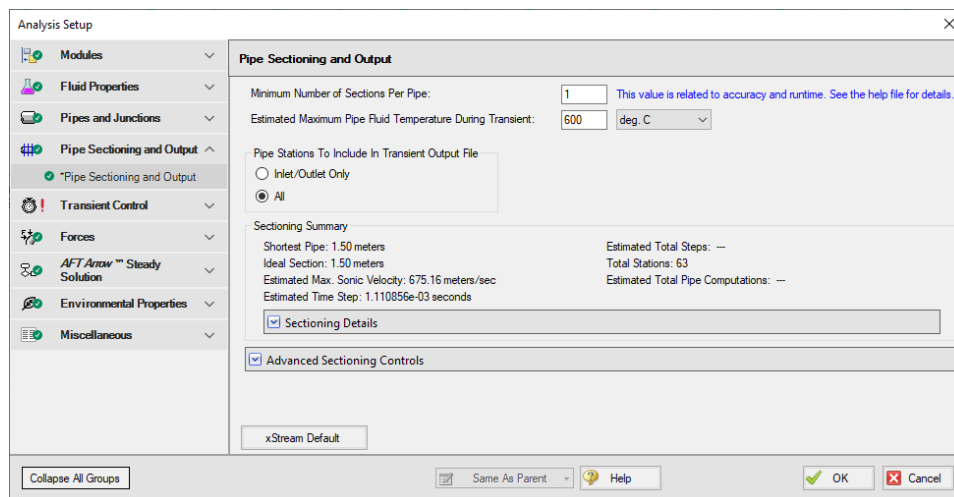
## Step 2. Specify Analysis Setup

### A. Specify the Fluid panel

- **Open the Analysis Setup window from the Common Toolbar** and open the Fluid panel.
- **Select Steam from the AFT Standard fluid library, and click the Add to Model button.**

### B. Specify the Pipe Sectioning and Output panel

- **Open the Pipe Sectioning and Output panel and specify the following:**
  1. Minimum Number of Sections Per Pipe = 1
  2. Estimated Maximum Pipe Temperature During Transient = 600 deg. C



**Figure 4: Pipe Sectioning and Output panel**

### C. Specify the Simulation Mode/Duration panel

➤ Enter a **Stop Time of 5 seconds** on the **Simulation Mode/Duration panel** from the Transient Control group. Verify the Time Simulation is set to Transient. Click the OK button.

### Step 3. Run the model

---

➤ Click the **Run Model button** on the **Common Toolbar** or select the option from the Analysis menu. AFT xStream will prompt you to save the model if you have not done so already, then will open the Solution Progress window.

### Step 4. Graph the results

---

#### A. Graph the transient pressures at the turbines

➤ **Go to the Graph Results window.** The first graph we will make will show the Static Pressure at the entrance to the steam turbines.

1. From the Graph Control tab on the Quick Access Panel, select the Transient Pipe tab
2. Select the Pipe P5 outlet station
3. Ensure that the Time Units are set to seconds
4. Select Pressure Static with units of bar in the Parameters definition area
5. Click Generate

This will generate the graph shown in Figure 6. The pressure variation is difficult to see using the default y-axis scale so we will adjust the scale using the following steps:

1. Right-click the y-axis
2. Un-check the Auto Scale box
3. Set the Minimum to 150 bar, the Maximum to 190 bar, and the Major Val to 10 (see Figure 7)

The graph should now match Figure 8. When the model contains multiple transient events, transient pipe graphs allow the user to see the cumulative impact of the events on system pressure. Note that the first peak contains an oscillation, which is attributable to the opening of the bypass valve.

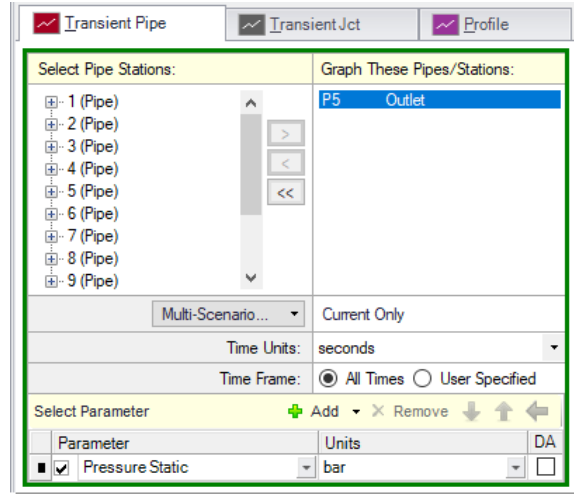


Figure 5: Graph Parameters for the inlet pressure to the turbines

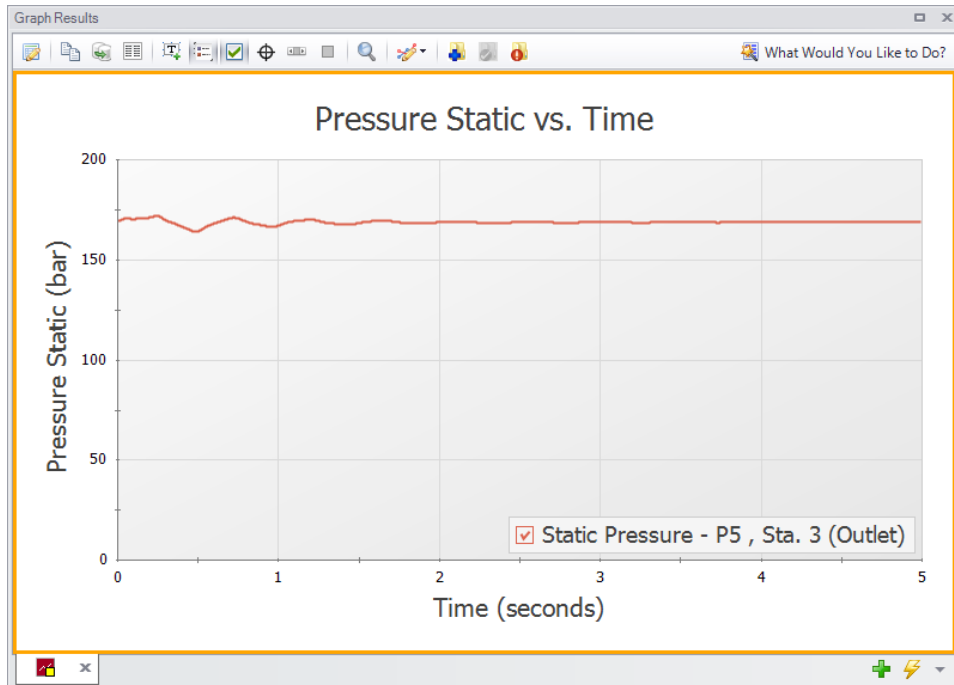


Figure 6: Static pressure at steam turbine 2 entrance

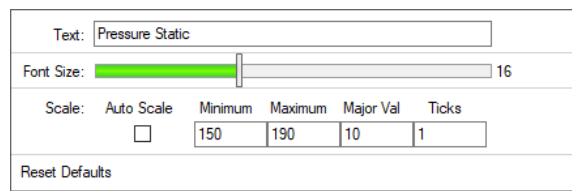


Figure 7: Axis Modification Window opened by right-clicking the y-axis

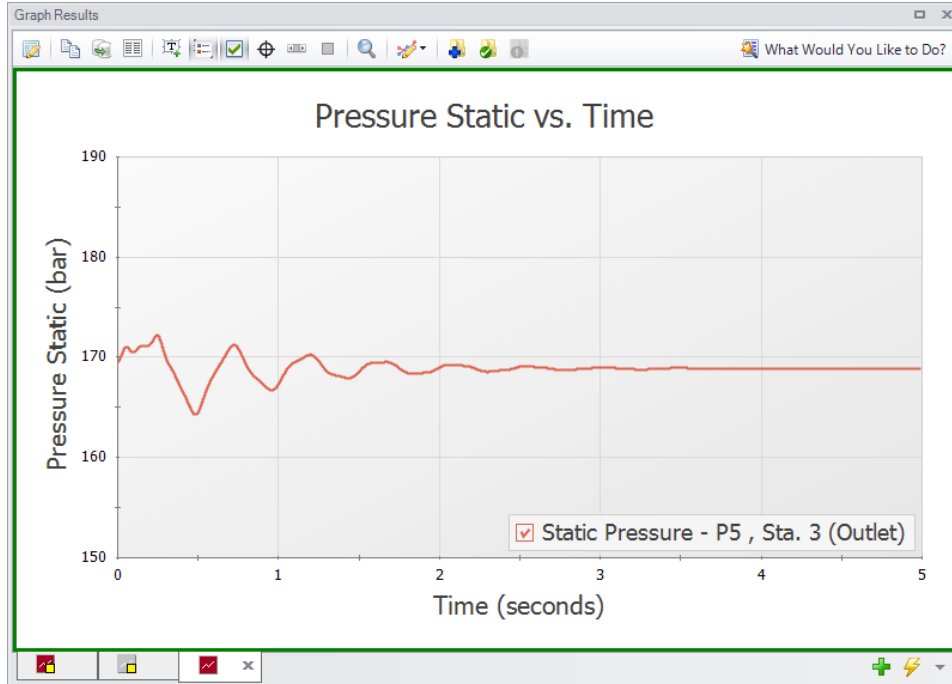


Figure 8: Static pressure at steam turbine 2 entrance with modified y-axis

## B. Animate the bypass line temperature and pressure

We will also animate the profile of the bypass line temperature and pressure in order to determine whether condensation occurs.

1. Create a new graph tab by clicking the New Tab button located in the bottom-right (green “+” icon).
2. Select the Profile tab under the Graph Control tab in the Quick Access Panel.
3. Select pipes 6, 8, 9, and 10.
4. Verify the Length Units are set to meters.
5. Select the option to Animate Using Output. To use this feature, data from all pipe stations must be saved (which is the default on the Pipe Sectioning and Output panel), otherwise the option will be grayed out.
6. Ensure that the Animation Time Units are set to seconds.
7. Add two additional parameter using the Add button, and specify the three parameters:
  - a. Pressure Static with units of bar
  - b. Temperature Static with units of deg. C
  - c. Temperature Saturation with units of deg. C
8. Make Temperature Saturation a secondary axis parameter by double-clicking in the empty box to the left of the name. Alternatively, select the parameter and click the right facing blue arrow above the list of parameters. If done successfully, the Temperature Saturation parameter should appear indented under Temperature Static.
9. Uncheck the Mx and Mn boxes only for the Temperature Saturation parameter.

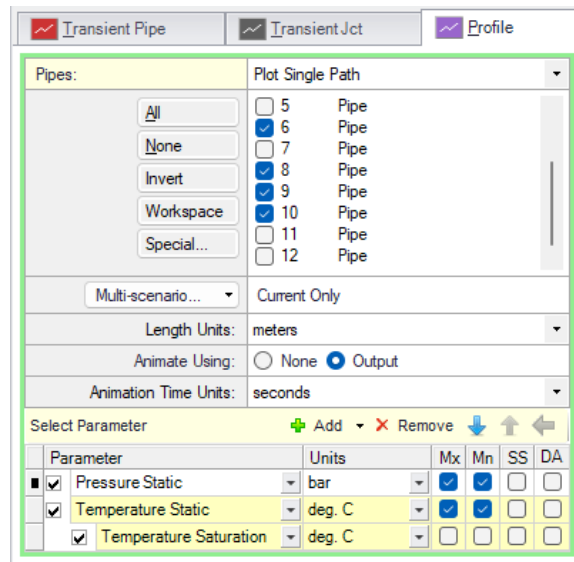
10. Click the Generate button.
11. Adjust the y-axis scaling for both Temperature Static and Temperature Saturation as follows:
  - a. Right-click on the temperature y-axis and uncheck the Auto Scale box
  - b. Set the Minimum to 300, Maximum to 600, and Major Val to 50.

---

**Important:** Ensure both the primary (left) and secondary (right) y-axes have the same max and min limits when both parameters have the same unit. This allows for easy visual comparison of the parameters.

---

12. Click the Play button to review the animation. The animation speed can be increased by using the Speed Slider at the right side of the playback controls.



**Figure 9: Graph Parameters for the bypass line animated profile**

Figure 10 is useful to determine whether or not condensation occurs in the system. Condensation can occur by a rise in pressure, a decrease in temperature, or a combination of both. Each fluid library contains saturation line data and warning messages will be given in the Output if the fluid properties drop into the two-phase region. In this model the temperatures stay well above the saturation line at all times, showing that the steam is not close to condensing.

---

**Reminder:** AFT xStream assumes all gases are superheated and does not model two-phase flow. Saturation data is only used for predicting the presence of saturation conditions, and the fundamental equations in the Solver assume the gas is single phase.

---



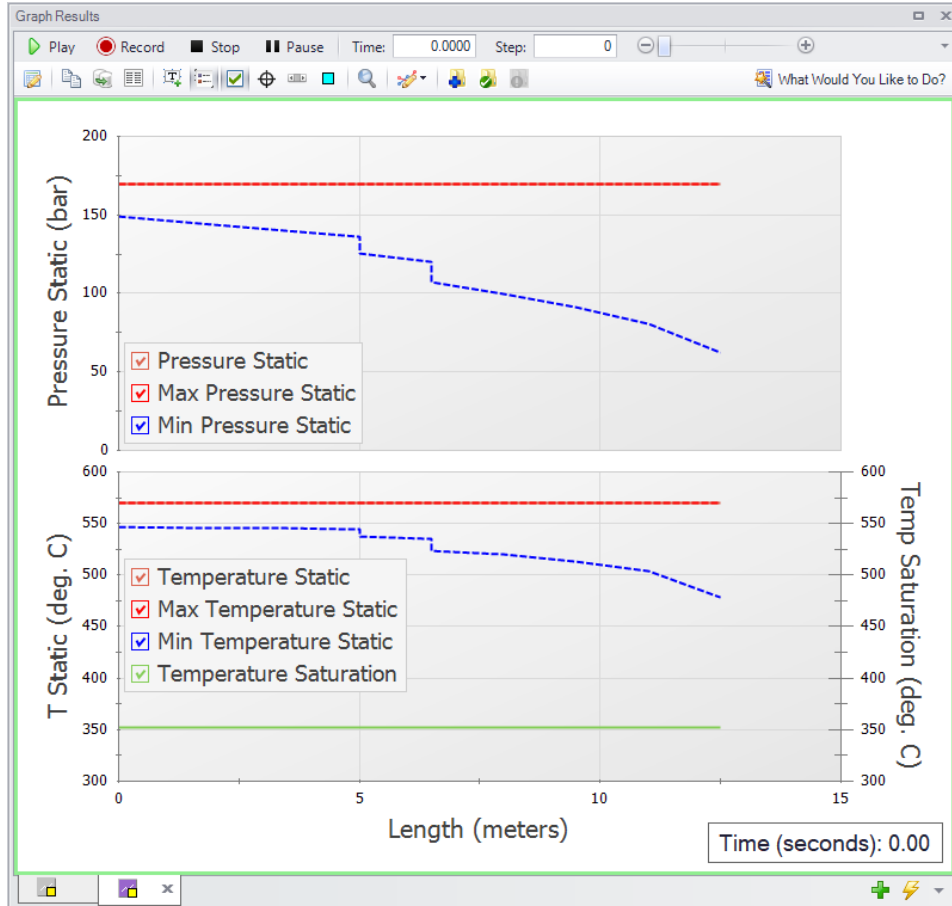


Figure 10: Bypass line animation profile

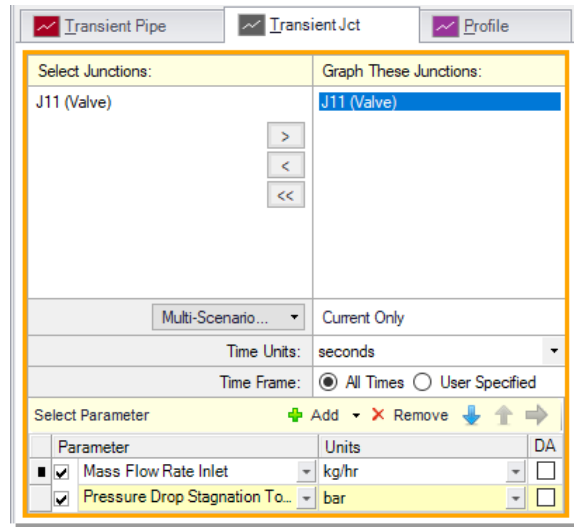
### C. Graph the mass flow rate and pressure drop through the bypass valve

Lastly, we shall create a graph to show the mass flow rate and the pressure drop through the bypass valve to gain insight into how the flow develops in the bypass line.

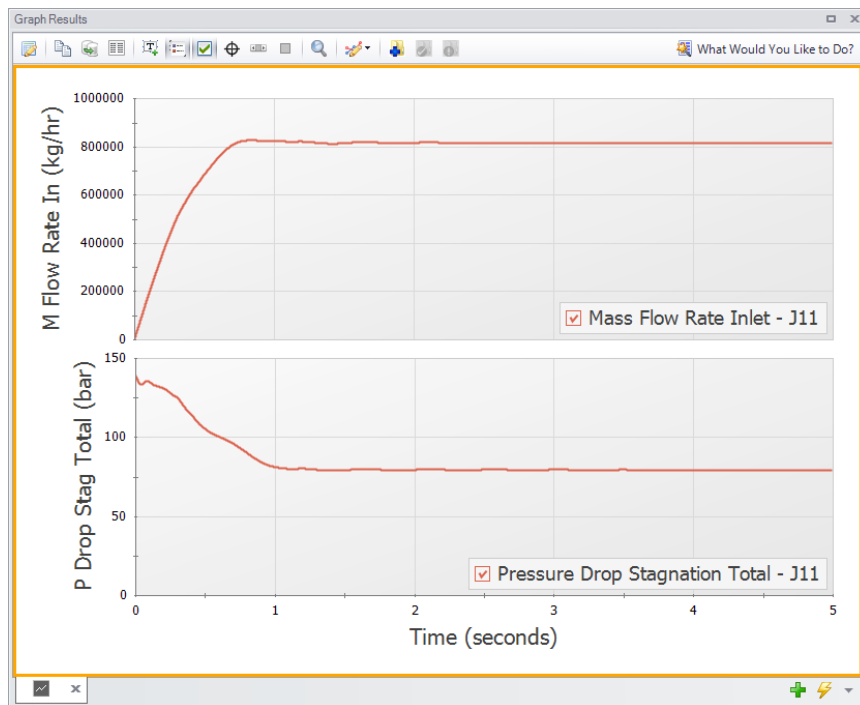
1. Create a new graph tab.
2. From the Graph Control tab on the Quick Access Panel, select the Transient Jct tab.
3. In the Select Junctions section, select J11 and click the “>” arrow.
4. Add a second parameter in the Select Parameters section, and specify the two parameters:
  - a. Mass Flow Rate Inlet with units of kg/hr
  - b. Pressure Drop Stagnation Total with units of bar
5. Click the Generate button.

As shown in Figure 12, the flow rate increases and the pressure drop decreases as the valve opens over time. While the observed trends for mass flow rate and pressure drop are typical for a valve that is opening, it is still worthwhile to make Transient Jct graphs when multiple transient events occur in order to fully understand the interactions between them.

**Note:**Figure 12 has modified axes titles and its legend removed for clarity.



**Figure 11: Graph Parameters for the Bypass Valve mass flow and pressure drop**



**Figure 12: Bypass valve mass flow rate and pressure drop transient**

## Step 5. Evaluate the effects of changing fluid libraries

### A. Review the results of other libraries

In addition to the AFT Standard fluid library, the model built in this example was run using Steam from the Chempak, NIST REFPROP, and ASME Steam libraries. Similar results to the AFT Standard fluid run were obtained for both the bypass valve and the turbine pressure vs time graph, so these results are not shown below. Nevertheless, it is sensible to run your system using different fluid libraries as they have been known to impact results. The temperature profile for the bypass line showed more significant differences, as is shown below comparing the results using NIST REFPROP vs AFT Standard (Figure 13). ASME Steam and Chempak gave nearly identical results to NIST, and therefore are not shown.

Note that the minimum temperature reached at J11 is about 32 degrees higher in the AFT Standard fluid library run compared to the NIST REFPROP run. The results for each fluid library showed that no condensation was found, and no warnings were given in the Output.

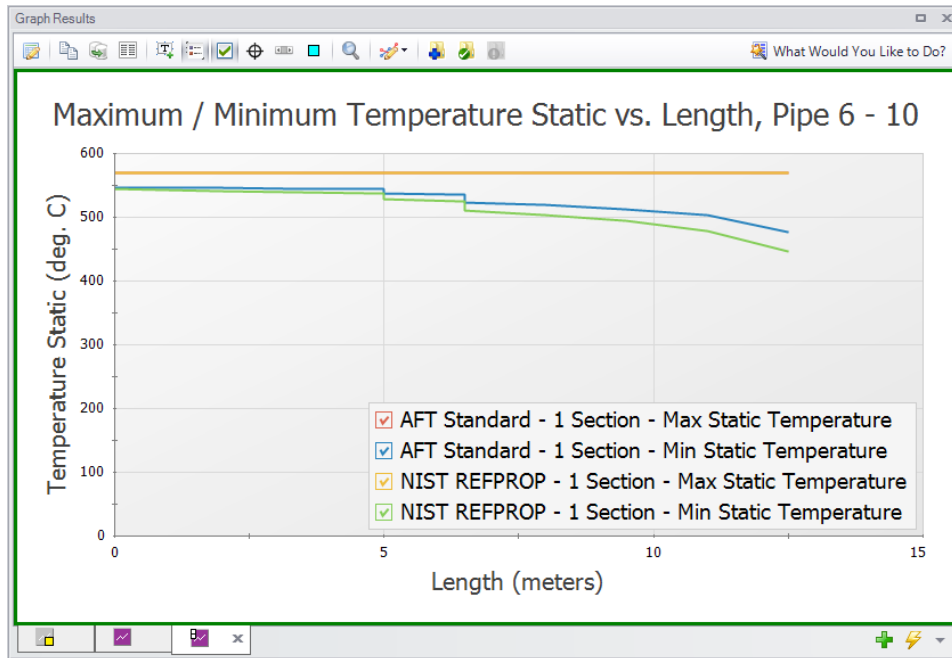


Figure 13: Bypass line profile using the NIST REFPROP fluid library

### B. Advice for preliminary runs

Obtaining final results from AFT xStream will require multiple iterations. To effectively work through those iterations, we recommend the iterative process begins with the following steps:

### **Step 1: Run the steady-state model with multiple fluid libraries**

It is recommended that the first runs are completed using a simplified version of your model in the *AFT Arrow™* Steady-State Solver with each fluid property model before considering transient conditions. This will allow for larger issues in the model to be caught before having to commit to Transient runs, which are more time consuming. This also can inform the types of variables that need to be considered during refined runs (i.e. if the *AFT Arrow™* steady-state runs are close to the saturation temperature, condensation is a phenomenon that should be monitored in subsequent runs).

### **Step 2: Complete a transient run using an AFT Standard fluid with minimal sectioning**

You should then run your simplified model in the Transient Solver using an AFT Standard fluid with minimal sectioning (i.e. a lower value for the Minimum Number of Sections Per Pipe setting on the Pipe Sectioning and Output panel in Analysis Setup). This is not always possible, such as when mixtures are present, but this step should always be attempted if allowable. This will allow for larger issues to be caught before committing to extended run times. Only after you are satisfied with the model should you attempt increasing the number of sections.

## **C. Selection of fluid property model for final runs**

A number of factors must be taken into consideration while selecting the fluid library for an AFT xStream model. If there are no mixtures, and the compound is present in the AFT Standard library, it is recommended that the AFT Standard library be the first library used. AFT Standard has the shortest run times of the different libraries and is an excellent choice for early runs focused on troubleshooting the model. However, if there is a need to model mixtures, and simplifying the system is not possible, initial runs should be conducted with NIST REFPROP or Chempak.

NIST REFPROP and Chempak both have greater accuracy than AFT Standard and can model mixtures. However, both have slower run times than AFT Standard. Chempak typically runs faster than NIST REFPROP with comparable accuracy. However, Chempak is an optional add-on for purchase while the NIST REFPROP library comes standard with AFT xStream.

Lastly, ASME Steam provides a well-established, trustworthy source of steam data. ASME Steam typically has the longest run times, and it is recommended that a high processing power computer paired with ample time be used to run your analysis if this library is selected. If either of those elements are unavailable, it is recommended that one of the other libraries be used, with the added note that NIST REFPROP is the most similar in accuracy to ASME Steam.

Table 1 summarizes the run times for this example with the different fluid libraries. The run times will be affected by the specifications of your computer as well as the memory available. Table 1 shows data for runs conducted on the laptops at AFT as of publication.

Additionally, Table 2 provides a library compatibility chart to assist with fluid property model selection.

It should be noted that if a tremendous amount of accuracy is needed, appropriate library selection will not be sufficient to give a high accuracy model. The single parameter that will have the greatest effect on accuracy is the Minimum Number of Sections per Pipe in the Sectioning panel. Ultimately, appropriate sectioning remains the single most important factor in creating a high accuracy model and should be weighed just as heavily as the fluid library when setting the parameters your transient gas system uses.

**Table 1: Comparison of run times with various fluid libraries**

Steam Turbine Shutdown Approx Run Times	
Fluid Property Model	Run Time
AFT Standard - 1 Section	2 - 3 minutes
Chempak - 1 Section	4 - 10 minutes
NIST REFPROP - 1 Section	0.9 - 1.4 hours
AFT Standard - 10 Sections	1 - 2 hours
ASME Steam - 1 Section	3 - 4 hours

**Table 2: Compatibility chart for fluid library selection for final runs**

Fluid Library	Model Requirements				
	Mixtures	Condensation Data	System Troubleshooting	Final Runs	Complex Systems*
AFT Standard	▪	✓	✓**	▪	✓
NIST REFPROP	✓	✓	▪	✓	✓
Chempak	✓	✓	▪	✓	✓
ASME Steam	▪	✓	▪	✓	▪

\* Evaluating the complexity of a system is most easily done in early runs of the model. It is suggested that all early runs should be run with a version of AFT Standard if they can, and the run time recorded. This run time should inform your decision on which fluid library is needed to run the more thorough runs when high accuracy results are needed. As seen from Table 1, the run times can vary wildly between the 4 libraries, so make sure you evaluate your time constraints and accuracy needs when weighing this decision. Factors that can impact the run time include the ratio in the length of the smallest pipe to the largest pipe, presence of mixtures, usage of Junctions containing resistance curves, and the presence of sonic choking. See the [Reducing Run Time](#) section for more information.

\*\* It is strongly recommended that all runs used to troubleshoot the design of the system are run using AFT Standard.

## Conclusion

Graphs were created showing the turbine inlet pressure over time, and animating the pressure and temperature in the bypass line. It was determined that no condensation occurs, regardless of which fluid library was used.

A standard procedure for initial and final runs was discussed:

- Step 1: Run the steady-state model with multiple fluid libraries
- Step 2: Complete a transient run using an AFT Standard fluid with minimal sectioning

Lastly, a comparison of the different fluid libraries was discussed. AFT Standard should be used for initial runs and troubleshooting (runs faster). NIST REFPROP, Chempak, and ASME Steam libraries should be used for final runs where higher accuracy is required; however, the trade off is that the model will have significantly longer run times.

# Pipe Forces from Turbine Shutdown Example

## Summary

---

Two turbines in a high pressure steam system have their valves simultaneously shut off, which causes the system to experience transient forces. This example describes how to determine the transient force data for the piping system during the steam flow transient. It also evaluates the effect of varying the time to close the valves on force magnitude. Lastly, this example shows how to export the force results to a force file to use in piping stress software.

## Topics Covered

- Calculating the forces associated with a transient event
- Using the Scenario Comparison Tool to compare scenarios
- Exporting force sets to CAESAR II®

## Required Knowledge

This example assumes that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the [Beginner - Tank Blowdown](#) example for more information on these topics.

## Model File

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

C:\AFT Products\AFT xStream 3\Examples

- ***Metric - Pipe Forces from Turbine Shutdown - Initial.xtr***
- ***Metric - Pipe Forces from Turbine Shutdown - Final.xtr***

## Step 1. Open the model file

---

For this example, we will be starting from a pre-built model file which has been partially set up for your convenience. From either the Startup window or the File menu, browse to the model file with the name shown above, using the variant with “- Initial” appended to the file name. The Workspace should appear as shown in Figure 1.

## Pipe Forces from Turbine Shutdown Example

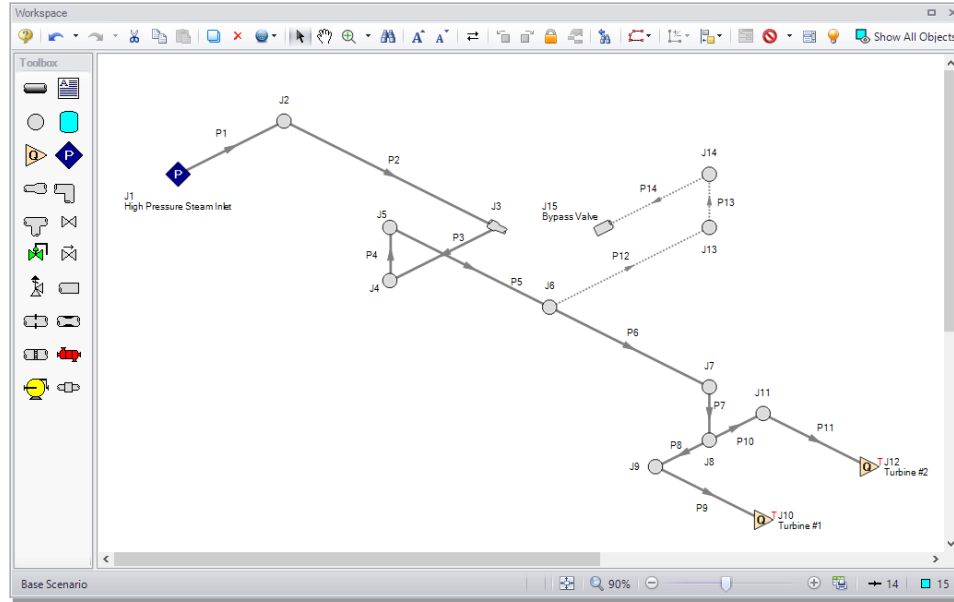


Figure 1: Model layout

### Step 2. Specify the Pipe Sectioning and Output panel

➤ **Open the Pipe Sectioning and Output panel in the Analysis Setup window.** Define the Minimum Number of Sections Per Pipe as 1 section and the Estimated Maximum Pipe Temperature During Transient as 600 degrees C.

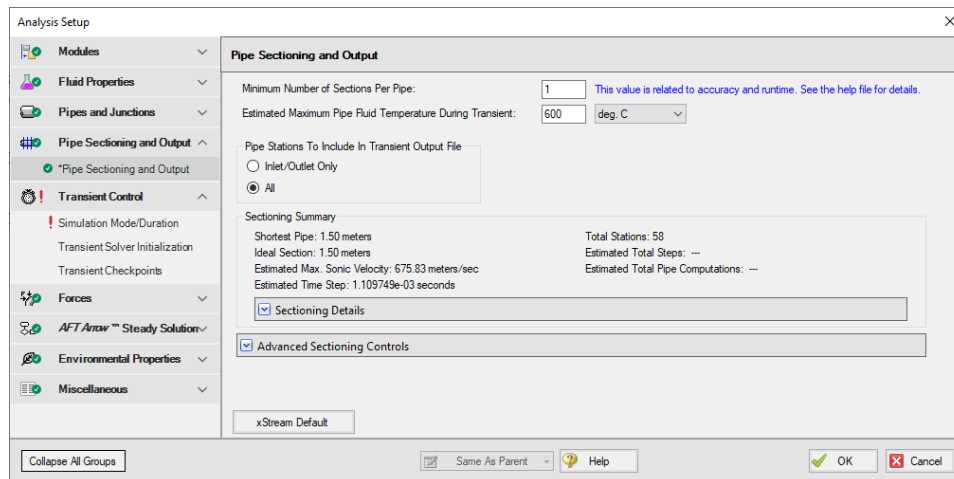


Figure 2: Pipe Sectioning and Output panel



### Step 3. Specify the Simulation Mode/Duration panel

---

➤ **Open the Simulation Mode/Duration panel from the Transient Control group.** Enter the Stop Time as 1 second. Click OK to save and close the Analysis Setup window.

### Step 4. Specify Force Sets

---

➤ **Open the Force Definitions panel from the Forces group.**

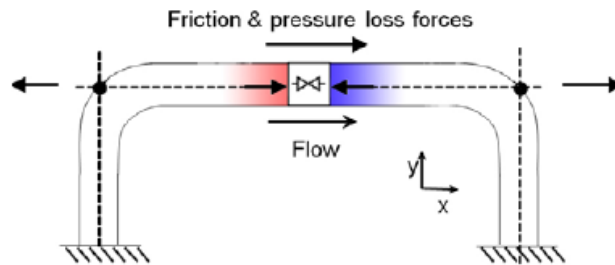
We will first add one Force Set from the Workspace.

#### A. Force Types

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

1. The force is between a pair of 90 degree elbows
2. The force is an axial pipe force
3. All pipes, fittings, and components between the elbow pair are in-line

The force equations in the Solver are specific to this geometry. The pipes and components between the elbow pair can be horizontal, vertical, or sloped, but the axial component of the fluid weight is not accounted for in the forces. AFT xStream only accounts for hydraulic forces, or those acting on the pipe from the fluid. Elevation and hydrostatic pressure are accounted for in the pressure and flow solution.



**Figure 3: Schematic of the Difference force type**

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

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

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

## B. Defining Force Sets

Force sets can be defined from the Workspace, or from within Analysis Setup. We will first add one Force Set from the Workspace. Generally it is best to represent each elbow with a unique junction on the Workspace (either an Elbow or Branch). This example assumes that the static pressure loss in the elbows is negligible, and therefore, they are modeled as Branch junctions which are lossless.

➤ **On the Workspace, right-click on Pipe P2** and from the right-click menu select Create a Force Set with Selected Pipe. Select the Use Pipe Information radio button and un-check the Include Appended Text option (Figure 4). Click Create. This will open the Force Definitions panel of the Analysis Setup window. The Force Definitions panel should appear as shown in Figure 5.

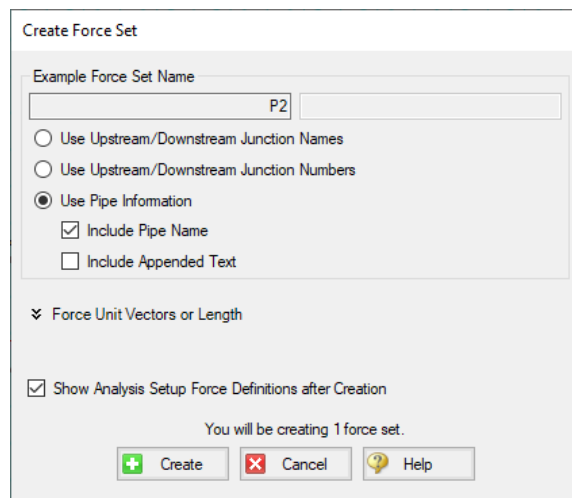
AFT xStream can only determine axial forces down a pipe in the direction defined as forward by the force set definition. As such, AFT xStream has no knowledge of the three-dimensional orientation of a given force set. For example, a vertical pipe and a horizontal pipe that both have the same force magnitude would be indistinguishable in AFT xStream. The force set definition informs the sign of the force, with a positive value indicating a force pushing from the start location to the end location, and a negative value indicating the converse.

If a more robust view of the 3D orientation of the pipes is required, directional information can be entered using the optional Force Unit Vector columns. The Force Unit Vectors do not impact AFT xStream's calculations, but may be useful to export the output to piping stress analysis software as discussed in Step 10

---

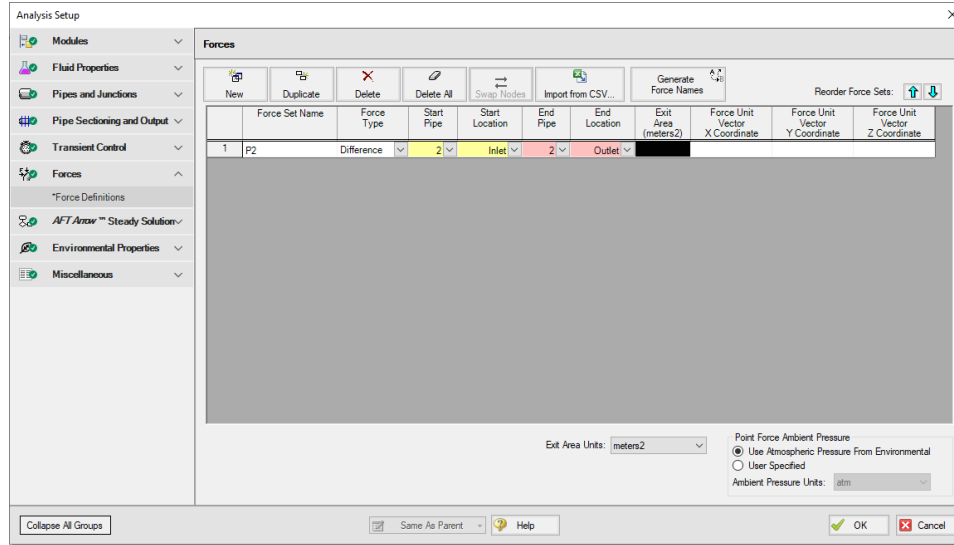
**Note:** Only single pipes can be added as a Difference force set type from the Workspace.

---



**Figure 4:** Adding a force set from the Workspace using the right-click menu

## Pipe Forces from Turbine Shutdown Example



**Figure 5: First Force Set added to the Force Definitions panel in the Analysis Setup window**

The remaining force sets will be defined inside the Force Definitions panel.

➤ **Define a force set by clicking New**, selecting the Force Type (in this example, we will be using Difference because all force sets are between 90 degree elbow pairs), entering a name, and defining the starting and ending locations of the force set. Set up the force sets as shown in Figure 6. Click OK to save and close the Analysis Setup window.

1. P2, Difference, P2 Inlet to P2 Outlet (previously created from the Workspace)
2. P3, Difference, P3 Inlet to P3 Outlet
3. P4, Difference, P4 Inlet to P4 Outlet

➤ **Double-Check Force Set Assumptions:** It is important to consider if the force sets adhere to the assumptions mentioned previously which apply to the Difference force type. All of the force sets are located between pairs of 90 degree elbows (represented as lossless Branch junctions). Furthermore, each force set spans a single pipe and there are no branching paths in the middle of the force sets. Therefore, the defined Difference force sets are valid and compatible with the equations programmed into AFT xStream.

---

**Note:** A Difference force set that spans Pipes P5-P6 would be between a pair of 90 degree elbows, and would be horizontally in-line; however, it would not be valid since there is a branching flow path in the middle. Even if the flow path from Pipe P12 to the closed bypass valve starts as stagnant in the steady-state, it can be affected by the transient, and could potentially serve as an anchoring support for Pipes P5 and P6. The same applies to Pipe P7, where the Difference force set would not be valid due to the branching flow at the endpoint.

---

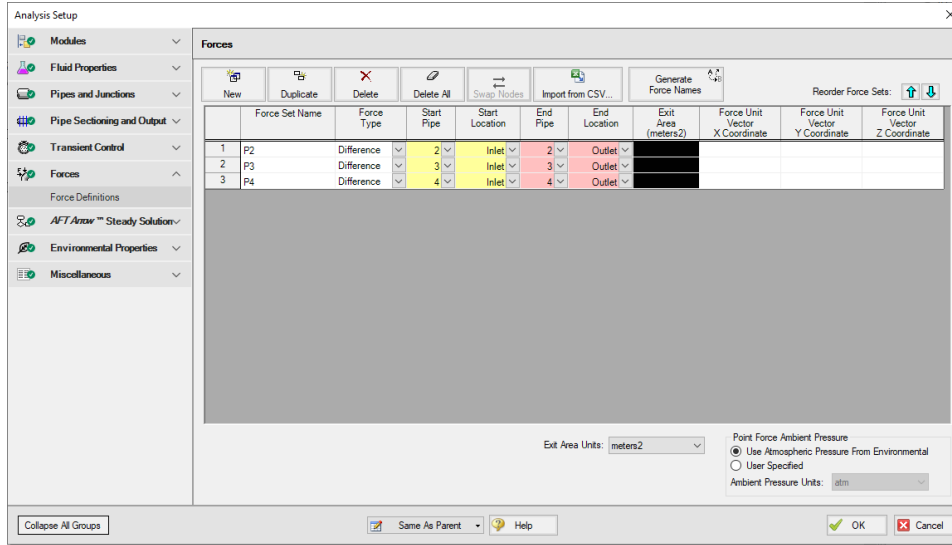


Figure 6: Completed Force Definitions panel in the Analysis Setup window

## Step 5. Create child scenarios using the Scenario Manager


### A. Create scenarios

In this model, we will evaluate three different valve closure durations for the system:

1. 0.1 Second Valve Closure
2. 0.25 Second Valve Closure
3. 0.5 Second Valve Closure

We will create three child scenarios to model the three cases.

➤ **Go to 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

and then selecting the Create Child icon . Enter the name "0.1 Second Valve Closure" in the Create Child Scenario window, and click OK. The new scenario should now appear in the Scenario Manager on the Quick Access Panel below the Base Scenario.

➤ **Right-click the Base Scenario in the Quick Access Panel.** Create another child and call it "0.25 Second Valve Closure". Finally, create one more child and call it "0.5 Second Valve Closure". The child scenarios should now be displayed as shown in Figure 7.

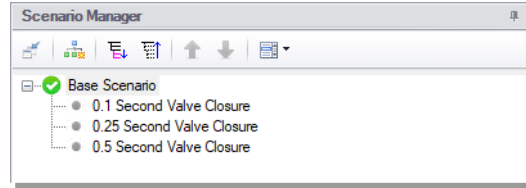


Figure 7: Scenario Tree with additional child scenarios created

## B. Define the child scenarios

Since the "0.1 Second Valve Closure" scenario uses the same transient definitions as the base scenario, we do not need to modify it. Having a duplicate of the parent scenario is useful for organizing Multi-Scenario graph selections and keeping your scenario tree intuitive. We will set up the remaining scenarios using the following steps:

1. Load the "0.25 Second Valve Closure" scenario by double-clicking the name in the Scenario Manager.
2. Open the Assigned Flow Properties window for Assigned Flow J10 and open the Transient tab. Enter the following data:

<b>Time (seconds)</b>	<b>Mass Flow(kg/sec)</b>	<b>Static Temperature (deg. C)</b>
0	62.75	570
0.25	0	570
1	0	570

3. Enter the same Transient Data for Assigned Flow J12.
4. Open the Child Scenario "0.5 Second Valve Closure" from the Scenario Manager.
5. Open the Assigned Flow Properties window for Assigned Flow J10 and open the Transient tab. Enter the following data:

<b>Time (seconds)</b>	<b>Mass Flow(kg/sec)</b>	<b>Static Temperature (deg. C)</b>
0	62.75	570
0.5	0	570
1	0	570

6. Enter the same Transient Data for Assigned Flow J12.

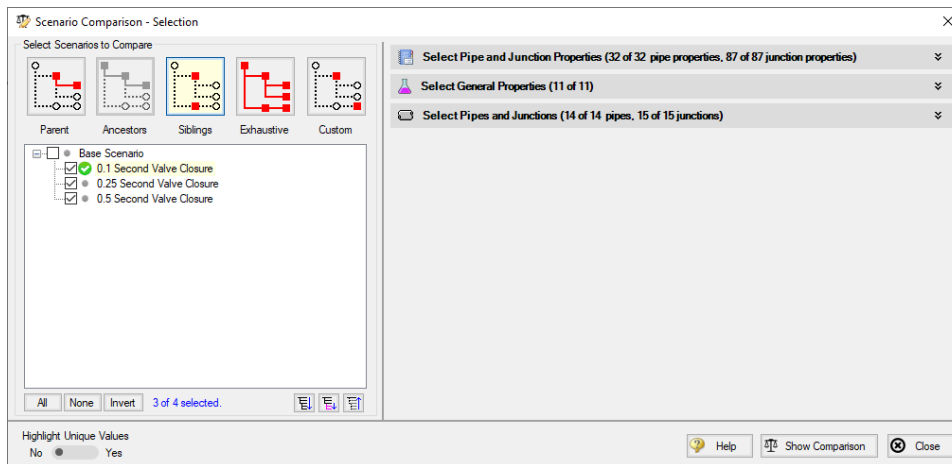
## Step 6. Compare child scenarios using the Scenario Comparison Tool

The three scenarios created should be identical except for the different transient data. To confirm the scenarios are otherwise identical, we will use the Scenario Comparison Tool.

## Pipe Forces from Turbine Shutdown Example

1. Load the "0.1 Second Valve Closure" scenario from the Scenario Manager.
2. In the Tools menu, select the Scenario Comparison Tool.
3. In the Scenario Comparison - Selection window (Figure 8), select the Siblings option.
4. Click Show Comparison.
5. In the Scenario Comparison Grid, set the Item and Parameter sliders to show Differences. Also set the Highlight Unique Values slider to Yes.

The Scenario Comparison Tool will run a comparison of all Pipe and Junction Properties as well as the General Properties used in each scenario and highlight any differences in the Scenario Comparison Grid. In this model, the only highlighted differences should be for the First Transient Data parameter for junctions J10 and J12 (Figure 9). Note that each column in the Scenario Comparison Grid is highlighted in a different color. This indicates that each scenario possesses a unique value for the given parameters. Click close to exit the Scenario Comparison Tool.



**Figure 8: Scenario Comparison Tool - Selection window**

Item	Parameter	Scenario		
		0.1 Second Valve Closure	0.25 Second Valve Closure	0.5 Second Valve Closure
Pipes (14 of 14 compared) - No Differences				
Junctions (15 of 15 compared)				
J10 (Turbine #1)	First Transient Data	Yes - Table	Yes - Table	Yes - Table
J12 (Turbine #2)	First Transient Data	Yes - Table	Yes - Table	Yes - Table
Modules (1 of 1 compared) - No Differences				
Fluid Properties (5 of 5 compared) - No Differences				
Pipe Sectioning (3 of 3 compared) - No Differences				
Transient Control (1 of 1 compared) - No Differences				
Steady Solution Control (3 of 3 compared) - No Differences				
Environmental Properties (2 of 2 compared) - No Differences				
Miscellaneous (1 of 1 compared) - No Differences				

**Figure 9: Scenario Comparison Grid of the sibling scenarios**

---

## Step 7. Run the first scenario

---

➤ **Double-click the "0.1 Second Valve Closure" child scenario in the Scenario Manager.** Select Run Model from the Analysis menu. This will open the Solution Progress window. Once finished, click Output on the Solution Progress window.

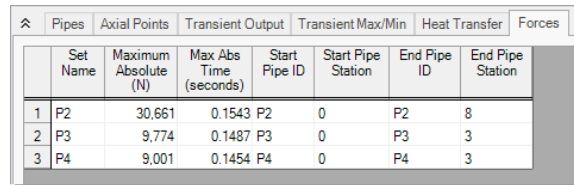
---

## Step 8. View the results

---

### A. Open the Forces tab

➤ **In the Output window, click on the Forces tab.** This will show the maximum force observed in the defined force sets, as well as provide information about how the force segments were defined.



	Set Name	Maximum Absolute (N)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	30,661	0.1543	P2	0	P2	8
2	P3	9,774	0.1487	P3	0	P3	3
3	P4	9,001	0.1454	P4	0	P4	3

*Figure 10: Forces tab in the Output window for the "0.1 Second Valve Closure" scenario*

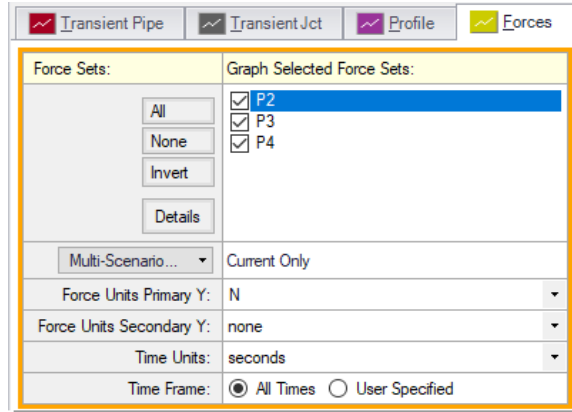
### B. Graph the force results

➤ **Open the Graph Results window.** We will create a graph that shows the transient forces over time for the force sets. To do so, perform the following steps:

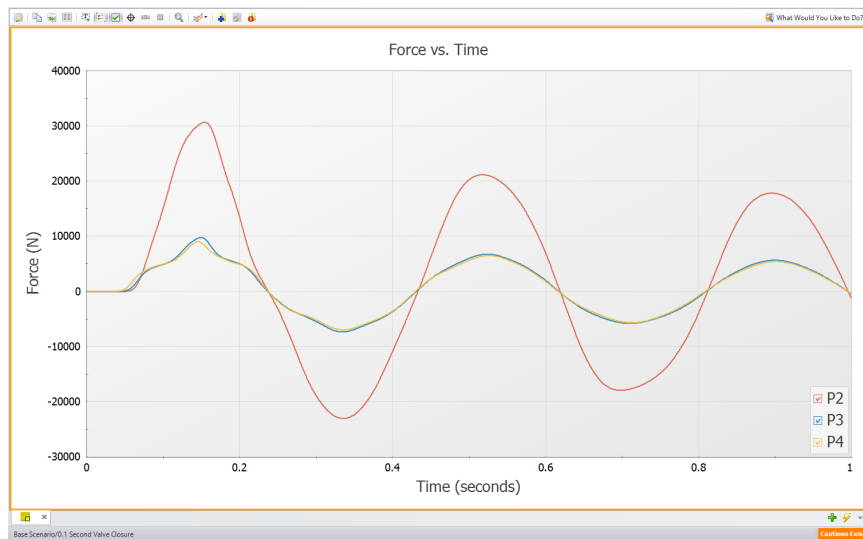
1. Select the Forces tab on the Graph Control tab on the Quick Access Panel, as shown in Figure 11.
2. Click the All button such that all force sets are selected from the list to be graphed.
3. Verify the Force Units Primary Y is set to N.
4. Verify that Time Units is set to seconds.
5. Verify that the Time Frame is set to All Times.
6. Click Generate.

This should generate the graph shown in Figure 12.

Add the graph to the Graph List Manager by right-clicking the graph results window and selecting "Add Graph To List...", or by using the icon on the toolbar. Name the graph "Force Sets" and click OK.



**Figure 11: Graph parameters for force sets**



**Figure 12: Force vs Time graph for the "0.1 Second Valve Closure" scenario**

Note that at Time = 0 seconds, there are no forces for any of the force sets. This is expected since the sum of forces across a piping run between two elbows equals zero in a steady-state system. If a Point or Exit type force set were used instead of a Difference type force set, then a non-zero force may be encountered during the steady-state.

Additionally, some traditional methods of analyzing force sets will not yield the same results as AFT xStream since they may not include friction or momentum effects in their force balances. Traditional methods can yield significantly different results and indicate incorrect, non-zero forces during steady-state conditions.

There are two important points to be observed here:

1. AFT xStream calculates transient fluid forces. This does not include weight from piping, components, or the fluid itself, or any other forces external to the piping. A comprehensive analysis of pipe loading must separately include these items.



## Pipe Forces from Turbine Shutdown Example

2. Ignoring friction and momentum force balance components will result in force imbalances that do not exist in reality.

### Step 9. Run the other child scenarios

➤ Run the other two child scenarios and load the Force Sets graph list item for each. The results are shown in the figures below.

	Set Name	Maximum Absolute (N)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	12,722	0.2164	P2	0	P2	8
2	P3	4,123	0.2175	P3	0	P3	3
3	P4	4,054	0.2109	P4	0	P4	3

Figure 13: Forces tab in the Output window for the "0.25 Second Valve Closure" scenario

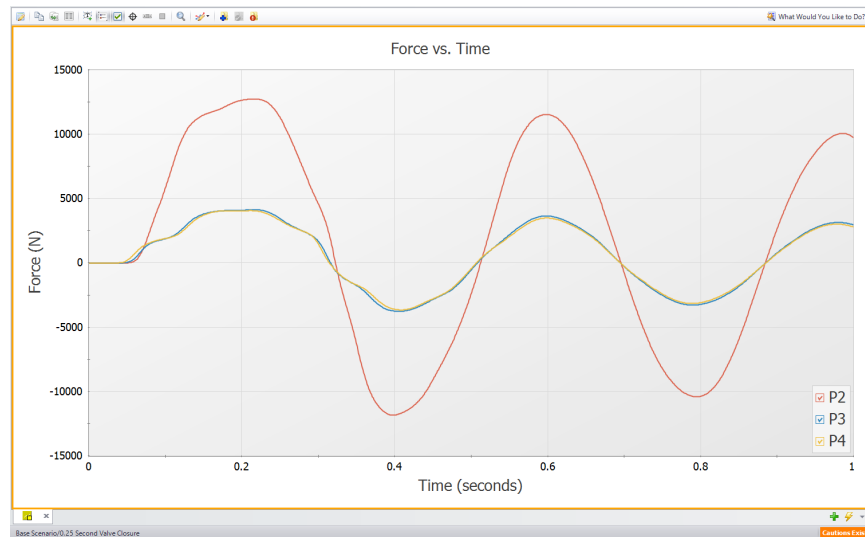
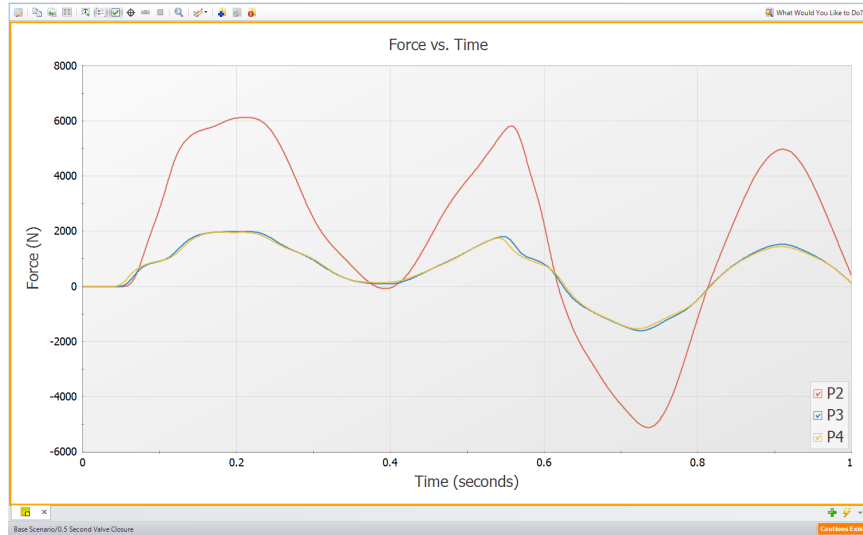


Figure 14: Force vs Time graph for the "0.25 Second Valve Closure" scenario

	Set Name	Maximum Absolute (N)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	6,126	0.2120	P2	0	P2	8
2	P3	1,984	0.2153	P3	0	P3	3
3	P4	1,967	0.1765	P4	0	P4	3

Figure 15: Forces tab in the Output window for the "0.5 Second Valve Closure" scenario

## Pipe Forces from Turbine Shutdown Example



**Figure 16: Force vs Time graph for the "0.5 Second Valve Closure" scenario**

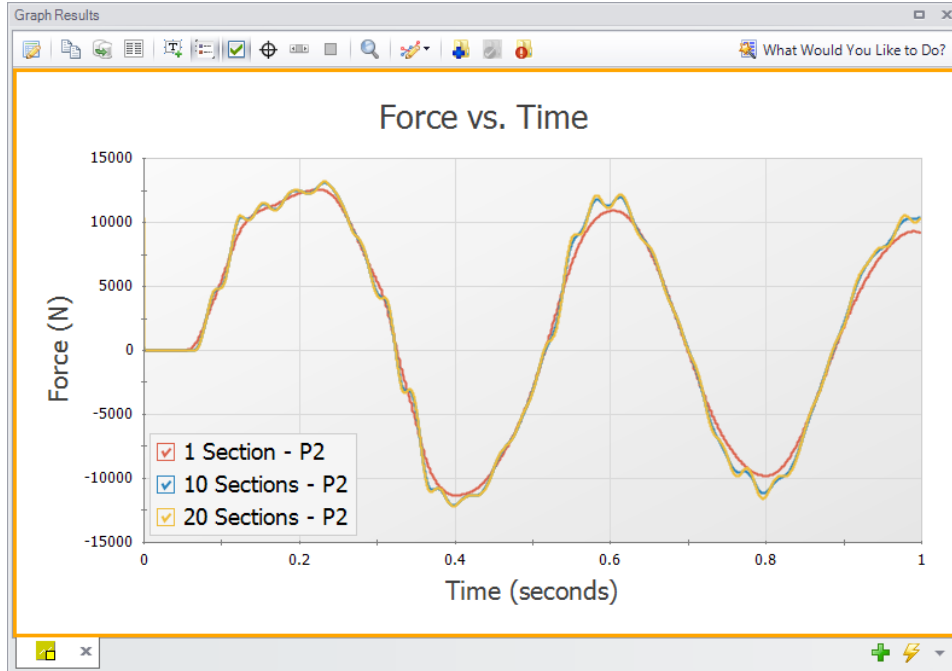
These figures show that the maximum force magnitude increases as the valve closes faster. Typically, the more rapidly a change occurs, the more severe the resulting forces are. Therefore, a utility of AFT xStream would be to evaluate the correlation between transient event duration (e.g., valve closing time) and the resultant force magnitudes.

### ***Factors affecting the timing and magnitude of forces***

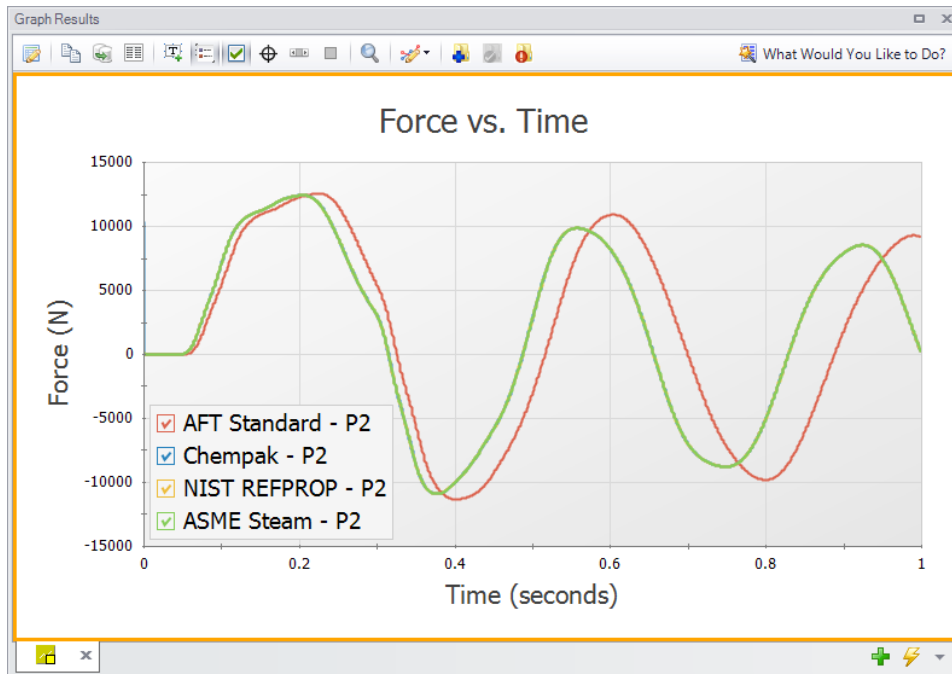
The pipe sectioning, fluid property model, and specification of the Maximum Pipe Mach Number During Transient setting can have a significant effect on the accuracy of the forces calculated by AFT xStream. These effects are not significant for the system built within this example. However, the results for altering each of these three parameters are shown in the figures below so that you may see the impact of changing these three parameters. Note that in Figure 18, Steam from the ASME Steam, Chempak, and NIST REFPROP fluid libraries are superimposed on each other, indicating no meaningful difference for this case.

If you wish to learn more about the effect of altering these 3 factors with regards to force, see the [Transient Sensitivity Analysis Tutorial](#).

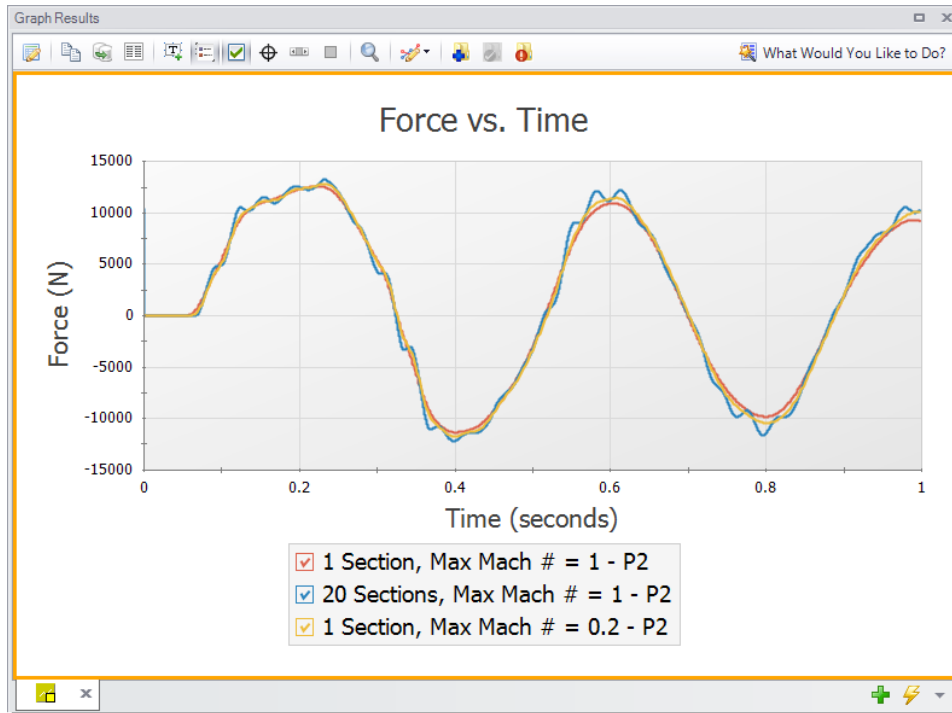
## Pipe Forces from Turbine Shutdown Example



**Figure 17: Effect of sectioning when all scenarios use the AFT Standard fluid library and a Maximum Pipe Mach Number During Transient of 1**



**Figure 18: Effect of the fluid property model when all scenarios use a Minimum Number of Sections Per Pipe of 1 and an Estimated Maximum Pipe Mach Number During Transient of 1**

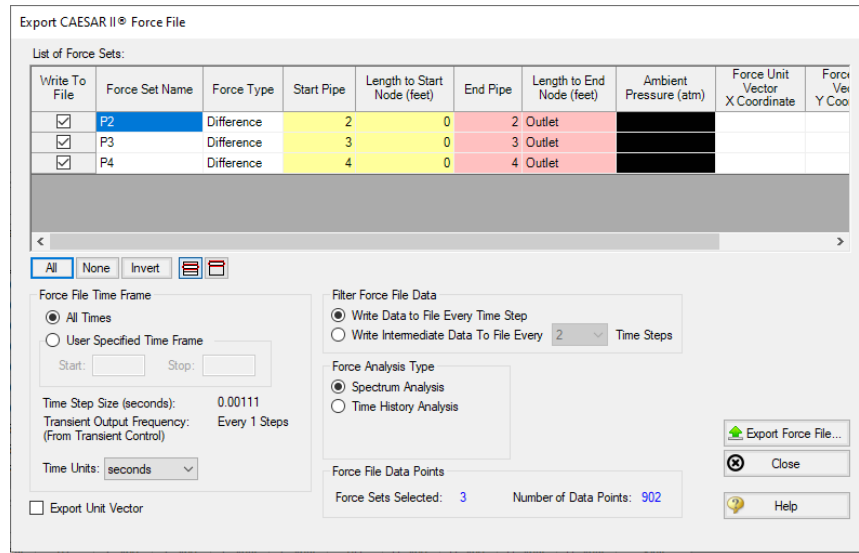


**Figure 19: Effect of Maximum Pipe Mach Number During Transient when all scenarios use the AFT Standard fluid library**

### Step 10. Export to a CAESAR II Force File

➤ **Open the Output window, and click on the File menu.** Select Export Force File... and choose CAESAR II® Force File. This will open the export file window shown in Figure 20.

## Pipe Forces from Turbine Shutdown Example



**Figure 20: Export CAESAR II® Force File window**

The force sets to include in the force file may be individually selected, along with how frequently data points are saved. The number of data points that will be in the file is also displayed for the given settings. This number should not exceed the maximum number of data points allowed for the relevant piping stress software, which in the case of CAESAR II® is 2500.

If unit vector information has been entered in the Force Definitions panel, the Export Unit Vector option can be used to include this information in the exported file. Similar methods to those described above can be used to export force files to ROHR2®, TRIFLEX®, and AutoPIPE®.

## Conclusion

Force sets were defined to calculate the transient forces between pairs of 90 degree elbows (represented by lossless Branch junctions) as a result of two simultaneous Turbine Safety Valve closures in a steam system. Several child scenarios were created to evaluate the effect of the closure time where we noted that a faster valve closure produced higher magnitudes of forces. Several important assumptions and takeaways were discussed:

1. The Difference [force type](#) is intended to model the specific case where all of the following criteria is true:
  - a. The force is between a pair of 90 degree elbows
  - b. The force is an axial pipe force
  - c. All pipes, fittings, and components between the elbow pair are in-line
2. AFT xStream calculates transient fluid forces. This does not include weight from piping, components, or the fluid itself, or any other forces external to the piping. A comprehensive analysis of pipe loading must separately include these items.
3. Ignoring friction and momentum force balance components will result in force imbalances that do not exist in reality.

Finally, we discussed how to export these forces into a force file to be used in pipe stress analysis software.

# Frequency Analysis - PFA Example

## Summary

---

The preliminary design for a system uses a positive displacement reciprocating compressor that is single-acting and has two heads (cylinders) with a 180 degree phase offset. It is suspected that the design may have pulsation resonance issues and needs to be checked for compliance with the API 618 standard. Perform a frequency analysis by placing a forcing function on the PD compressor and observing the frequency response of the system. Determine the worst acoustic natural frequency of the system, run the compressor at the worst-case RPM, and determine if the API 618 limits are violated.

## Topics Covered

- Using the PFA Module to identify natural acoustic frequencies
- Defining valid sectioning and simulation duration settings for pulsation analysis
- Identifying problematic compressor speeds based on pulsation frequency and cylinder arrangement
- Using periodic flow to represent a reciprocating compressor
- Checking API 618 compliance

## Required Knowledge

This example assumes that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the [Beginner - Tank Blowdown](#) example for more information on these topics.

## Model File

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

C:\AFT Products\AFT xStream 3\Examples

- *Metric - Frequency Analysis - PFA.xtr*

## Step 1. Activate the PFA module

---

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

## Step 2. Build the model

### A. Place the pipes and junctions

The next step is to define the pipes and junctions. In the Workspace window, assemble the model as shown in Figure 1.

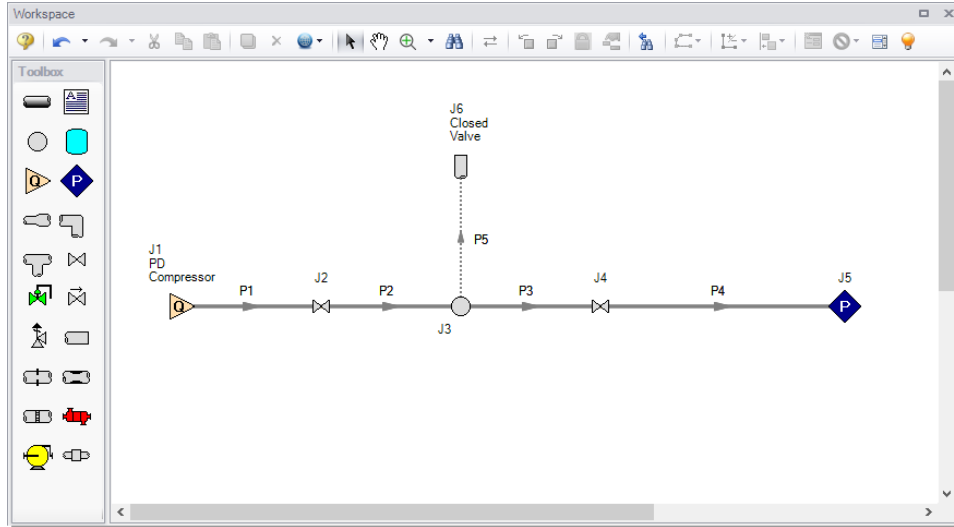


Figure 1: Layout of frequency analysis model

### B. Enter the pipe and junction data

The system is in place but now we need to enter the input data for the pipes and junctions. Enter the following data:

All Pipes are Steel - ANSI, STD (schedule 40), and have standard roughness. The rest of the pipe information is as follows:

Pipe #	Length (meters)	Size (inches)
1	8	10
2	8	10
3	8	10
4	80	16
5	3	6

1. Assigned Flow J1
  - a. Elevation = 3.5 meters
  - b. Name = PD Compressor
  - c. Mass Flow Rate = 2 kg/sec

---

**Note:** This is the average bulk flow produced by the compressor.

---
- d. Temperature = 38 deg. C
2. Valve J2
  - a. Elevation = 3.5 meters
  - b. Loss Source = User Specified
  - c. Loss Model = Kv
  - d. Kv = 385
  - e. xT = 0.8
3. Branch J3
  - a. Elevation = 3.5 meters
4. Valve J4
  - a. Elevation = 0 meters
  - b. Loss Source = User Specified
  - c. Loss Model = Kv
  - d. Kv = 1380
  - e. xT = 0.6
5. Assigned Pressure J5
  - a. Elevation = 0 meters
  - b. Stagnation Pressure = 0 barG
  - c. Temperature = 38 deg. C
6. Dead End J6
  - a. Elevation = 3.5 meters
  - b. Name = Closed Valve

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

➤ **Turn on List Undefined Objects** from the Main Toolbar to verify if all data is entered. If it is not, the undefined pipes and junctions window will show a list of incomplete items. If there are undefined objects, go back to the incomplete pipes or junctions and enter the missing data.

---

## Step 3. Specify Analysis Setup

---

### A. Fluid panel

- **Open the Analysis Setup window** and select the Fluid panel under the Fluid Properties group.
- **Select Methane from the AFT Standard fluid library, and click the Add to Model button.**



## **B. Pipe Sectioning and Output panel**

➤ **Open the Pipe Sectioning and Output panel and define the following settings:** in

1. Minimum Number of Sections Per Pipe = 8 sections
2. Estimated Maximum Pipe Fluid Temperature During Transient = 65 deg. C

## **C. Simulation Mode/Duration panel**

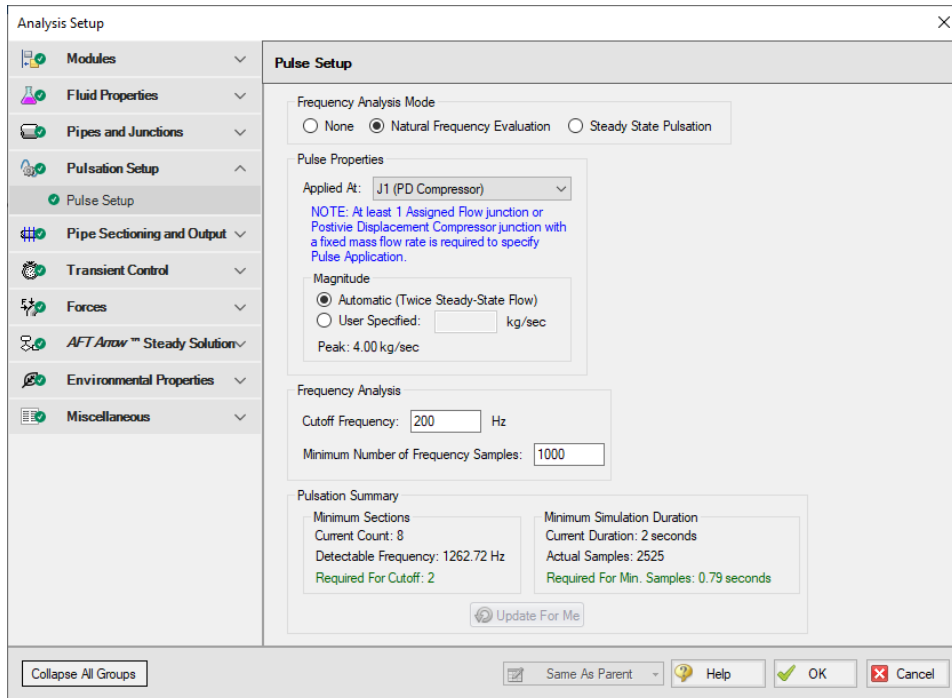
➤ **Enter a Stop Time of 2 seconds on the Simulation Mode/Duration panel** in the Transient Control group.

## **D. Pulse Setup panel**

➤ **Open the Pulse Setup panel** from the Pulsation Setup group. The Pulse Setup panel is used to specify the location and magnitude of the pulse applied to the system, as well as the parameters required to generate the forcing function for frequency analysis.

➤ **Define the Pulsation inputs** as follows (Figure 2):

1. Select Natural Frequency Evaluation for the Frequency Analysis Mode.
2. Select junction J1 as the location for the pulse to be applied at.
3. For Magnitude, ensure that Automatic (Twice Steady-State Flow) is selected.
4. Under Frequency Analysis, define the Cutoff Frequency as 200 Hz and verify that the Minimum Number of Frequency Samples is set to the default value of 1000.



**Figure 2: Pulse Setup panel in the Analysis Setup window with all pulsation settings defined**

Now that all of the pulsation inputs are defined, look at the Pulsation Summary at the bottom of the window. The defined Cutoff Frequency and the Minimum Number of Frequency Samples will affect the number of sections and the simulation duration that are required to run the analysis. If the number of sections is insufficient, then the time step will be too large to capture the specified Cutoff Frequency. Similarly, if the simulation duration is too small, then the number of time steps will not accommodate the Minimum Number of Frequency Samples.

Checking the Pulsation Summary for this example, the Minimum Sections Required for Cutoff are 2, and the Minimum Simulation Duration Required for Min. Samples is 0.79 seconds. Both of these requirements are satisfied by the current model settings. The Update For Me button can be used to automatically change the minimum number of sections and/or the minimum simulation duration if they are insufficient. Click OK to save and close the Analysis Setup window.

## Step 4. Run the model

➤ **Select Run Model** from the Common Toolbar. This will open the Solution Progress window. This model has an estimated run time of 5 minutes, but the run time is dependent on the speed of your PC.

## Step 5. Ensure the pressure response stabilizes

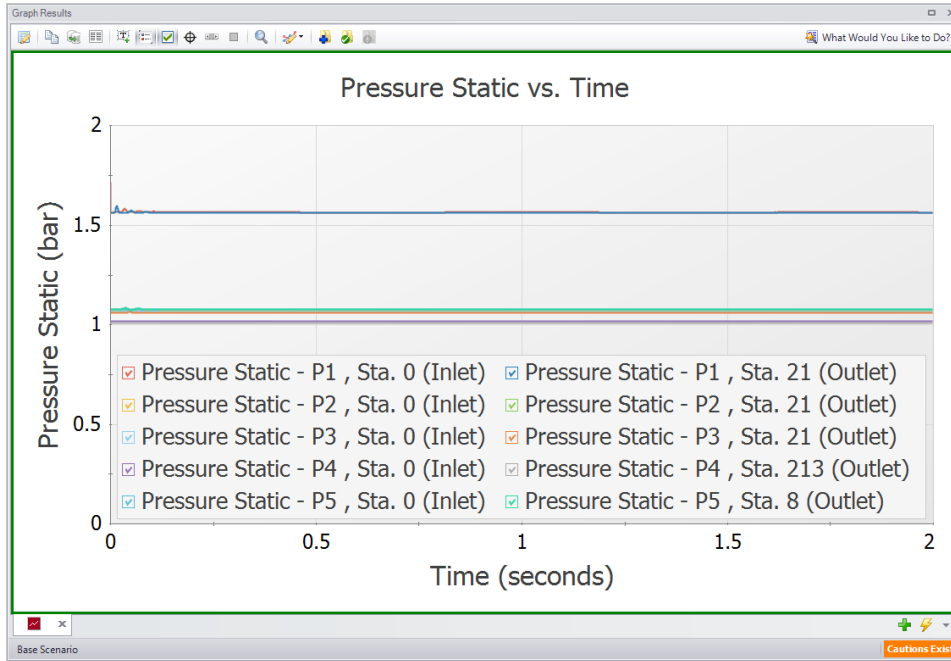
➤ **Open the Graph Results window.** The first step will be to generate a plot of the pressure in each of the pipes to ensure that the simulation duration was long enough for the response to stabilize.

➤ **Create the pressure graph as follows:**

## Frequency Analysis - PFA Example

1. Select the Transient Pipe tab from the Graph Control tab of the Quick Access Panel
2. Add the Inlet and Outlet station for each of the pipes
3. Choose Pressure Static as the Parameter in units of bar
4. Click Generate

The pressure graph should appear as shown in Figure 3. Note that there is a small perturbation in the pressure when the pulse is applied to the system, though the pressure results completely stabilize by the end of the run. If the pressure results were still changing at the end of the run, the model should be re-run with a longer simulation duration.



**Figure 3: Static Pressure vs Time in each pipe shows that the Simulation Duration is sufficient**

### Step 6. Graph the frequency response

➤ Next, graph the acoustic frequency response as follows:

1. Click the New Tab button (green "+" icon in the bottom-right)
2. Select the Frequency tab in the Graph Parameter section of the Quick Access Panel (Figure 4)
3. Add the Inlet and Outlet station for each of the pipes
4. Verify that the pressure units are set to bar
5. Click the Generate button

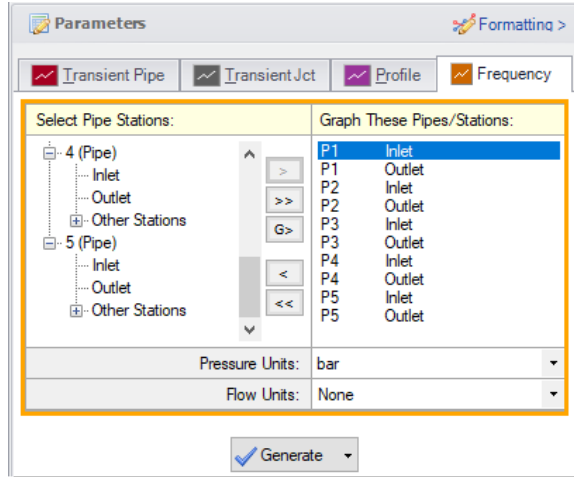


Figure 4: Graph Parameters section for the frequency analysis

**Identify excitation frequencies to study**

From the graph in Figure 5, there are several frequencies that produce a large pressure response. Points on the plot with a large response magnitude are easily identified by finding local maxima on the graph. The frequency and amplitude associated with these maxima can be highlighted on the graph by clicking and dragging the cursor from left to right highlighting the location of the local maximum. These maxima are the acoustic natural frequencies that are potentially the most damaging when excited by a positive displacement compressor. These frequencies can also be caused by equipment that contributes large pressure drops, such as control valves.

Two natural frequencies are identified for the system based on the peaks in the graph at roughly 28 Hz and 56 Hz. Note that the number of sections used for the analysis may affect the location of the peaks, so it is prudent to run the analysis with more sections to reduce uncertainty in the results. It is also recommended to analyze frequencies near identified peaks (for example, analyzing 55 and 56 Hz for the peak at 56 Hz) to account for potential shifting of the frequency peaks that can result from the assumptions made for the frequency analysis in AFT xStream.

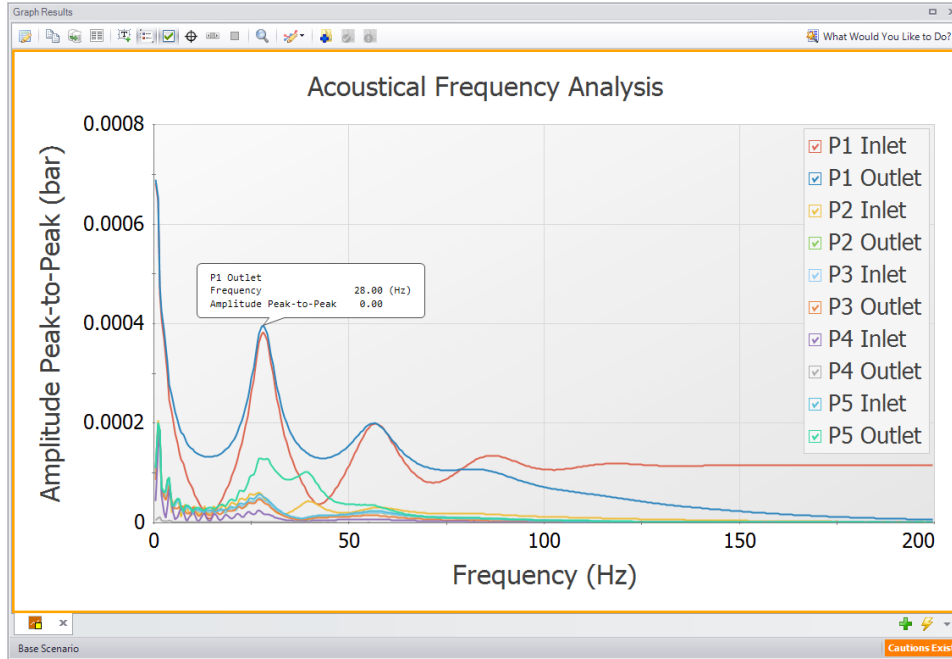


Figure 5: Acoustical frequency graph for the positive displacement compressor system

## Step 7. Simulate the worst-case scenario

### A. Determine the worst-case RPM

The 28 Hz excitation frequency resulted in the highest magnitude pressure response for the system; therefore, it will be the focus for the remainder of this example. In order to run a worst-case scenario, we need to translate the excitation frequency to a compressor speed in RPM. The equation below relates compressor RPM and the number of heads ( $N$ ) to the fundamental frequency, also called the **base pulsation frequency** ( $f_b$ ), and only applies when the number of heads is even and equally offset.

$$f_b = \frac{RPM * N}{60}$$

In this case 28 Hz pulsation frequency corresponds to 840 RPM. We will use this RPM in the next step. A real analysis might also include lower speeds that have a harmonic multiple falling on an excitation frequency of the system, such as 420 RPM, if these speeds are near the desired operating speed of the compressor.

### B. Create a scenario with Steady State Pulsation

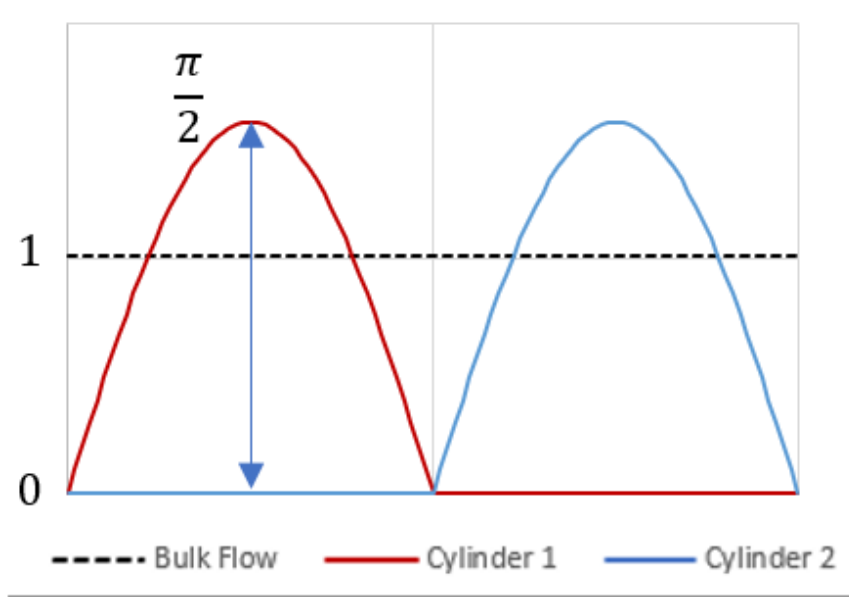
➤ Create a child scenario and switch to Steady State Pulsation analysis mode:

1. In the Scenario Manager, right-click on the Base Scenario and choose Create Child. Name the child scenario "840 RPM".
2. Open the Analysis Setup window and go to the Pulse Setup panel.
3. Switch the Frequency Analysis Mode to the Steady State Pulsation option.
4. Under the Pulsation Source section, select the Reciprocating Compressor option. The Compressor dropdown menu will show User Defined since this example is using an Assigned Flow junction to represent the discharge of the compressor.
5. Enter the Speed as 840 RPM and the Number of Heads as 2.
6. Ensure the checkbox for Evaluate with API 618 is enabled. Click OK to save and close Analysis Setup.

### C. Approximate the compressor flow

Since the PD compressor is single-acting and has 2 compression cylinders with a 180 degree phase offset, we can take advantage of the Periodic Flow option in the Assigned Flow junction to create a modified sine wave that approximates the mass flow leaving the compressor. In this case, having 2 heads with a 180 degree offset results in an easy, symmetrical chopped sine wave with two peaks per period.

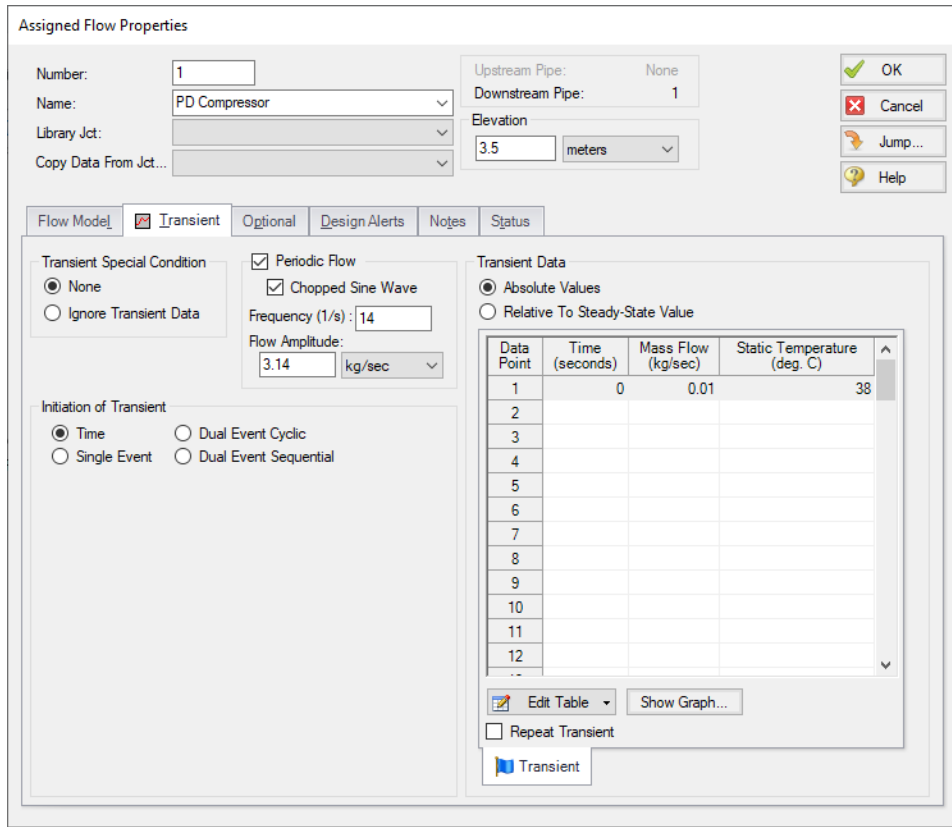
The Periodic Flow with Chopped Sine Wave option will require a frequency and amplitude. For the Chopped Sine Wave sub-option, the frequency represents two peaks. It is equivalent to taking the absolute value of a full period of a normal sine wave. Assuming that the minimum flow that occurs between the peaks is zero, the flow amplitude must scale the steady-state flow rate by  $\pi/2$  to achieve an equivalent bulk flow rate. This is shown in Figure 6 for one crankshaft rotation, and the areas under the steady-state and periodic mass flow curves are equal to the other. This scaling results in a flow amplitude of 3.14 kg/sec. Because the compressor flow has two peaks per crankshaft revolution, the frequency of the sinusoid is equal to the frequency of the crankshaft, which at 840 RPM is 14 1/s (Hz).



**Figure 6: Illustration of transient and steady mass flow rates with identical areas under the curves, resulting in equivalent bulk flow rates**

➤ **Modify the Assigned Flow junction to represent steady-state pulsation:**

1. Open the Assigned Flow Properties window for junction J1.
2. In the Flow Model tab, set Mass Flow Rate to 0.01 kg/s. This is chosen to be near zero since the Assigned Flow junction does not accept an input steady-state flow of zero.
3. In the Transient tab, clear the pulse transient data by clicking Edit Table under the transient table and choosing Clear All Data.
4. Check the boxes next to Periodic Flow and Chopped Sine Wave.
5. Set the Frequency to 14 1/s.
6. Set the Flow Amplitude to 3.14 kg/sec (Figure 7).
7. Click OK to save and close the Assigned Flow Properties window.



**Figure 7: Assigned Flow Properties window with Periodic Flow transient applied**

The model is now updated to represent the pulsation at the positive displacement compressor that will excite the system.

## Step 8. Check for compliance with API 618

### A. Check the Output for warnings

➤ Run the model, then navigate to the Output window.

A warning will be shown stating that the maximum pressure limit for API 618 was exceeded as is shown in Figure 8. The caution message stating that lowering the estimated maximum pipe Mach number may increase accuracy is useful for improving sectioning, but iterating on this value is not necessary for the purpose of this example. The caution message about the maximum transient pipe Mach number exceeding the steady pipe Mach number can be ignored, since this is to be expected based on the flow amplitude.

When the API 618 evaluation is performed, the Pulsation Summary tab is shown in the Output window with additional information on where pressure limit violations occurred, and what inputs were used to calculate the API 618 pressure limits for the scenario (Figure 9).

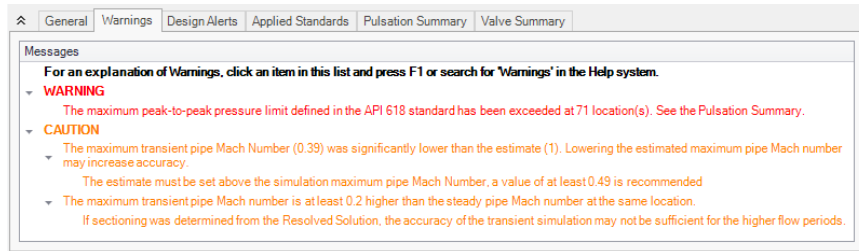


Figure 8: Warning showing that the API 618 pressure limits have been exceeded at multiple locations

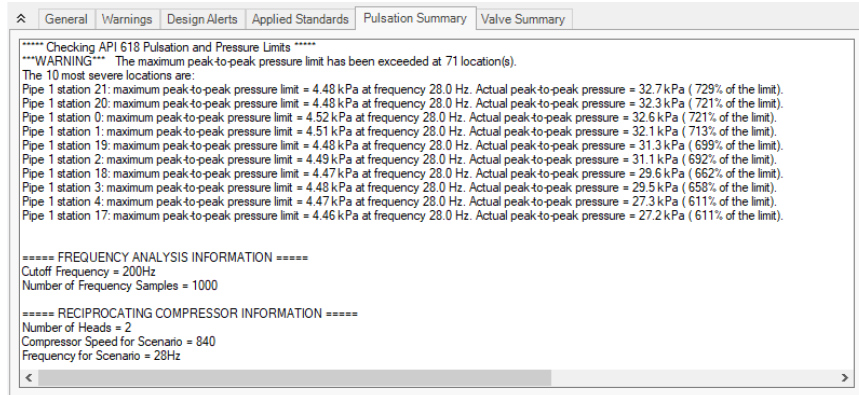


Figure 9: Pulsation Summary tab listing the pipes with the largest API 618 pressure violations

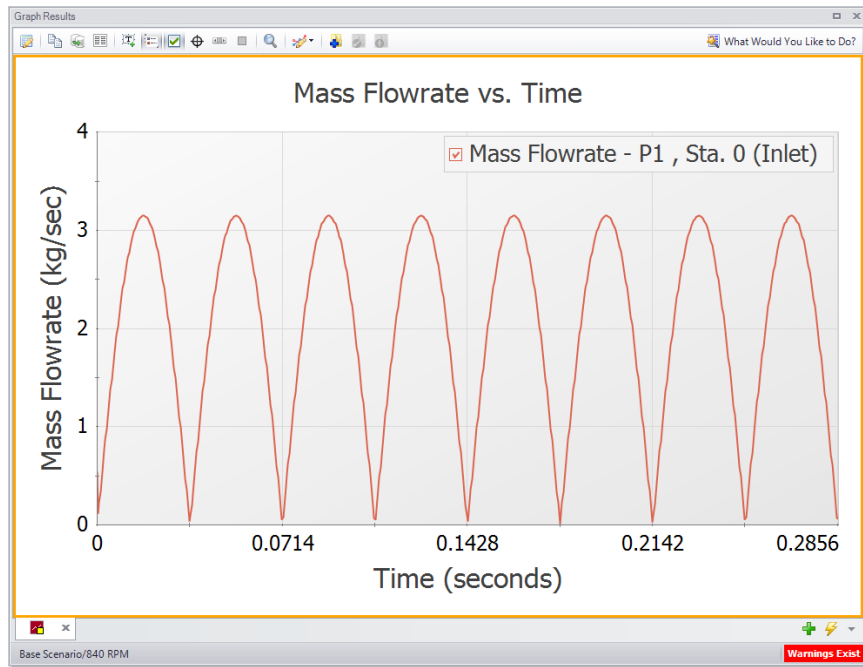


## B. Examine the pulsation at the compressor

The pulsation behavior at the outlet of the compressor can be illustrated by graphing the flow at the inlet of Pipe P1.

1. Select the Transient Pipe tab in the Graph Parameter section of the Quick Access Panel
2. Add the Outlet station from Pipe P1
3. Select Mass Flowrate as the graphing parameter with units of kg/sec
4. Ensure that Time Units are set to seconds
5. Click the Generate button
6. Right-click on the x-axis and uncheck the Auto Scale option
7. Set the Maximum to 0.286, which corresponds to 4 crankshaft rotations
8. Set the Major Val to 0.0714, which is the period of a single crankshaft rotation (60/840)

The resulting graph is shown in Figure 10. There should be 8 pulses each with a maximum flow near 3.14 kg/sec and a minimum flow near zero. The period of each pulse (a single peak on the graph) is equal to 0.0357 seconds, which translates to a pulsation frequency of 28 Hz. Being equal to the first major natural frequency of the system, this should result in maximum excitation of the pressure amplitude in the system at this frequency and its associated harmonic integer multiples.



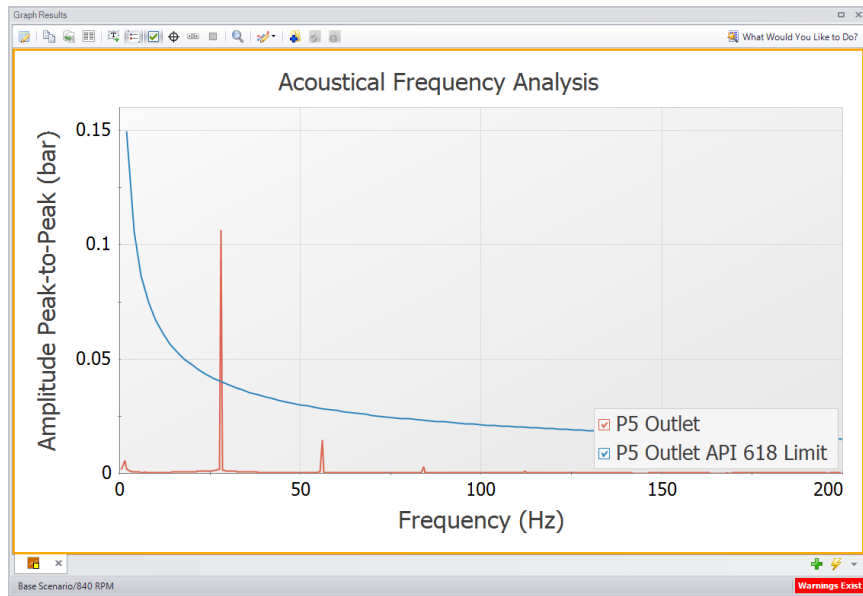
**Figure 10: Periodic mass flowrate supplied by the compressor over 4 crankshaft revolutions**

### C. Graph the API 618 limits

There were many locations within the model where the API 618 limits were exceeded. Since this example focused on the closed valve at the J6 Dead End, we will graph the API 618 limits for this location. The same procedure could be applied for any location in the model.

1. Select the Frequency tab in the Graph Parameter section of the Quick Access Panel
2. Add the Pipe P5 Outlet station
3. Ensure the Pressure Units are set to bar
4. Click the Generate button

Figure 11 shows the resulting graph in which AFT xStream has overlaid the calculated API 618 limits. A noticeable spike is seen at 28 Hz which exceeds the API 618 limit for the outlet station. Keen observers will notice very small spikes at the harmonic multiples of the compressor frequency (56 Hz).



**Figure 11: Frequency graph showing the API 618 limits**

Because the peak-to-peak pulsation level recommended by API 618 is exceeded, the system design may need to be changed in order to minimize vibration and maximize system longevity. This could be achieved by changing the compressor selection such as one with more cylinders or replacing the compressor with several smaller capacity compressors. Other approaches may include changing pipe geometry or adding pulsation suppression devices such as acoustic filters, pulsation bottles, finite volume surge tanks, or choking tubes.

### Conclusion

The PFA Add-on Module for AFT xStream can be used to apply a pressure pulse to the system in order to identify acoustic frequencies which may result in damage. These frequencies can be used in further

pulsation analysis to determine if pulsation mitigation is required. The Steady State Pulsation analysis mode and Assigned Flow junction with a periodic chopped sine wave allowed us to simulate the PD compressor running at a specified RPM.

The piping layout and geometry of the system was found to have an excitation frequency of 28 Hz (the first harmonic). With a single-acting, 2 head, 180 degree phase offset compressor, the fundamental frequency or base pulsation frequency of 28 Hz corresponded to a compressor speed of 840 RPM. The compressor outlet flowrate was approximated with a chopped sine wave where the frequency was 14 Hz. The two peaks per period represented the flow from each compression cylinder during one full rotation of the crankshaft.

The peak-to-peak pressure oscillations were found to violate the API 618 standard. Future modifications would need to be made to the system such as changing the compressor, altering the piping layout, or utilizing pulsation suppression devices.

## Index

---

### A

AFT Arrow Steady Solver 28, 76

AFT Standard library See Fluid libraries, AFT Standard library

Analysis Setup window 22, 51

Animation

    Graph 58-59, 71

    Speed adjustment 72

Annotation Tool See Toolbox, Annotation tool

API 618 94, 104

ASME Steam Tables library See Fluid libraries, ASME Steam Tables library

### C

Change Units window 32

Chempak library See Fluid libraries, Chempak library

Condensation 72, 76

Copy Data From Jct... 14

### D

Defining force sets See Forces, Force sets, Definition

Defining objects

    List Undefined Objects window 13, 51, 67, 96

### E

Estimated Maximum Temperature During Transient See Pipe Sectioning panel, Estimated Maximum Temperature During Transient

Export force files See Forces, Exporting force files

**F**

Fluid libraries

AFT Standard library 23, 51, 67, 76, 96

ASME Steam Tables library 23, 76

Chempak library 23, 76

Effect of 75, 90

NIST REFPROP library 24, 76

Selection 76-77

Fluid panel 23, 51, 67, 96

Forces

Exporting force files 92

Force sets 81

    Definition 82-83

    Direction 82

    Force Unit Vectors 82

Methodology 88

Forces panel 93

**G**

Graph Animations See Animation, Graph

Graph Control tab

    Forces 87

    Profile tab 58, 71

    Transient Jet tab 35, 73

    Transient Pipe tab 33, 56, 68

Graph List Manager 57, 60, 87

Graph Results window 32  
    Modifying axes of graphs 68

**H**

Highlight feature 15

**I**

ID numbers 8, 13

Inspection feature 16, 20

Isometric See Pipe Drawing Mode, Isometric

**J**

Junction library 14

Junctions 7

    Tank 7

    Valve 16

**L**

List Undefined Objects window See Defining objects, List Undefined Objects window

Lock Object 12

**M**

Method of Characteristics 25, 54

Minimum Number of Sections Per Pipe See Pipe Sectioning panel, Minimum Number of  
    Sections Per Pipe

Mixtures 23, 76

Model building process 75

Model Data window 21

Model Status Light 22

**N**

NIST REFPROP library See Fluid libraries, NIST REFPROP library

**O**

Objects 8

Open Pipe/Jct Window 14

Output Control window 29, 31

Output file 29

Output window 29, 54

    Event Messages By Time 30

    Forces tab 87

    Transient Max/Min tab 55

**P**

PFA Module

    Excitation frequencies 100

Pipe Drawing Mode

    2D Freeform 63

    2D Orthogonal 63

    Isometric 63

        Adding segments 63

Pipe Drawing Tool See Toolbox, Pipe Drawing Tool

Pipe Properties window 19-20

Pipe Sectioning panel 25, 51, 55, 67, 80, 97

    Estimated Maximum Pipe Mach Number During Transient 25

        Effect of 90

- Estimated Maximum Temperature During Transient 25, 51, 67, 80, 97
  - Effect of 54
- Minimum Number of Sections Per Pipe 25, 51, 67, 80, 97
  - Effect of 42, 90
- Sectioning Summary 26
- Preferred Units 14
- Properties windows
  - Notes tab 15
  - Optional tab 15-16
  - Status tab 15, 21
  - Transient tab
    - Specification 17, 85, 103
- Pulsation panel 97
  - Cutoff Frequency 97
  - Minimum Number of Frequency Samples 97
  - Pulsation Summary 98

**Q**

- Quick Access Panel 7
  - Graph List Manager 57
- Quick Graph Formatting 34

**R**

- Reverse Direction 12
- Reverse flow 12
- Rotating Icons 63
- Run Model 27, 54, 68, 87, 98, 104



**S**

- Scenario Comparison Tool 85-86
- Scenario Manager 55, 60, 84, 102
- Select Graph Parameters 33
- Selecting a fluid library See Fluid libraries, Selection
- Show Grid 63
- Show Object Status 13, 16
- Simulation Mode/Duration panel 24, 26, 52, 68, 81, 97
- Sonic choking 26, 35, 60
- Stacked Graphs 33
- Startup window 5
  - Modeling Preferences 5
- Status Bar 22

**T**

- Tank junction See Junctions, Tank
- Toolbars 22
- Toolbox 7
  - Annotation tool 7
  - Pipe Drawing Tool 7, 10-11
- Tooltips 7
- Transient Control group 26
- Transient event specification See Properties windows, Transient tab, Specification
- Transient Jct tab See Graph Control tab, Transient Jct tab
- Transient Pipe tab See Graph Control tab, Transient Pipe tab
- Transient Solver 28, 76

**U**

Undefined objects list See Defining objects, List Undefined Objects window

User Options 13

**V**

Valve junction See Junctions, Valve

**W**

Workspace window 7, 47

Workspace Layers 39