

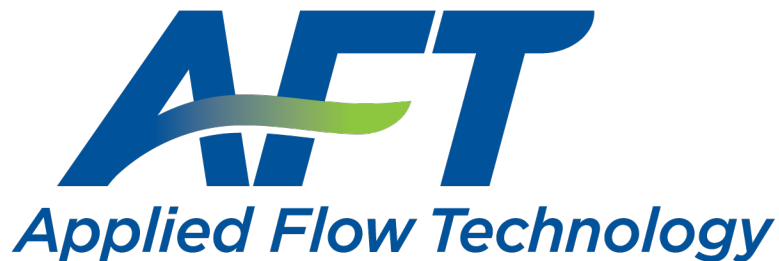
# ***AFT xStream***<sup>™</sup>

## **Examples - English Units**

**AFT xStream Version 2**

**Gas Transient Modeling in Piping Systems**

Published: May 01, 2023



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

# Contents

---

<b>Examples Overview .....</b>	<b>3</b>
<b>Tank Blowdown Example (English Units) .....</b>	<b>4</b>
<b>Gas Turbine Fuel System Example .....</b>	<b>43</b>
<b>High Pressure Steam - Fluid Property Models Example .....</b>	<b>58</b>
<b>High Pressure Steam - Forces Example .....</b>	<b>74</b>
<b>Frequency Analysis - PFA Example .....</b>	<b>91</b>
<b>Index .....</b>	<b>102</b>

## Examples Overview

Example	Fluid	Pipes	Descriptions
<a href="#">Tank Blowdown - Beginner</a>	Nitrogen	2	An in depth discussion of model building in AFT xStream. Models a simple valve opening during a tank blowdown.
<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="#">High Pressure Steam - Fluid Property Models</a>	Steam	12	Two turbines in a high pressure steam system are simultaneously valved shut as a bypass line valve is opened. This model demonstrates factors to know to evaluate which fluid database to use in your system, as well as demonstrating how to construct a model in the isometric drawing mode.
<a href="#">High Pressure Steam - Forces</a>	Steam	14	Two turbines in a high pressure steam system are simultaneously valved shut. 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="#">Frequency Analysis - PFA</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.

## Tank Blowdown Example (English Units)

### Summary

---

This example will give you the big picture of AFT xStream's layout and structure. To accomplish this purpose, a model with two pipes and three junctions will be built to capture the system effects when a valve is opened.

### Topics covered

- Model building basics
- Entering pipe and junction data
- Entering transient data
- Graphing output results

### 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

The file listed below contains a completed version of this example and can be located in the Examples folder as part of the AFT xStream installation. This file can also be downloaded from the AFT xStream help site from the Tank Blowdown Example page:

- ***US - Tank Blowdown.xtr***

### Step 1. Start AFT xStream

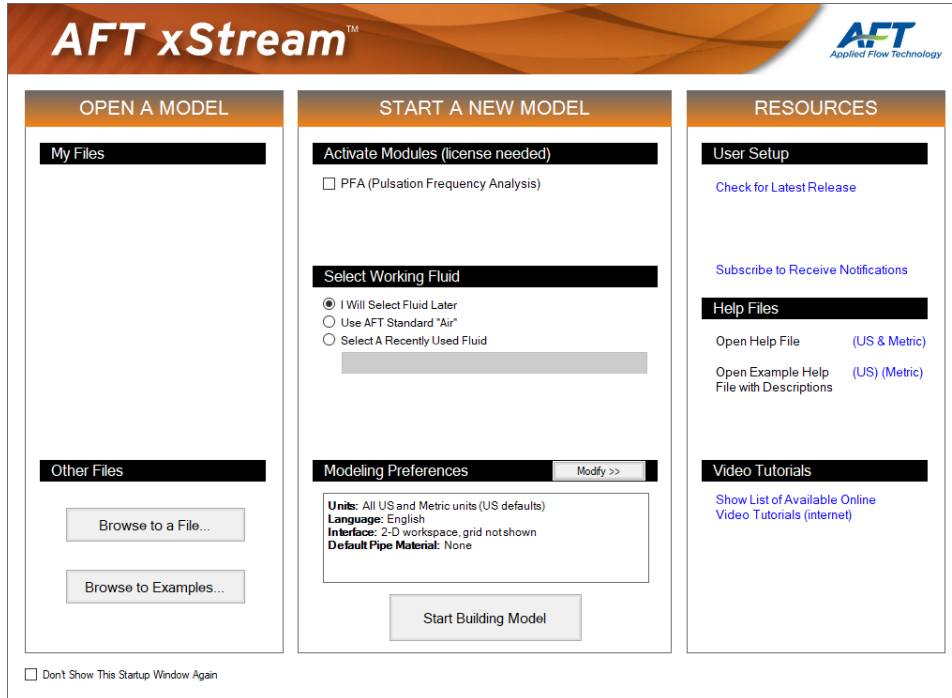
---

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

After AFT xStream starts, the Startup window, as shown in Figure 1, appears. This window provides you with several options before you start building a model: You can open a model that was recently worked on, browse to a different xStream file, or browse to the xStream Example files. If you have a license for the xStream PFA add-on module, it can be activated here. The Unit System can also be specified to use US Only, Metric Only, Both with US Defaults, or Both with Metric Defaults. They can be set as the user default at this time as well. You can also set the working fluid to AFT Standard Air or another recently used AFT fluid. You can click the link to "Check for Latest Release" to go to the AFT website to see if you

## Tank Blowdown Example

have the latest release of AFT xStream. You can also "Subscribe to Receive Notifications" of when product maintenance releases are available, receive the AFT newsletter, etc. Finally, you can open available learning resources such as examples.



**Figure 1: AFT xStream Startup window**

### **Click "Start Building Model".**

The Workspace window is initially the active window, as seen in Figure 2. The five tabs in the AFT xStream window represent the five Primary Windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

AFT xStream supports dual monitor usage. You can click and drag any of the five primary window tabs off of the main xStream window. Once you drag one of the primary windows off of the xStream window, you can move it anywhere you like on your screen, including onto a second monitor in a dual monitor configuration. To add the primary window back to the main xStream primary tab window bar, simply click the "X" button in the upper right of the primary window.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT 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. The Workspace takes up the rest of the window.

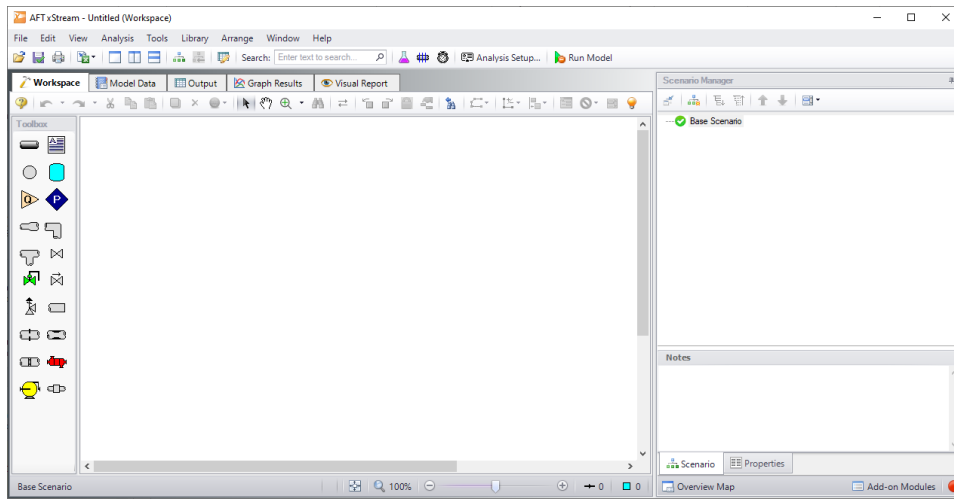
## Tank Blowdown Example

---

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

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

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



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

## Step 2. Lay out the model

---

To lay out the tank blowdown model, you will place a Tank junction, a Valve junction, and an Assigned Pressure junction 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 3 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. For junctions, this number is, by default, placed directly above the junction and prefixed with the letter "J". Pipe ID numbers are prefixed with the letter "P". You can optionally choose to display either or both the ID number and the

## Tank Blowdown Example

---

name of a pipe or junction. You also can 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 one. You can change this to any desired number greater than zero, but less than 100,000.

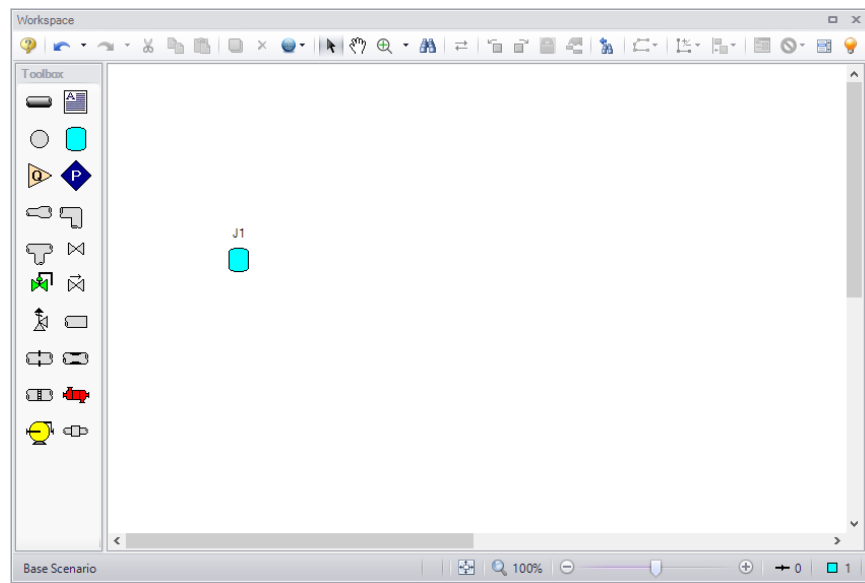
### ***Editing on the Workspace***

Once on the Workspace, junctions can be moved to new locations and edited using the features on the Edit menu. Cutting, copying, and pasting are all supported. Undo is available for most formatting and arrangement actions on the Workspace.

---

**Note:** The relative location of objects in AFT xStream is not important. Distances and heights are defined through dialog boxes. The relative locations on the Workspace establish the connectivity of the objects, but have no bearing on the actual length or elevation relationships.

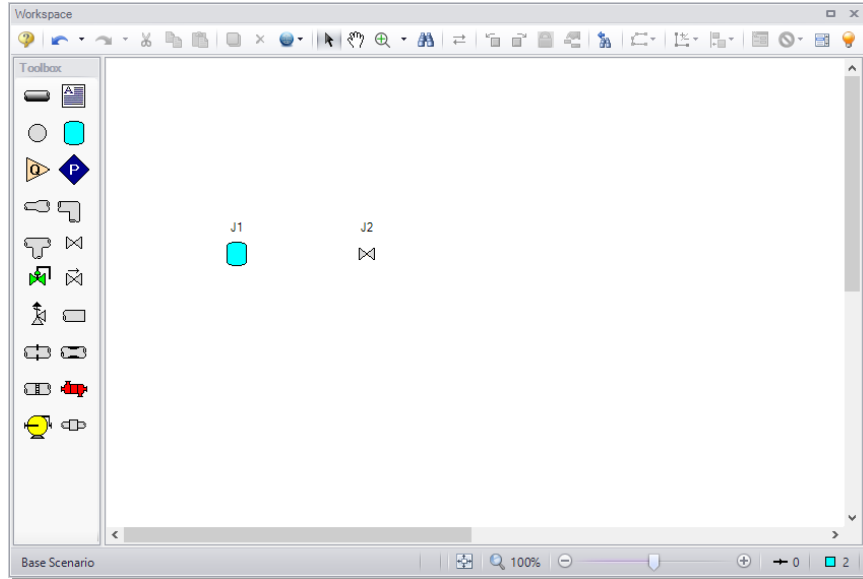
---



***Figure 3: 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 4. The Valve will be assigned the default number "J2".



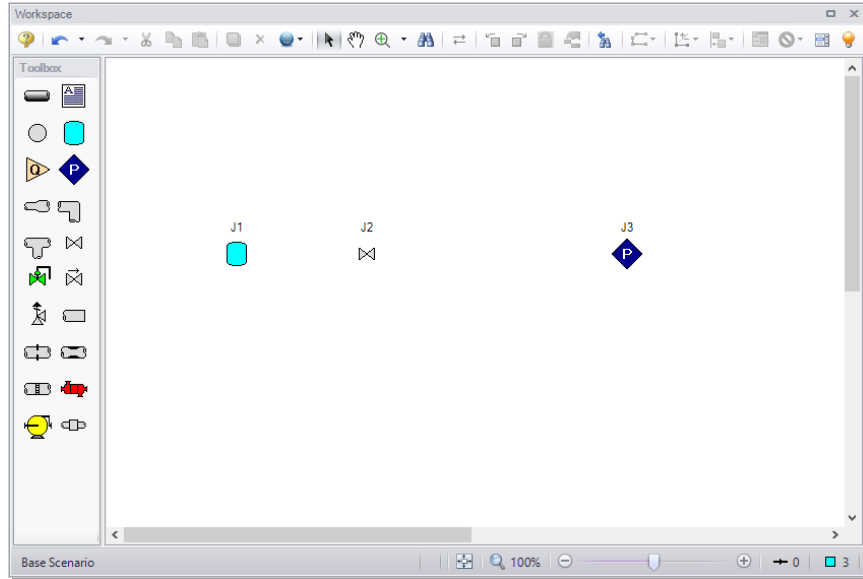
**Figure 4: Tank Blowdown model with Valve placed**

### 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 5. The Assigned Pressure junction will be assigned the default number "J3".

➤ **Before continuing, save the work you have done so far.** Choose Save As... from the File menu and enter a file name (Tank Blowdown, perhaps) and AFT xStream will append the ".xtr" extension to the file name.





**Figure 5: All junctions placed**

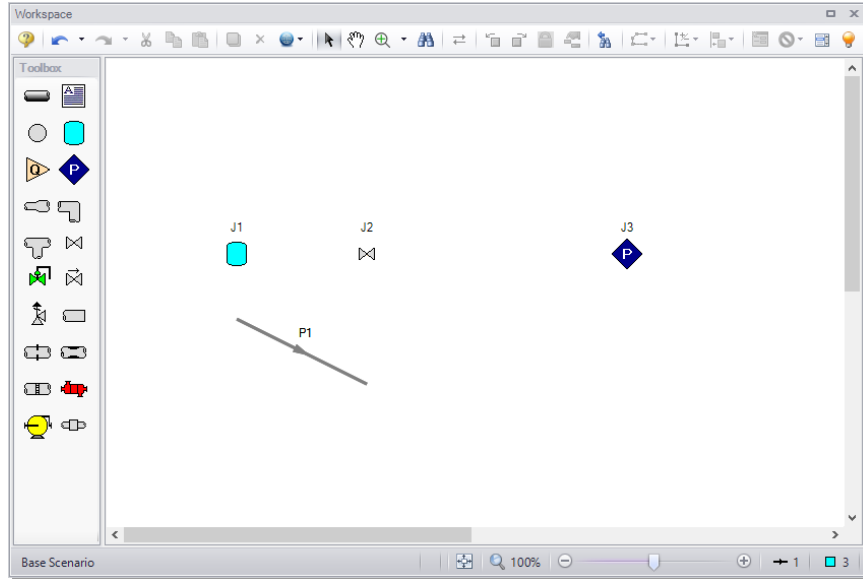
## IV. Draw a pipe between J1 and J2

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

➤ **To create a pipe, click the Pipe Drawing Tool icon on the Toolbox.** The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe above the junctions by clicking and dragging the mouse, similar to that shown in Figure 6.

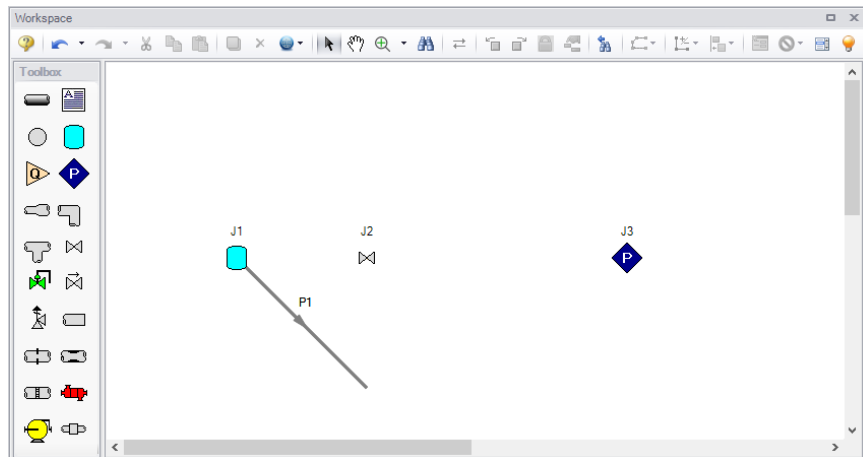
The pipe object on the Workspace has an ID number (P1) that is shown near the center of the pipe.

## Tank Blowdown Example



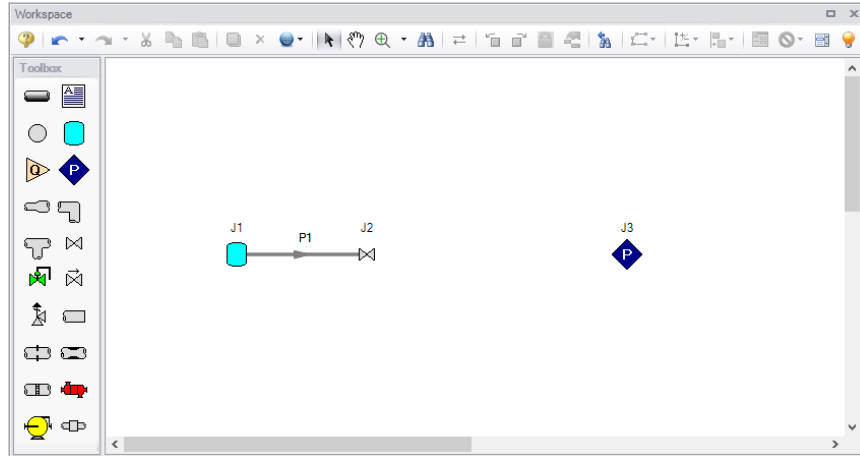
**Figure 6: First pipe drawn**

➤ **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 7). 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 8).



**Figure 7: First pipe partially connected**

## Tank Blowdown Example



**Figure 8: First pipe fully connected**

### **Reference positive flow direction**

Located on the pipe is an arrow that indicates the reference positive flow direction for the pipe. AFT xStream assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the Arrange menu or selecting the Reverse Pipe Direction button on the Workspace Toolbar.

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

### **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 9.

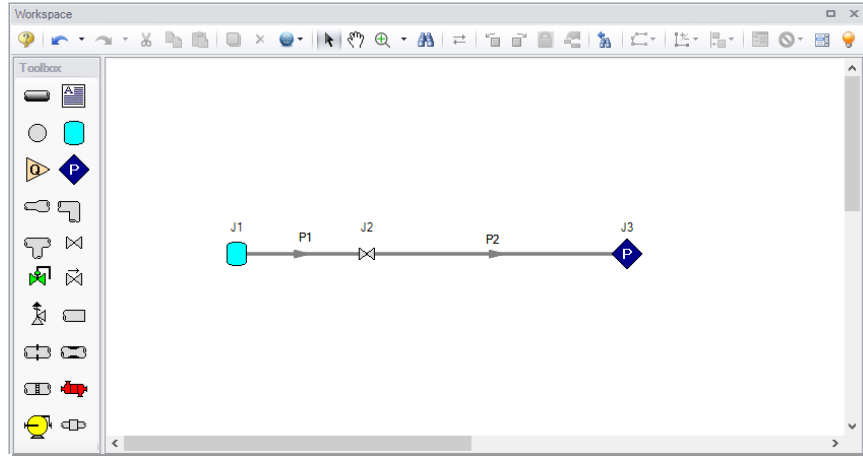
Save the model by selecting "Save" in the File menu, by clicking the "Save" icon on the Toolbar, or by entering Ctrl+S.

---

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

---

## Tank Blowdown Example



**Figure 9: All pipes and junctions placed**

### A. Define the model components

#### **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.

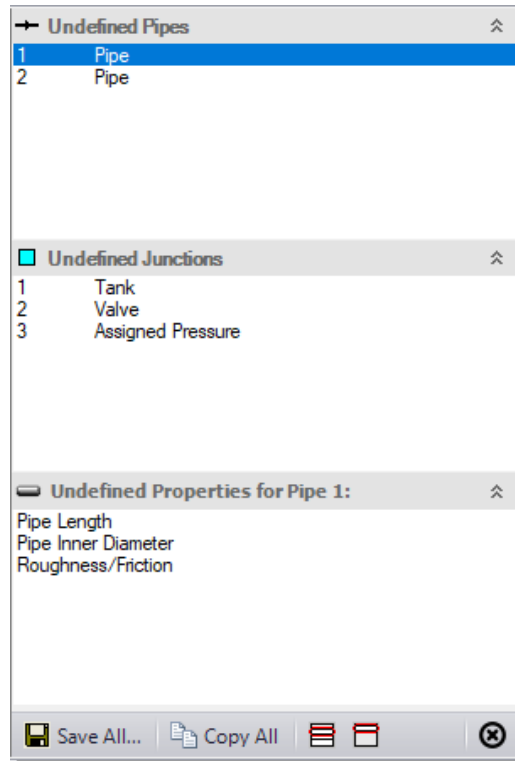
When "Show Object Status" is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. (These colors are configurable through User Options from the Tools menu.)

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

#### **Showing undefined objects**

Another useful feature is the List Undefined Objects window (Figure 10). 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 10:** The List Undefined Objects window lets you see the undefined properties for each undefined object

### I. Enter data for Tank J1

As shown in Figure 9, the J1 junction is a Tank junction.

➤ **To define the first tank**, open the J1 Tank Properties window (See Figure 11) by double-clicking on the J1 icon. Choose the Finite Tank Model, then enter an Elevation of 0 feet. You can assign any unit of length found in the adjacent drop down list.

---

**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 a Pressure of 600 psia**, a Temperature of 300 deg. F, and a Volume of 300 ft<sup>3</sup>. Also set the Initial Conditions to Known and the Thermodynamic Process to 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.

---

By default, the junction's name is the junction type. The name can be updated by entering it in the Name field at the top of the window. In Figure 11, the name of Tank J1 is "Supply Tank." The name can be displayed on the Workspace, Visual Report, or in the Output.

## Tank Blowdown Example

---

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.

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

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

The highlight feature displays all the required information in the Properties window in light blue. The highlight feature is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window or by pressing the F2 key. The highlight feature can also be turned on or off in the User Options window, or accessed from the View menu by selecting the "Highlight in Pipe and Jct Windows" option.

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.

## Tank Blowdown Example

Tank Properties

Number:

Name:

Library Jct:

Copy Data From Jct...:

Elevation:

OK Cancel Jump... Help

Tank Model Optional Design Alerts Notes Status

Tank Model

Infinite

Finite

Initial Conditions

Pressure:

Temperature:

Enthalpy:

Finite Tank Parameters

Fixed Volume:

Initial Conditions

Known

Calculate From Steady State

Thermodynamic Process

Adiabatic

Isothermal

Known Heat and Volume vs. Time

**Figure 11: Properties window for Tank J1**

### ***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 12.

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

## Tank Blowdown Example

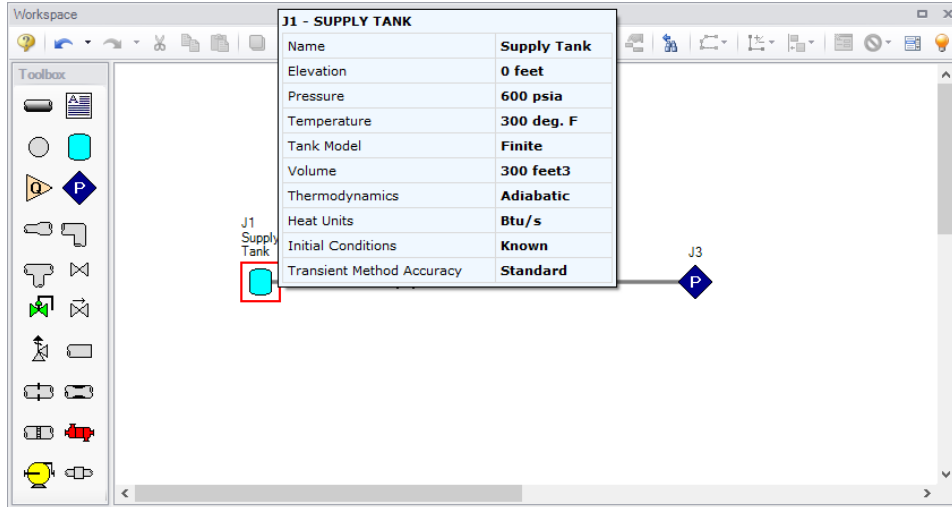


Figure 12: Inspect junction properties from the Workspace by clicking and holding the right mouse button

## II. Define Valve J2

- **Open the J2 Valve Properties window** (see Figure 13) and enter an elevation of 0 feet.
- **Choose the loss model as  $C_v$**  and enter a  $C_v$  value of 60. This typically represents the valve's  $C_v$  during steady-state. When the valve is closed during the steady state then opens during the transient (as is the case for this model), the loss model represents the valve's loss at the end of the transient. Enter an  $X_t$  of 0.7.
- **Click on the Optional tab** and, under Special Condition, select Closed. This indicates the valve will be closed in the steady state.
- **Click the Transient tab** and enter the data below for  $C_v$  (See Figure 14).

Time (seconds)	$C_v$	$X_t$
0	0	0.7
0.1	60	0.7
240	60	0.7

The first data point ( $C_v = 0$ ) must match the steady-state value. The transient data represents the valve initially closed. The valve then opens over a period of 0.1 seconds and stays open.

---

**Note:** The initial transient data point is  $C_v = 0$  because we set the Special Condition as Closed under the Optional Tab. Had we selected none for the special condition, the first data point would have to be  $C_v = 60$ .

---

The J2 Valve is the element which causes 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.



## Tank Blowdown Example

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

Valve Properties

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

OK  
Cancel  
Jump...  
Help

Loss Model Transient Optional Design Alerts Notes Status

Subsonic Loss Model  
 Cv (ANSI/ISA) ?  
 Kv

Loss Source  
 Fixed Cv  
 From Open Percent Table (on Optional Tab)  
% Open:

Cv: 60  
Xt: 0.7 Guide for Xt...

Open Percentage Data

	Open Pct.	Cv	Flow Area	Xt	
1					
2					
3					
4					
5					

Edit Table  
Show Graph...  
Create Cv/Open%...  
Optional Data

Flow Area Units: feet2 Open Pct. Increment: 10 Autofill

Exit Valve (optional)  
Exit Pressure:   
Exit Temperature:

Figure 13: Input data for Valve junction J2

## Tank Blowdown Example

Valve Properties

Number: 2 Upstream Pipe: 1  
Name: Valve Downstream Pipe: 2  
Library Jct: Elevation: 0 feet  
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)	Cv	Xt
1	0	0	0.7
2	0.1	60	0.7
3	240	60	0.7
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			

Xt Source  
 User  
 Open % Table  
Fill Xt from table

Edit Table Show Graph...  
 Repeat Transient  
Transient

OK Cancel Jump... Help

Figure 14: Transient input data for Valve junction J2

### III. Define Assigned Pressure J3

- Open the J3 Assigned Pressure Properties window (see Figure 15) and enter an Elevation of 0 feet.
- Enter a Stagnation Pressure of 14.7 psia and a Stagnation Temperature of 70 F.
- Save the model again before proceeding.

## Tank Blowdown Example

Assigned Pressure Properties

Number:

Name:

Library Jct:

Copy Data From Jct...:

Elevation:

OK Cancel Jump... Help

Pressure Model Loss Coefficient Transient Optional Design Alerts Notes Status

Boundary Specification

Stagnation  Static

Stagnation Pressure:

Stagnation  Temperature  Enthalpy

*Figure 15: Enter the data for Assigned Pressure J3 in the Assigned Pressure Properties window*

### IV. Define Pipe P1

The next step is to define all the pipes. Double-click the pipe object on the Workspace.

➤ **First, open the Pipe Properties window for Pipe P1.** For Pipe P1, enter a length of 20 feet, and select the Material as Steel - ANSI, nominal 2 inch size, STD (schedule 40) (see Figure 16).

## Tank Blowdown Example

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 '2.067 inches'. The 'Length' is '20 feet'. The 'Data Set' is 'Standard' and 'Absolute Roughness' is '0.0018 inches'. The 'Load Default' button is visible next to the roughness field.

**Figure 16: Properties window for Pipe P1**

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

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

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

Click OK to finish defining P1.

## V. Define Pipe P2

➤ **Open the Pipe Properties window for Pipe P2.** Enter a length of 1500 feet, and select the Material as Steel - ANSI, nominal 2 inch size, STD (schedule 40).

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

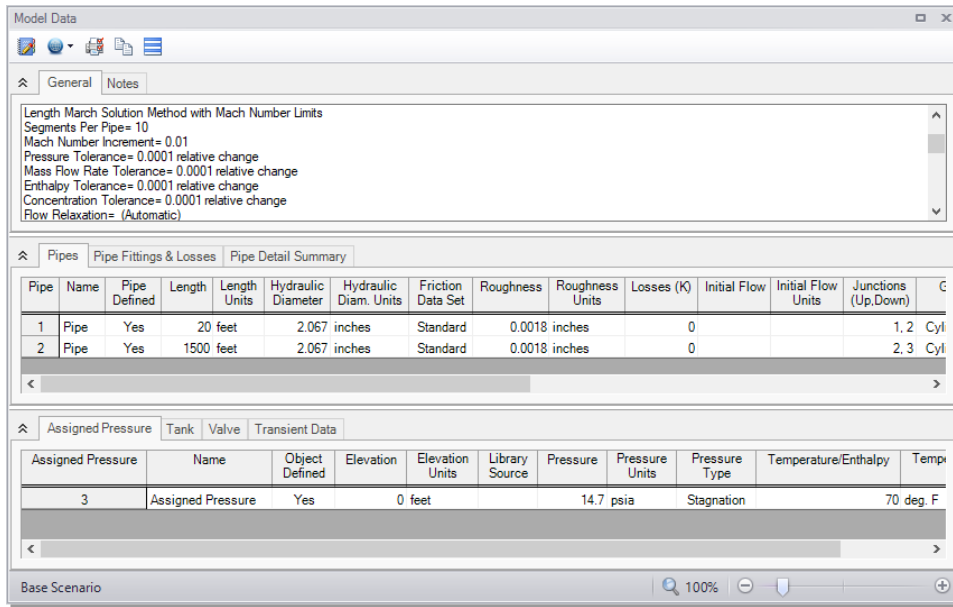
➤ **Save the model.** It is also a good idea to review the input using the Model Data window.

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

The Model Data window is shown in Figure 17. To change to this window, you can select it from the Window menu, you can click on the Model Data tab, or you can press Ctrl+M. The Model Data window gives you a text based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, exported into Excel, saved to a formatted file, printed to an Adobe™ PDF, or printed out for review. Figure 18 shows an expanded view of the Transient Data tab from Figure 17. 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 17: The Model Data window displays all input in text form**

## Tank Blowdown Example

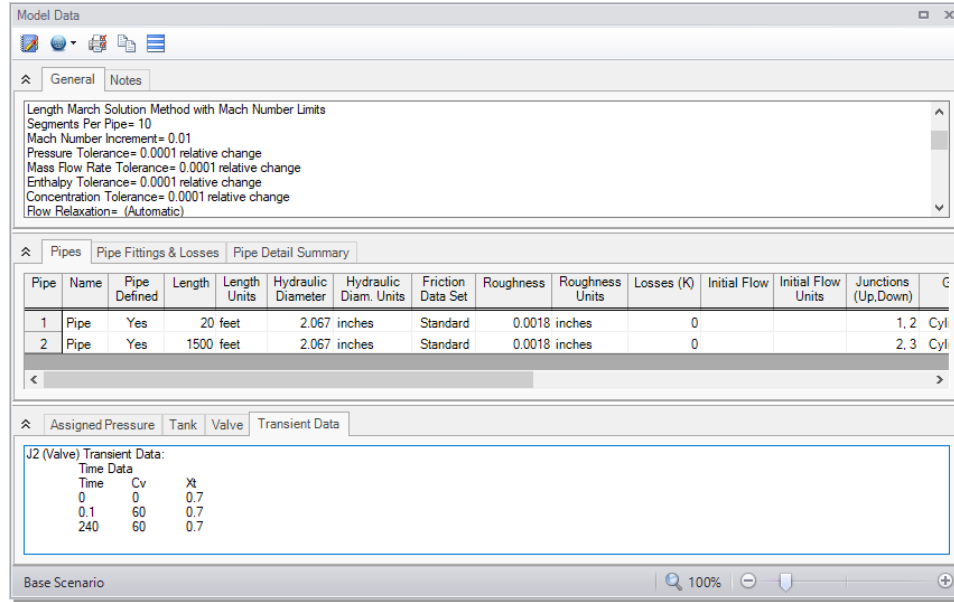


Figure 18: The Transient data tab in the Model Data window displays transient input data

### Step 3. Specify Analysis Setup

➤ **Next, click Analysis Setup on the Main Toolbar** to open the Analysis Setup window (see Figure 19). The Analysis Setup window contains six or more groups depending on which modules you may be using. Each group has one or more items. A fully defined group has every item within it defined with all required inputs. Complete every group to run the Solver. A group or item that needs more information will have a red exclamation point to the left of the group or item name. After completing a group, the group icon shows a green circle with a checkmark. After defining all groups, thus completing the Analysis Setup, the Model Status light in the lower right corner 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.

# Tank Blowdown Example

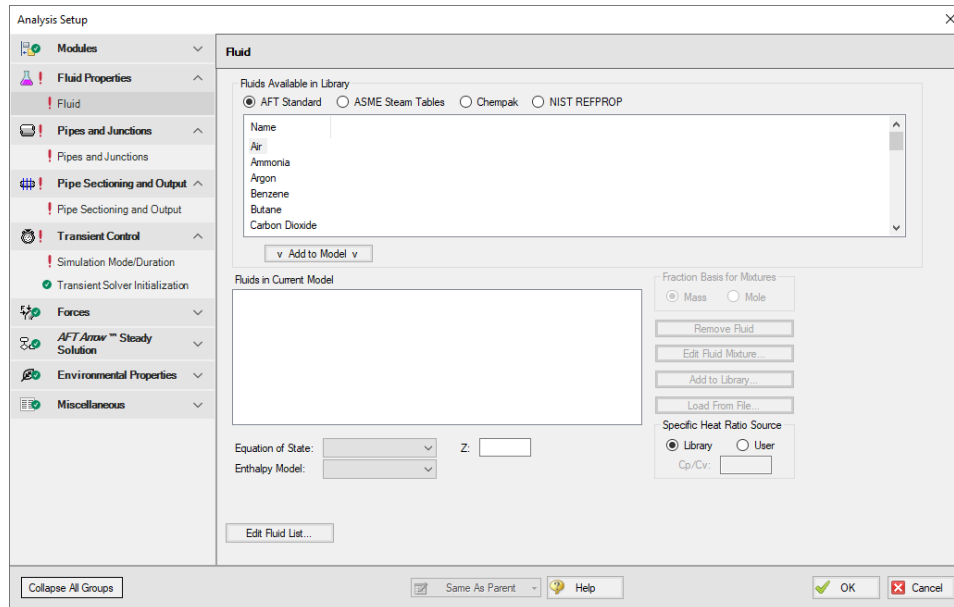


Figure 19: The Analysis Setup window tracks the defined and undefined model input

## A. Specify the Modules group

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 19. Here you can enable and disable the Pulsation Frequency Analysis add-on module. By default, the model will have all modules disabled. Therefore, make no changes to this panel for this example.

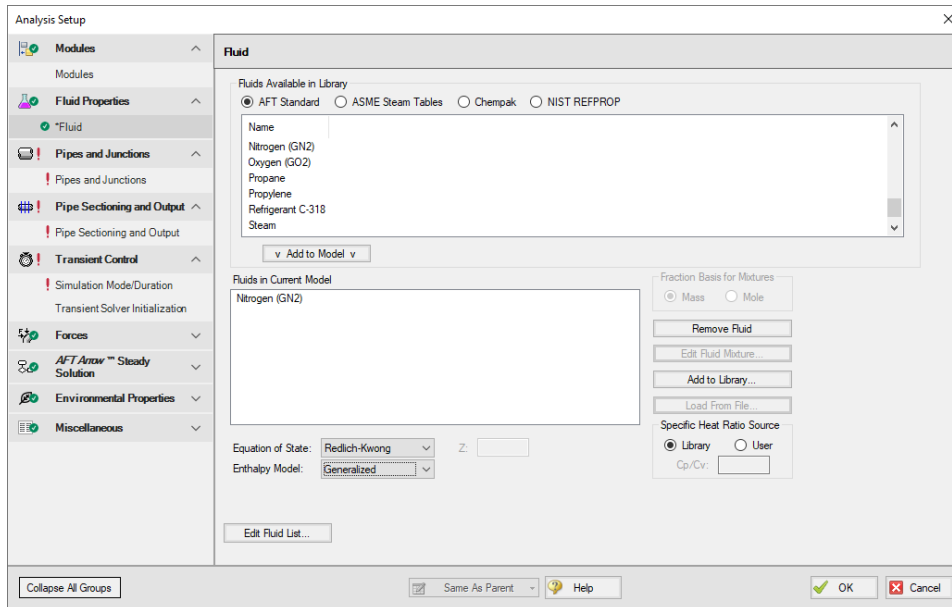
## B. Specify the Fluid Properties group

➤ **Select the Fluid item** to open the Fluid panel (Figure 20).

This panel allows you to specify the fluid used in the model.

You can select a fluid from the standard AFT xStream fluid library (AFT Standard), or select ASME Steam Tables. Additionally, you can select multiple fluids and/or create mixtures using the included NIST REFPROP library or optional add-on Chempak library, or create custom fluids. 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...".

➤ **Select the AFT Standard fluid option**, then choose "Nitrogen (GN2)" from the list and click the "Add to Model" button.



**Figure 20: The Fluid panel is where you specify the fluid**

## C. Inspect the Pipes and Junctions group

The Pipes and Junctions group in the Analysis Setup window contains the Pipes and Junctions panel, which 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 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, and a steady state run is trivial.

## D. Specify the Pipe Sectioning and Output group

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 satisfied:



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

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

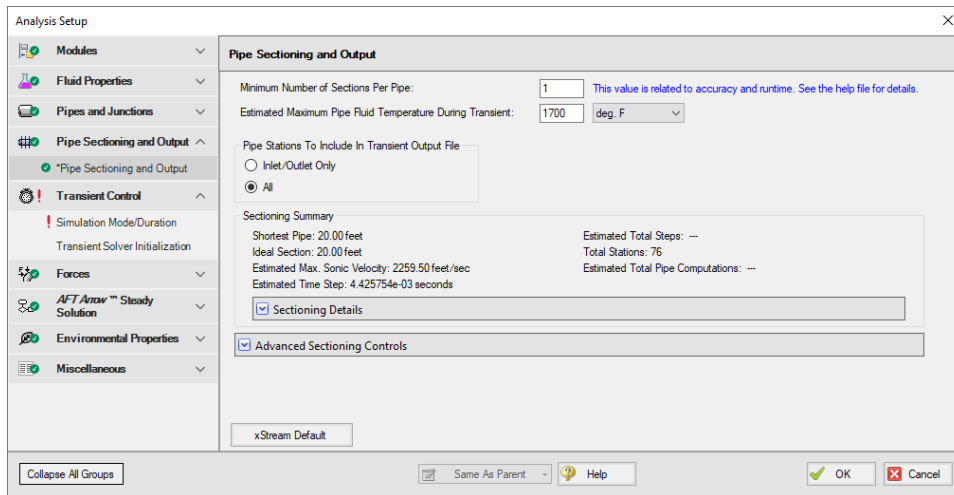
➤ **Enter "1" for the Minimum Number of Sections Per Pipe**

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. The number of sections used will impact both the accuracy and the run time - a larger number of sections will increase the result accuracy, but will also quadratically increase the run time for the model. The Minimum Number of Sections will be defined as 1 in this example for expediency, though this may result in a lower accuracy than when using more sections. The error in the results due to this sectioning choice will be explored further in a later step.

➤ **Define the Estimated Maximum Pipe Temperature During Transient as 1700 deg. F.**

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 Sectioning Summary underneath the inputs will be partially filled in. Certain properties, such as total steps and total pipe computations, need the Simulation Mode/Duration panel completed to be defined.



**Figure 21: The Sectioning panel in the Analysis Setup window is used to calculate the time step of the model**

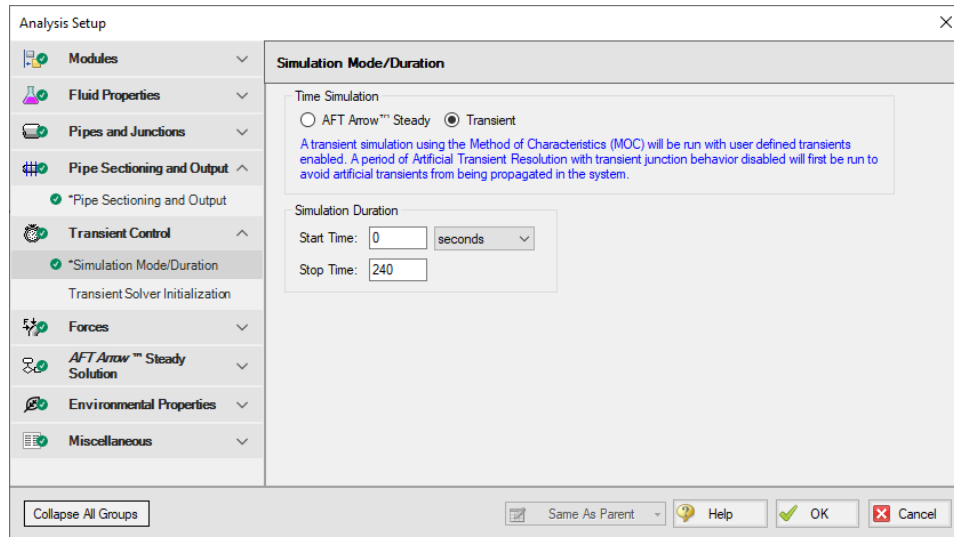
## E. Specify the Transient Control group

The Transient Control group has two items, the Simulation Mode/Duration panel and the Transient Solver Initialization panel. The Simulation Mode/Duration panel allows you to choose the simulation type, and to specify the time that the simulation starts and stops.

The Transient Solver Initialization panel contains advanced settings related to the resolving artificial transients in the model, and typically should not be changed.

➤ **Select the Simulation Mode/Duration panel** (Figure 22). Enter 0 seconds for Start Time and 240 for Stop Time.

➤ **Click OK to accept the current settings.** The last Analysis Setup panel should be completed. The model is ready to be solved.



**Figure 22: 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 solver

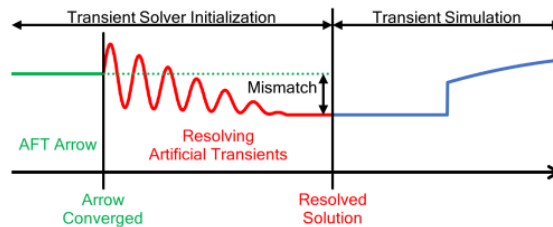
➤ **Choose "Run Model..." from the Analysis menu** or click the "Run Model" button on the Main Toolbar. During execution, the Solution Progress window displays the state of the simulation (Figure 24). You can use this window to pause or cancel the solver's activity.

Note the run time may take 5-10 minutes for this model, depending on your computer's speed.

### 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.

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. To allow artificial waves caused by these discrepancies to die out, AFT xStream runs the Transient solver for a period of time before the transient simulation begins. Once the artificial transients have died out 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 23 as a visual representation of the process described above.



**Figure 23: Illustration of the Transient Solver Initialization process with artificial transient resolution**

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

## Tank Blowdown Example

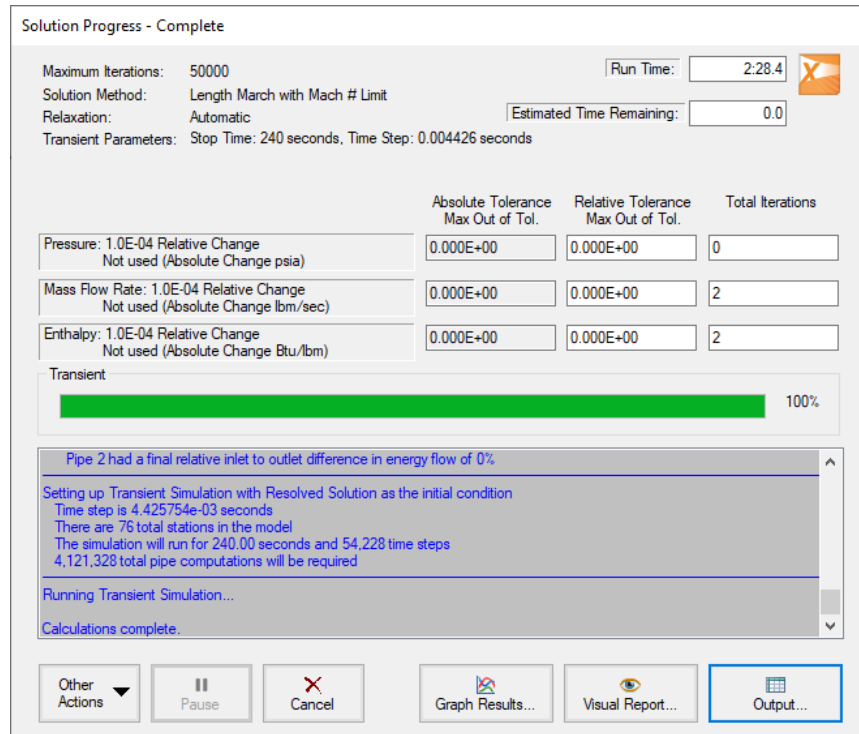


Figure 24: The Solution Progress window displays the state of the simulation

### The transient output file

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

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

## Step 5. Review the Output window

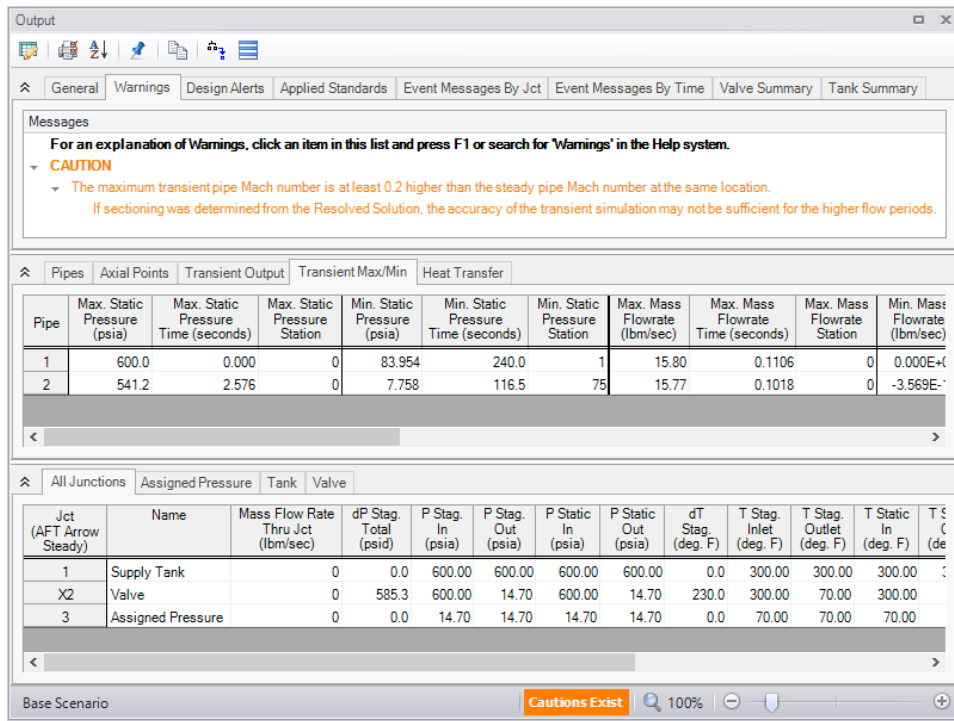
Click Output after the solver is finished. The Output window (Figure 25) is similar in structure to the Model Data window. Three areas are shown, and you can minimize or enlarge each section by clicking the arrow next to the General, Pipes, and All Junctions tabs. The items displayed in the tabs can be customized with the Output Control window from the Tools drop-down 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 later in Step 10.

## Tank Blowdown Example

The Output window allows you to review both the steady-state and transient results. The Pipes tab, All Junctions tab, and any specific junction tabs in the Junction section (such as Branch, Tank, Valve, etc.) show the steady-state results. You can review the solutions for each time step (i.e., a time history) for which data was written to file. Also, a summary of the maximum and minimum transient results for each computing station is given on the Transient Max/Min tab in the pipe area. These two data sets are located on the Transient Output tab and Transient Max/Min tab in the pipe area of the Output window (see Figure 26 and Figure 27). Note that in order to display all pipe stations as shown in Figure 26, 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.

Click on the Event Messages By Time tab (See Figure 28). There should be messages indicating that Valve J2 experienced a near instantaneous choking that dissipated after about 0.06 seconds while Assigned Pressure J3 experienced choking after 2.52 seconds that lasted until approximately T=116.5 seconds. Choking is an important phenomenon to pay attention to as it indicates that the fluid has reached sonic velocity. This has significant implications for the MOC calculation method as will be discussed later.



**Figure 25: The Output window displays steady and transient output in text form**

# Tank Blowdown Example

Step: 0 Time: 0.00000E+00 Pipe -- Station	Static Pressure (psia)	Mass Flowrate (lbm/sec)	Velocity (feet/sec)	Static Temperature (deg. F)
1 -- 0	600.00	0	0	300.00
2 -- 0	14.70	0	0	70.00

Figure 26: The Output window displays transient data from each time step

Pipe (AFT Arrow Steady)	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)
1	Pipe	0	0	0	0	0	0	600.00	600.00	600.00	600.00	0	300.00
2	Pipe	0	0	0	0	0	0	14.70	14.70	14.70	14.70	0	70.00

Figure 27: The Output window displays the values of steady-state parameters in the pipes tab

Time	Event Message
4.426E-03	Valve Junction 2 became sonically choked at Time: 0.004425754 seconds
0.05753	Valve Junction 2 lost sonic choking at Time: 0.0575348 seconds
2.523	Assigned Pressure Junction 3 became sonically choked at Time: 2.52268 seconds
116.5	Assigned Pressure Junction 3 lost sonic choking at Time: 116.4593 seconds

Figure 28: The Output window displays event messages about system behavior in the Event Messages by Time tab

## A. Modify the output format

Go to the Transient Max/Min tab in the Output window. If you selected the default AFT xStream Output Control, the Max. Mass Flowrate and Min. Mass Flowrate should have units of lbm/sec (pounds mass per second).

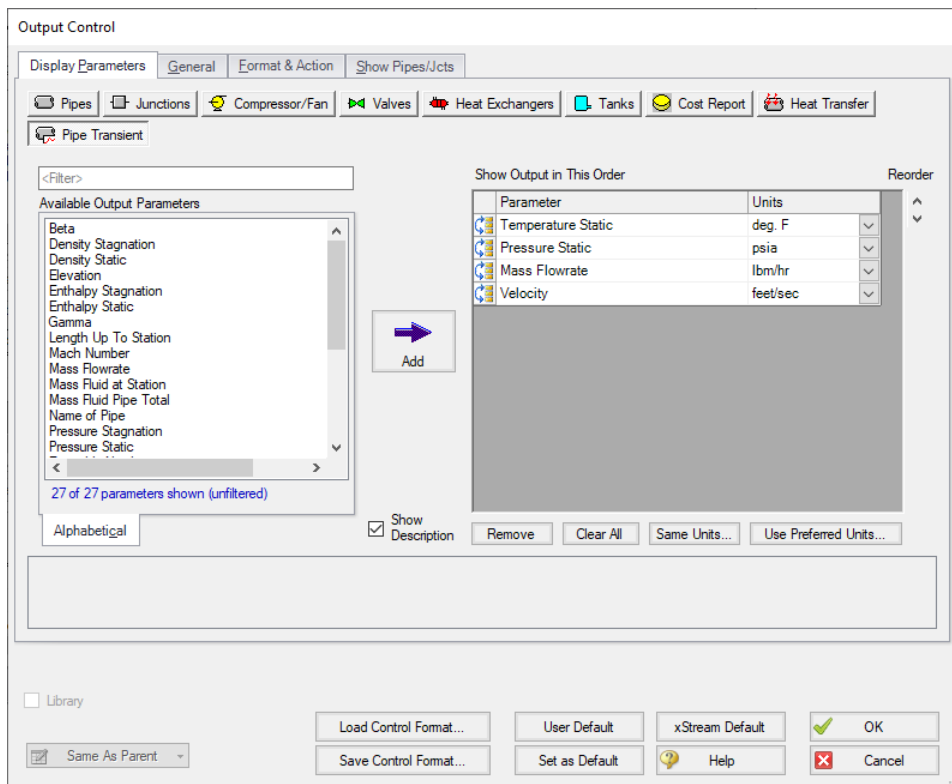
➤ **Select Output Control from the Tools menu or the main Toolbar.** Open the Pipe Transient results section. On the right side of the Pipe Transient results section is the list of currently selected output parameters (see Figure 29). Click and change the units by clicking the arrow beside the units, and then selecting lbm/hr (pounds mass per hour) from the drop down list.

➤ **Click OK to display changes to the current results.** You should see both the Max. Mass Flowrate and Min. Mass Flowrate in units of lbm/hr.

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

## Tank Blowdown Example

- **Select the Temperature Static** parameter and use the Reorder scroll bar to move it up to the top of the parameter list.
- **Click OK to display the changes to the current results.** You will see in the Transient Max/Min table that the first column now contains Temperature Static. The Output Control window allows you to obtain the parameters, units and order you prefer in your output. This flexibility will help you work with AFT xStream in the way that is most meaningful to you, reducing the possibility of errors.
- **Lastly, double-click the column header Max. Mass Flowrate** in the Transient Max/Min tab. This will open a window in which you can change the units once again if you prefer. These changes are extended to the Output Control parameter data you have previously set.




**Figure 29: The Output Control window allows for the modification of Parameters shown, Units, and the Order of Results**

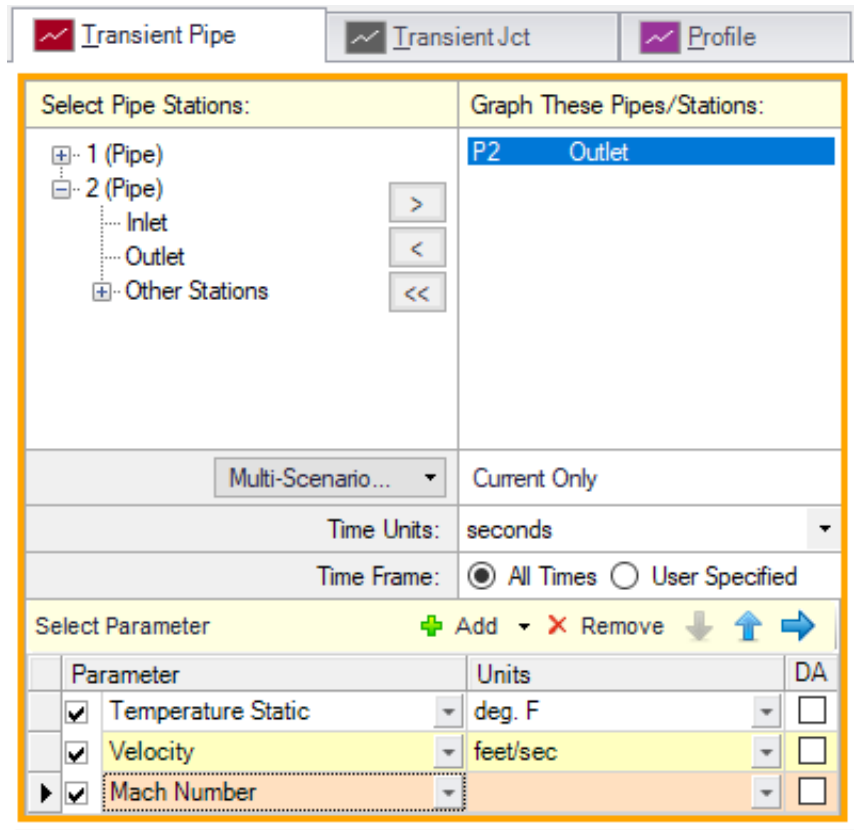
### B. Graph the results

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 choosing it from the Windows menu, clicking the Graph Results tab, by clicking "Graph Results" from the Solution Progress window after running the model, or by pressing Ctrl+G.

## Tank Blowdown Example

The Graph Parameters menu will automatically be displayed in the Quick Access Panel on the far right of the Graph Results window, and is where you can specify which graphs to generate. On the Transient Pipe tab under Select Pipe Stations, expand P2 pipe stations and double-click, or click  (Add button) the outlet station. This pipe computing station correlates to the pipe discharge for the Blowdown Tank system. Select the Graph Parameter as Temperature Static and set the units to deg F. Add the Parameters Velocity and Mach Number (See Figure 30).



**Figure 30:** 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

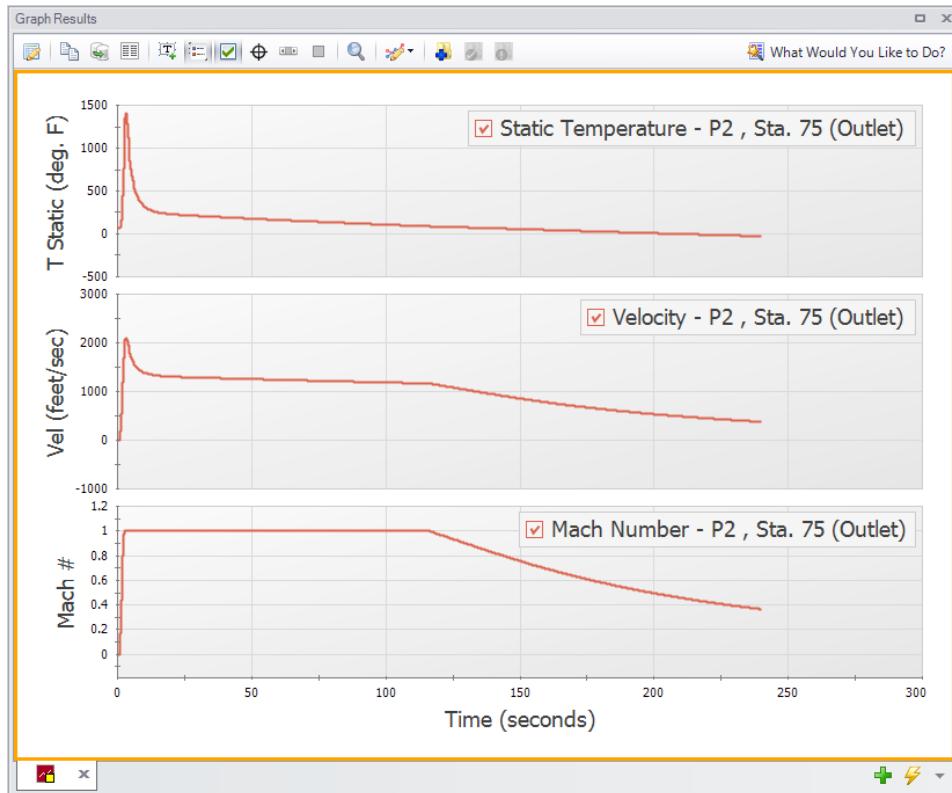
➤ **Click the Generate button.** The graph shows the temperature, velocity, and Mach number at the exit for Pipe 2 (Figure 31).

You can use the other buttons in the Graph Results window to change the graph appearance and to save and import data for cross-plotting. The Graph Results window can be printed, saved to file, copied to the clipboard, or printed to an Adobe PDF file. The graph's x-y data can also be exported to a file or copied to the clipboard.

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



## Tank Blowdown Example



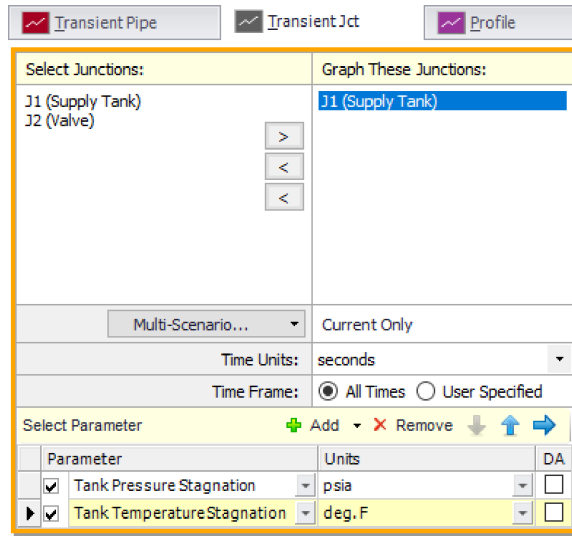
**Figure 31: The Graph Results window offers full-featured plot generation. The static temperature, velocity, and Mach number of Pipe P2 outlet are shown.**

Further review of the graph results in Figure 31 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 115 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. It is good to keep in mind this correlation between temperature and sonic velocity. This relationship is important to understand when looking at systems that encounter choking.

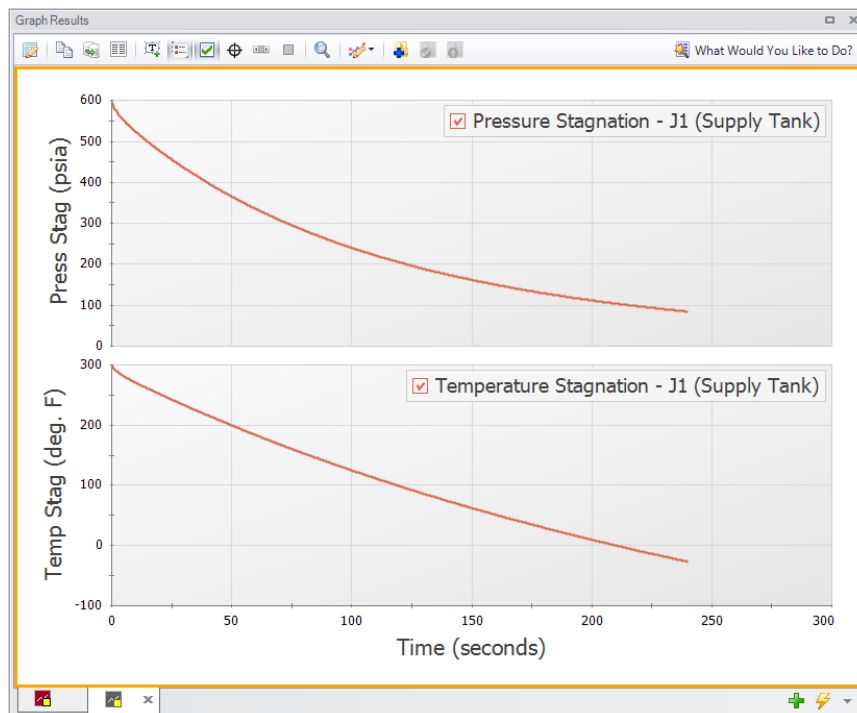
➤ **In the Quick Access Panel**, open the Transient Jct Tab. Under Select Junctions, add J1 (Supply Tank). Select the Graph Parameters as Tank Pressure Stagnation and Tank Temperature Stagnation (see Figure 32). Set the units to psia and deg. F. Click the Generate button. This will show the temperature and pressure in the supply tank (Figure 33).

**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.

## Tank Blowdown Example



**Figure 32: Graph Parameters window for the Transient Junction graph**



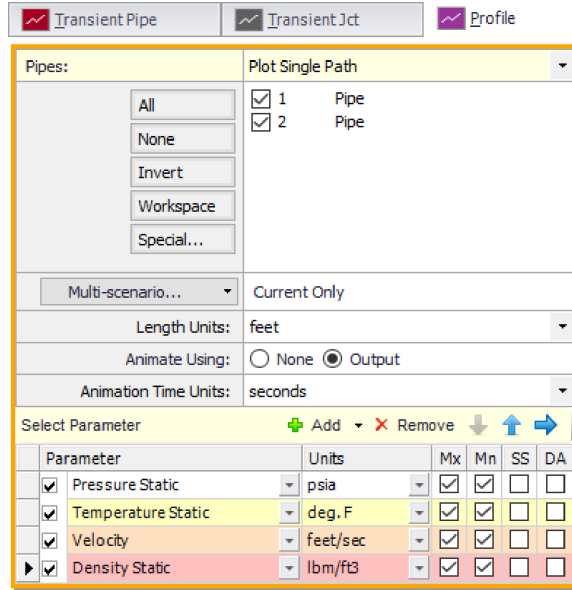
**Figure 33: The Transient Junction graph shows how variables within a junction change over time. The temperature and pressure of Tank J1 over time are plotted.**

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 300 psia. The junction graph would indicate the tank pressure would reach such a point around 75 seconds after beginning of the tank blowdown. Also note that the rate of decrease in the tank pressure decays at later time points as the pressure gradient driving flow out of the tank becomes smaller as time progresses.

## Tank Blowdown Example

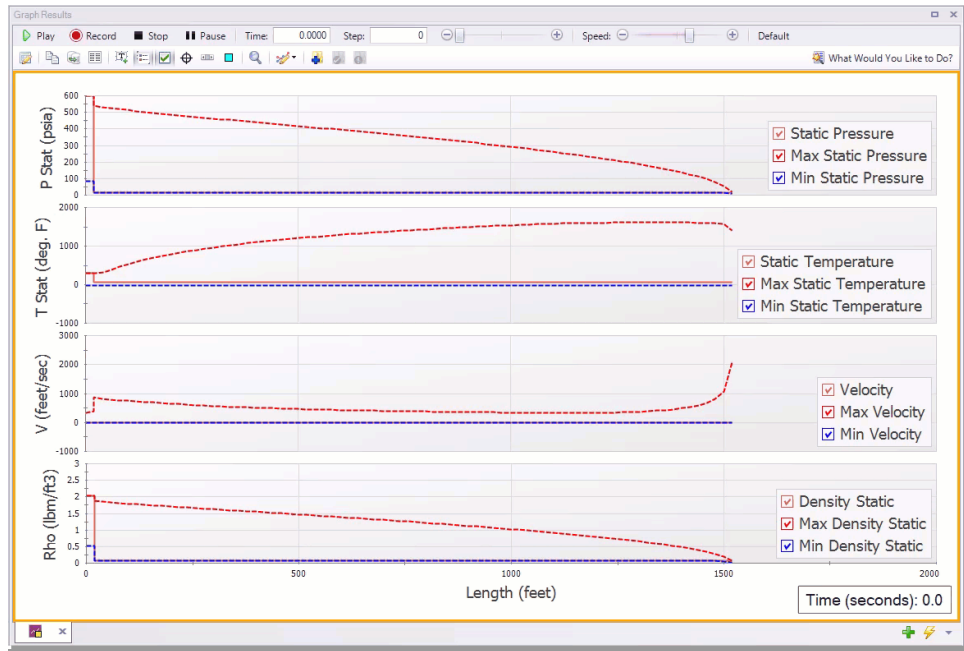
Lastly, we will animate the profile of the Blowdown Tank system to show the wave caused by opening the valve. In the Quick Access Panel, open the Profile tab. Under Plot Single Path, select Pipe 1 and Pipe 2. After ensuring your length is feet, select the option to Animate Using Output and ensure the Animation Time Units are set to seconds. Add Parameters until you have 4, then set them to Pressure Static, Temperature Static, Velocity, and Density Static. Set the units to psia, Deg. F, feet/sec, and lbm/ft<sup>3</sup>, respectively (see Figure 34).

➤ **Click the Generate button.** This will show an animated profile of the pipe system over the specified time frame (see Figure 35).



**Figure 34: Profile Graph Parameters window**

## Tank Blowdown Example



**Figure 35:** The Profile Graph show the Maximum and Minimum values of all parameters shown in the Graph over the specified time Frame. It can also be set to animate the values so that the behavior within the system as a whole can be observed. The blue line correlates to minimum values, the bright red line shows maximum value, and the dark red line shows the value at the current timestep.

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 that occurs over a shorter length of time than the communication time of the system is considered to happen instantaneously.

### C. View the Visual Report

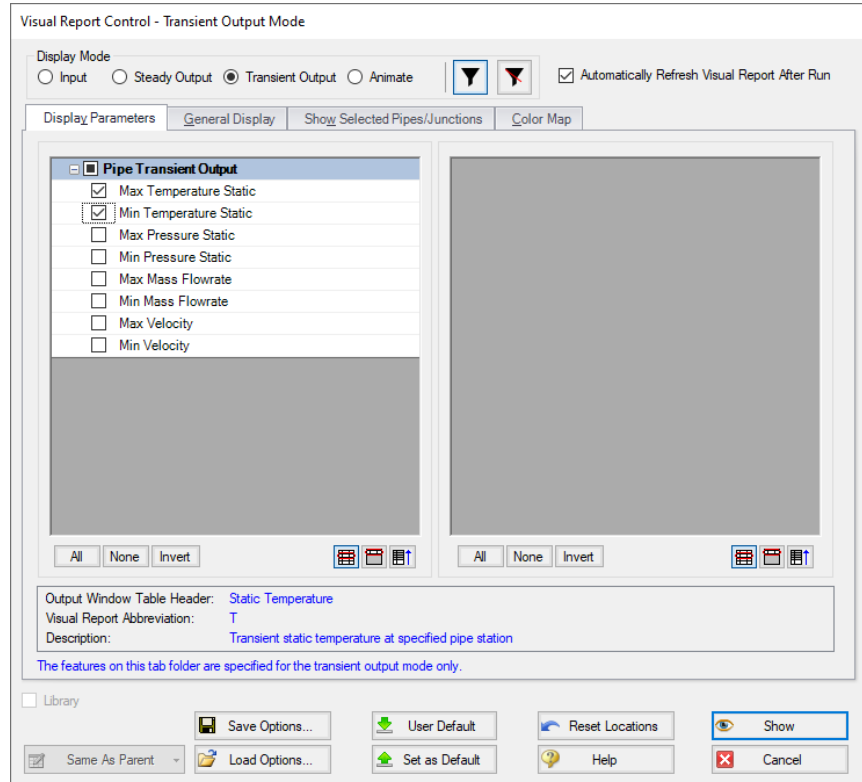
The Visual Report allows you to show text output alongside the model schematic. This is useful to quickly show pertinent information in relation to location in the model. The Visual Report can also animate the transient pipe results in a color animation overlaid on the model.

➤ **Change to the Visual Report window** by choosing it from the Window menu, clicking the Visual Report tab or by pressing Ctrl+I. The Visual Report window allows you to integrate your text results with the graphic layout of your pipe network. The Visual Report Control window should open automatically when accessing the Visual Report window for the first time. The Visual Report Control window can also be opened from the Visual Report Toolbar or the Tools menu. Note that the parameters present in Visual Report Control are determined by the parameters selected in Output Control.

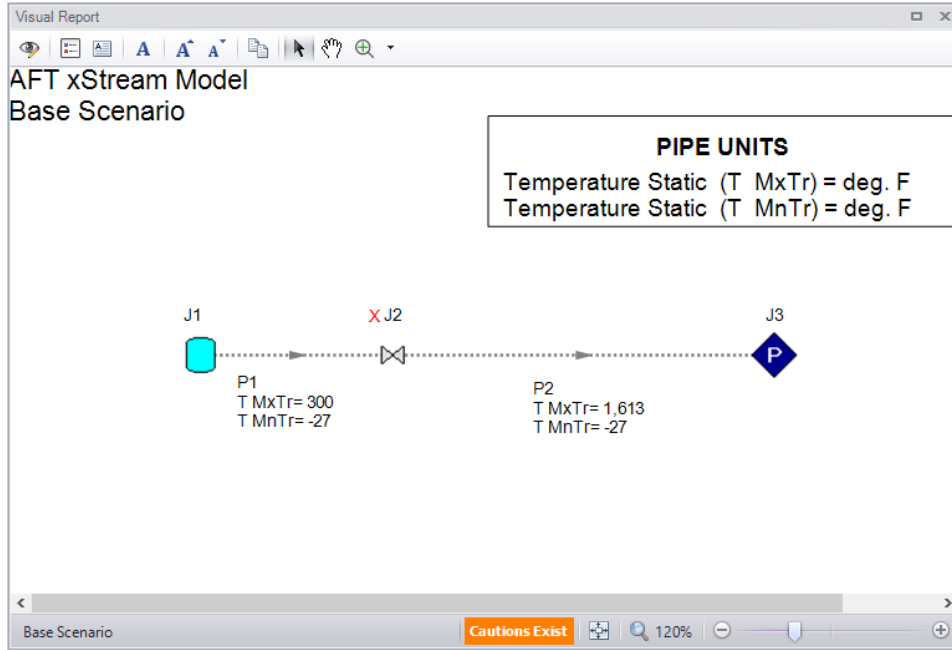
➤ **Select "Max Temperature Static" and "Min Temperature Static"** in the Pipe Transient Output area of the Visual Report Window as shown in Figure 36. Click the Show button. This will generate the Visual Report window graphic seen in Figure 37.

## Tank Blowdown Example

It is common for the text in the Visual Report window to overlap when first generated. You can change this by selecting smaller fonts or by dragging the text to a new area to increase clarity. You can also use the Visual Report Control window to display units in a legend to increase the clarity of the display. These adjustments have already been done in Figure 37. This window can be printed or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.



**Figure 36: The Visual Report Control window selects content for the Visual Report window**



**Figure 37: The Visual Report integrates results with model layout**

## D. 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.

To see the impact of convective heat transfer on the results create a child scenario named “Convective Heat Transfer.”

In the “Convective Heat Transfer” scenario update pipes P1 and P2. Open the Pipe Properties window for each pipe and enter the following on the Heat Transfer tab (Fig):

1. Heat Transfer Model = Convective Heat Transfer
2. External Convection Coefficient Correlation = Free-Horizontal (Churchill-Chu)
3. Ambient Temperature = 70 deg. F

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 350 deg. F and click OK to accept the changes.

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

In the Transient Max/Min tab shown in Figure 38 it can be seen that the maximum temperature in pipe P2 is only 228 deg. F as opposed to 1613 deg. F which was calculated with adiabatic heat transfer.

Go to the Graph results window and recreate the pipe P2 outlet graph from Figure 31 as is shown below in Figure 39.

## Tank Blowdown Example

Pipe	Max. Static Temperature (deg. F)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. F)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station
1	300.0	0.000	0	-21.896	240.0	0	600.0	0.000	1
2	228.4	16.99	0	2.577	0.02110	0	513.8	2.982	0

Figure 38: Transient Max/Min temperature output with convective heat transfer modelled

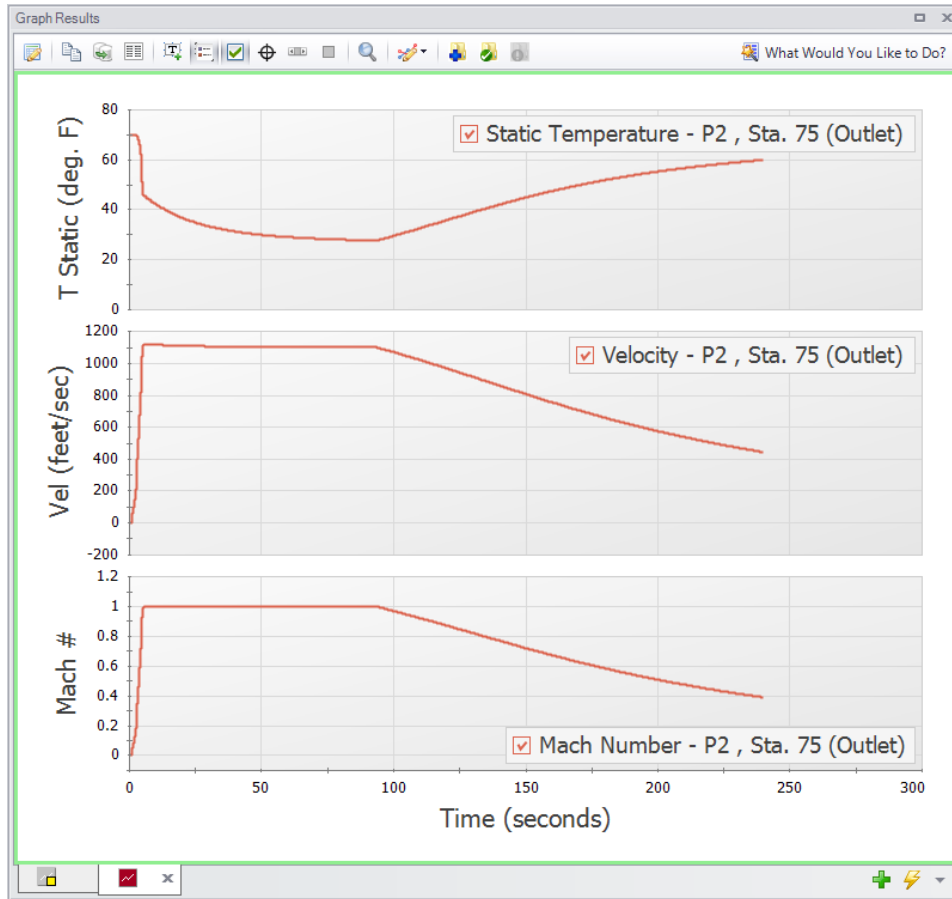


Figure 39: Temperature, velocity, and Mach number for the discharge of Pipe P2 with convective heat transfer

### E. 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. Comparing Figure 33 to Figure 42, values such as the tank 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 2 (Figure 25 and Figure 40), changed substantially. This value increased from 1613 deg. F to 2097 deg. F. This change in the maximum temperature indicates that the difference in sectioning has significant implications if your interest is

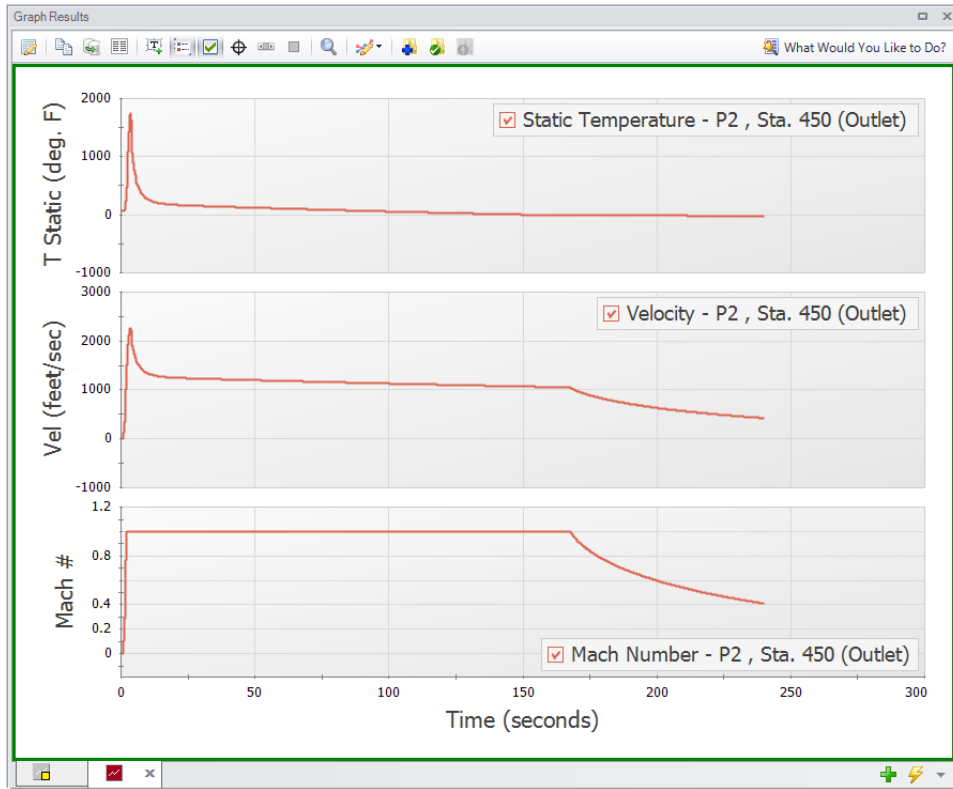
## Tank Blowdown Example

in the conditions in the blowdown pipe. It should be noted that changing the Minimum Number of Sections Per Pipe from one to six caused the computation time of the model to increase from 5-6 minutes to 3.5 hours. Therefore, it is of critical importance to balance accuracy and computation time when using AFT xStream. This idea is discussed in more detail [here](#).

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.

Pipe	Max. Static Temperature (deg. F)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. F)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station
1	300.0	5.163E-03	0	-23.69	240.0	0	600.0	0.000	0
2	2,098.2	3,049	365	-33.33	240.0	450	540.4	2,625	0

**Figure 40: Transient Min/Max window for modified sectioning**



**Figure 41: Temperature, velocity, and Mach number for the discharge of Pipe P2 with 6 sections minimum per pipe**



# Tank Blowdown Example

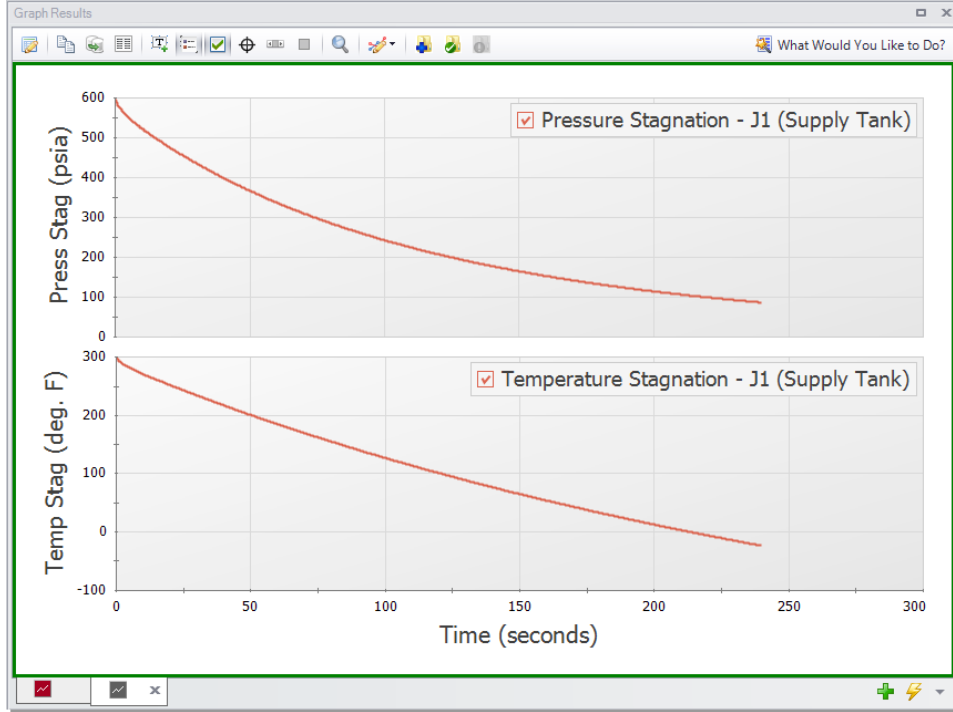


Figure 42: Temperature and pressure stagnation for Tank J1 with 6 sections minimum per pipe

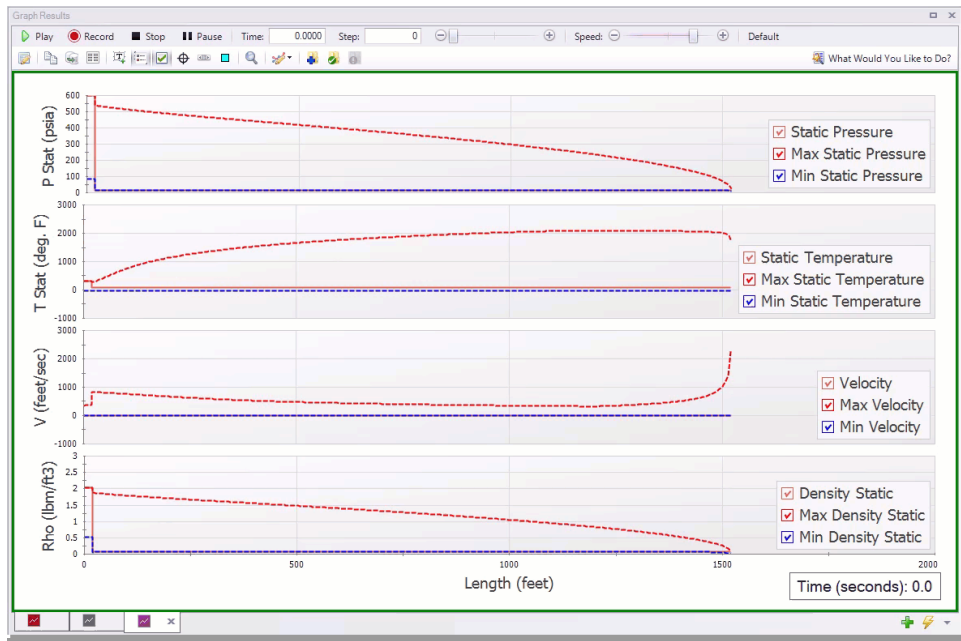


Figure 43: Profile Graph for Tank Blowdown system with 6 Sections Minimum Per Pipe where the blue line represents the minimum value, the bright red line represents the maximum value and the dark red line represents the value at the current time step

## **Conclusion**

---

You have now used AFT xStream's five Primary Windows to build and analyze a simple gas transient model.

# Gas Turbine Fuel System Example

## Summary

---

A compressor feeding natural gas to combustion baskets in a Turbine Fuel System has one of its 3 control valves close instantaneously. The effect of this transient event on the temperature, pressure, and velocity of the system is observed.

## Topics covered

This example will cover the following topics:

- Modeling a valve closure as a transient event
- Modeling a heat exchanger
- Using the Scenario Manager in AFT xStream
- Indicating the importance of the Estimated Maximum Temperature During Transient
- Creating animated profile graphs

## Required knowledge

This example assumes the user has already worked through the [Beginner: Tank Blowdown](#) example, or has a level of knowledge consistent with that topic.

## Model file

The file listed below contains a completed version of this example and can be located in the Examples folder as part of the AFT xStream installation. This file can also be downloaded from the AFT xStream help site from the Gas Turbine Fuel Example page:

- *US - Gas Turbine Fuel System.xtr*

## Step 1. Start AFT xStream

---

Open AFT xStream. Click "Start Building Model".

## Step 2. Build the model

### A. Place the pipes and junctions

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

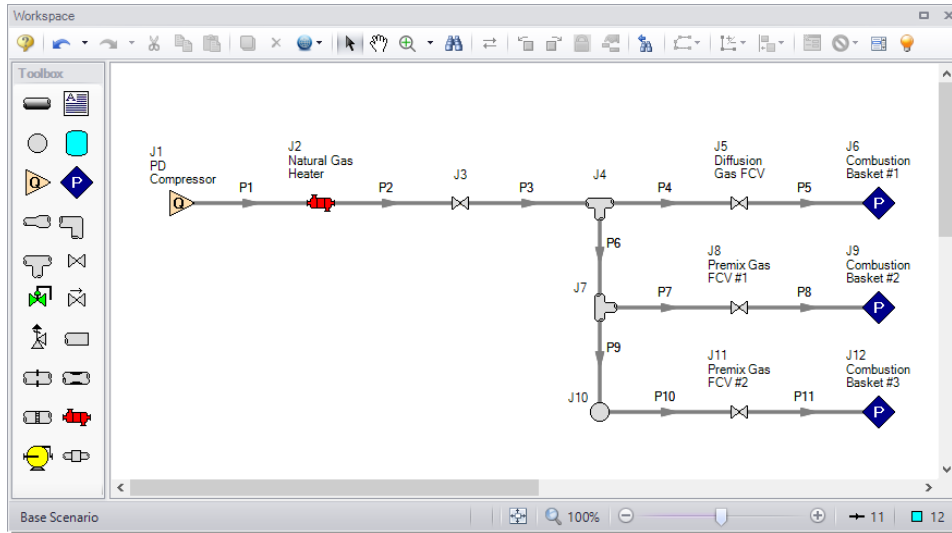


Figure 1: Layout of gas turbine fuel system

### B. Enter the pipe and junction data

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

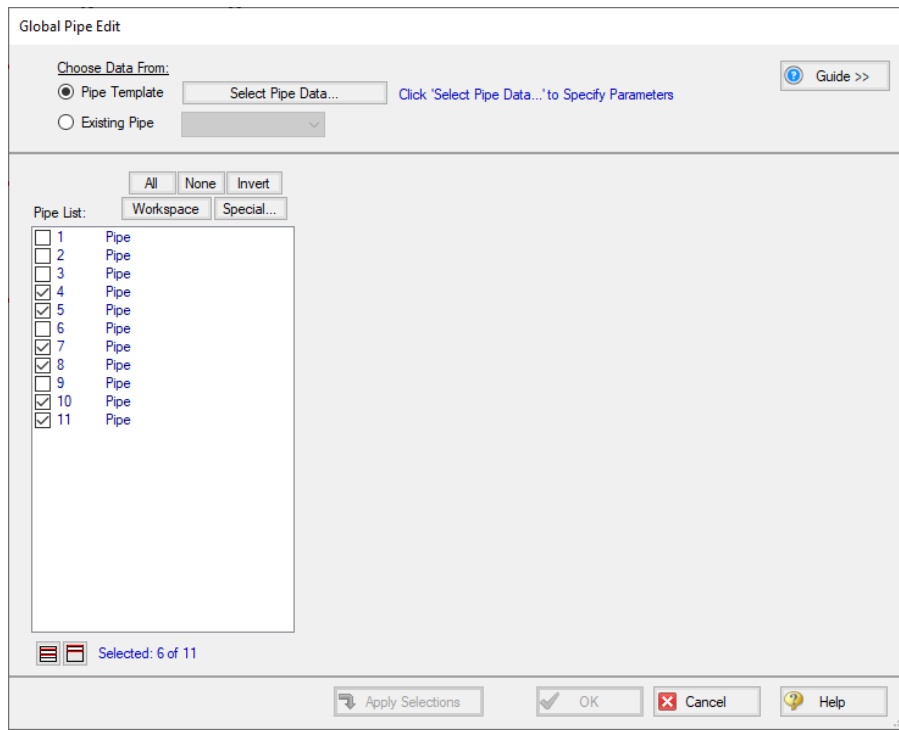
All Pipes are Stainless Steel - ANSI, Schedule 10S, and have standard roughness. The rest of the pipe information is as follows:

Pipe #	Size (inches)	Length (feet)
1	8	75
2	8	25
3	8	5
4	5	5
5	5	5

## Gas Turbine Fuel System Example

<b>Pipe #</b>	<b>Size (inches)</b>	<b>Length (feet)</b>
6	8	5
7	5	5
8	5	5
9	8	5
10	5	5
11	5	5

Because several of the pipes are the same, you can use the Global Pipe Editing tool to speed up the process of entering the data. The Global Pipe Editing 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" drop-down on the toolbar 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.

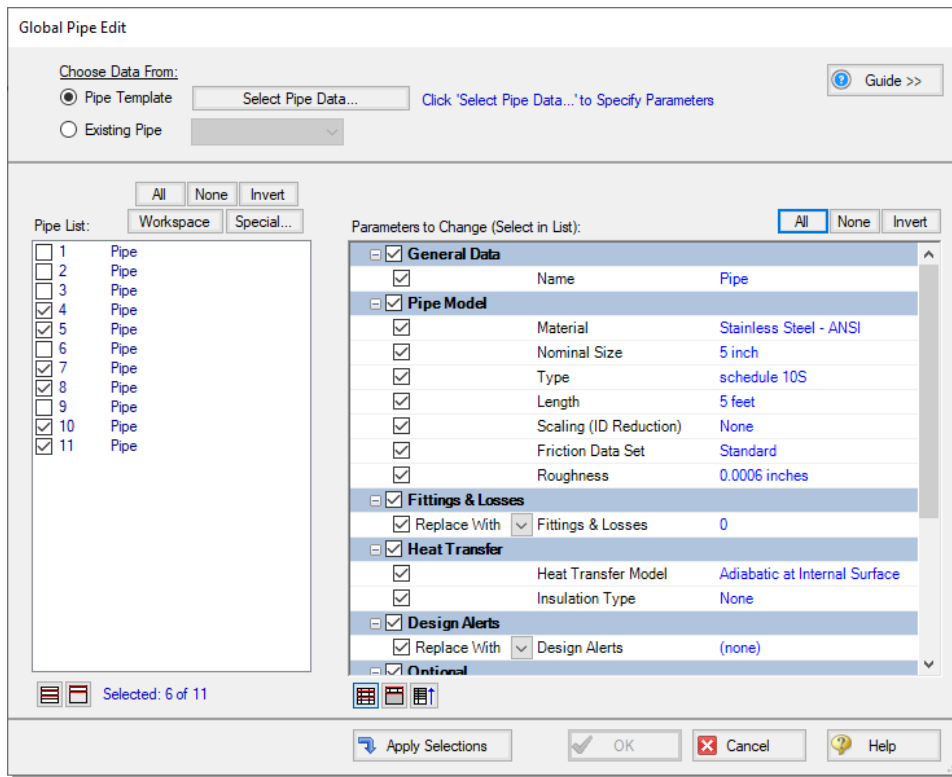


**Figure 2: Global Pipe Edit window**

After selecting the pipes, 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. For this example, you want all of the parameters for all of the selected pipes to be updated to the

## Gas Turbine Fuel System Example

specified values. This can be accomplished by selecting the "All" button above the list of parameters. 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. Select OK. 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.

---

**Notes:** 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 feet
  - c. Mass Flow Rate = 90,000 lbm/hr

---

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

---

- d. Temperature = 70 deg. F
2. Heat Exchanger J2

## Gas Turbine Fuel System Example

---

- a. Name = Natural Gas Heater
- b. Elevation = 0 feet
- c.  $K = 6.6$
- d. Steady Heat Rate = 3,535,200 Btu/hr

---

**Note:** Be advised that the unit for heat rate is not the default as it is Btu/hr as opposed to Btu/s.

---


- 3. Valve J3
  - a. Elevation = 0 feet
  - b.  $C_v = 10,000$
  - c.  $X_t = 0.3$
- 4. Valve J5
  - a. Name = Diffusion Gas FCV
  - b. Elevation = 0 feet
  - c.  $C_v = 95$
  - d.  $X_t = 0.7$
- 5. Valve J8
  - a. Name = Premix Gas FCV #1
  - b. Elevation = 0 feet
  - c.  $C_v = 195$
  - d.  $X_t = 0.7$
- 6. Valve J11
  - a. Name = Premix Gas FCV #2
  - b. Elevation = 0 feet
  - c.  $C_v = 195$
  - d.  $X_t = 0.7$
  - e. Click on the Transient Tab and enter the following data

<b>Time (seconds)</b>	<b><math>C_v</math></b>	<b><math>X_t</math></b>
0	195	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 feet
  - e. Stagnation Pressure = 15.5 atm
  - f. Temperature = 140 deg F.

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

## 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 3. Specify Analysis Setup

### A. Specify the Fluid panel

➤ **Open the Analysis Setup window by clicking Analysis Setup** on the Main Toolbar. Under the "AFT Standard" fluid option, select "Methane" from the list and click the "Add to Model" button. Click OK.

**Note:** Methane is used to model natural gas in order to minimize the run time. If you need to model a real natural gas mixture, then instead of using the AFT Standard fluid library, it would be better to use the NIST REFPROP library or the optional Chempak add-on.

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

➤ **Open the Pipe Sectioning and Output panel.** Set the Minimum Number of Sections Per Pipe to 8 sections and the Estimated Maximum Pipe Temperature During Transient to 140 deg. F.

**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 Steps 8 and 9.

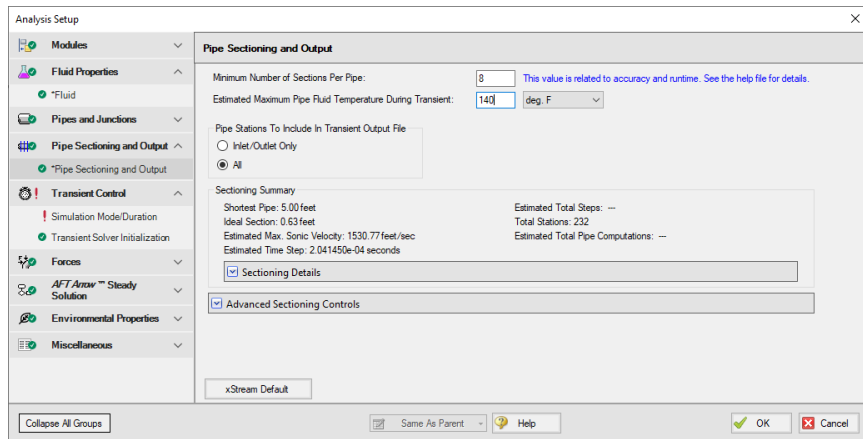


Figure 4: Sectioning panel for Turbine Fuel System



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

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

## Step 4. Create scenarios to model different sectioning setups

---


### A. Create scenarios

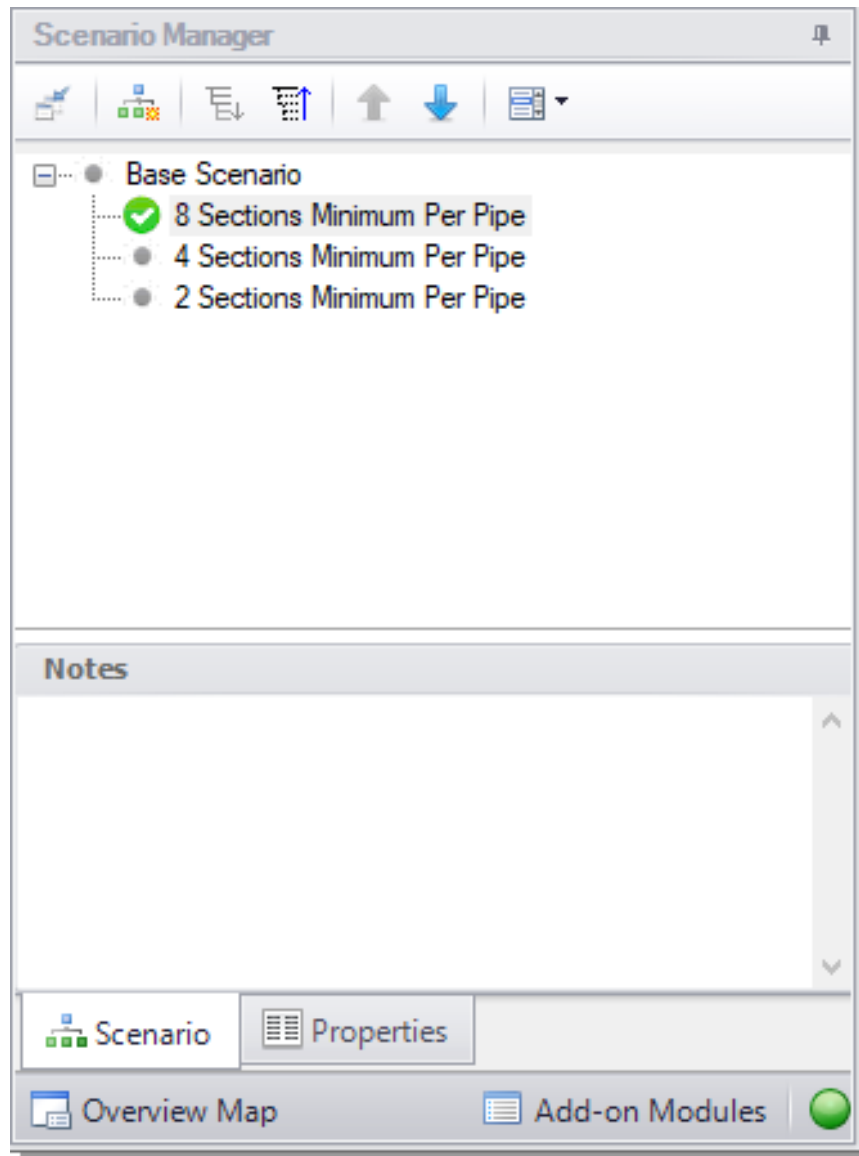
In this model, the following three types of Sectioning cases for the system are to be evaluated:

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. Create another child and call it "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: The Scenario Manager on the Quick Access Panel displays the scenario tree which allows you to create scenarios and keep them organized within the same model file*

## **B. Set up scenarios**

Since the "8 Sections Minimum Per Pipe" scenario has the same setup as the Base Scenario, we do not need to modify it. Select the "4 Sections Minimum Per Pipe" scenario by double-clicking the name in the Scenario Manager.

➤ **Open the Analysis Setup window** from the Main Toolbar. Click on the Pipe Sectioning and Output panel and change the Minimum Number of Sections Per Pipe to 4 then click OK.

➤ **Open the Child Scenario "2 Sections Minimum per Pipe"**. Open the Analysis Setup window from the Main Toolbar. Open the Pipe Sectioning and Output panel and change the Minimum Number of Sections Per Pipe to 2 then click OK.

### Step 5. Run the first scenario

---

➤ **Double-click the "2 Sections Minimum Per Pipe" scenario** in the Scenario Manager. Select "Run Model" on the toolbar. AFT xStream will prompt you to save, then will open the Solution Progress window. This model is estimated to run in approximately one minute, but the run time is dependent on the speed of your computer.

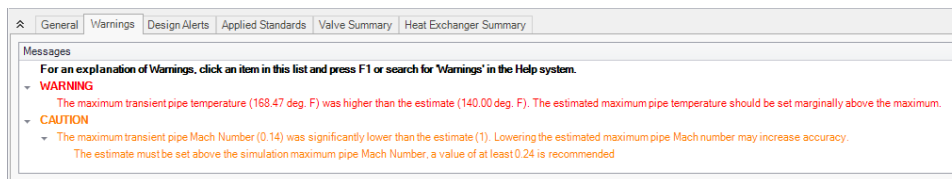
### Step 6. Review the results

---

➤ **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 error 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 7. Adjust the Pipe Sectioning panel

➤ **Double-click "Base Scenario"** in the Scenario Manager. Open the Analysis Setup window from the Main Toolbar and select the Pipe Sectioning and Output panel. Change the Estimated Maximum Pipe Temperature During Transient to 180 deg. F. Click OK.

**Note:** It is essential that this change is made in the Base Scenario so all scenarios are updated.

## Step 8. Re-run the model and review results

### A. Review Output window

➤ **Double-click the "8 Sections Minimum Per Pipe" scenario.** Select "Run Model" from the Analysis menu, 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 error as discussed previously. Select the Transient Max/Min tab (Figure 7). Here we can see that the maximum temperature in the inlet lines (P5, P8, & P11) to the combustion baskets is 163.6 deg. F and occurs in the inlet line to combustion basket 1.

Pipes		Axial Points		Transient Output		Transient Max/Min		Heat Transfer		
Pipe	city	Min. Velocity (feet/sec)	Min. Velocity Time (seconds)	Min. Velocity Station	Max. Static Temperature (deg. F)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. F)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station
1	120	61.50	0.04360	120	80.41	0.8113	120	69.89	0.2390	2
2	40	66.73	0.02556	40	173.19	0.04360	0	139.28	0.000	40
3	8	66.25	0.02180	8	166.12	0.3738	0	139.27	0.000	8
4	0	49.35	0.02735	0	165.67	0.4283	0	139.59	3.963E-03	8
5	8	55.58	0.000	0	163.62	0.4974	0	138.40	0.02933	4
6	8	35.39	0.02041	2	165.74	0.4283	0	139.42	0.000	8
7	8	106.48	0.000	0	164.08	0.5091	0	139.27	0.000	8
8	8	113.50	1.982E-04	0	162.31	0.5428	0	137.77	0.02517	5
9	0	-16.40	0.01724	0	164.97	0.5093	0	139.62	0.000	8
10	8	-28.64	0.01407	0	157.97	2.000	0	139.27	0.000	8
11	8	-55.13	0.01348	8	159.26	0.01665	0	134.58	0.01070	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:

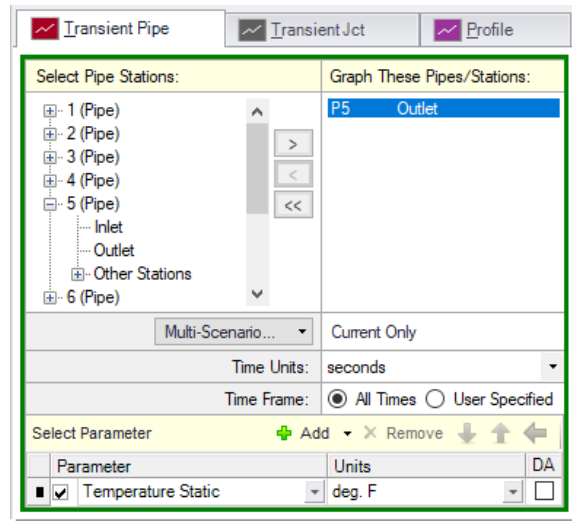
1. From the Graph Control tab on the Quick Access Panel, select the Transient Pipe tab as shown in Figure 8

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" and specify "deg. F" in the drop-down box under "Units"
5. Click the Generate button

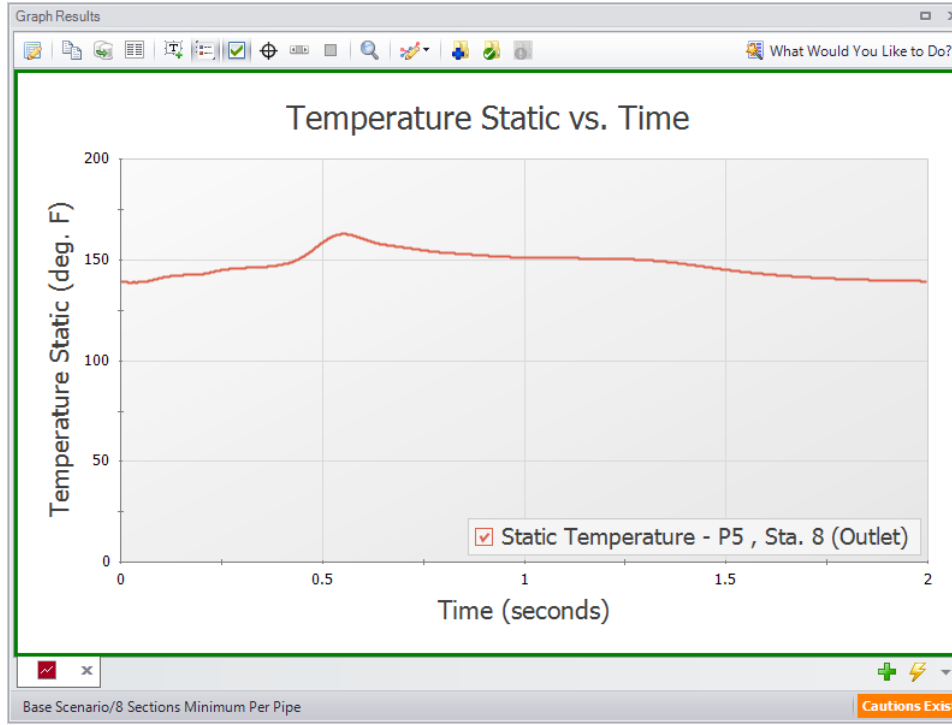
The results (see Figure 9) show a maximum temperature of 162.9 deg. F at the entrance to combustion basket #1. Note this temperature is lower than the temperature shown on the Transient Max/min tab because this graph is showing the outlet of pipe P5, while the maximum temperature in pipe P5 occurs at the inlet of the pipe. The maximum temperature for the outlet of P5 occurs at about 0.6 seconds before returning to the previous steady state temperature of 140 deg. F after the flow has normalized.

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. After generating the graph, click the "Add Graph to List" icon  on the Graph Results Toolbar (or simply right-click the graph itself and choose "Add Graph to List"). 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 templates are saved as opposed to the graph data.



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



**Figure 9: Combustion Basket #1 Temperature vs Time Graph**

### C. Animate the results

We will create a graph that models the temperature, pressure, and velocity along the flow path between the positive displacement compressor and Combustion Basket #1. To do this:

1. Create a new graph tab by clicking the "New Tab" button
2. Select the Profile tab under the Graph Control tab in the Quick Access Panel. Alternatively, you can access the Select Graph Parameters window by clicking on the "Select Graph Parameters" icon located at the top left of the Graph Results window.
3. Select pipes 1, 2, 3, 4, and 5
4. Select the option to Animate Using Output.
5. Ensure that the animation time used is seconds.
6. Select Temperature Static as a parameter.
7. Add Pressure Static and Velocity Parameters to the Parameters Window.
8. Click Generate.

*To view the animated profile*

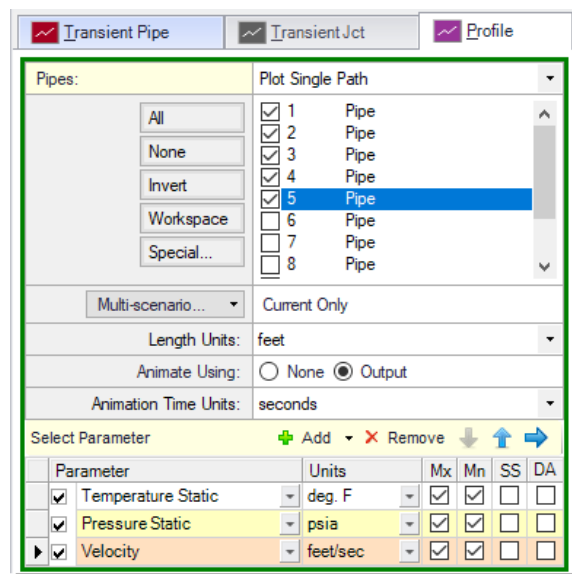
*follow this link: <https://bit.ly/3NbaokZ>*

You should generate a graph that looks like the one shown in Figure 10. 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. This heightened temperature is not immediately seen by the combustion baskets downstream, however, as 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  $T = 0.5$  seconds, at which point the inlet to the combustion basket reaches its maximum temperature.

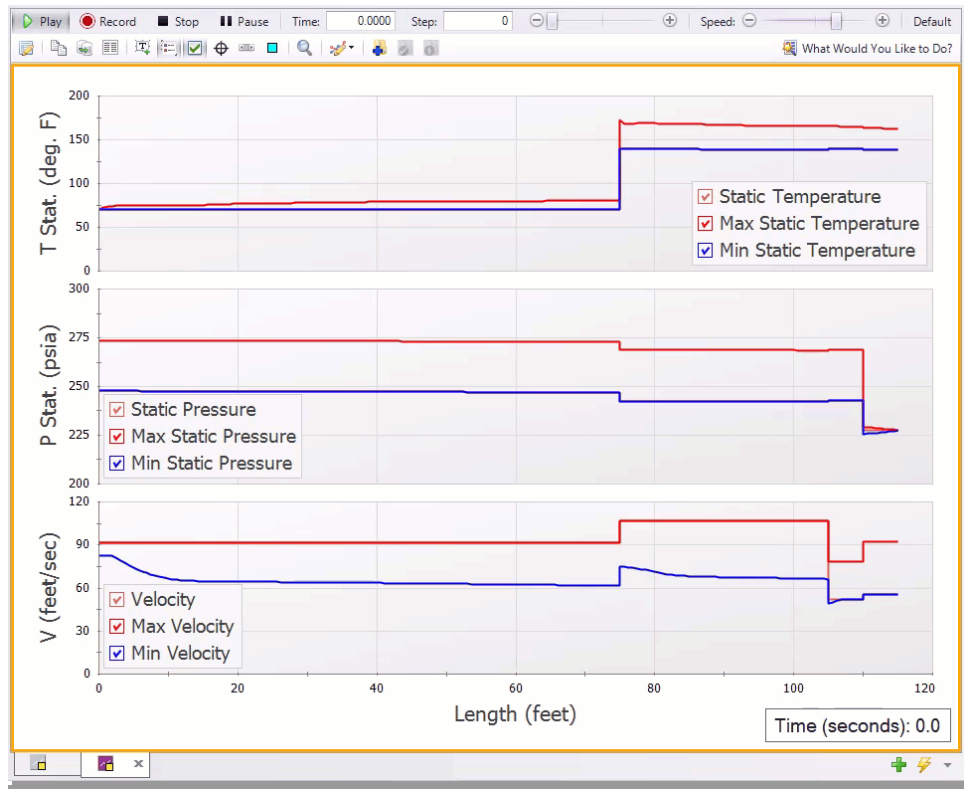
Click the "Add Graph to List" icon and give it a name such as "Combustion Basket #1 Profile". Click OK to add the Graph to the Graph List.

**Note:**Figure 10 has had the axes titles modified to increase clarity.



**Figure 10: Graph Parameters for Combustion Basket #1 Profile**

## Gas Turbine Fuel System Example



**Figure 11: Combustion Basket #1 Profile**

**Note:** You can make similar graphs for combustion basket #2. To do so, change the pipe used in the temperature vs time graph to pipe 8, and change the pipes in the animated profile graph to 1, 2, 3, 6, 7, and 8. This would generate similar graphs to the ones seen in and Figure 10.

### Step 9. Run the other scenarios and graph the results

Using the Scenario Manager, load the other two scenarios and run them.

Create the same graphs you created for Step 10 for each of the scenarios by double-clicking on the graph names in the Graph List Manager. This will reveal that the general behavior of the graph is the same for all 3 scenarios, but that the maximum temperature rises as the section count is increased.

The numerical maximum temperature can be found in the Output window of each scenario and is summarized in Table 1. For these cases the maximum temperatures differ by 4.7 degrees.



**Table 1: Summary of maximum Static Temperature for the three cases**

<b>Case</b>	<b>Max Transient Temperature (deg F)</b>
8 Sections	173.2
4 Sections	170.2
2 Sections	168.5

## **Conclusion**

---

Use of transient conditions within a valve junction allowed us to model the effects of an unexpected valve closure in a system. While this example was focused on temperature, it also showed that values such as pressure and velocity can be tracked by AFT xStream.

Additionally, this example identifies 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 the Method of Characteristics uses. If sonic choking occurs at a temperature higher than the 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.

# High Pressure Steam - Fluid Property Models Example

## Summary

---

This example evaluates two turbines in a high pressure steam system that are simultaneously valved off as a bypass line valve is opened. The focus will be the transient behavior in the bypass line in response to simultaneous transient events. Whether or not condensation occurs will also be evaluated through the use of different Fluid Property Models.

## Topics covered

- Using the isometric grid to create a model
- Creating systems with multiple transients
- Evaluating the selection of a fluid property model

## Required knowledge

This example assumes the user has already worked through the [Beginner: Tank Blowdown](#) example, or has a level of knowledge consistent with that topic.

## Model file

The file listed below contains a completed version of this example and can be located in the Examples folder as part of the AFT xStream installation. This file can also be downloaded from the AFT xStream help site from the High Pressure Steam - Fluid Property Models Example page:

- ***US - High Pressure Steam - Fluid Property Models.xtr***

## Step 1. Start AFT xStream

---

Open AFT xStream. Click "Start Building Model".

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.

## Step 2. Build the model

---

### A. Place the pipes and junctions

The next step is to define 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 pipe layout and provide a better understanding of the system when analyzing calculated forces.

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.

---

## High Pressure Steam - Fluid Property Models Example

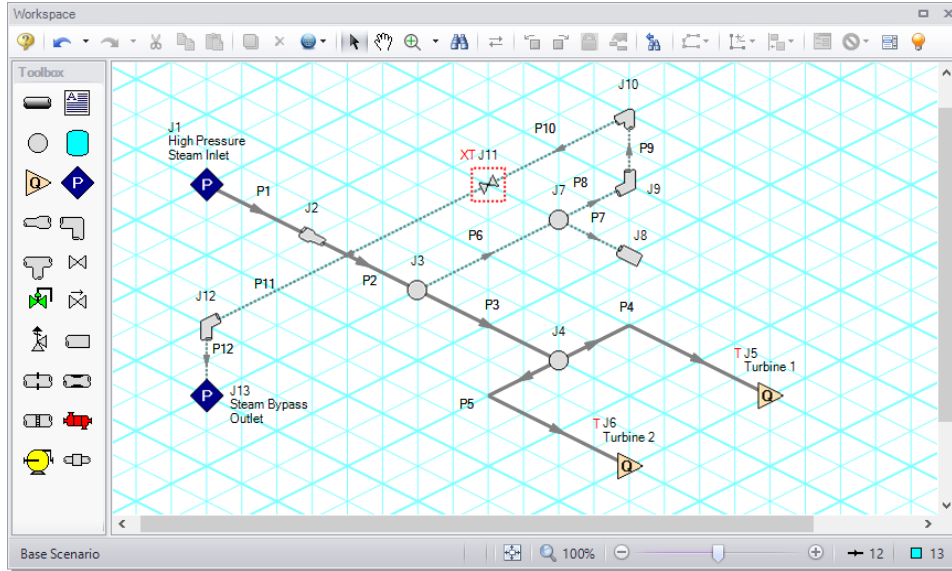


Figure 1: The isometric layout with the grid shown

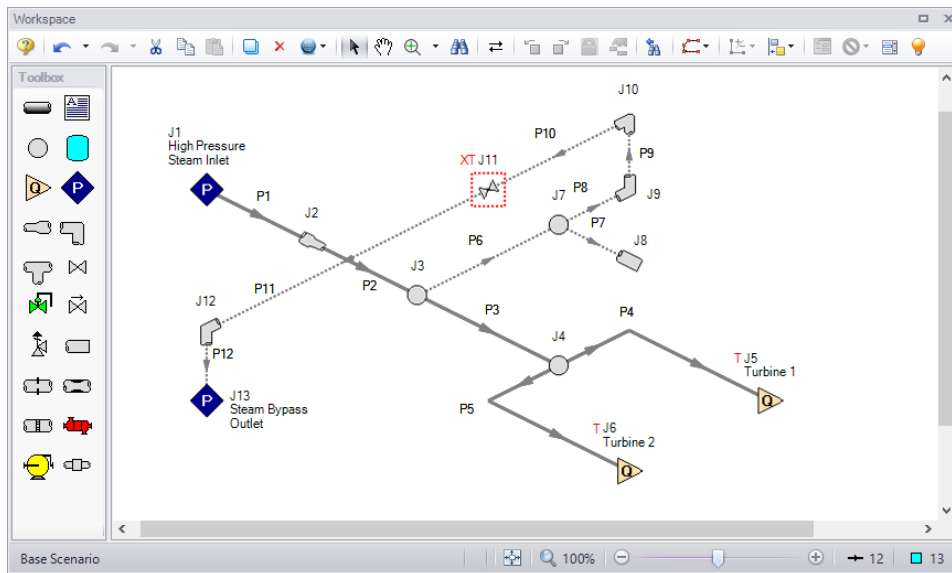


Figure 2: The isometric layout with the grid turned off

### 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 (0.0018 in) as the Friction Model, along with the following data:

## High Pressure Steam - Fluid Property Models Example

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

<b>Pipe #</b>	<b>Inner Diameter (inches)</b>	<b>Length (feet)</b>
1	13.25	100
2	12	60
3	12	45
4	12	15
5	12	15
6	6	6
7	4	6
8	6	10
9	6	5
10	6	20
11	19.25	20
12	19.25	5

The screenshot shows the 'Pipe Properties' dialog box with the following settings:

- Number: 1
- Name: Pipe
- Upstream Junction: 1
- Downstream Junction: 2
- Copy Data From Pipe...: (empty)
- Copy Previous...: (button)
- Size: Pipe Material: (User Specified), Size: (empty), Type: (empty), Inner Diameter: 13.25 inches, ID Reduction (Scaling): (unchecked)
- Length: 100 feet
- Friction Model: Data Set: (empty), Absolute Roughness: 0.0018 inches, Load Default: (button)

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

## High Pressure Steam - Fluid Property Models Example

---

1. Assigned Pressure J1
  - a. Name = High Pressure Steam Inlet
  - b. Elevation = 0 feet
  - c. Static Pressure = 2425 psia
  - d. Temperature = 1060 deg. F
2. Area Change J2
  - a. Type = Conical Transition
  - b. Angle = 45 degrees
3. Branch J3,J4,J7
  - a. Elevation = 0 feet
4. Assigned Flowrate J5,J6
  - a. J5 Name = Turbine 1
  - b. J6 Name = Turbine 2
  - c. Elevation = 0 feet
  - d. Mass Flowrate = 504000 lbm/hr

---

**Note:** Ensure that you change the unit in the Mass Flowrate from lbm/s to lbm/hr, as lbm/hr is not the default unit.

---

- e. Temperature = 1060 deg. F
5. Dead End J8
  - a. Elevation = 0 feet
6. Bend J9
  - a. Elevation = 0 feet
  - b. Type = Standard Elbow (knee, threaded)
7. Bend J10, J12
  - a. Elevation = 5 feet
  - b. Type = Standard Elbow (knee, threaded)
8. Valve J11
  - a.  $C_v = 3000$
  - b.  $X_t = 0.7$
  - c. Elevation = 5 feet
  - d. In 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><math>C_v</math></b>	<b><math>X_t</math></b>
0	0	0.7
2	3000	0.7
5	3000	0.7

9. Assigned Pressure J13
  - a. Name = Steam Bypass Outlet
  - b. Elevation = 0 feet
  - c. Static Pressure = 435 psia
  - d. Static Temperature = 660 deg. F

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

➤ 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 3. Specify Analysis Setup

### A. Specify Fluid panel

➤ Open the **Analysis Setup** window from the **Main Toolbar** and open the Fluid panel. Under the "AFT Standard" fluid option, choose "Steam" from the list and click the "Add to Model" button. Click OK.

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

➤ Open the **Pipe Sectioning and Output** panel. Specify the Minimum Number of Sections Per Pipe as 1 and the Estimated Maximum Pipe Temperature During Transient to 1100 deg. F.

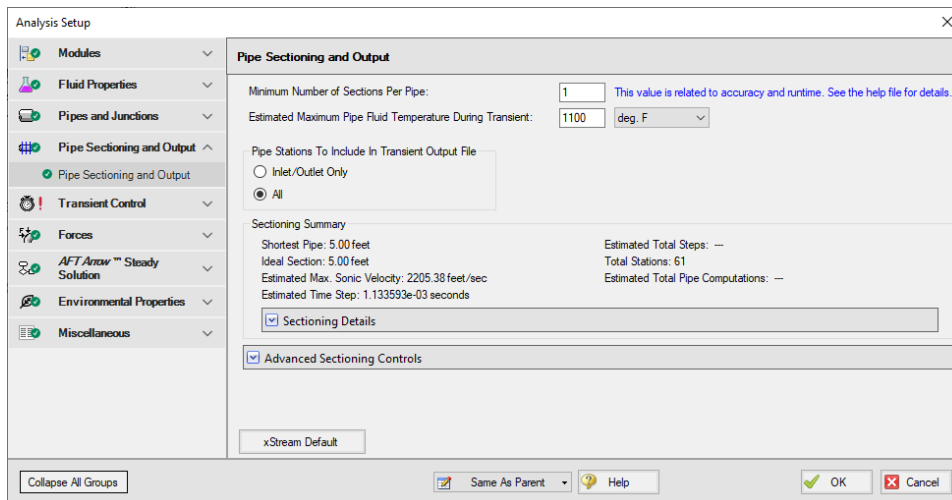


Figure 4: Sectioning panel for High Pressure Steam System

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

➤ **Open the Simulation Mode/Duration panel from the Transient Control group** from the Analysis Setup window. Enter the Stop Time as 5 seconds. Click the OK button.

### Step 4. Run the model

---

➤ **Select "Run Model"** from the Main Toolbar. The model has an estimated run time of 5 minutes, but the run time is dependent on the speed of your PC.

### Step 5. Graph the results

---

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

➤ **Go to the Graph Results window.** The first graph we 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 time units are set to seconds
4. In the Parameters definition area, select "Pressure Static" and specify "psia" in the dropdown box under Units
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. Uncheck the Auto Scale box
3. Set the Minimum to 2200 psia, the Maximum to 2600 psia, and Major Val to 100 (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.



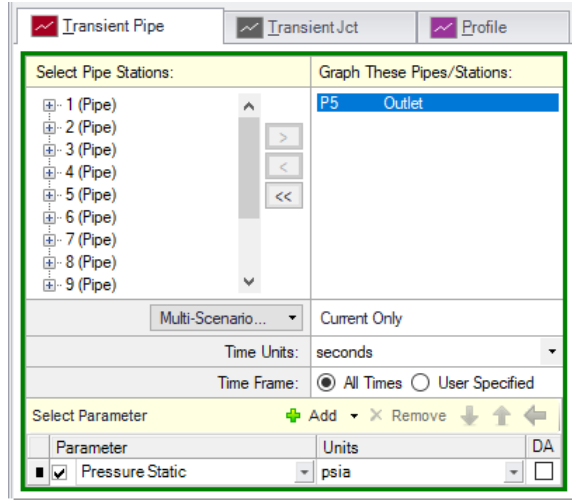


Figure 5: Graph Parameters for Inlet Pressure to Turbines

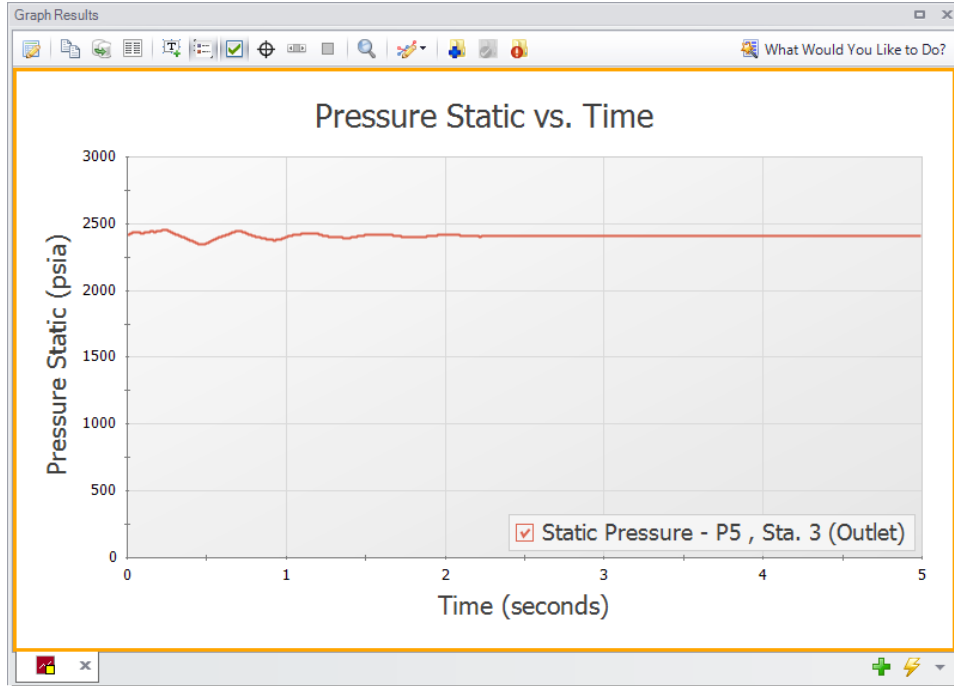


Figure 6: Static pressure at steam turbine 2 entrance

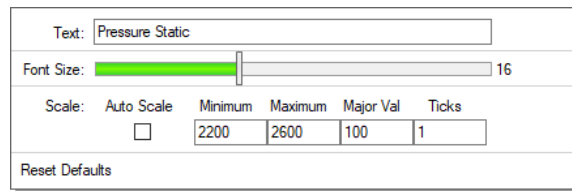
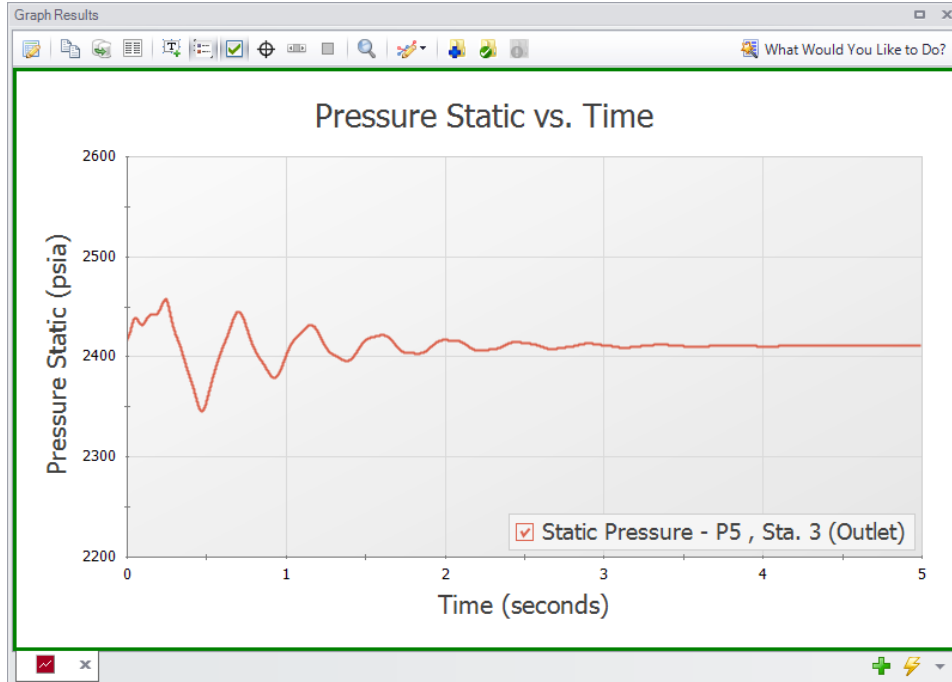


Figure 7: Axis Modification Window opened by right-clicking the y-axis



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

### B. Create a profile of the bypass line temperature and pressure

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

1. Click the "New Tab" button to create a new graph.
2. From the Graph Control tab on the Quick Access Panel, select the Profile tab (Figure 9)
3. Select pipes 6, 8, 9, and 10
4. Verify the Length unit is set to feet
5. Next to Animate Using, verify "Output" is selected, and verify "seconds" are the selected Animation Time Units
6. Add a second parameter in the parameters definition area, then specify one parameter to be "Pressure Static" and the other to be "Temperature Static"
7. Change the Pressure Static units to "psig" and the Temperature Static units to "deg. F"
8. Click the Generate button
9. Click the Play button to review the graph

**Note:** If you wish to adjust the speed of the transient in the profile, you can use the Speed Slider in the upper right-hand corner of the Graph Results window to speed up or slow down the animation.

Figure 10 is useful to determine whether or not condensation occurs in the system, as well as understanding the cause of condensation. Condensation can occur by a rise in pressure, a decrease in temperature, or a combination of both. Saturation line data was added for AFT Standard fluids in AFT

## High Pressure Steam - Fluid Property Models Example

xStream 2. In this model the temperatures stay well above the saturation line at all times, showing that the steam is not close to condensing.

It should be noted that the saturation data is limited to predicting the presence of saturation conditions as opposed to modeling the behavior of the multi-phase mixture.

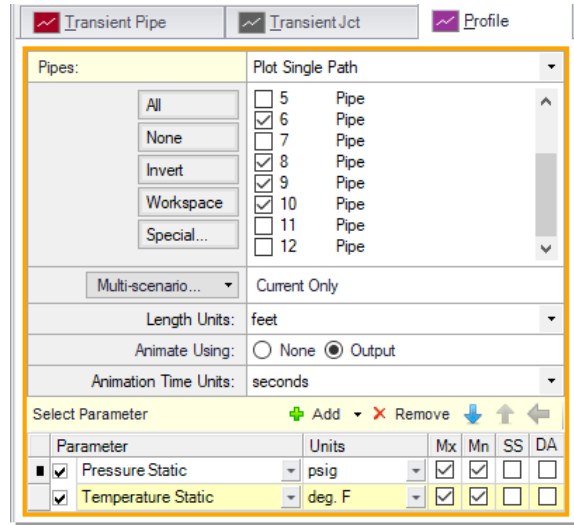


Figure 9: Graph Parameters for bypass line profile

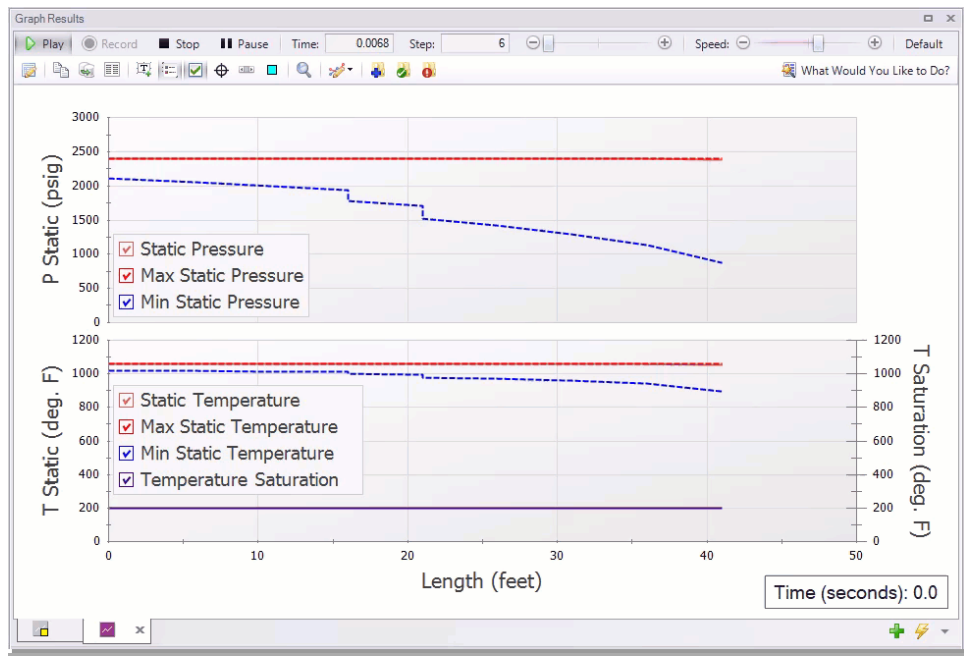


Figure 10: Bypass line animation profile

To view the animated profile, follow this link: <https://bit.ly/38Xiz5F>

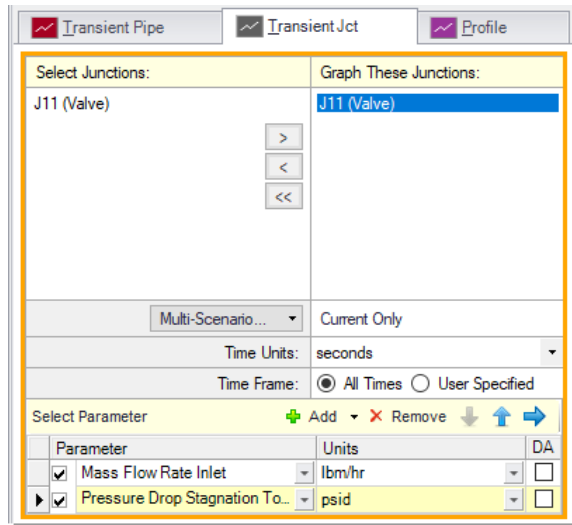
### C. Graph the mass flow rate and pressure drop through the bypass valve

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. Click the "New Tab" button to add 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. Specify the parameters as "Mass Flow Rate Inlet" and "Pressure Drop Stagnation Total".
5. Set the units for Mass Flow Rate Inlet to "lbm/hr" and Pressure Drop Stagnation Total to "psid"
6. Click Generate

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: Bypass valve mass flow and pressure drop Parameters**

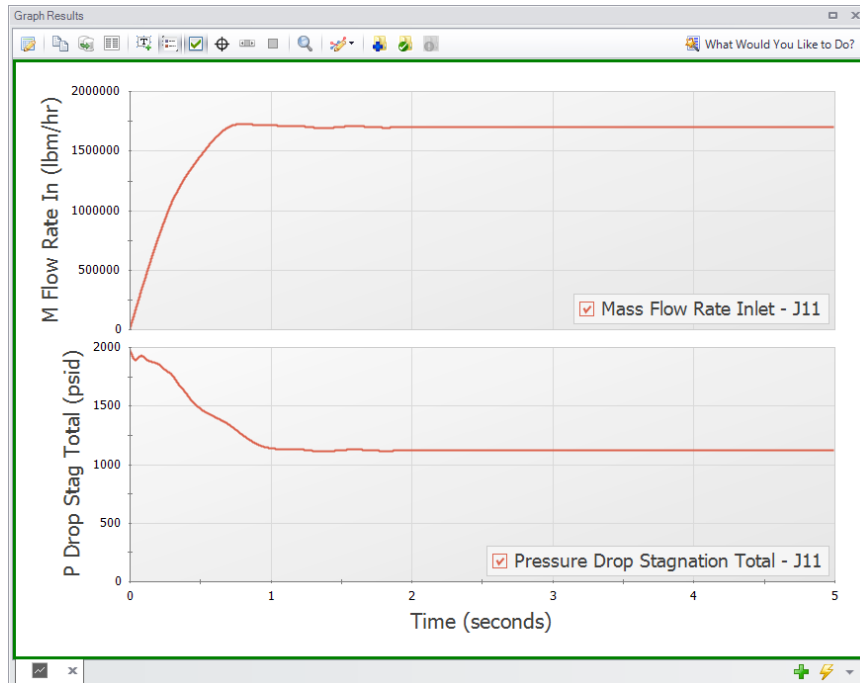


Figure 12: Bypass valve mass flow rate and pressure drop transient

## Step 6. Results of Other Libraries and Discussion

### A. Review the results of other libraries

In addition to the AFT Standard fluid library, the model built in this example was run using 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 property models 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 to the AFT Standard fluid library. ASME Steam and Chempak gave nearly identical results to NIST, and therefore are not shown.

Note that the minimum temperature reached at J11 is 56 degrees Fahrenheit higher in the AFT Standard fluid library run compared to the NIST REFPROP run.

**Note:** The axes of the below graphs were modified for clarity.

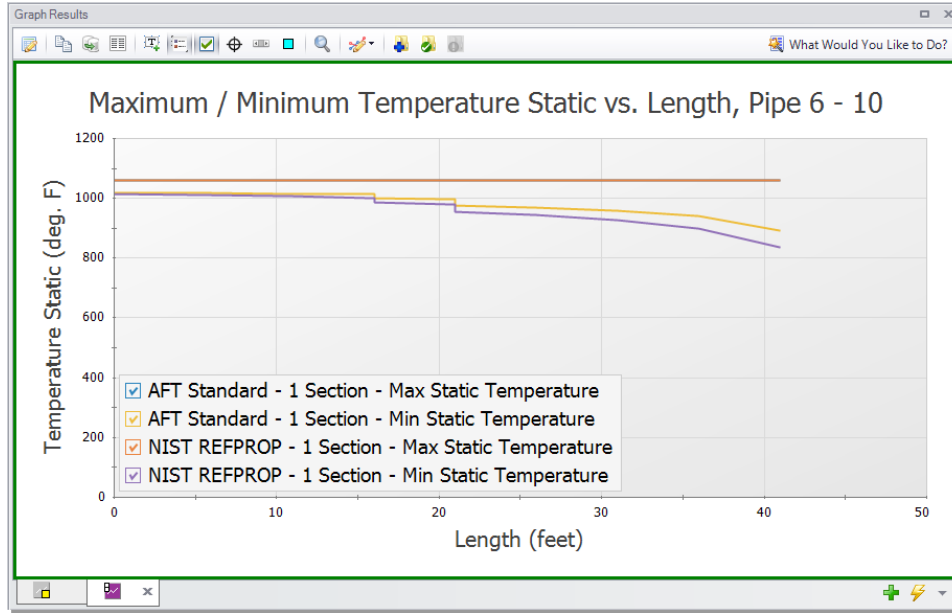


Figure 13: Bypass line profile obtained using NIST REFPROP

## 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

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. 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 property model 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 four 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 represent mixtures. However, both have slower run times than AFT Standard. Chempak runs faster than NIST REFPROP with comparable accuracy. However, Chempak is an optional add-on while the NIST REFPROP library comes standard with AFT xStream.

Lastly, ASME Steam provides a well-established, trustworthy source of steam data. ASME Steam has the longest run times of the four, 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 is unavailable, it is recommended that one of the three other libraries be used, with the added note that NIST REFPROP is the most similar in accuracy to ASME steam. Table 1 summarizes this example's run time with the four different fluid property models. 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 computers 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 High Pressure Steam System run times**

<b>High Pressure Steam System Run Times</b>	
<b>Fluid Property Model</b>	<b>Run Time</b>
AFT Standard - 1 Section	1 - 2 minutes
Chempak - 1 Section	8 - 10 minutes
NIST REFPROP - 1 Section	0.9 - 1.4 hours
AFT Standard - 10 Sections	1.6 - 2 hours
ASME Steam - 1 Section	3.9 - 4 hours



**Table 2: Compatibility chart for fluid property model selection for final runs**

Fluid Library	Model Requirements		
	Mixtures	Complex Systems*	User Specified Fluids
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.

## Conclusions

---

The mass flow rate through the bypass line was modeled and the cumulative effect of multiple transient events was observed.

# High Pressure Steam - Forces Example

## Summary

---

Two turbines in a high pressure steam system are simultaneously valved off, which causes the system to experience transient forces. This example describes how to determine the imbalanced 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 the user has already worked through the [Beginner: Tank Blowdown](#) example, or has a level of knowledge consistent with that topic.

## Model file

This example uses the following file, which is installed in the Examples folder as part of the AFT xStream installation. This file can also be downloaded from the High Pressure Steam - Forces Example topic on the Product Help Site:

- *US - High Pressure Steam - Forces.xtr*

## Step 1. Start AFT xStream

---

Open AFT xStream. Click the Start Building Model button.

## Step 2. Build the model

---

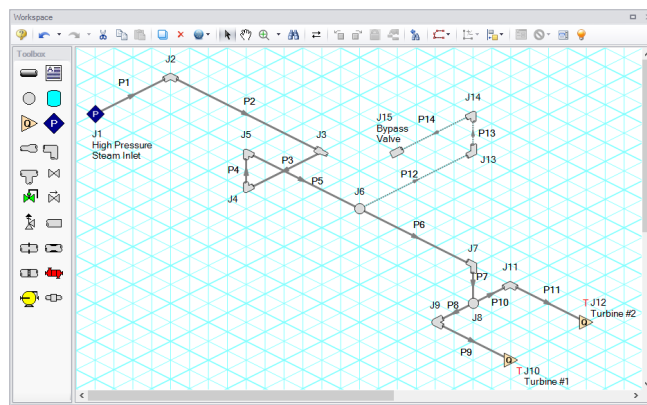
### A. Place the pipes and junctions

This example will be created in the isometric view in order to get a greater feel for the pipe layout.

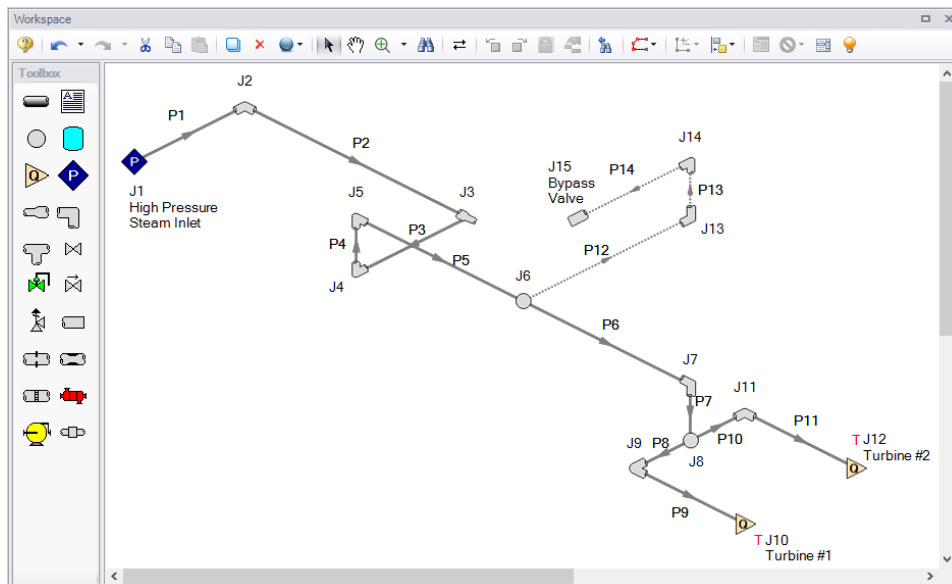
## High Pressure Steam - Forces Example

1. From the Arrange menu, select the option to Show Grid
2. Open the Arrange menu again and select the Isometric option under Pipe Drawing Mode
3. Place the junctions as shown in Figure 1
4. Click the pipe tool and draw the pipes in the model as shown
5. Use the Rotate Icons button, or right click on the junction and select Auto-rotate Icon to align the junction with the connected pipes. Alternatively, the Customize Icon option 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: Model layout with grid**



**Figure 2: Model layout without Grid**

## B. Enter the pipe 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 windows.

All pipes in the model are User Specified and utilize a User Specified Friction Model with an Absolute Roughness of 0.002 in.

<i>Pipe</i>	<i>Internal Diameter (inches)</i>	<i>Length (feet)</i>
1	13.2	13
2	13.2	40
3	10.6	13
4	10.6	13
5	10.6	33
6	10.6	43
7	10.6	6.5
8	10.6	5
9	8.1	20
10	10.6	5
11	8.1	20
12	6.5	50
13	6.5	10
14	6.5	10

## C. Enter the junction data

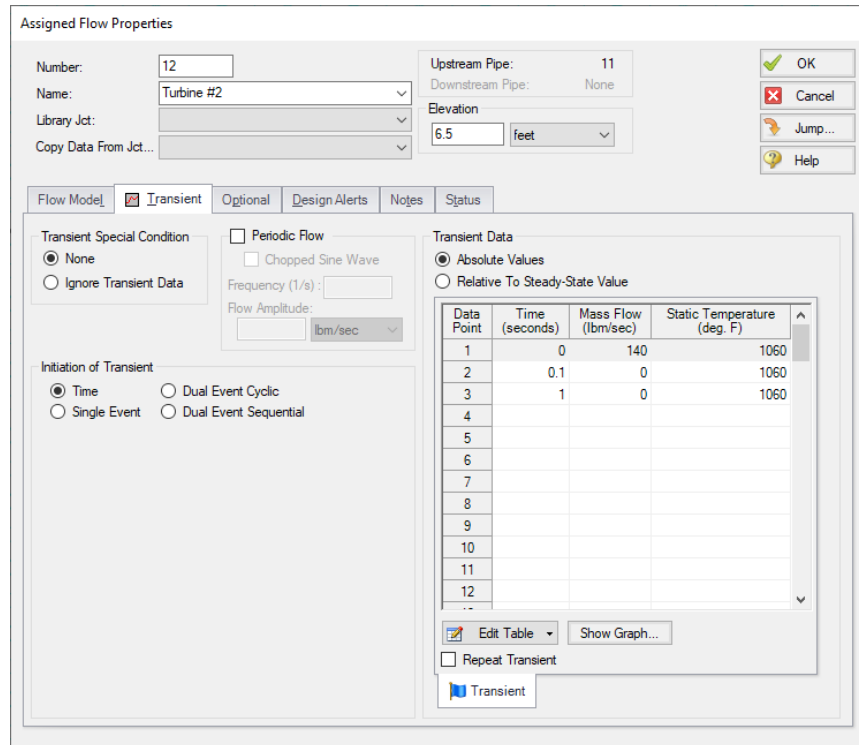
1. Assigned Pressure J1
  - a. Name = High Pressure Steam Inlet
  - b. Elevation = 0 feet
  - c. Static Pressure = 2420 psia
  - d. Static Temperature = 1060 deg.F
2. Bend J2, J4
  - a. Elevation = 0 feet
  - b. Type = Standard Elbow (knee, threaded)
3. Area Change J3

## High Pressure Steam - Forces Example

---

- a. Elevation = 0 feet
- b. Type = "Conical Transition"
- c. Angle = 45 degrees
4. Bend J5, J7, J13
  - a. Elevation = 13 feet
  - b. Type = Standard Elbow (knee, threaded)
5. Branch J6
  - a. Elevation = 13 feet
6. Branch J8
  - a. Elevation = 6.5 feet
7. Bend J9, J11
  - a. Elevation = 6.5 feet
  - b. Type = Standard Elbow (knee, threaded)
8. Assigned Flow J10, J12
  - a. J10 Name = Turbine #1
  - b. J12 Name = Turbine #2
  - c. Elevation = 6.5 feet
  - d. Mass Flow Rate = 140 lbm/sec (to each turbine)
  - e. Temperature = 1060 deg. F
  - f. On the Transient tab enter the following data for both Assigned Flow junctions:

<b><i>Time (seconds)</i></b>	<b><i>Mass Flow Rate (lbm/sec)</i></b>	<b><i>Static Tem- perature (deg. F)</i></b>
0	140	1060
0.1	0	1060
1	0	1060



**Figure 3: Transient data for Assigned Flow junctions J10 and J12**

9. Bend J14
  - a. Elevation = 23 feet
  - b. Type = Standard Elbow (knee, threaded)
10. Dead End J15
  - a. Name = Bypass Valve
  - b. Elevation = 23 feet

## D. Check if the pipes and junctions are specified

➤ **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 3. Specify Fluid

➤ **Select Fluid from the Analysis menu** to open the Fluid panel in the Analysis Setup window. For this example, select the AFT Standard fluid option, then choose "Steam" from the list and click the "Add to Model" button.

## Step 4. 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 1100 degrees F.

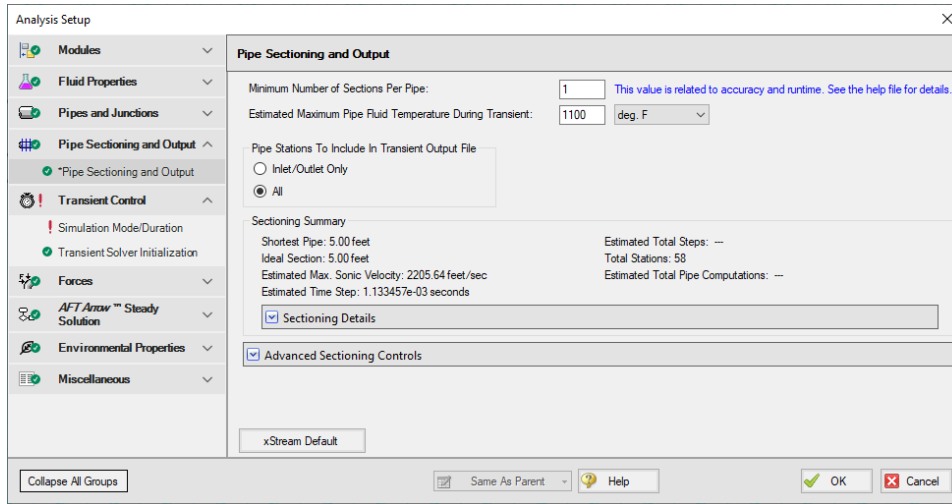


Figure 4: Sectioning panel for high pressure steam system in the Analysis Setup window

## Step 5. 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 close the Analysis Setup window.

## Step 6. Specify Force Sets

We will first add one Force Set from the Workspace.

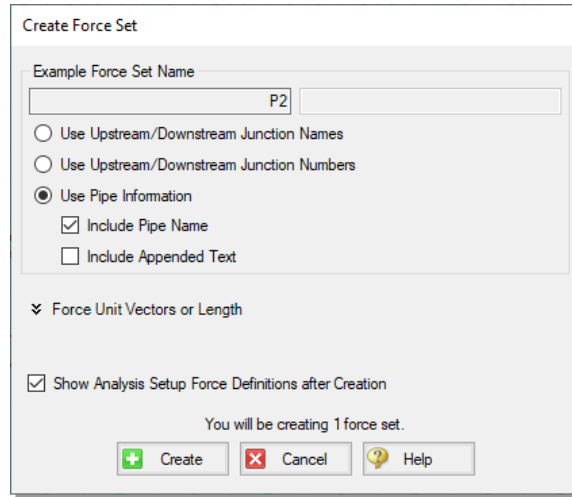
➤ **On the Workspace, right-click on pipe P2** and from the right-click menu select "Create a Force Set with Selected Pipe." Select the Use Pipe Information radio button and uncheck the "Include Appended Text" option (Figure 5). Click Create. This will open the Force Definitions panel of the Analysis Setup window. The Force Definitions panel should appear as shown in Figure 6.

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.

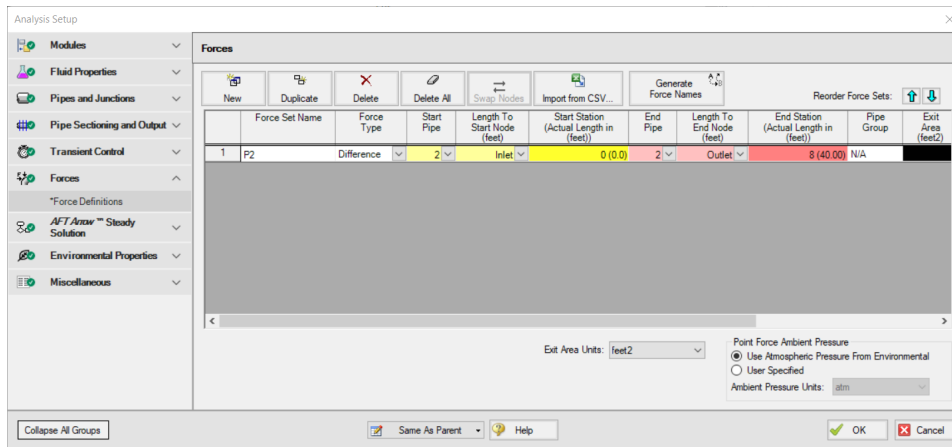
## High Pressure Steam - Forces Example

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 12

**Note:** Only single pipes can be added as a "Difference" force set type from the Workspace. Also note that difference forces should strictly be used between two 90 degree angle pipe fittings.



**Figure 5: Adding force set from Workspace using the right-click menu**



**Figure 6: High Pressure Steam System Force input 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"), entering a name, and defining the starting and ending locations of the force set. Set up the force sets as shown in Figure 7. Click OK to exit the Analysis Setup window.



## High Pressure Steam - Forces Example

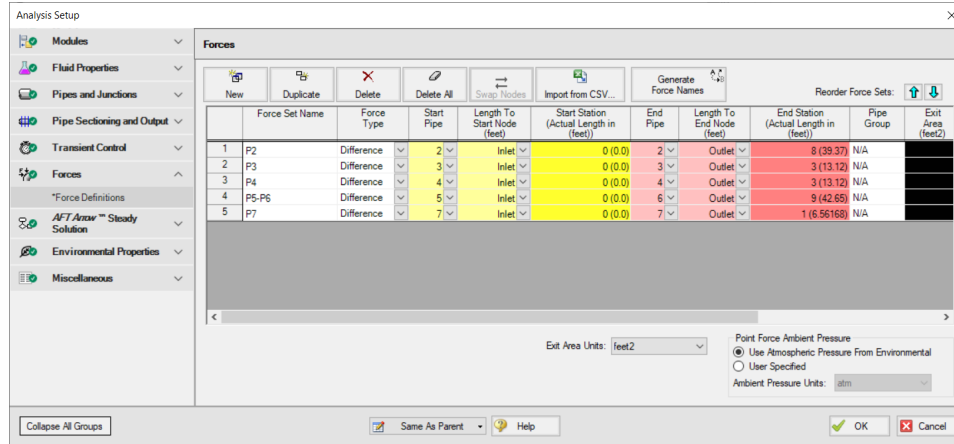


Figure 7: Completed Forces panel in the Analysis Setup window

## Step 7. Add child scenarios using 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 on

the Scenario Manager on the Quick Access Panel 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 "0.1 Second Valve Closure" 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 8.

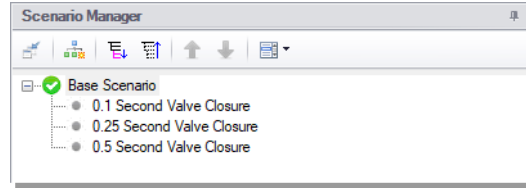


Figure 8: Completed Scenario Manager for High Pressure Steam - Forces

## 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. We will set up the remaining scenarios using the following steps:

1. Open 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:

<i>Time (seconds)</i>	<i>Mass Flow(lbm/sec)</i>	<i>Static Temperature (deg. F)</i>
0	140	1060
0.25	0	1060
1	0	1060

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:

<i>Time (seconds)</i>	<i>Mass Flow(lbm/sec)</i>	<i>Static Temperature (deg. F)</i>
0	140	1060
0.5	0	1060
1	0	1060

6. Enter the same Transient Data for Assigned Flow J12

## Step 8. Compare child scenarios using the Scenario Comparison Tool

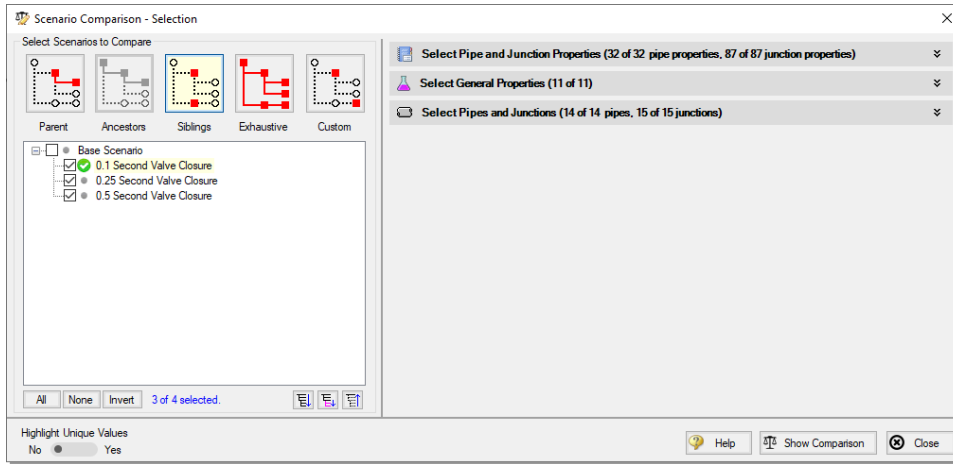
The three scenarios created should be identical except for the different transient data. To confirm the scenarios are otherwise identical, we will use the Scenario Comparison Tool.

1. Load the "0.1 Second Valve Closure" scenario from the Scenario Manager
2. In the Tools menu, select the Scenario Comparison Tool

## High Pressure Steam - Forces Example

3. In the Scenario Comparison - Selection window (Figure 9), select "Siblings"
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 10). 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 9: 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 10: Scenario Comparison Grid of the 3 scenarios**

## Step 9. Run the first scenario

➤ **Double-click the "0.1 Second Valve Closure" child scenario in the Scenario Manager.** Ensure that all force sets are set to be applied in the Force Definitions panel of the Analysis Setup window as shown in Figure 7. Select "Run Model" from the Analysis menu. This will open the Solution Progress window. This model has an estimated run time of 1 minute, but the run time is dependent on the speed of your computer.

## Step 10. 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 (lbf)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	6,940.3	0.15642	P2	0	P2	8
2	P3	2,134.2	0.14962	P3	0	P3	3
3	P4	1,958.1	0.14622	P4	0	P4	3
4	P5-P6	7,051.6	0.10881	P5	0	P6	9
5	P7	635.3	0.07708	P7	0	P7	1

*Figure 11: Forces tab in the Output window for the "0.1 Second Valve Closure" scenario*

### B. Graph the force results

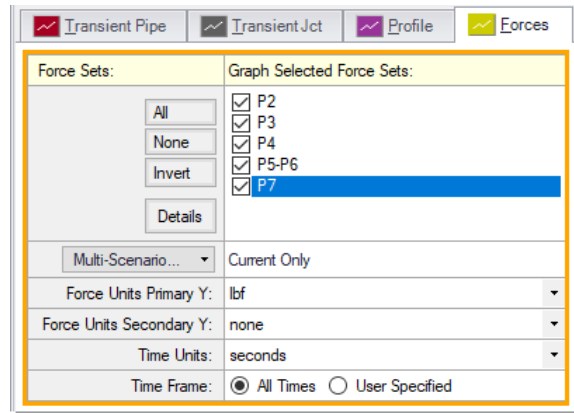
➤ **Open the Graph Results window.** We will create a graph that shows the transient forces over time for the force sets. To do so, perform the following steps:

1. Select the Forces tab on the Graph Control tab on the Quick Access Panel, as shown in Figure 12.
2. Select "All" force sets to be graphed.
3. Verify the Force Units Primary Y is set to "lbf."
4. Verify that Time Units is set to "seconds."
5. Verify that the Time Frame is set to "All times."
6. Click Generate.

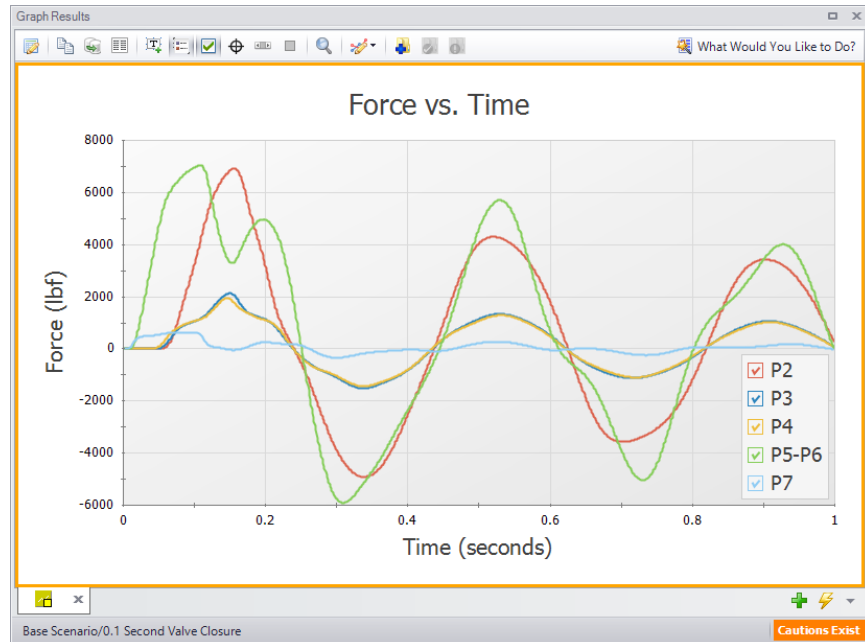
This should generate the graph shown in Figure 13.

Add the graph to the Graph List Manager by right-clicking the graph results window and selecting "Add Graph To List...". Name the graph "Force Sets" and click OK.

## High Pressure Steam - Forces Example



**Figure 12: Graph parameters for force sets**



**Figure 13: Graph for forces of the "0.1 Second Valve Closure" scenario**

Note that at  $t = 0$  seconds, there are no force imbalances 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:

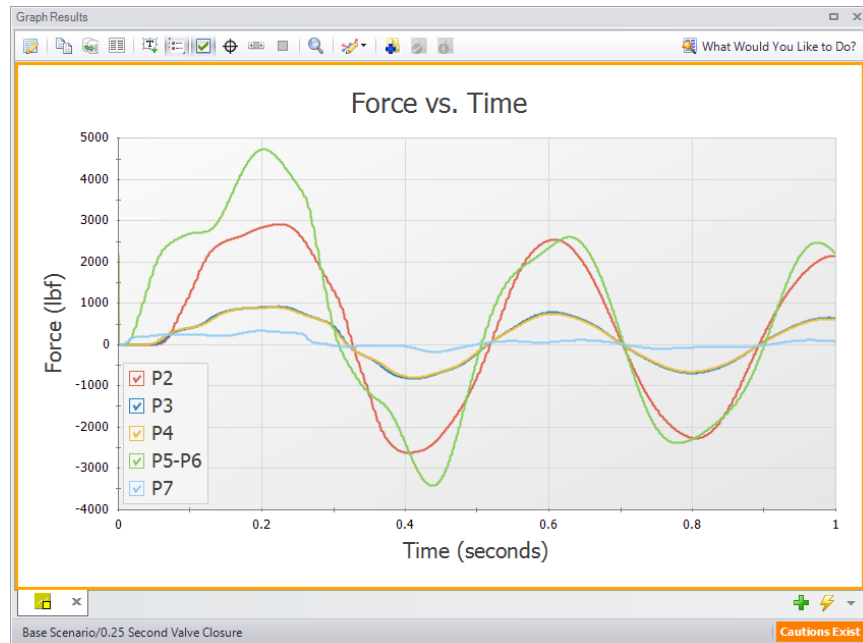
1. AFT xStream calculates transient fluid forces. This does not include piping, component, or fluid weight, or any other forces external to the piping. A comprehensive analysis of pipe loading must separately include these items.
2. Ignoring friction and momentum force balance components will result in force imbalances that do not exist.

### Step 11. Run the other child scenarios

➤ Run the other two child scenarios and load the Forces tab and the "Force Sets" graph list item for each. The results are seen below in Figures 14-17.

	Set Name	Maximum Absolute (lbf)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	2,920.6	0.2267	P2	0	P2	8
2	P3	920.2	0.2233	P3	0	P3	3
3	P4	900.3	0.2176	P4	0	P4	3
4	P5-P6	4,733.4	0.2029	P5	0	P6	9
5	P7	331.4	0.1984	P7	0	P7	1

**Figure 14: Forces tab in the Output window for "0.25 Second Valve Closure" scenario**



**Figure 15: Graph for forces of the "0.25 Second Valve Closure" scenario**

## High Pressure Steam - Forces Example

	Set Name	Maximum Absolute (lbf)	Max Abs Time (seconds)	Start Pipe ID	Start Pipe Station	End Pipe ID	End Pipe Station
1	P2	1,360.0	0.2210	P2	0	P2	8
2	P3	429.1	0.2199	P3	0	P3	3
3	P4	422.7	0.2131	P4	0	P4	3
4	P5-P6	2,272.6	0.2006	P5	0	P6	9
5	P7	162.2	0.1972	P7	0	P7	1

Figure 16: Forces tab in the Output window for the "0.5 Second Valve Closure" scenario

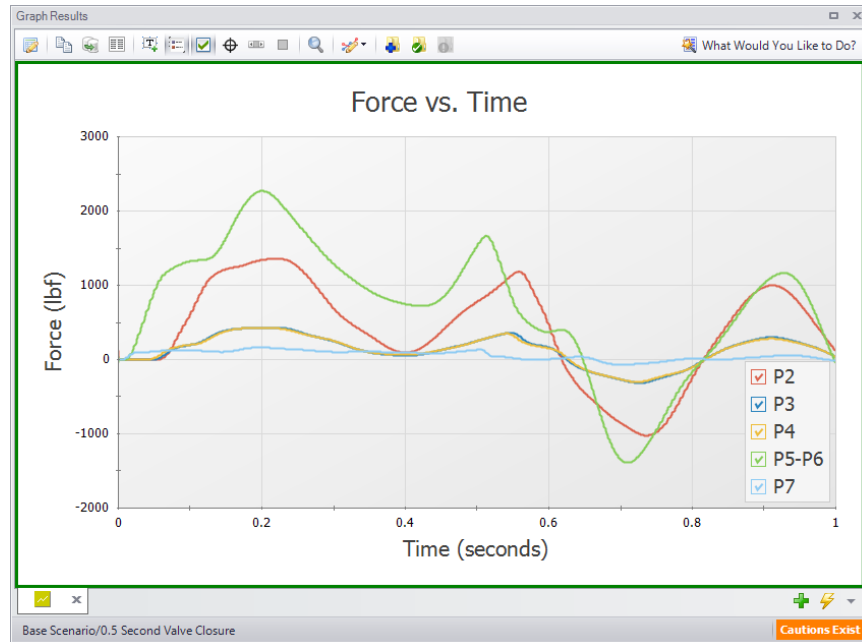


Figure 17: Graph for forces of the "0.5 Second Valve Closure" scenario

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

Pipe sectioning, the selection of fluid property model, and specification of Maximum Pipe Mach Number During Transient can have a significant effect on the accuracy of the forces calculated by AFT xStream. This effect was not significant in the system built within this example. However, the results for altering each of these three parameters are shown in Figures 18-20 so that you may see the impact of changing these three parameters. Note that in Figure 19, ASME Steam, Chempak, and NIST REFPROP 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, the click [here](#) to go to the Transient Sensitivity Analysis Tutorial.

## High Pressure Steam - Forces Example

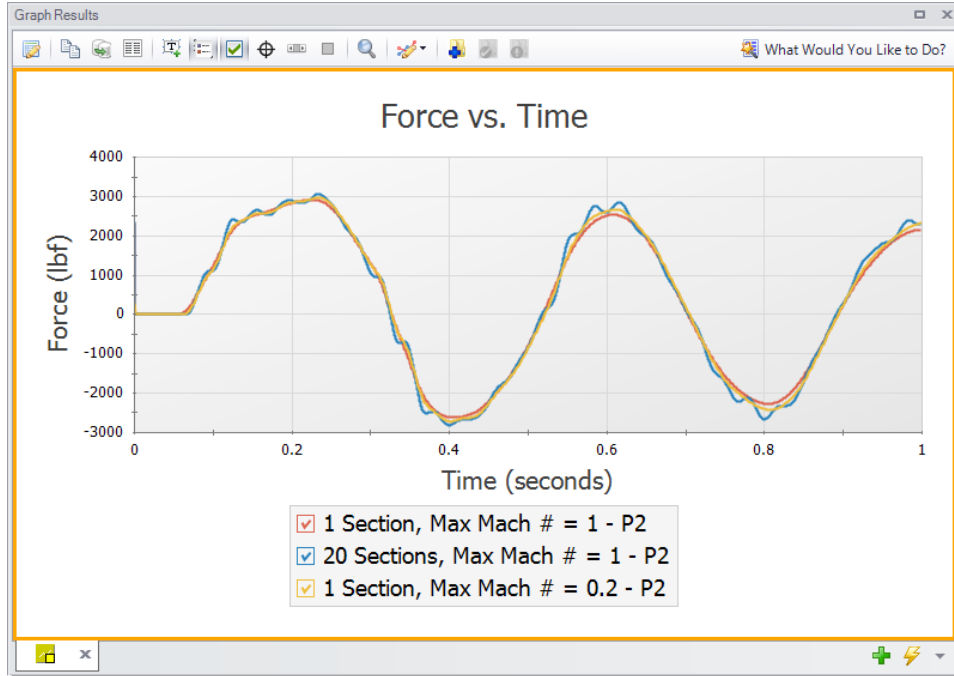


**Figure 18: Effect of sectioning on High Pressure Steam - Forces model. All runs use the AFT Standard fluid library and a Maximum Pipe Mach Number During Transient of 1.**



**Figure 19: Effect of fluid property model on High Pressure Steam - Forces model. All runs use a Minimum Number of Sections Per Pipe of 1 and an Estimated Maximum Pipe Mach Number During Transient of 1.**

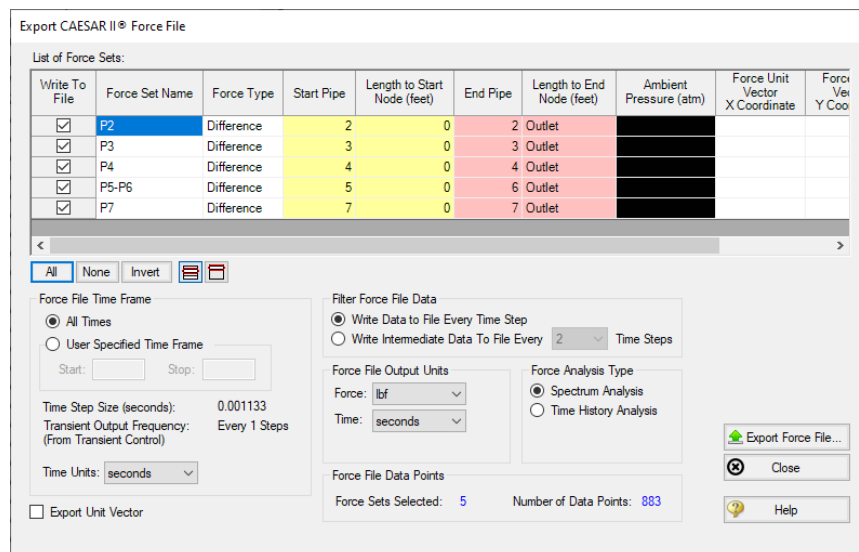




**Figure 20: Effect of Maximum Pipe Mach Number During Transient on High Pressure Steam - Forces model. All runs use the AFT Standard fluid library.**

## Step 12. Export 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 21.



**Figure 21: 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 setting. 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**

---

We created a force model to calculate the forces experienced as a result of two simultaneous closures in a steam header. We also identified how the length of the transient can have a significant impact on the forces a system experiences. 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

---

A frequency analysis is performed on a system with a positive displacement compressor to determine the acoustical frequencies that need to be evaluated for the existing piping configuration.

## Topics covered

- Using the PFA module to identify natural acoustic frequencies
- Defining valid sectioning and simulation duration settings for pulsation analysis
- Checking API-618 compliance

## Required knowledge

This example assumes the user has already worked through the [Beginner: Tank Blowdown](#) example, or has a level of knowledge consistent with that topic.

## Model file

The file listed below contains a completed version of this example and can be located in the Examples folder as part of the AFT xStream installation. This file can also be downloaded from the AFT xStream help site from the Frequency Analysis - PFA Example page:

- *US - Frequency Analysis PFA.xtr*

## Step 1. Start AFT xStream

---

Open AFT xStream. Click the Start Building Model button.

From the Startup window under the "Activate Modules" section in the center panel, select the PFA (Pulsation Frequency Analysis) option to activate the PFA module. If AFT xStream is already running, choose Activate Modules from the Tools menu, then check the box next to PFA to activate the module on the Modules panel.

## 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.

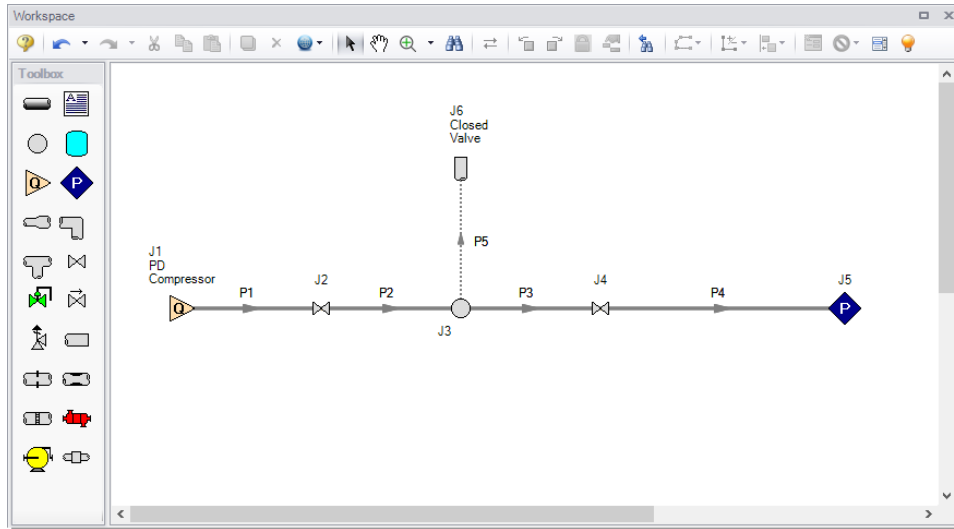


Figure 1: Layout of frequency analysis model

### 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 (feet)	Size (inches)
1	25	10
2	25	10
3	25	10
4	250	16
5	10	6

1. Assigned Flow J1
  - a. Elevation = 12 feet
  - b. Name = PD Compressor
  - c. Mass Flow Rate = 100 lbm/s
  - d. Temperature = 100 deg. F
2. Valve J2
  - a. Elevation = 12 feet
  - b.  $C_v = 446$
  - c.  $X_t = 0.8$
3. Branch J3
  - a. Elevation = 12 feet
4. Valve J4
  - a. Elevation = 0 feet
  - b.  $C_v = 1570$
  - c.  $X_t = 0.6$
5. Assigned Pressure J5
  - a. Elevation = 0 feet
  - b. Stagnation Pressure = 0 psig
  - c. Temperature = 100 deg F
6. Dead End J6
  - a. Elevation = 12 feet
  - 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. Specify the Fluid panel**

➤ **Open the Analysis Setup window** and select the Fluid panel under the Fluid Properties group. Under the "AFT Standard" fluid option, choose "Methane" from the list and click the "Add to Model" button. Click OK.

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

➤ **Open the Pipe Sectioning and Output panel** in the Pipe Sectioning and Output group. Define the Minimum Number of Sections per Pipe as 8 sections and the Estimated Maximum Pipe Temperature During Transient as 150 deg. F.

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

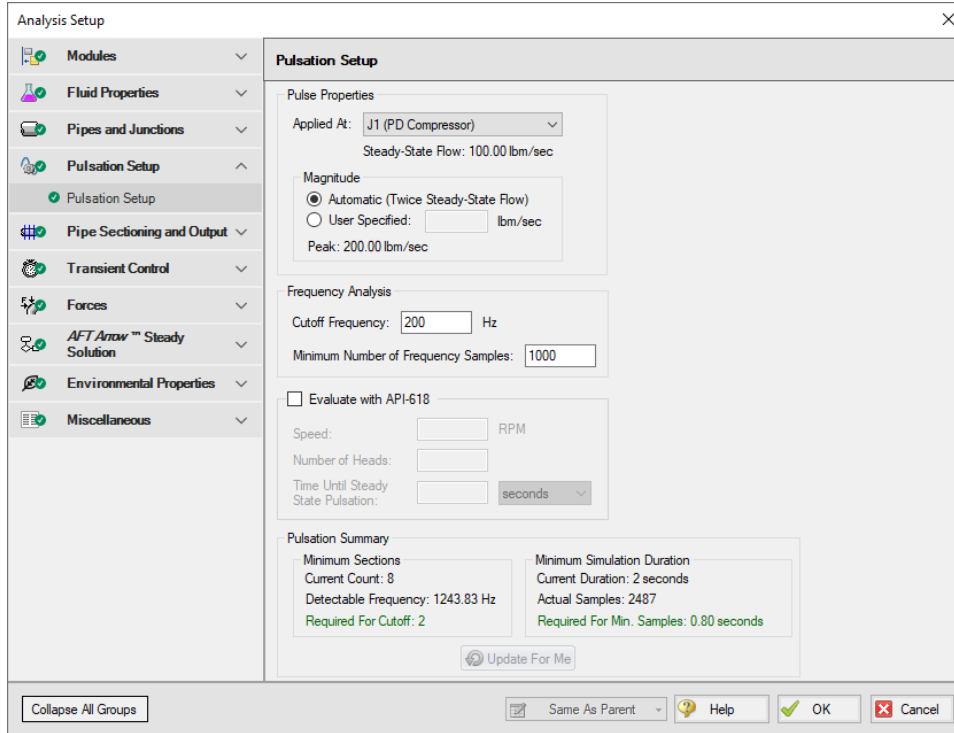
➤ **Open the Simulation Mode/Duration panel** in the Transient Control group. Enter the Stop Time as 2 seconds.

## **D. Specify the 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. Check that junction J1 is selected for the pulse to be applied at. Note that the pulse may only be applied at Assigned Flow or Compressor junctions.
2. For Magnitude, ensure that Automatic (Twice Steady-State Flow) is selected.
3. 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.
4. Leave the Evaluate with API-618 option unchecked. This option will be used in a later step once the acoustic frequencies have been determined.



**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 0.80 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 exit the Analysis Setup window.

#### Step 4. Run the model

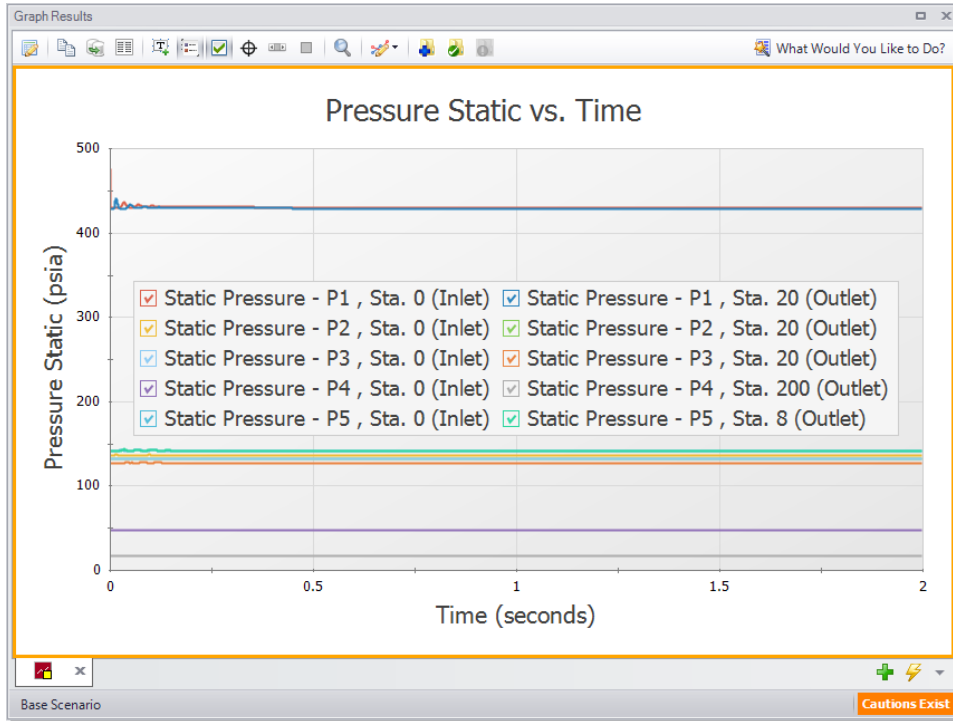
➤ **Select "Run Model"** from the Main Toolbar. This will open the Solution Progress window. This model has an estimated run time of 5 minutes, but the run time is dependent on the speed of your PC.

#### Step 5. Review the frequency results

➤ **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:

1. Select the Transient Pipe tab in the Graph Control tab in the Quick Access Panel
2. Add the inlet and outlet station for each of the pipes
3. Under Select Parameter choose "Pressure Static" and verify "psia" are the selected units
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: Pressure Static in each of the pipes for the frequency analysis**

Next, graph the acoustic frequency response as follows:

1. Click the "New Tab" button
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 psia
5. Click Generate



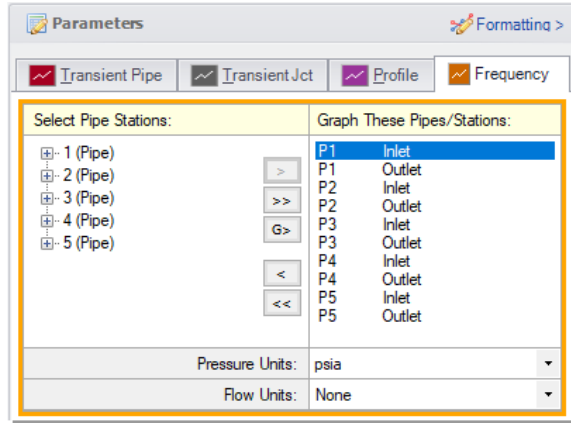
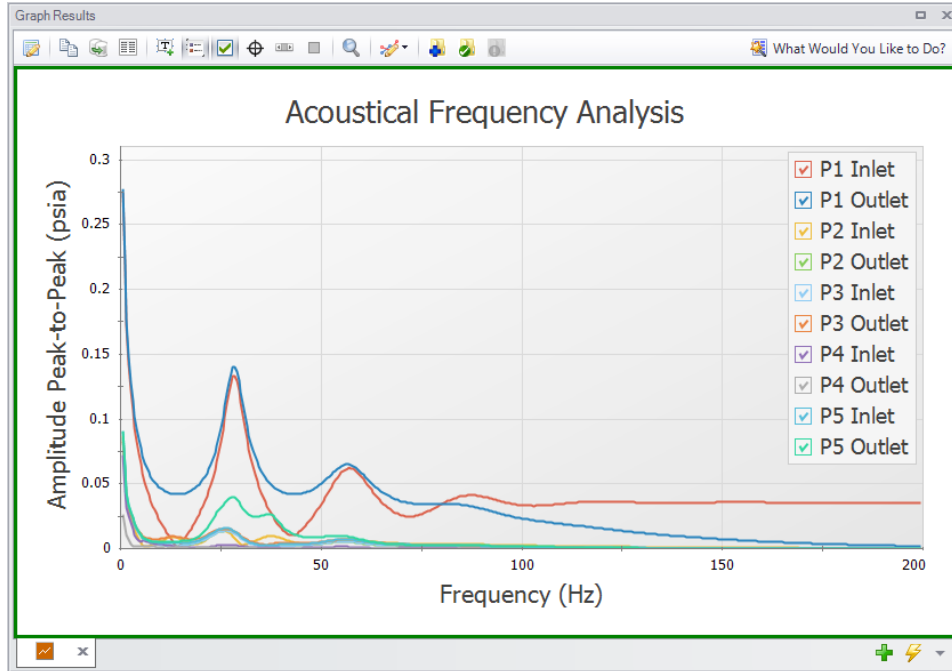


Figure 4: Graph Parameters section for the frequency analysis

**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. 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 57 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 59 Hz for the peak at 57 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 6. Determine the pressure response for API-618

The identified excitation frequencies can now be used to determine the pressure response in the system caused by the pulsation at the compressor. The pressure response can then be evaluated for compliance with API-618.

To determine the pressure response of the excitation frequencies, complete the following steps:

1. In the Scenario Manager, right-click on the base scenario and choose "Create Child". Name the child scenario "28 Hz".
2. Open the Analysis Setup window and go to the Pulse Setup panel.
3. Check the box to "Evaluate with API-618 and enter the Speed as 750 rpm and the Number of Heads as 2.
4. Enter the Time Until Steady State Pulsation as 0.25 seconds. The method to find this value is discussed later in the example. Click OK to close Analysis Setup.
5. Open the Assigned Flow Properties window for junction J1.
6. In the Transient tab, clear the pulse transient data by clicking Edit Table under the transient table and choosing Clear All Data.
7. Check the box next to Periodic Flow. Define the Frequency as 28 1/s and the Flow Amplitude as 10 lbm/sec (Figure 6).
8. Click OK to close the Properties window.

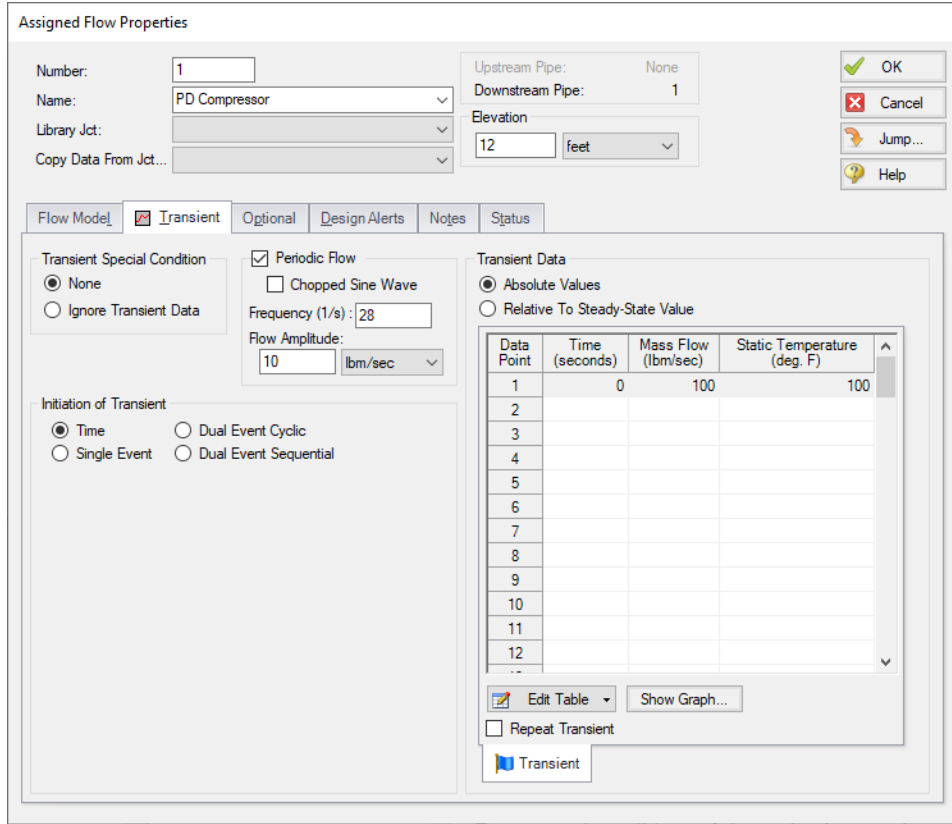


Figure 6: Assigned Flow Properties window with Periodic Flow transient applied

The model is now updated to represent the pulsation at the positive displacement compressor that will excite the system. Run the model, then navigate to the Output window. This scenario will run faster than the base scenario (about 2 minutes).

In the Output window there is now a warning stating that the maximum pressure limit for API-618 was exceeded as is shown in Figure 7. 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 8).

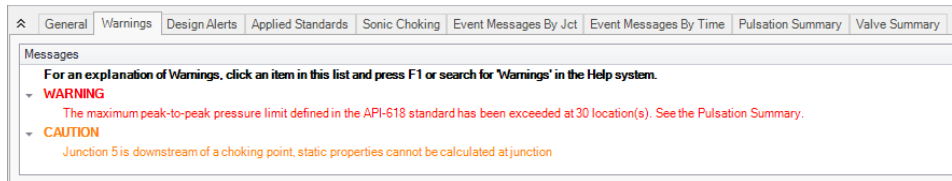
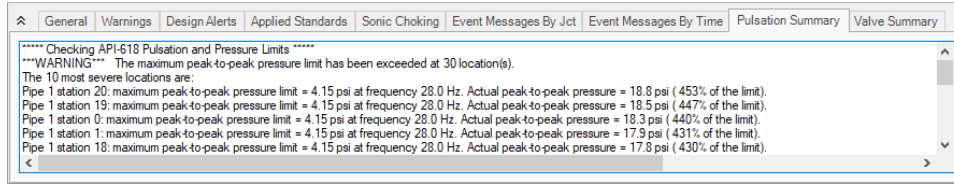


Figure 7: Warning showing that the API-618 pressure limits have been exceeded at multiple locations



**Figure 8: Pulsation Summary tab listing the pipes with the largest pressure violations**

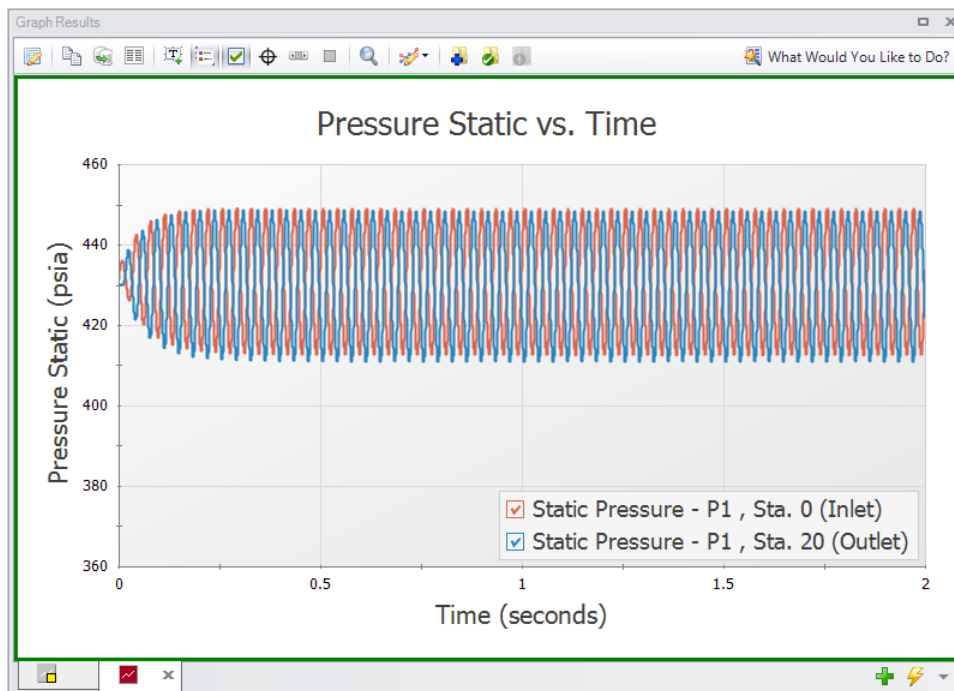
In the Graph Results window, generate a graph of the static pressure following the same process used to create Figure 3. However, only include the pipe P1 inlet and outlet station. The resulting graph is shown in Figure 9. These pressure results can be used to check for compliance with standards such as API-618.

The pressure results for the compressor outlet (Pipe P1 inlet) can also be used to determine the Time Until Steady State Pulsation, which is one of the inputs needed to accurately evaluate API-618. The Time Until Steady State Pulsation is the time at which the pressure oscillations have a consistent amplitude. In Figure 9 below the wave heights appear to become consistent at about 0.25 seconds.

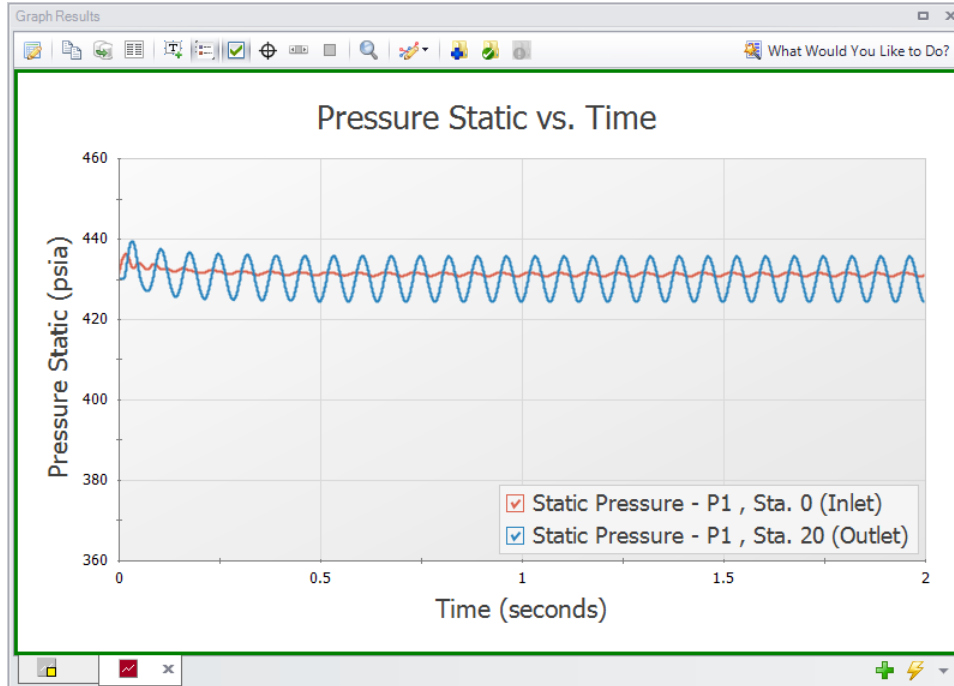
Note for this example the value of 0.25 seconds was given in advance to expedite the process, but normally the user would need to run the model once with the "Time Until Steady..." set to 0, then determine the time input and re-run the model with the correct value.

For comparison, this model can also be run at one of the frequencies which gave a lower amplitude, such as 14 Hz. At 14 Hz, a lower amplitude pressure response is produced, as can be seen in Figure 10.

**Note:** The pulsation observed in this model represents a pure sine wave. Real systems might have a different forcing function.



**Figure 9: Pressure Static response at 29 Hz frequency**



**Figure 10: Pressure Static response at 14 Hz frequency**

## 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, and what mitigation would be effective.

## Index

---

### A

AFT Arrow Steady Solver 27, 71

AFT Standard library See Fluid libraries, AFT Standard library

Analysis Setup window 22, 48, 50

Animation

    Graph 67

    Speed adjustment 67

Annotation Tool See Toolbox, Annotation tool

API 618 91, 100

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

### C

Chempak library See Fluid libraries, Chempak library

Condensation 67, 71

Copy Data From Jct... 14

### D

Defining force sets See Forces, Force sets, Definition

Defining objects

    List Undefined Objects window 12, 48, 63, 78, 93

### E

Edit menu 7

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, 48, 63, 71, 93

ASME Steam Tables library 23, 72

Chempak library 23, 72

Effect of 70, 87

NIST REFPROP library 23, 72

Selection 71, 73

Fluid panel 23, 48, 63, 93

Forces

Exporting force files 89

Force sets 79

    Definition 79-80

    Direction 79

    Force Unit Vectors 80

Methodology 85

Forces panel 90

**G**

Graph Animations See Animation, Graph

Graph Control tab

    Forces 84

    Profile tab 54, 67

    Transient Jet tab 69

    Transient Pipe tab 52, 64

Graph List Manager 53, 56, 84

Graph Results window 31  
    Modifying axes of graphs 64

**H**

Highlight feature 14

**I**

ID numbers 6, 12

Inspection feature 15

Isometric See Pipe Drawing Mode, Isometric

**J**

Jump button 20

Junction library 14

Junctions 6

    Tank 6, 13

**L**

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

**M**

Method of Characteristics 24, 51

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

Mixtures 23, 71-72

Model building process 71

Model Data window 21

**N**

NIST REFPROP library See Fluid libraries, NIST REFPROP library



**O**

- Objects 6
- Open Pipe/Jct Window 13
- Output Control window 28
- Output file See Transient output file
- Output window 27-28, 51
  - Forces tab 84, 86
  - Transient Max/Min tab 52

**P**

- PFA Module
  - Excitation frequencies 97-98
- Pipe Drawing Mode
  - 2D Freeform 59
  - 2D Orthogonal 59
  - Isometric 59, 75
    - Adding segments 59
- Pipe Drawing Tool See Toolbox, Pipe Drawing Tool
- Pipe Properties window 19
- Pipe Sectioning panel 24, 48, 52, 63, 79, 94
  - Estimated Maximum Pipe Mach Number During Transient
    - Effect of 87
  - Estimated Maximum Temperature During Transient 25, 48, 63, 79, 94
    - Effect of 51
  - Minimum Number of Sections Per Pipe 25, 48, 63, 79, 94
    - Effect of 39, 87

Sectioning Summary 25

Properties windows

Notes tab 14

Optional tab 14

Status tab 14, 20

Transient tab

Specification 82, 98

Pulsation panel 94

Cutoff Frequency 94

Minimum Number of Frequency Samples 94

Pulsation Summary 95

## Q

Quick Access Panel 5

Graph List Manager 53

## R

Rotating Icons 59, 75

Run Model 26, 51-52, 64, 84, 95, 99

## S

Scenario Comparison Tool 82-83

Scenario Manager 52, 56, 81, 98

Selecting a fluid library See Fluid libraries, Selection

Show Grid 59, 75

Show Object Status 12, 14

Simulation Mode/Duration panel 24, 26, 49, 64, 79, 94

Solution Progress window 26

Sonic choking 33, 57

Status Bar 22

## T

Tank junction See Junctions, Tank

Toolbars 22

Toolbox 5-6

- Annotation tool 6

- Pipe Drawing Tool 6, 9, 11

Transient Control group 26

Transient event specification See Properties windows, Transient tab, Specification

Transient Jct tab See Graph Control tab, Transient Jct tab

Transient output file 28

Transient Pipe tab See Graph Control tab, Transient Pipe tab

Transient Solver 27, 71

## U

Undefined objects list See Defining objects, List Undefined Objects window

User Options 12

## V

Visual Report Control 36

Visual Report window 36

## W

Workspace window 5, 44