AFT xStream™ 2
Quick Start Guide
English Units

Dynamic solutions for a fluid world™

AFT
Applied Flow Technology

AFT xStream™
Quick Start Guide

English Units

AFT xStream Version 2

Gas Transient Modeling in Piping Systems

Published: May 01, 2023

Dynamic solutions for a fluid world ™

CAUTION!

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

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

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

Contents

Introducing AFT xStream

Welcome to AFT xStream™, Applied Flow Technology's powerful gas transient modeling tool. With AFT xStream you can model transients caused by a wide range of pipe system behavior. This tool will allow you to understand transient pressure extremes.

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

AFT xStream has one available add-on module which extends AFT xStream's modeling capabilities into new areas.

- The Pulsation Frequency Analysis (PFA) module allows the engineer to determine resonant frequencies that can result in excessive vibration and damage to systems.

Modeling capabilities

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

- Transients in open and closed (recirculating) systems
- Network systems that branch or loop
- Systems with valve transients
- Systems with compressor transients

- Systems that experience sonic choking, including multiple sonic choking points and transient sonic choking

- Multiple design cases in a single model file

Add-on module capabilities (optional)

- Identification of natural acoustic frequencies (PFA module)

- Evaluation of pressure results with API-618

The steady-state solver

Before a transient model can be run, the initial steady-state conditions are required. AFT xStream obtains the steady-state solution using multiple stages. First, the solver from AFT Arrow™, AFT's steady state compressible flow modeling software, is used to calculate the steady solution to initialize the second stage.

Second, the transient solver is run with user-specified transients disabled. The AFT Arrow solver will typically have slight differences from the transient solver, and the transient solver must be run to find a new steady solution to avoid introducing artificial transients into the transient analysis. This solution is run in a series of time step blocks. After each block, the velocity results from the beginning and end of the block are compared to determine if the results are sufficiently steady.

If desired, AFT xStream can be run in "AFT Arrow Steady" mode where only the steady flow pipe pneumatics are modeled. When run in "Transient" mode, the steady flow solution is used to assist with initializing the transient solution. This convenience helps the user avoid the often error-prone process of manually setting up the initial conditions, ensures the solution is steady before the transients are run, and allows the user to quickly and safely modify and rerun the model as needed.

The transient solver

AFT xStream employs the Method of Characteristics (MOC) to solve the transient equations of gas flow. The fundamental mass, momentum, and energy equations are solved at all computing stations in each pipe to

accurately represent the propagation of transient waves throughout the system.

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

Thermophysical property data

AFT xStream derives physical properties from one of four sources. The first is the standard AFT xStream set of compressible fluids which contains data for 28 common fluids (called the AFT Standard library).

The second is steam data from the ASME Steam Tables library.

The third is the NIST REFPROP library. REFPROP is licensed from the National Institute of Standards. REFPROP has a library of approximately 150 fluids and supports user specified fluid mixtures.

The fourth is the Chempak™ library. Chempak is licensed from Madison Technical Software and is offered as an optional add-on to AFT xStream. Chempak has a library of approximately 584 gasses and supports user specified fluid mixtures. AFT xStream is restricted to non-reacting mixture calculations.

Engineering assumptions in AFT xStream

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

- Superheated gas flow only (no two-phase flow)
- One-dimensional flow
- No chemical reactions
- The pipes are rigid

AFT xStream Primary Windows

AFT xStream has five windows that work in an integrated fashion. Each is located on a separate, movable tab. You work exclusively from one of

these windows at all times. For this reason, they are referred to as *Primary Windows*.

Of the five Primary Windows, two are input windows, two are output windows, and one displays both input and output information. Figure 1.1 shows the relationships between the Primary Windows.
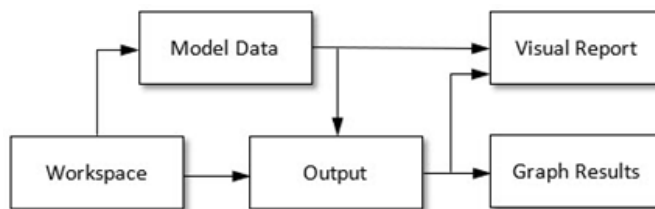
Figure 1.1   Primary Window workflow in AFT xStream

Input windows

The two windows that primarily function as input windows are the Workspace window and the Model Data window. These two windows, one graphical and the other text based, work together to process model input data with immense flexibility. The tools provided in these two windows allow you to model a large variety of pipe networks.

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

Output windows

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

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

Note: AFT xStream supports multiple monitors. You can click and drag any of the five Primary Window tabs off of the main AFT xStream window. Once you drag one of the Primary Windows off of the AFT 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 AFT xStream Primary Window tab bar, simply click the X button in the upper right of the Primary Window.

C H A P T E R   2

Tank Blowdown Example

This chapter 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.

A number of other example model discussions are included on the AFT help site. It can be opened from the Help Menu by choosing "Show Examples" and selecting "English Units....".

Topics covered

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

Required knowledge

No prior knowledge is required for this example.

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*

This example is also provided in metric units under the Metric Units file name, *Metric - Tank Blowdown.xtr*.

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

Problem statement

This problem evaluates the initial design for a tank that undergoes a blowdown. Nitrogen flows out of a supply tank into a 2 inch line as a part of the blowdown. The flow is controlled by opening and closing a valve in the 2 inch line.

This model will simulate opening a valve in 0.1 seconds. The valve will remain open for the remainder of the simulation.

Graph the pressure and temperature of the tank to observe how both parameters change with time. Also, graph the Mach number, temperature, and velocity at the discharge of the second pipe to understand how the flow properties change as the blowdown progresses.

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 2.2, appears. This window provides you with several options before you start building a model:
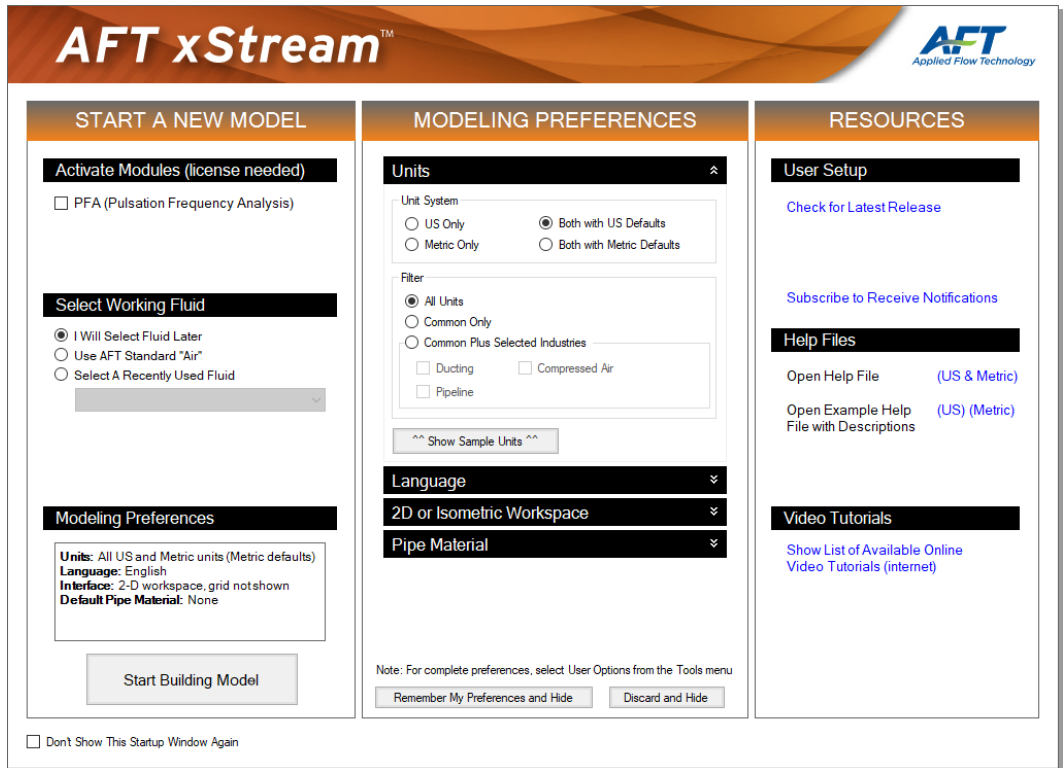
Figure 2.1   Startup window with Modeling Preferences expanded

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

- Open a previous model, browse to a model, or browse to an Example

- Activate an Add-on Module

- Select "AFT Standard "Air" " or a recently used fluid to be the Working Fluid

- Review or modify Modeling Preferences

  - Select a Unit System

  - Filter units to include Common Only or Common Plus Selected Industries

- Choose a Grid Style

- Select a Default Pipe Material

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

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

When collapsed, you can view your current Modeling Preferences at the bottom of Start a New Model. To further review or adjust your preferences, click the "Modify >>" button (Figure 2.2).
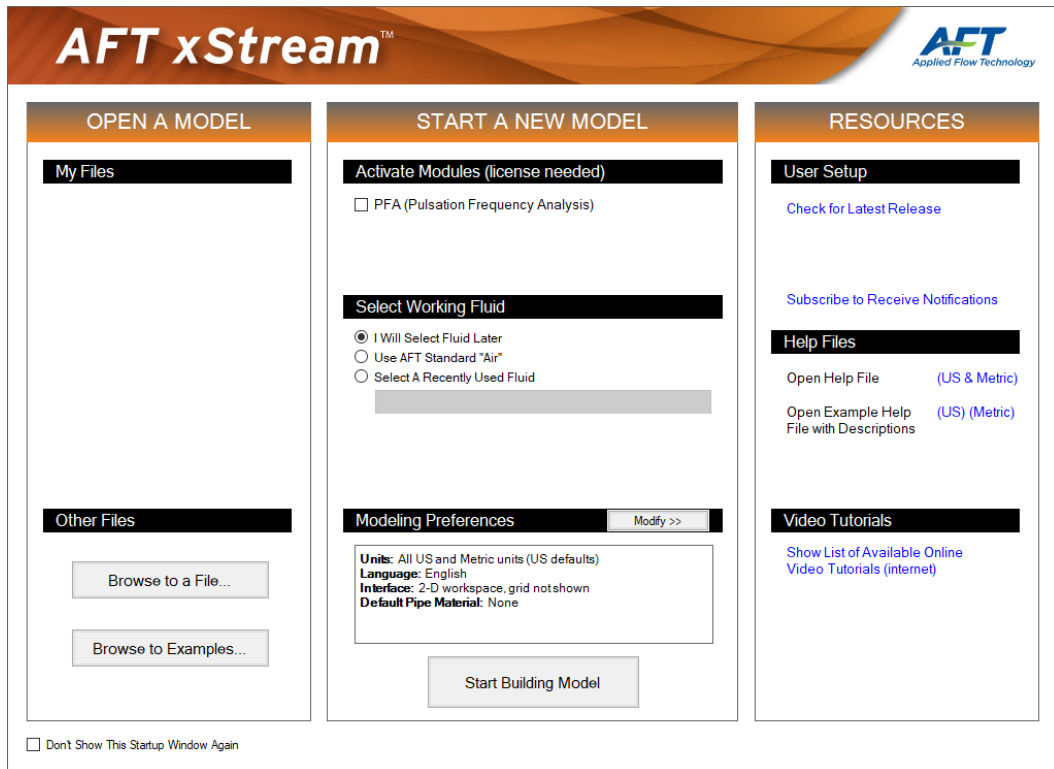
Figure 2.2   AFT xStream Startup window

With Modeling Preference expanded, as in Figure 2.1, select "Both with US Defaults" under Unit System. Select "Common Only" under Filter to show only commonly used units, instead of "All Units."

You can "Show Sample Units" to see which units will be included based on your selections, as shown in Figure 2.3.
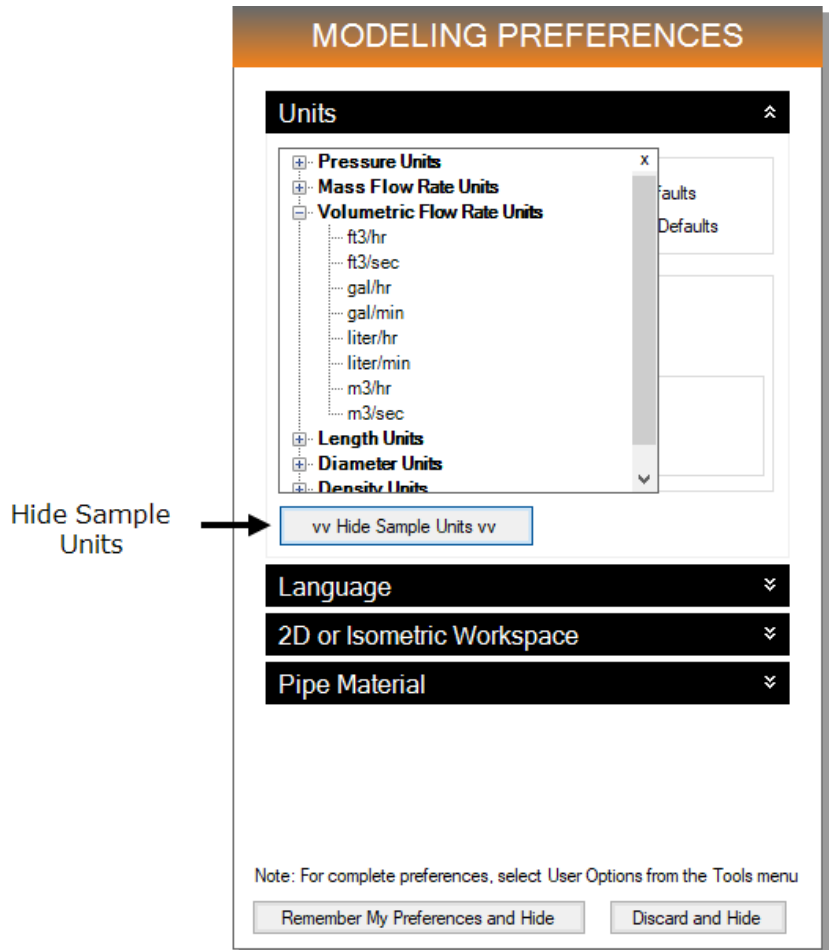
Figure 2.3   **"Show Sample Units" with "Common Only" selected**

➢ **Click "Start Building Model"**.

The Workspace window is initially the active window, as seen in Figure 2.4.

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.
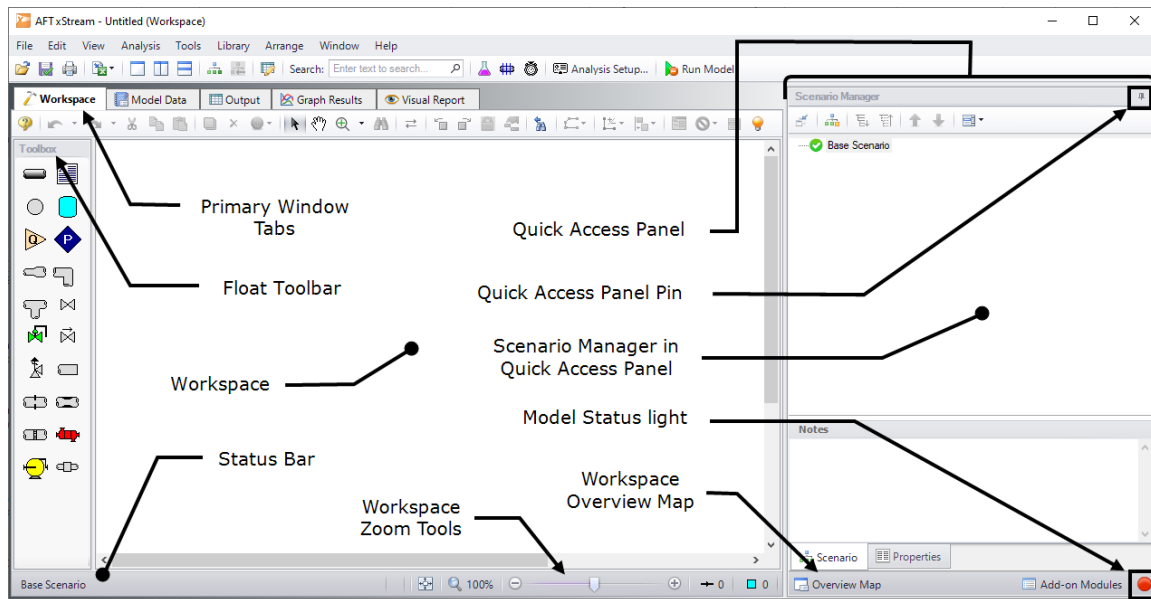
Figure 2.4   **The Workspace window is where the model is built. The Status Bar shows the model status.**

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.

Unpinning the Quick Access Panel

By default, the Quick Access Panel is pinned to the Workspace window so that it is constantly displayed. The Quick Access Panel can be unpinned so it is only displayed when the mouse is hovered over the tab displayed on the right edge of the Workspace window. You can unpin the Quick Access Panel by clicking on the pin icon displayed in the top right corner of the Quick Access Panel (Figure 2.4).

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.

A. Place the Tank junction

➢ **To start, click and drag a Tank junction from the Toolbox** and drop it on the Workspace. Figure 2.5a 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 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.
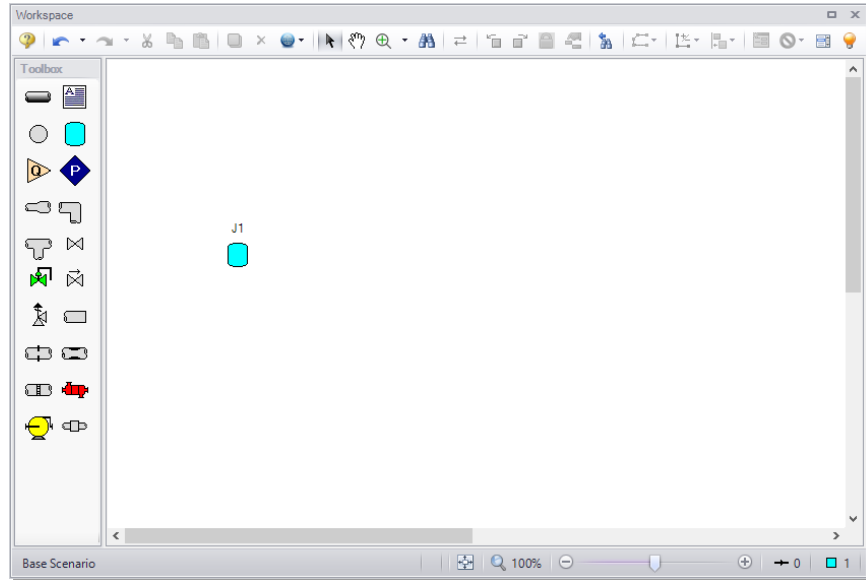
Figure 2.5a   Tank junction placed

B. Place the other junctions

Drag a Valve and an Assigned Pressure junction from the Toolbox and drop them on the Workspace so that your model appears similar to Figure 2.5b.

If the icons do not line up exactly, the calculations will not be affected. You can align the icons by using the Align feature selected from the Arrange menu.

Note: The relative location of objects in AFT xStream is not important. Distances and heights are defined through Property window 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.
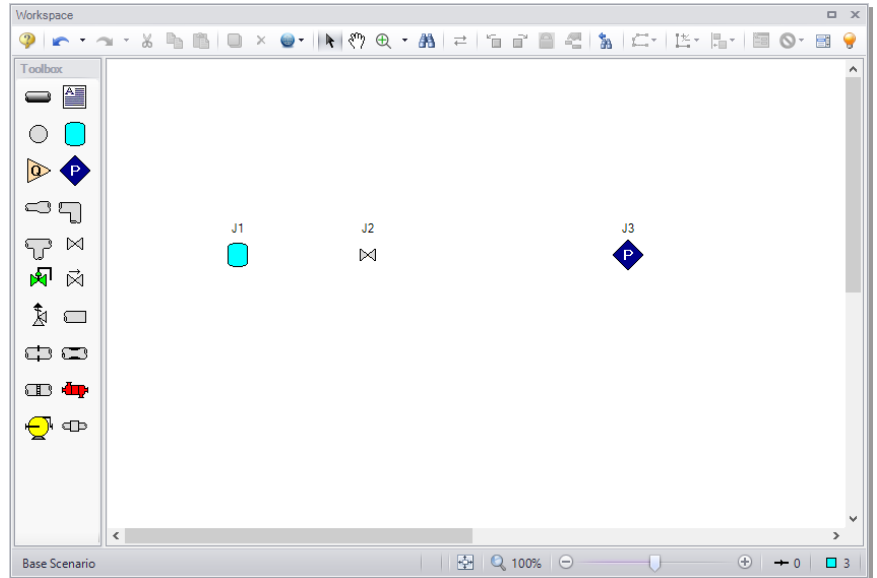
Figure 2.5b   All junctions placed

➢ **Before continuing, save the work you have done so far** with "Save As" from the File menu. 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.

C. 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 2.5c.

The pipe object on the Workspace has an ID number (P1) that is shown near the center of the pipe.
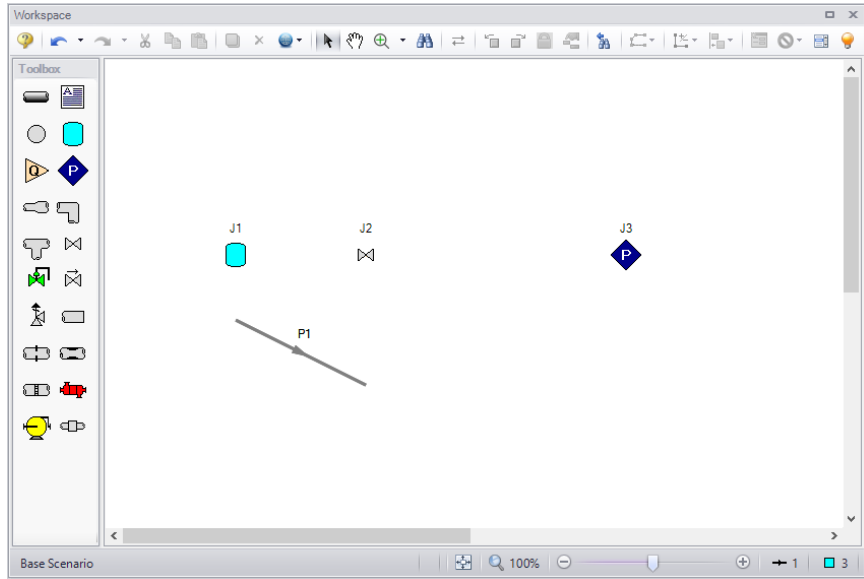
Figure 2.5c   First pipe drawn

➢ **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 2.5d). 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 2.5e).
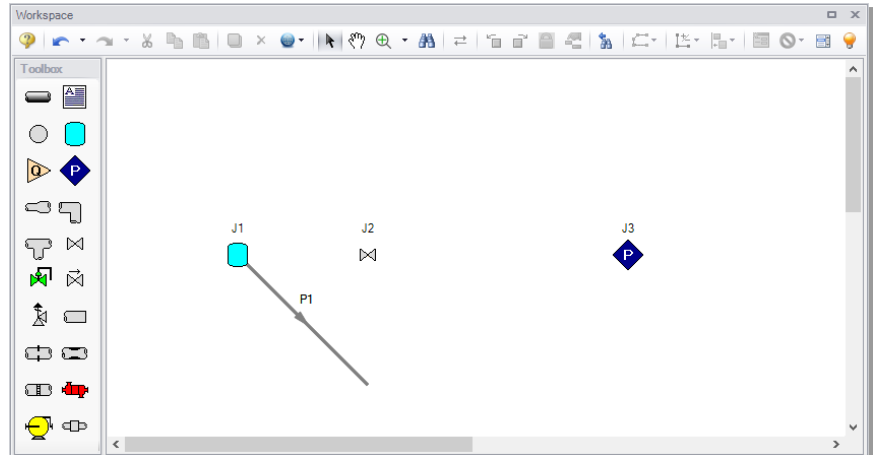
Figure 2.5d   First pipe partially connected
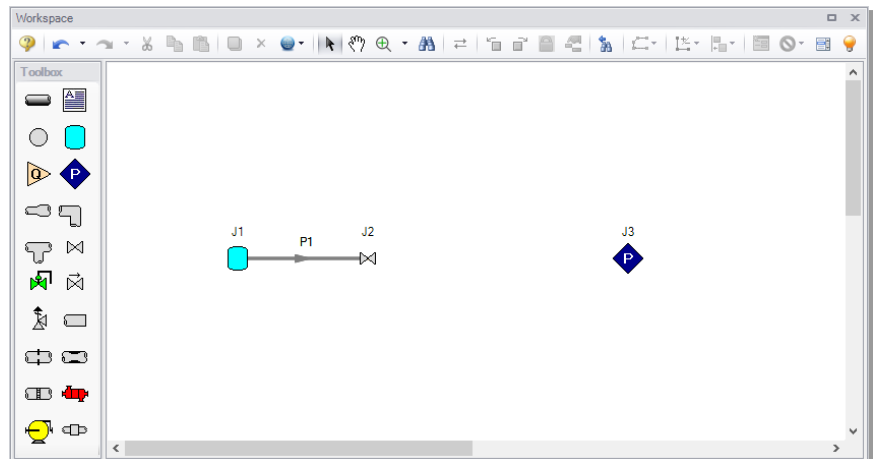
Figure 2.5e   First pipe fully connected

D. 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 2.5f.

Save the model by selecting "Save" in the File menu, by clicking the "Save" icon on the Toolbar, or by entering Ctrl+S.
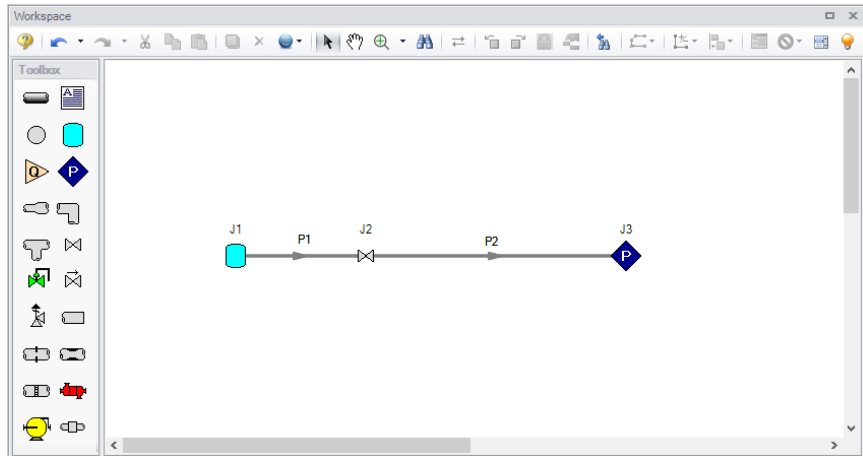
Figure 2.5f   All pipes and junctions placed

Reference positive flow direction

There is an arrow on each pipe that indicates the reference positive flow direction for the pipe. AFT 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 is used for reference purposes only and need not be the actual flow direction. If the reference positive flow direction is the opposite of that obtained by the solver, the output will show the flow rate as a *negative number*. However, when used with compressors and certain other junction types the pipes must be in the appropriate forward flow direction because that is how AFT xStream determines which side is suction and which is discharge.

Note: Some users find it desirable to lock objects to the Workspace once they have been placed. This prevents accidental movement and disruption of the connections. Locking also helps prevent accidental deletion of your output once a solution has been reached. 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.

Step 3. Define the model components

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

Object status

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

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

Because you have not yet defined the pipes and junctions in this model, all the 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 2.6). 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.
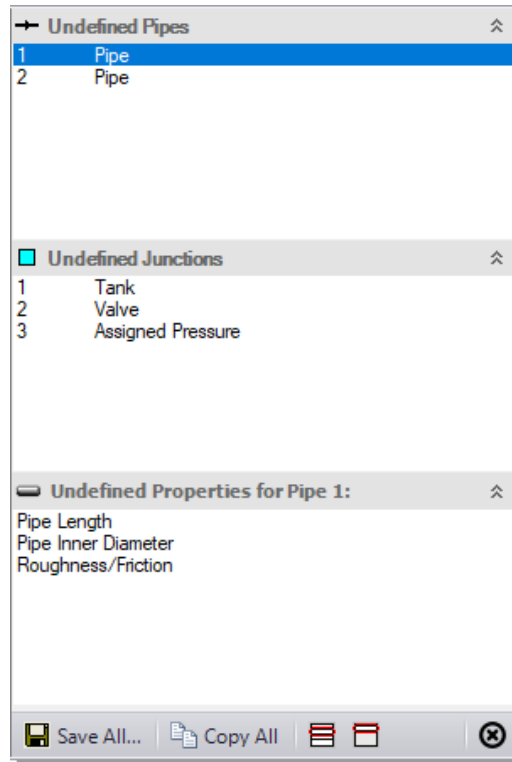
Figure 2.6   The List Undefined Objects window lets you see the undefined properties for each undefined object

A. Enter data for the Tank

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

➢ **To define the first tank**, open the J1 Tank Properties window (Figure 2.7) by double-clicking on the J1 junction

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

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

1. Enter 0 feet for Elevation

2. Set the Tank Model to "Finite"

3. Set 600 psia for Pressure

4. Set the Temperature to 300 deg. F

5. Set the Volume to 300 feet3

6. Set the Initial Conditions to "Known"

7. Set the Thermodynamic Process to "Adiabatic"

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

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 the top part of 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.

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 2.7, the name of Tank J1 is "Supply Tank." The name can be displayed on the Workspace, Visual Report, or in the Output.

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.

Figure 2.7   Properties window for Tank J1

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.

Using the tabs in the Properties windows

The information in the Properties windows is grouped into several categories and placed on separate tabs. Click the tab to bring its

information forward. Figure 2.7 is an example of a Tank's Properties window.

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

The Optional tab allows you to 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.

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

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

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

The Inspection feature

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspection feature. Position the cursor on the Tank J1 junction and hold down the right mouse button. An information box appears, as shown in Figure 2.8.

Inspecting is a faster way of examining the input (and output, if output results are available) than opening the Properties window.
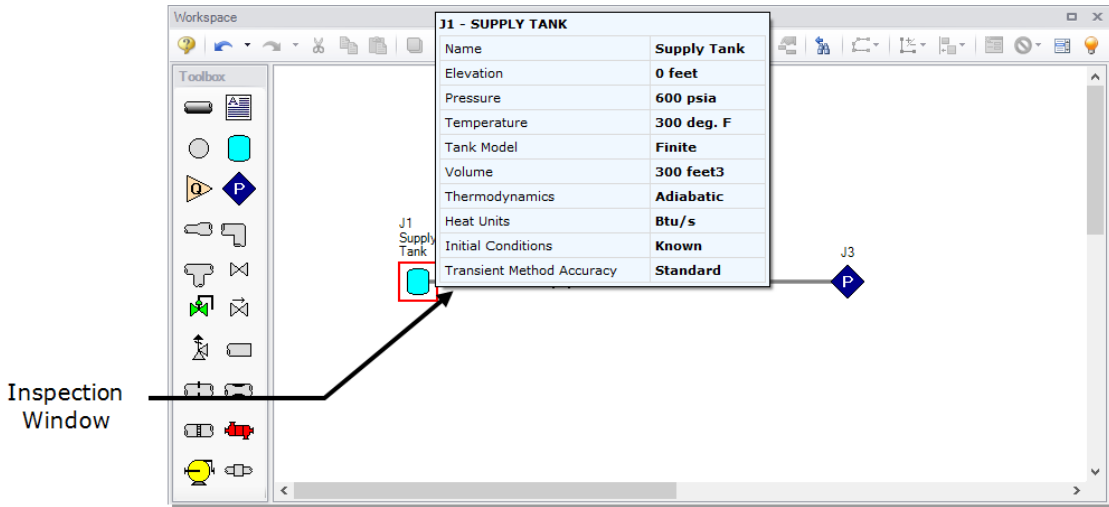
Figure 2.8   Inspect junction properties from the Workspace by clicking and holding the right mouse button

B. Enter Valve data

The J2 Junction is a Valve junction. This junction will be the initiator of the transient.

➢ **Double-click the J2 junction icon** to open the Valve Properties window (Figure 2.9).

1. Verify the Elevation is 0 feet

Note: This value by default will be equal to the last elevation used.

2. Choose the loss model as Cv and enter a Cv value of 60. This typically represents the valve's Cv during steady-state.

3. Enter an Xt of 0.7

4. Click the Optional tab and click the radio button labeled "Closed" under Special Condition. This will set the steady state Cv equal to 0.

Figure 2.9   Input data for Valve junction J2

5. Click the Transient tab and select "Time" under "Initiation of Transient" to indicate that the valve closure will be initiated at the time you specify, then enter the following Cv vs. Time data (Figure 2.10).

Time (seconds)	Cv	Xt
0	0	0.7
0.1	60	0.7
240	60	0.7

The first data point of the table must match the steady-state value of the junction. In this case, the valve is closed in the initial steady-

state due to the Special Condition specified in Step 4. Therefore, the initial transient data point automatically populates the valve as being closed (Cv of zero means the valve is closed). The valve then fully opens over a period of 0.1 seconds. It remains in this position for the rest of the simulation.
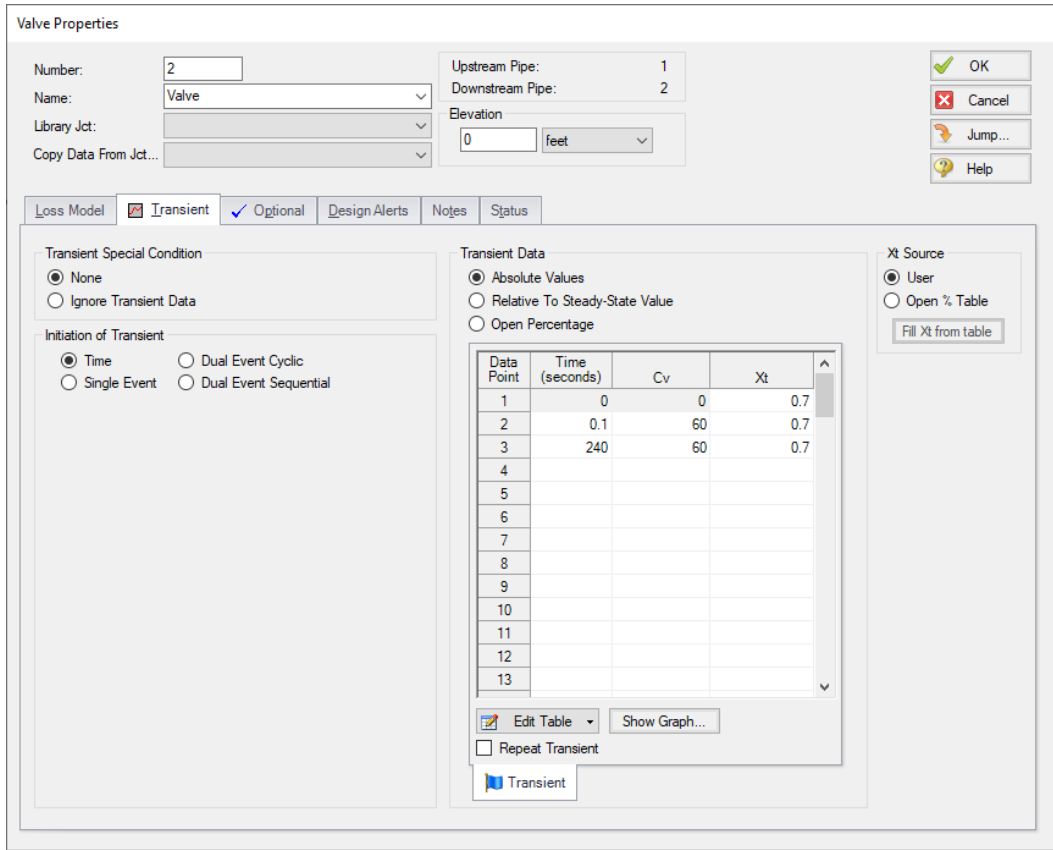
Figure 2.10   Transient input data for Valve junction J2

6. Click the OK button. There should now be a "T" symbol next to the Valve junction in the Workspace, indicating that transient data is entered for the junction. The Valve junction in the Workspace will additionally have a red "X" next to it indicating that the junction contains a special condition.

C. Enter Assigned Pressure data

The J3 Junction is an Assigned Pressure junction.

➢ **Double-click the J3 junction icon** to open the Assigned Pressure Properties window (Figure 2.11).

1. Verify the Elevation is 0 feet

2. Verify the Stagnation radio button is selected for "Boundary Specification"

3. Enter a Stagnation Pressure of 14.7 psia

4. Verify the Temperature radio button is selected and enter a value of 70 deg. F

5. Click the OK button. The Assigned Pressure junction should be fully defined.

Figure 2.11   Input data for Assigned Pressure junction J3

D. Enter pipe data for P1

Data for pipes and junctions can be entered in any order. In this example, the junctions were done first. The next step is to define all the pipes. Double-click the pipe object on the Workspace.

➢   **First, open the Pipe Properties window for Pipe P1.** The Pipe Model tab opens by default (Figure 2.12).

1. In the Pipe Material field, choose "Steel - ANSI"

2. In the Size field, select "2 inch"

3. In the Type field, select "STD (schedule 40)"

4. Specify the Length as 20 feet

5. For the Friction Model field, verify "Standard" is selected

Figure 2.12   Properties window for Pipe P1

The Pipe Properties window

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

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

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

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

Click OK to finish defining P1.

E. Enter pipe data for P2

Similar to Pipe P1, enter the following data for pipe P2:

1. In the Pipe Material field, select "Steel - ANSI"
2. In the Size field, verify "2 inch" is selected
3. In the Type field, verify "STD (schedule 40)" is selected
4. Specify the Length as 1500 feet
5. For the Friction Model field, verify "Standard" is selected
6. Click OK to finish defining P2

➢ **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.

F. 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 2.13. 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.

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.
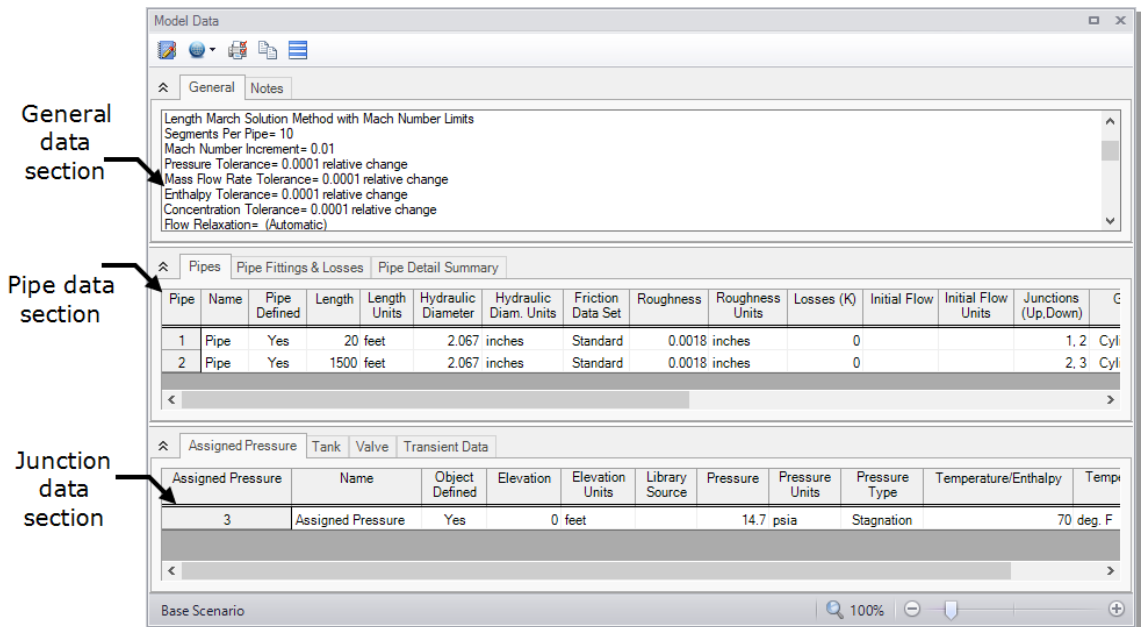
Figure 2.13   The Model Data window displays all input in text form

Step 4. Specify Analysis Setup

➢ **Next, click Analysis Setup on the Main Toolbar** to open the Analysis Setup window (see Figure 2.14). 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 (Figure 2.4).
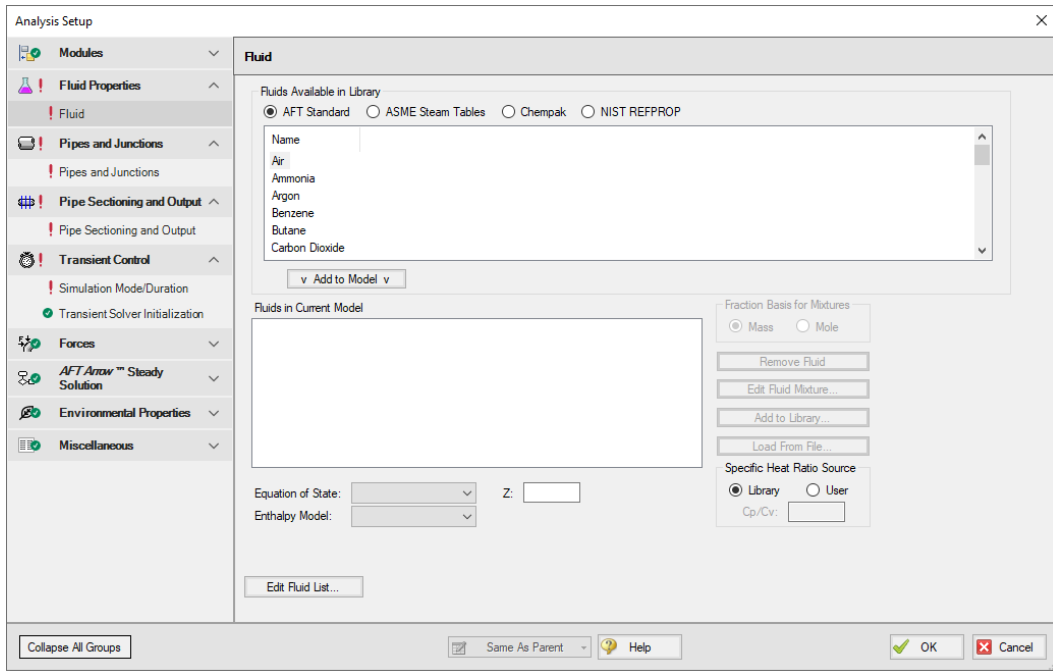
Figure 2.14   The Analysis Setup window tracks the defined and undefined model input

A. Specify the Modules group

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 2.14. 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 2.15).

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. Users can also add their own custom fluids to the AFT Standard library.

➢ **Select the AFT Standard fluid option**, then choose "Nitrogen (GN2)" from the list and click the "Add to Model" button.
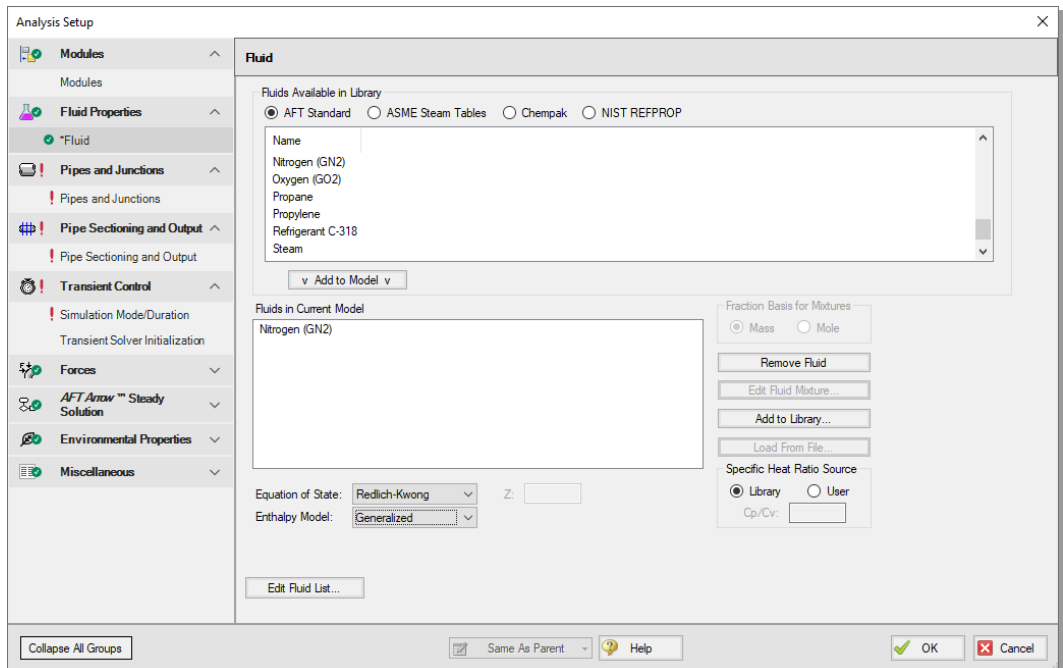
Figure 2.15   The Fluid panel is where you specify the fluid

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 *Δt* is the time step, *Δx* is the section length, and *c* is the sonic velocity.

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

The Minimum Number of Sections Per Pipe determines the section length by taking the quotient of the shortest pipe's length and the Minimum Number of Sections. Increasing the number of sections in a

model will always have a positive correlation to the model accuracy as the error inherent to the Method of Characteristics decreases when the step size is made smaller. However, the run time of the model increases quadratically as well, since each added section to the shortest pipe increases the number of calculations for each time step. This increase in run time is exacerbated when there is a large disparity between the minimum pipe length and the length of the longest pipe, such as exists in this model. The Minimum Number of Sections Per Pipe for this example will be set to 1 for expediency, but note that it would be wise to use more computational sections if you are running an AFT xStream model for real projects.

➢ **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. 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.
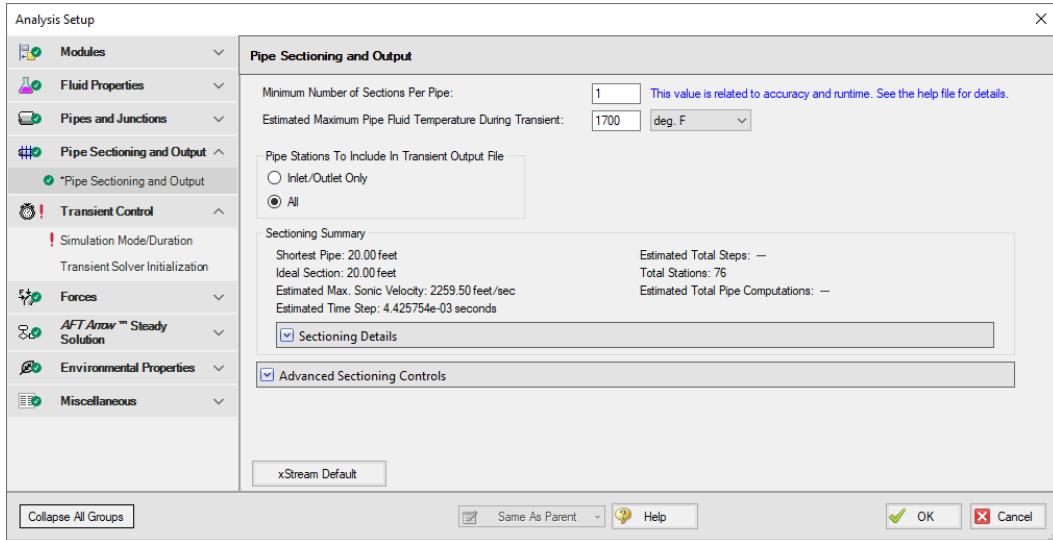
Figure 2.16   The Sectioning panel in the Analysis Setup window is used to calculate the time step of the model

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 2.17). 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.
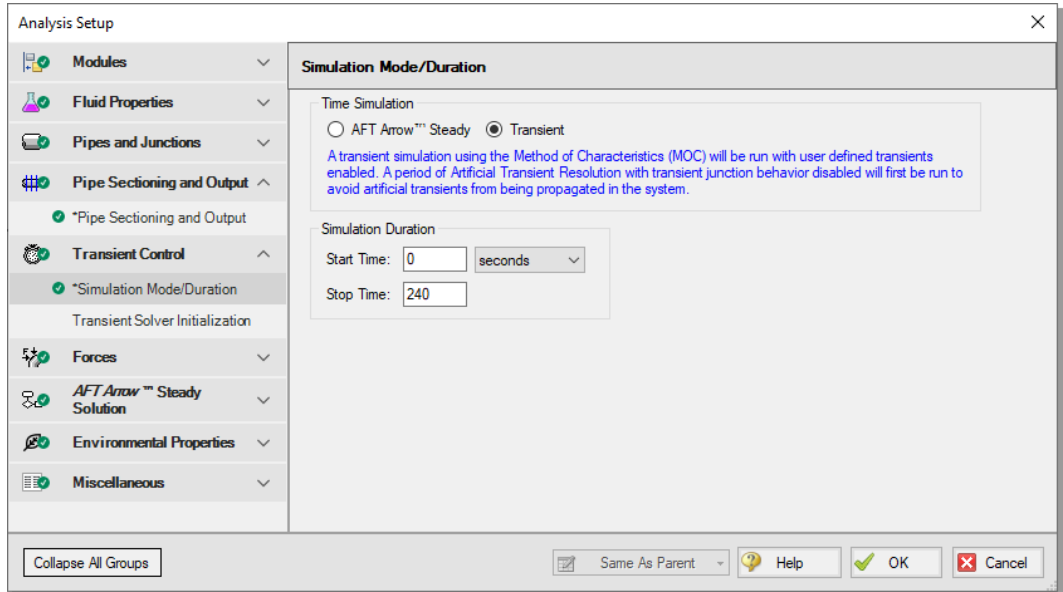
Figure 2.17   The Simulation Mode/Duration panel is used to specify the simulation type and the time span for the transient

➢   **Save the model.**

Step 5. 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 2.18). 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.
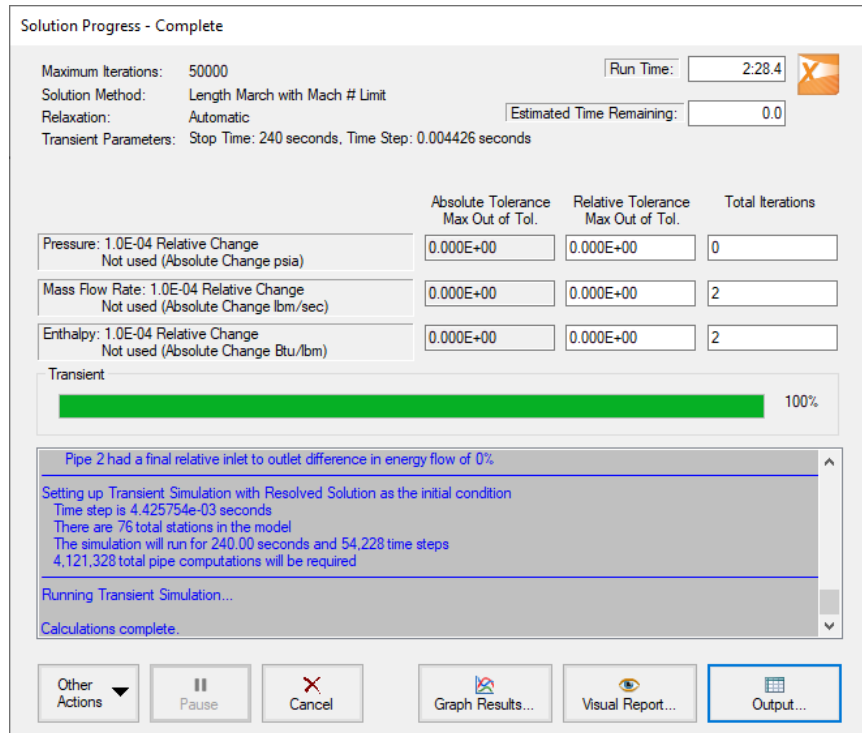
Figure 2.18   The Solution Progress window displays the state of the simulation

The two solvers

AFT xStream has two solvers. The first is called the *AFT Arrow™* Steady Solver, which as its name suggests, obtains a steady-state solution to the pipe network. The second solver is called the Transient Solver and solves the gas transient equations.

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 2.19 as a visual representation of the process described above.
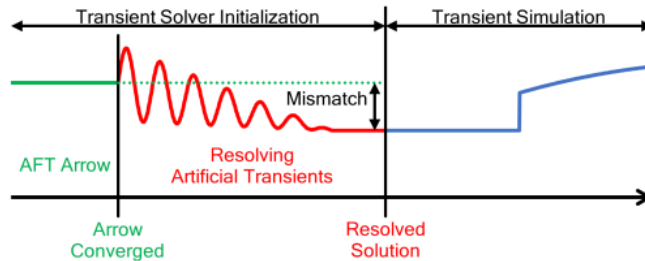
**Figure 2.19   Illustration of the Transient Solver Initialization
                 process with artificial transient resolution**

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.

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 6. Review the Output window

Click Output after the solver is finished. The Output window (Figure 2.20) 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.

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. A summary of the maximum and minimum transient results for each pipe is given on the Transient Max/Min tab in the Pipe section and is the active tab displayed in Figure 2.20.
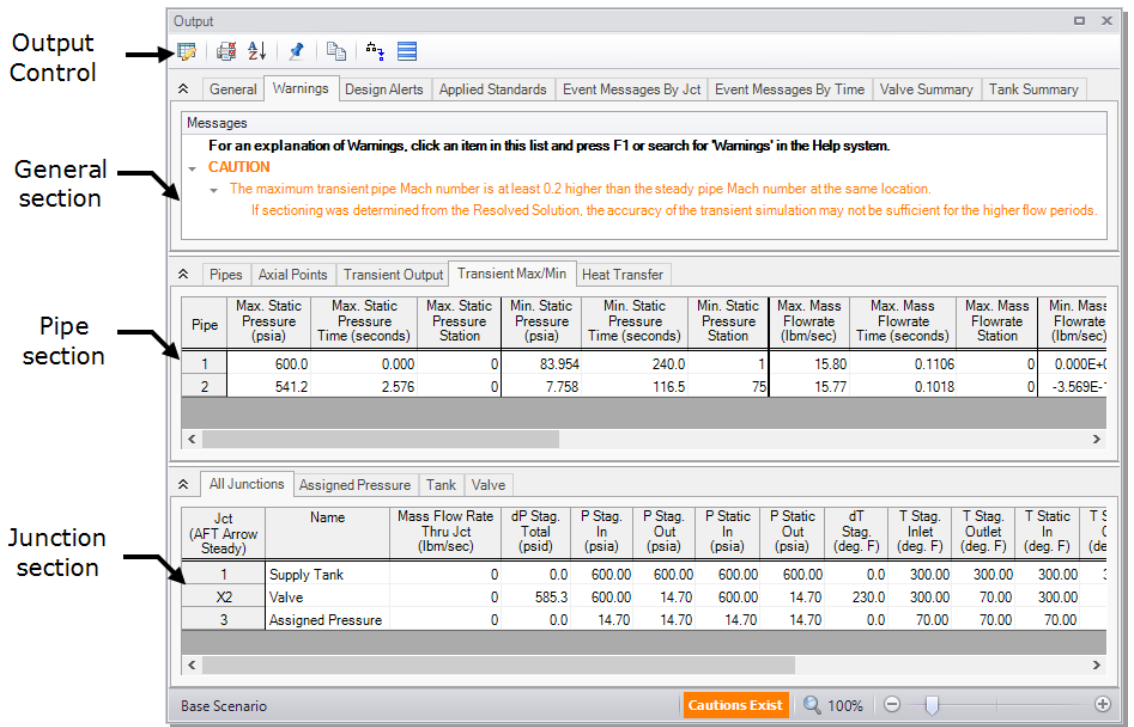
Figure 2.20   The Output window displays steady and transient output in text form
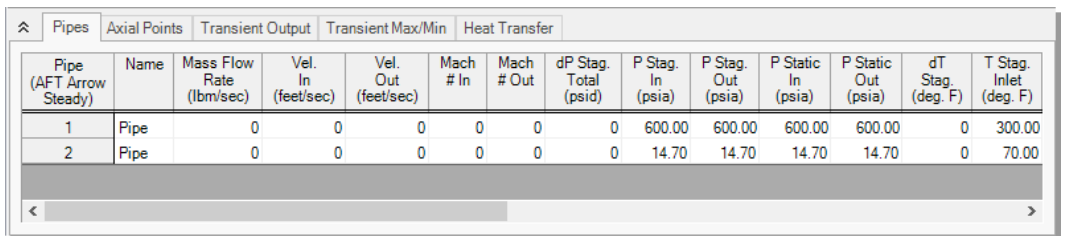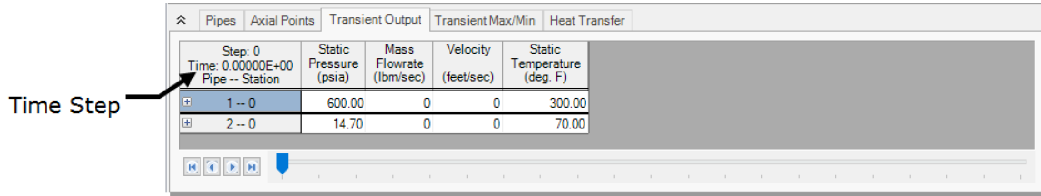
Figure 2.21   The Output window displays the values of steady-state parameters in the pipes tab
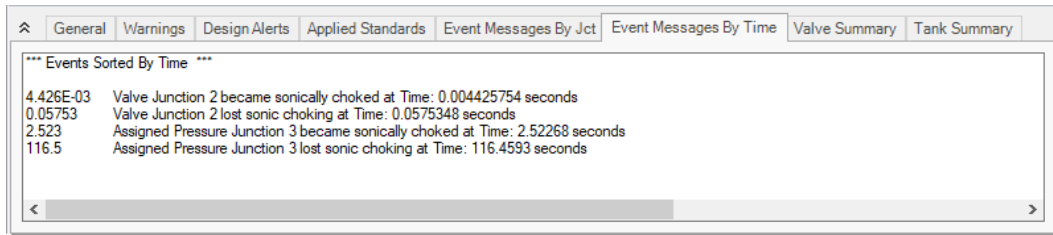
You can also review the solutions for each time step (i.e., a time history) for which data was written to file on the Transient Output tab in the Pipe section (Figure 2.22). Moving the slider along the bottom will change the time step shown in the table.

Figure 2.22   The Output window displays transient data for each time step in the Transient Output tab

The Event Messages By Time tab in the General section (Figure 2.23) allows you to review changes of states at junctions, such as the onset of choking or the initiation of transient events. This is important in gas transients as sonic choking changes the computational method used by the solver. Therefore, knowing where choked flow occurs can provide important insights about how the final results were tabulated.

Figure 2.23   The Output window displays event messages about system behavior in the Event Messages by Time tab

The Output Control window can be accessed from either the Tools menu or from the Output window Toolbar (Figure 2.24). The Output Control window allows you to select the specific output parameters you want to display in the Output window. You also can choose the units. If you do not change any of the Output Control settings, the default Output Control parameters are assigned.
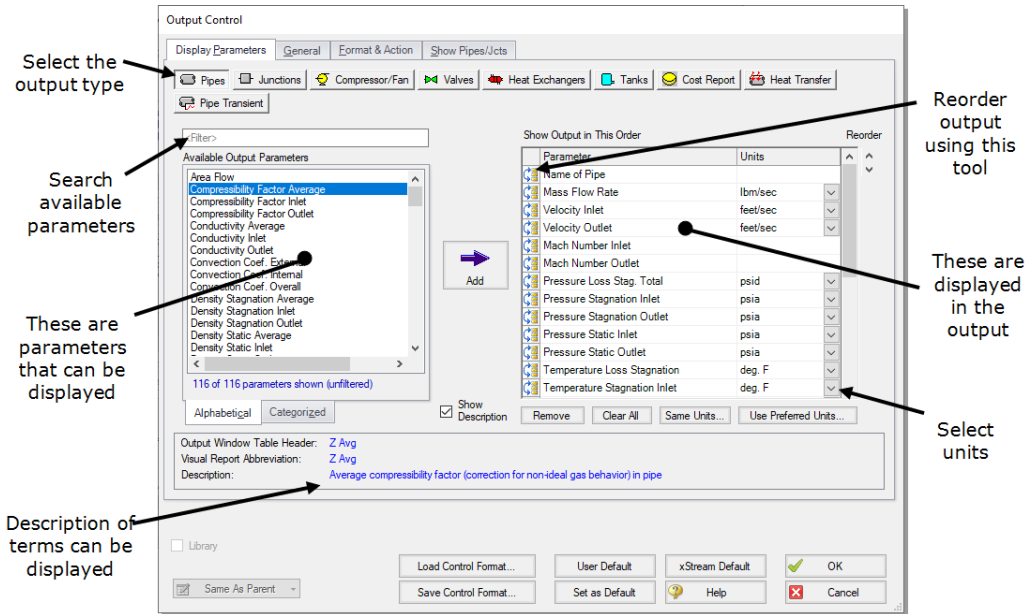
Figure 2.24   The Output Control window lets you customize the output

Units for each column in the Output window can also be changed by double-clicking the column header. This will open a window in which you can select the units you prefer (Figure 2.25). These changes are extended to the Output Control parameter data that is set.
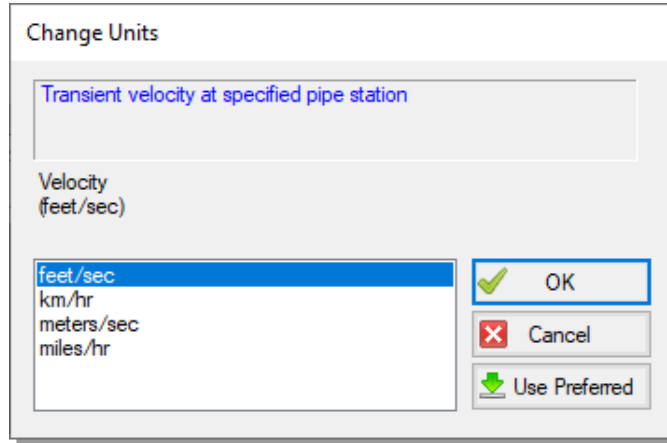
**Figure 2.25   The Change Units window is opened from the Output
window tabs by double-clicking the column header**

Step 7. View the Graph Results

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

➢ **Open the Graph Results window** by 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.

AFT xStream's Graph Guide feature, accessed by clicking on the "What
Would You Like to Do?" button located at the top right of the Graph
Results window, provides assistance by guiding you through the creation
of a "Quick and Simple" graph, or an "Advanced" graph (Figure 2.26).
You can create a graph by following the prompts on the Graph Guide, or
by manually specifying the graphing parameters on the Graph Control
tab on the Quick Access Panel, or from the "Select Graph Parameters"
icon located in the top left corner of the Graph Results Toolbar. For the
remainder of this Quick Start Guide, the Graph Guide will be hidden,
but keep in mind you can use it whenever you would like assistance in
creating a graph.

Figure 2.27 shows the Graph Results window with the Graph Control
tab selected on the Quick Access Panel.
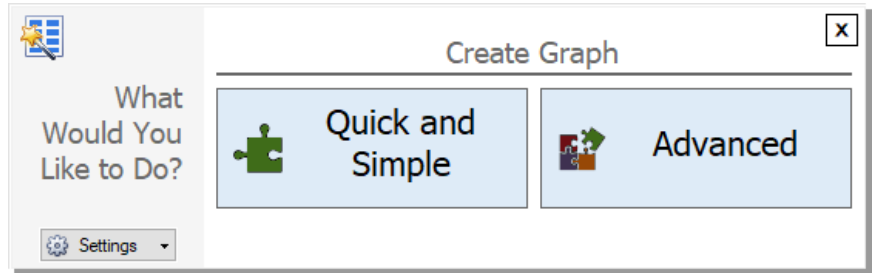
Figure 2.26 The Graph Guide
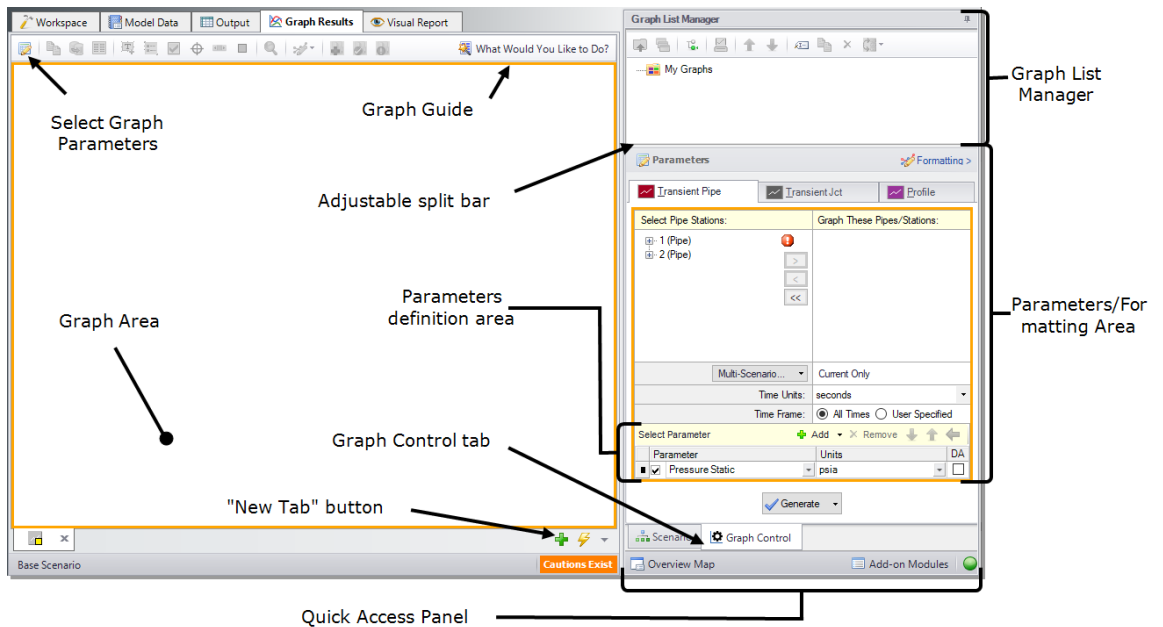
Figure 2.27 The Graph Results window and Quick Access Panel Graph Control tab is where various system parameters (both steady-state and transient) can be graphed

AFT xStream gives you the ability to create "stacked graphs". These are graphs that are vertically aligned with each other that use the same x-axis but with different parameters on the y-axis of each graph. This feature is helpful when you want to look at how different parameters

vary with time at a given position without having to create separate graphs.

In this example, you will create stacked graphs of the Temperature Static, Velocity, and Mach number at the exit of pipe P2 to see how these parameters change as the blowdown proceeds.

➢ **Access the graphing parameters by opening the Graph Control tab on the Quick Access Panel (this tab is automatically selected when the Graph Results window is opened)**. Because we are interested in seeing how the pressures and flows in specific pipe sections respond over time, ensure that the Transient Pipe tab is selected in the Parameters/Formatting area on the Quick Access Panel. Alternatively, you can open the Select Graph Parameters window by clicking on the corresponding icon on the Graph Results Toolbar (Figure 2.27).

➢ **Under "Select Pipe Stations" on the Transient Pipe tab, expand the P2 pipe stations and double-click on Outlet**, which is the pipe computing station at the discharge of the blowdown pipe. Alternatively, you can click on the right arrow button after selecting the pipe station you want to graph to add it to the "Graph These Pipes/Stations" list.

➢ **Verify "seconds" is selected for the Time Units** and "All Times" is selected for the Time Frame.

➢ **In the Parameters definition area, select "Temperature Static" and under "Units" select "deg. F" in the dropdown box** to graph the static temperature in deg. F at the pipe discharge over time.

➢ **In order to add the stacked graphs under the static temperature graph showing the velocity and Mach number at the outlet of pipe P2, click the "Add" button twice**, which is the green "+" icon next to "Select Parameter". Two new rows will appear under the Parameter definition area (Figure 2.28).

➢ **Choose "Velocity" in the first new parameter row** and select units of "feet/s". Choose "Mach Number" for the second row.

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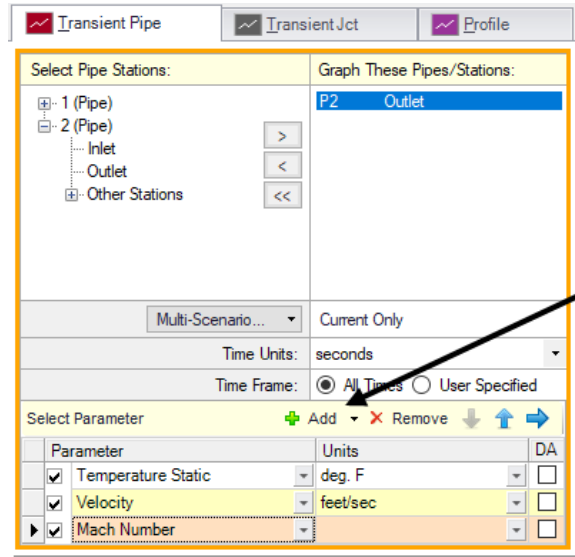
Figure 2.28 **The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area**

➢ **Click Generate at the bottom of the Quick Access Panel** to create the static temperature, velocity, and Mach number graphs at the pipe discharge over the duration of the simulation.

➢ **To format the legend font size, right-click on each legend and use the scroll bar** to decrease the font size to 14 for all three graphs. Drag the legend to the bottom right corner of the graph

➢ **To format each axis font size, right-click on each axis title and use the scroll bar** to decrease the size until the font size on each axis appears as you want it.

Figure 2.29 shows the stacked graphs detailing the static temperature, velocity, and the Mach number at Pipe P2 outlet over the course of 240 seconds. Here you can see that the maximum temperature at the discharge is about 1400 deg. F. Also note that the maximum temperature in Figure 2.29 does not approach the value indicated in the Transient Max/Min tab. This is because the Transient Pipe graph shows only the outlet station (station 75), while the maximum temperature in pipe 2 is reached at station 65.

Note: The axis titles in the stacked graphs in Figure 2.29 have been modified for clarity.
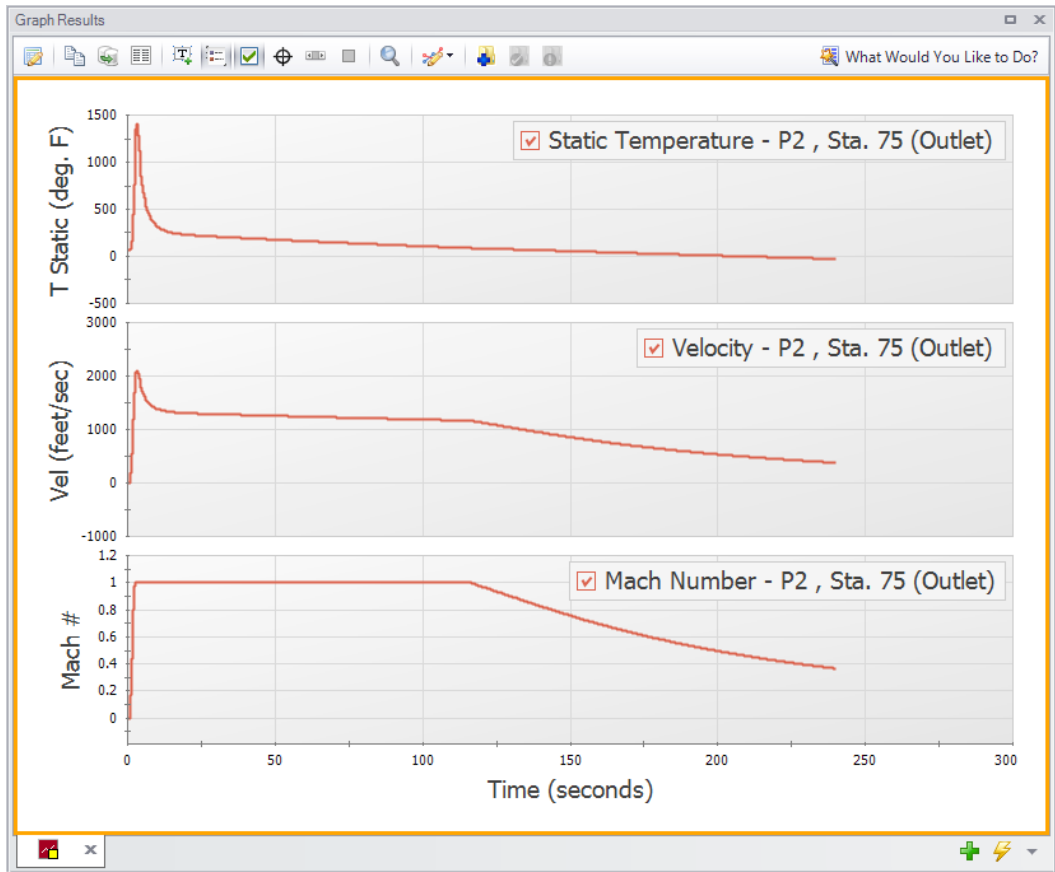
Figure 2.29   **The Graph Results window offers full-featured plot generation. The static temperature, velocity, and Mach number of Pipe P2 outlet are shown.**

Further review

Further review of the graph results in Figure 2.29 shows that sonic choking occurs after several seconds 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. This relationship is important to understand when looking at systems that encounter choking.

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

➢ **Create a new graph tab** by clicking the "New Tab" button which is the green plus icon located on the bottom right, immediately below the graph area (Figure 2.27).

➢ **Open the "Transient Jct" tab** on the Graph Control tab of the Quick Access Panel (Figure 2.30). Add the J1 (Supply Tank) Junction.

➢ **Verify "seconds" is selected in the Time Units section and "All Times" is selected in the Time Frame section** to graph the Tank Temperature and Pressure Stagnation over the entire simulation duration.

➢ **Click the "Add" button** to add a new row under the Parameter definition area.

➢ **Select "Tank Temperature Stagnation"** and specify the units as deg. F. For the second row, select "Tank Pressure Stagnation" and specify the units as psia.

➢ **Click Generate** at the bottom of the Quick Access Panel to display the graph. These graphs show the tank pressure and temperature initially decrease quickly, then decrease more slowly as time progresses. This change in slope is due to a decreasing pressure gradient between the tank and the atmosphere as the tank is emptied, which slows the mass flow rate out of the tank (Figure 2.31).

Note: AFT xStream assumes all pipes are adiabatic (perfectly insulated). Over a 240 second run time this assumption may not be accurate. Consider this when evaluating simulation results over longer times.
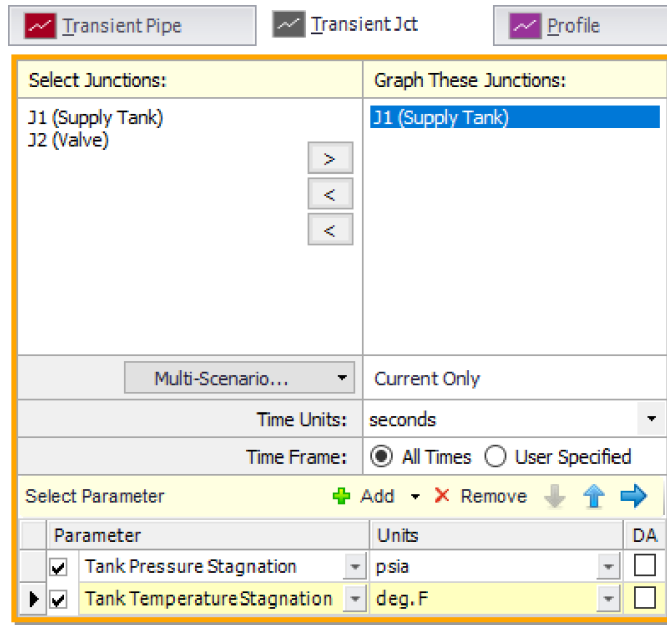
Figure 2.30   Graph Parameters window for the Transient Junction graph
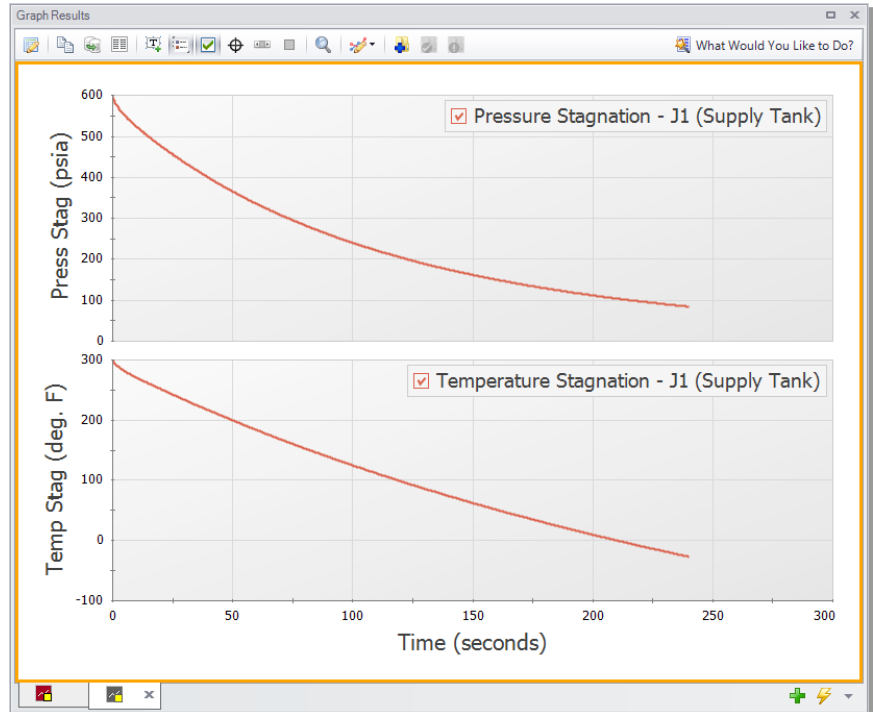
Figure 2.31   The Transient Junction graph shows how variables within a junction change over time. The temperature and pressure of Tank J1 over time are plotted.

Step 8. 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 2.32. Click the Show button. This will generate the Visual Report window graphic seen in Figure 2.33.
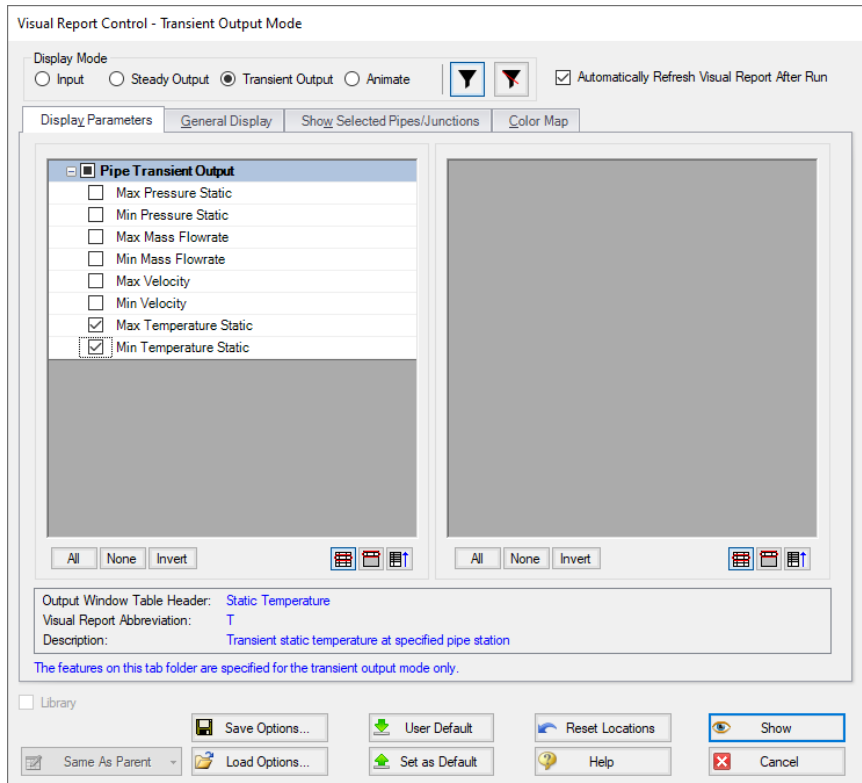
Figure 2.32   The Visual Report Control window selects content for the Visual Report window

It is common for the text in the Visual Report window to overlap when first generated. You can change this 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 been made in Figure 2.33. This window can be printed

or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.
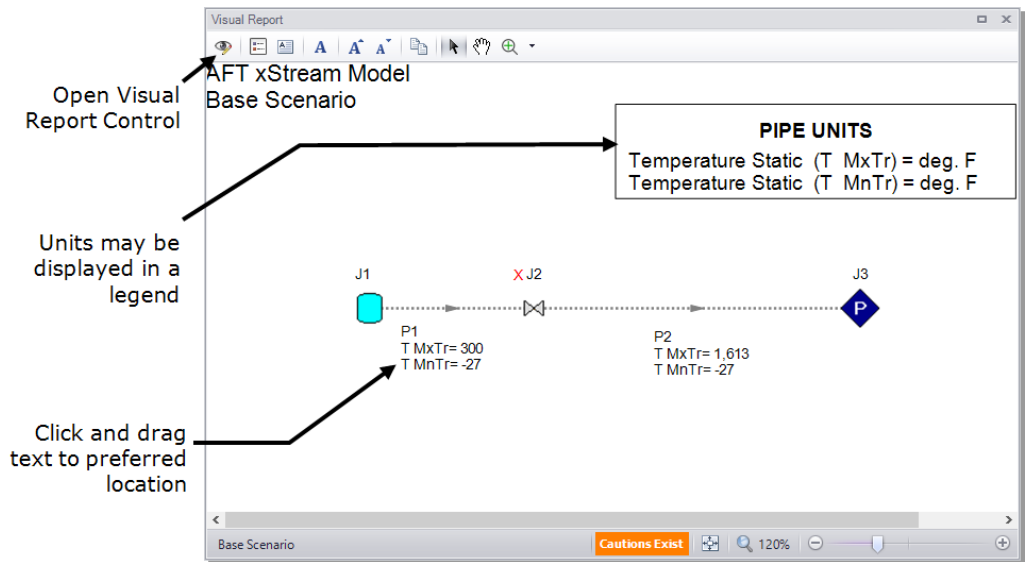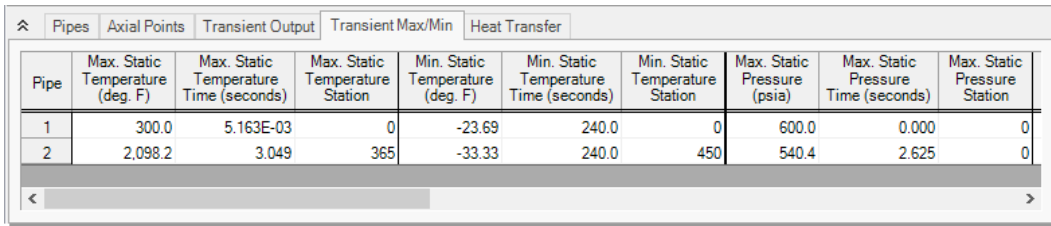
Figure 2.33   The Visual Report integrates results with model layout

Step 9. Examine the effect of sectioning

In AFT xStream, sectioning plays an important role in model accuracy. Due to how the MOC operates, accuracy increases with the number of sections you have in the model. To illustrate this, the model was re-run with a Minimum Number of Sections Per Pipe of 6. Comparing Figure 2.31 to Figure 2.36, 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 2.20 and Figure 2.34), 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 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.

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 2.34   Transient Min/Max window for modified sectioning
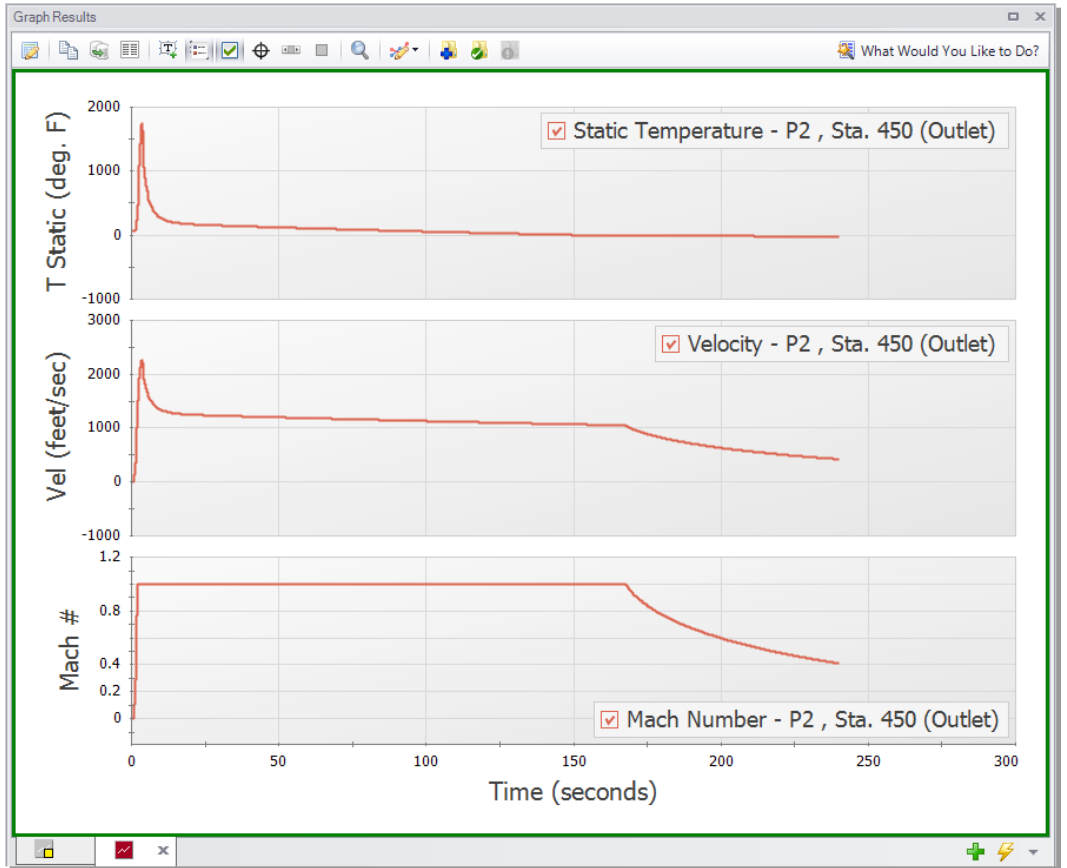
Figure 2.35   Temperature, velocity, and Mach number for the discharge of Pipe P2 with 6 sections minimum per pipe
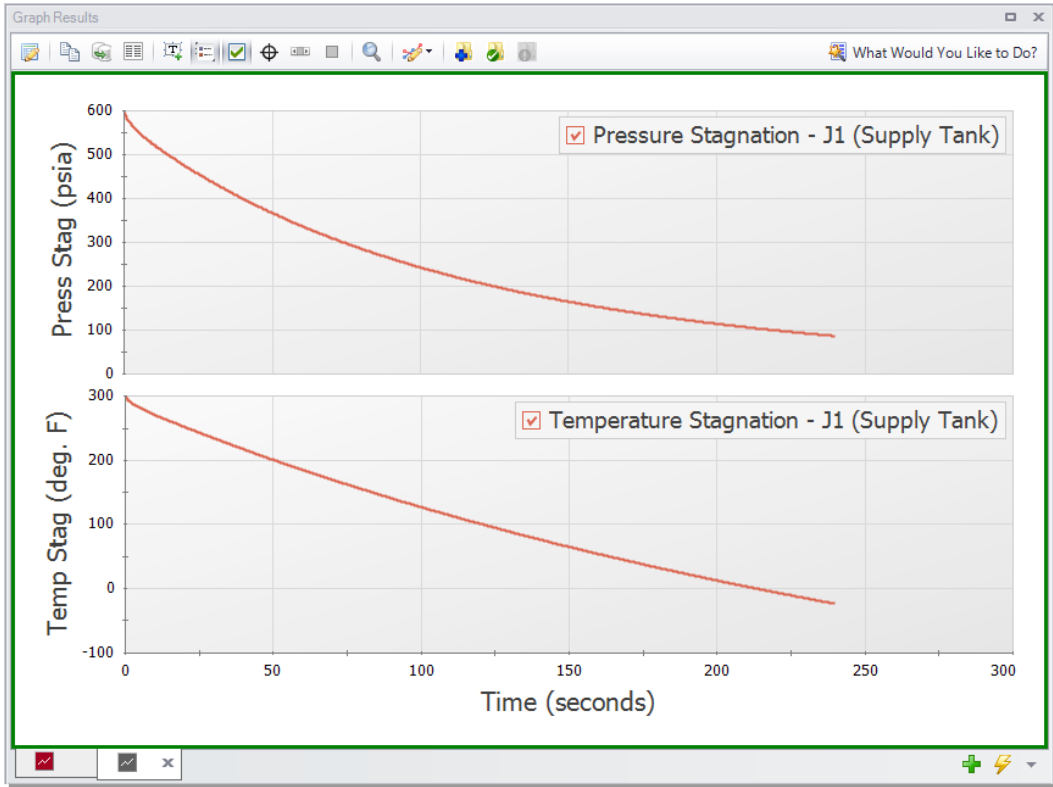
Figure 2.36   Temperature and pressure stagnation for Tank J1 with 6 sections minimum per pipe

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

This example looks at an instantaneous valve closure in front of the combustion baskets in a natural gas fuel system. The objective is to determine the maximum fluid temperature reached during the transient.

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 that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, entering pipe and junction properties, and entering transient data. Refer to the Tank Blowdown Example in Chapter 2 for more information on these topics.

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*

Problem statement

The objective of this example is to evaluate the maximum temperature that the natural gas in a turbine fuel system will reach if one of three control valves were to instantaneously shut. The natural gas is preheated by a heat exchanger before passing through the three control valves, with each feeding a separate combustion basket.

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

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 3.1.
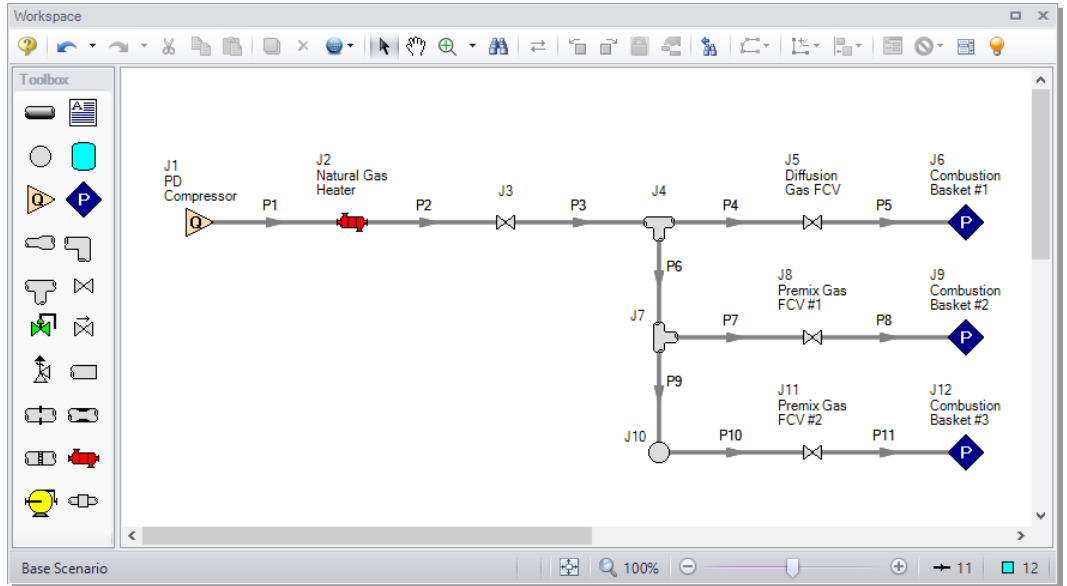
Figure 3.1   Layout of gas turbine fuel system

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
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" dropdown 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 3.2.
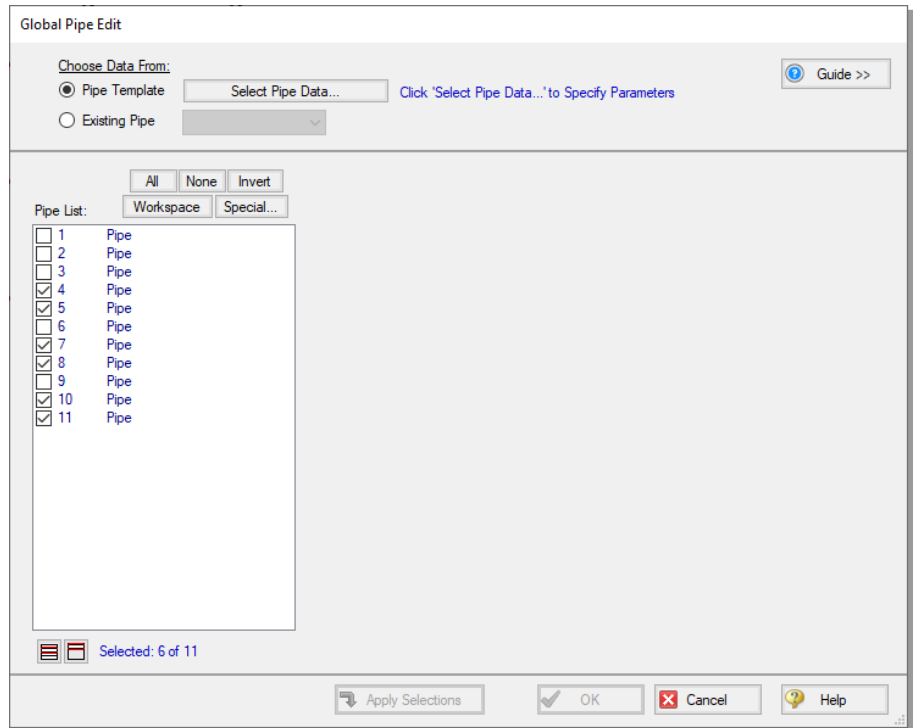
Figure 3.2   Global Pipe Edit window

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 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 3.3.
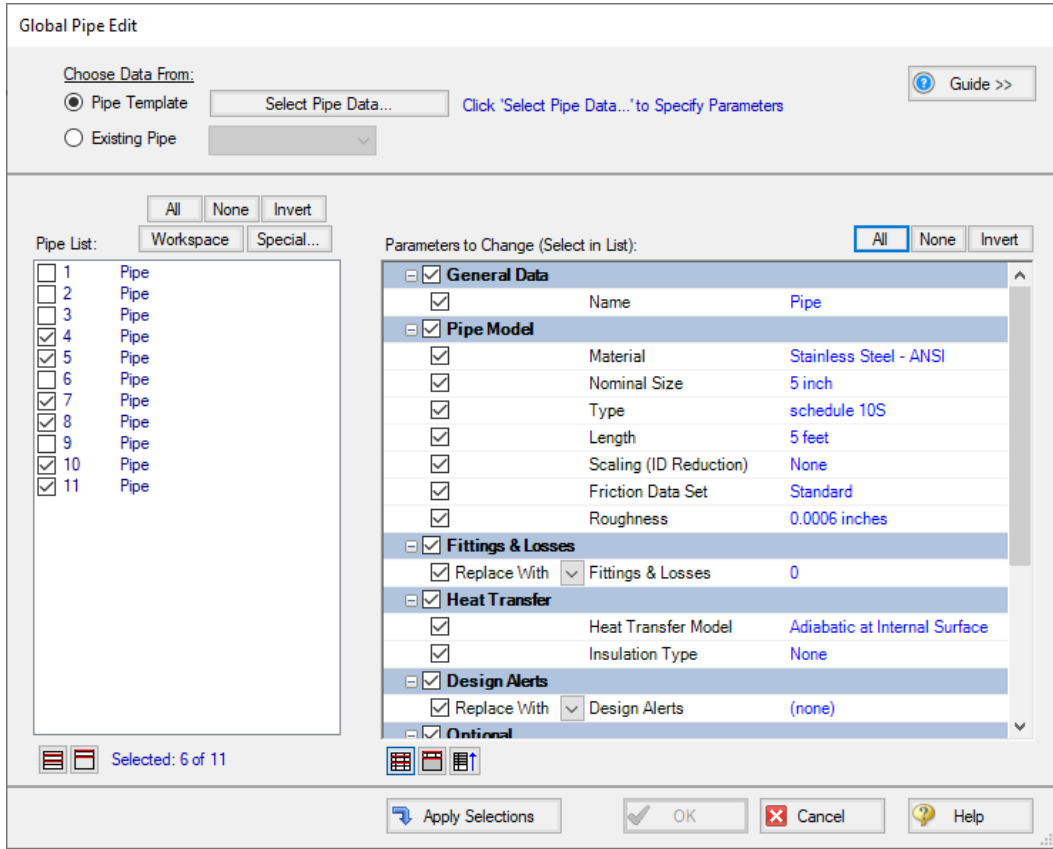
Figure 3.3   Global Pipe Edit window with all pipe parameters selected for updating

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.

J1 - Assigned Flow

1. Name = PD Compressor

2. Elevation = 0 feet

3. Mass Flow Rate = 90000 lbm/hr

4. Temperature = 70 deg. F

J2 - Heat Exchanger

In the Loss Model tab enter the following:

1. Name = Natural Gas Heater

2. Elevation = 0 feet

3. K = 6.6

In the Thermal Data tab enter the following:

Steady Heat Rate = 3535200 Btu/hr

J3 - Valve

1. Elevation = 0 feet

2. Cv = 10000

3. Xt = 0.3

J4 and J7 - Tee

Elevation = 0 feet

J5 - Valve

1. Name = Diffusion Gas FCV

2. Elevation = 0 feet

3. Cv = 95

4. Xt = 0.7

J6, J9 and J12 - Assigned Pressure

Because all of the Assigned Pressures have similar input data, the
Global Junction Edit tool could be used to input the data for all of the

junctions at the same time. The Global Junction Edit tool works in much the same way as the Global Pipe Edit tool.

Another tool that can be used to speed up data entry for items with the same input data is the Copy Data From Junction tool. Enter the data for Assigned Pressure junction J6, as listed below, by double clicking the junction icon on the Workspace.

1. J6 Name = Combustion Basket #1

2. Elevation = 0 feet

3. Stagnation Pressure = 15.5 atm

4. Temperature = 140 deg. F

Click OK to close the junction J6 Properties window. Open the junction J9 Properties window. Near the top of the Properties window is a drop-down box labeled "Copy Data From Jct". This box lists all of the Assigned Pressure junctions on the Workspace. Select junction J6 from the list. This will open the Copy Data From Junction window, which shows a list of all of the parameters for junction J6, and their values. By checking the box beside a parameter, the current junction will copy the parameter value from J6. In this case, you want all of the parameters for J9 to be the same as J6 except for the name. Select the All button at the top of the list to select all of the parameters, then uncheck the name parameter, as shown in Figure 3.4.
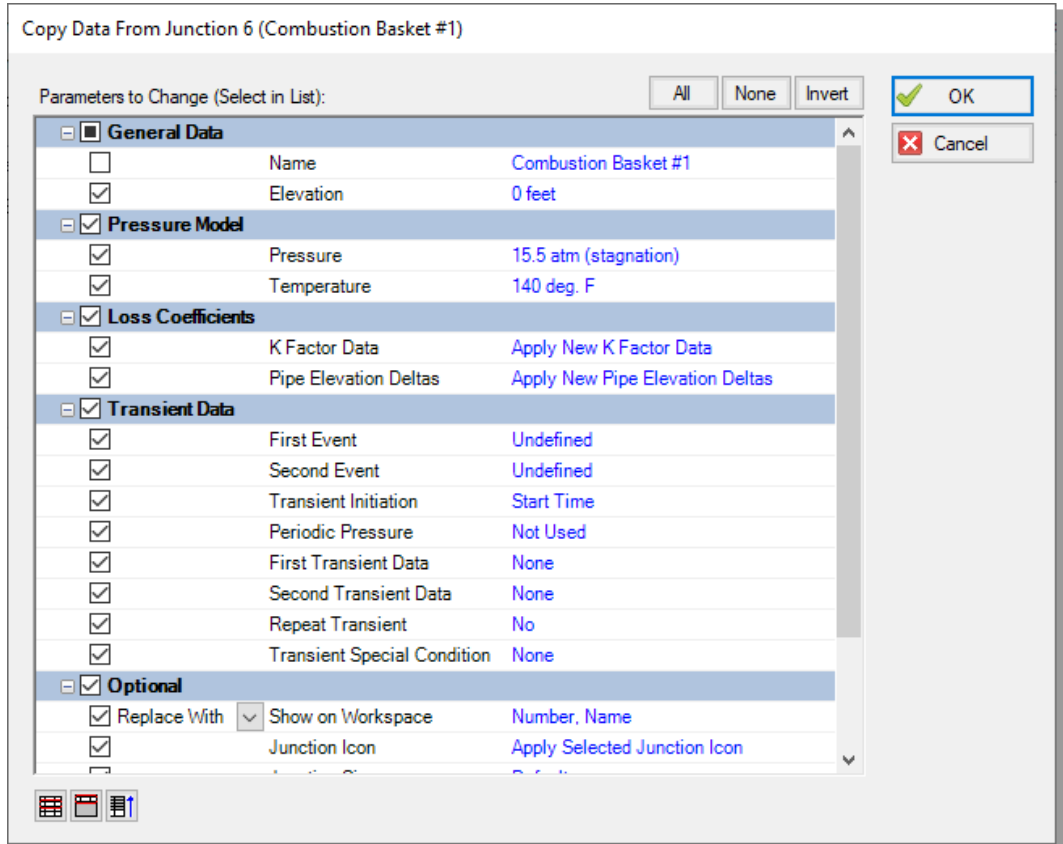
Figure 3.4   The Copy Data From Junction window

After clicking the OK button in the Copy Data From Junction window, all of the junction J9 parameters are updated to the selected J6 parameter values. Set the name of junction J9 to "Combustion Basket #2" and click OK. Repeat this process for J12, giving junction J12 the name "Combustion Basket #3".

J8 - Valve

1.   Name = Premix Gas FCV #1

2.   Elevation = 0 feet

3. Cv = 195

4. Xt = 0.7

J10 - Branch

Elevation = 0 feet

J11 - Valve

1. Name = Premix Gas FCV #2

2. Elevation = 0 feet

3. Cv = 195

4. Xt = 0.7

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

Time (seconds)	Cv	Xt
0	195	0.7
0.01	0	0.7
2	0	0.7

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.

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.

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

- Create, name, and organize scenarios

- Select the scenario to appear in the Workspace (the current scenario)

- Delete, copy, and rename scenarios

- Duplicate scenarios and save them as separate models

- Review the source of a scenario's properties

- Pass changes from a scenario to its children

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

➢ **Click the "Create Child" button on the Scenario Manager** on the Quick Access Panel. Name the child "8 Sections Minimum Per Pipe". A new scenario will appear below the Base Scenario in the scenario tree. Select the Base Scenario, create another child, and call it "4 Sections Minimum Per Pipe". Finally, select the Base Scenario again and create a third child called "2 Sections Minimum Per Pipe" (see Figure 3.5). Note that the new child scenario is loaded when it is created using the Scenario Manager on the Quick Access Panel.

"Create Child Scenario" button
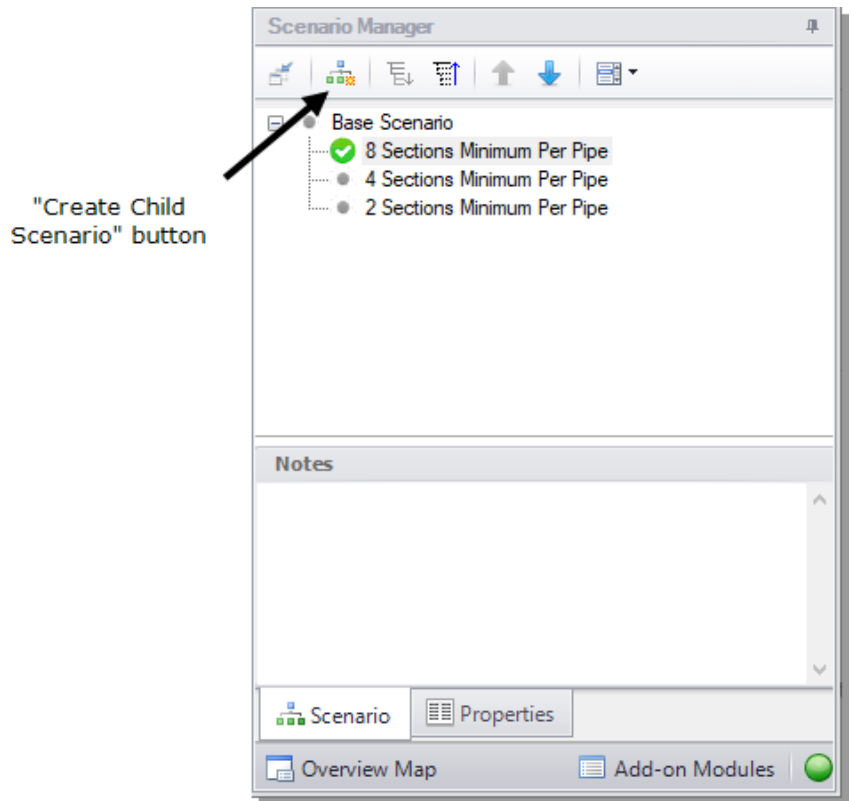
Figure 3.5   The Scenario Manager on the Quick Access Panel displays the scenario tree which allows you to create scenarios and keep them organized within the same model file

B. Set up scenarios

Child scenarios "inherit" data from their ancestor scenarios. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent. Since the Base Scenario already has been setup with 8 Minimum Sections Per Pipe, the "8 Sections Minimum Per Pipe" child does not need to be modified.

4 Sections Minimum Per Pipe
Double-click the "4 Sections Minimum Per Pipe" scenario in the scenario tree of the Scenario Manager on the Quick Access Panel. This

will load the "4 Sections Minimum Per Pipe" scenario onto the Workspace as the current scenario. The currently loaded scenario is indicated in the scenario tree with a green circle with a checkmark (Figure 3.5).

➢ **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. The required input for this scenario is now complete.

2 Sections Minimum Per Pipe

Double-click the "2 Sections Minimum Per Pipe" scenario from the scenario tree in the Scenario Manager on the Quick Access Panel. This will load the "2 Sections Minimum Per Pipe" scenario onto the Workspace as the current scenario. This step can also be accomplished by opening the Scenario Manager from the Tools menu, selecting the "2 Sections Minimum Per Pipe" scenario and clicking the "Load As Current Scenario" button.

➢ **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. The required input for this scenario is now complete.

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 3.6). The

Estimated Maximum Pipe Temperature During Transient is critical to the calculation of transient conditions due to its necessity in the Method of Characteristics.

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.
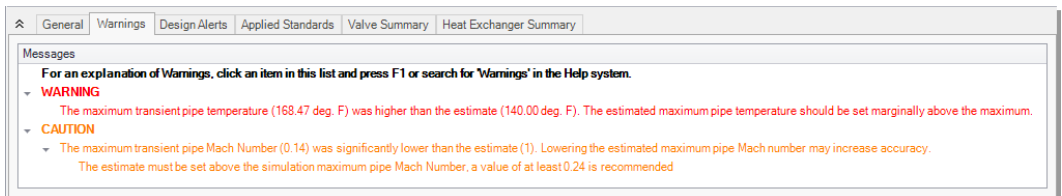
Figure 3.6   The maximum transient pipe temperature exceeds estimated maximum temperature warning indicates that the model is at risk of instability if changes are made

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

➢ **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 3.7). 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.

Pipe	city (feet/sec)	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

Tabs: Pipes · Axial Points · Transient Output · Transient Max/Min · Heat Transfer

Figure 3.7   Transient Max/Min tab in the Output window

Step 9. 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

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 dropdown box under "Units"

5. Click the Generate button

The results (see Figure 3.8) 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.
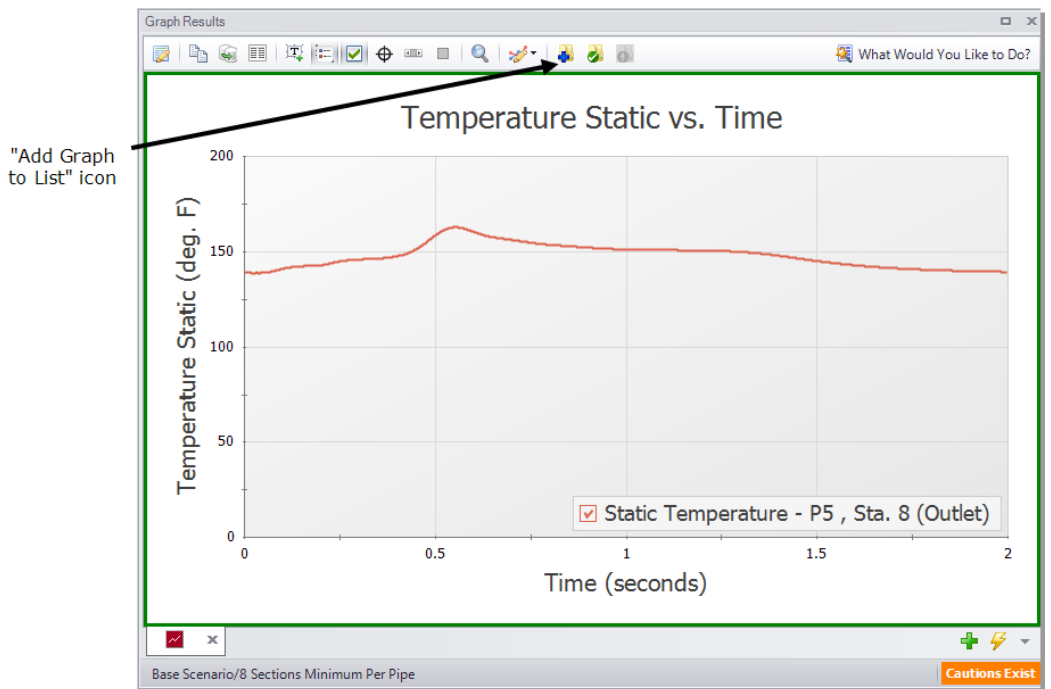

Figure 3.8   Inlet to combustion basket #1 Temperature Static vs Time

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.

Step 10. Animate the results

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

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

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. Add two additional parameters in the Parameters definition area. Specify the three parameters as "Temperature Static", "Pressure Static", and "Velocity"

5. Verify the Length Units are feet

6. Next to Animate Using, select "Output", and verify "seconds" is selected next to Animation Time Units (see the following paragraph for an explanation about this option)

7. Verify the temperature units are "deg. F", the pressure units are "psia", and velocity units are "feet/sec"

8. Check the Mx and Mn boxes next to the graph parameters to plot the maximum and minimum values on the graph (see Figure 3.9)

To use the "Animate Using: Output" option for animation, which is done here, all pipe stations need to be saved in the Pipe Sectioning and Output panel (this was already verified in Step 5. If All is not selected, "Output" will be grayed out).
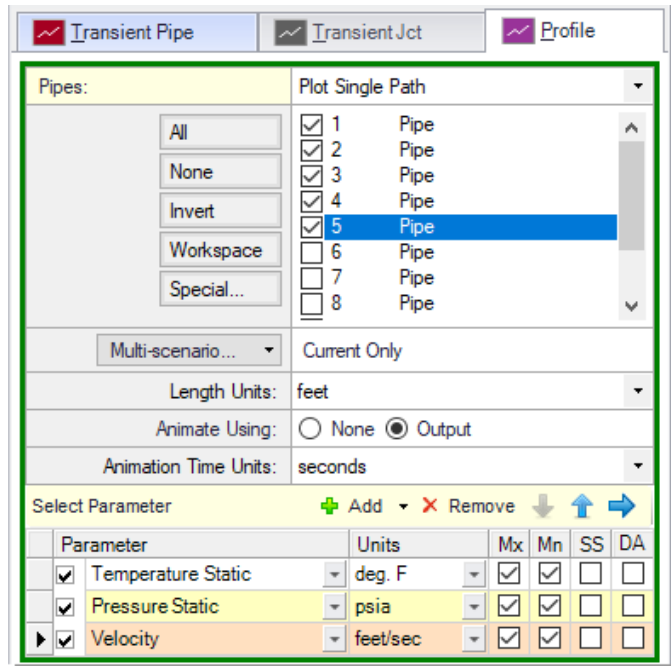

Figure 3.9   Graph Parameters for combustion basket #1 profile

Click the Generate button. Additional animation control features appear on the Graph Results window (Figure 3.10). Press the Play button and watch the temperature, pressure, and velocity responses move between the maximum and minimum values. This animation can also be recorded to a file. Click the "Add Graph to List" icon and give it a name, such as "Combustion Basket #1 Profile".

Note: Figure 3.10 has had the axes titles and scales modified to increase clarity.
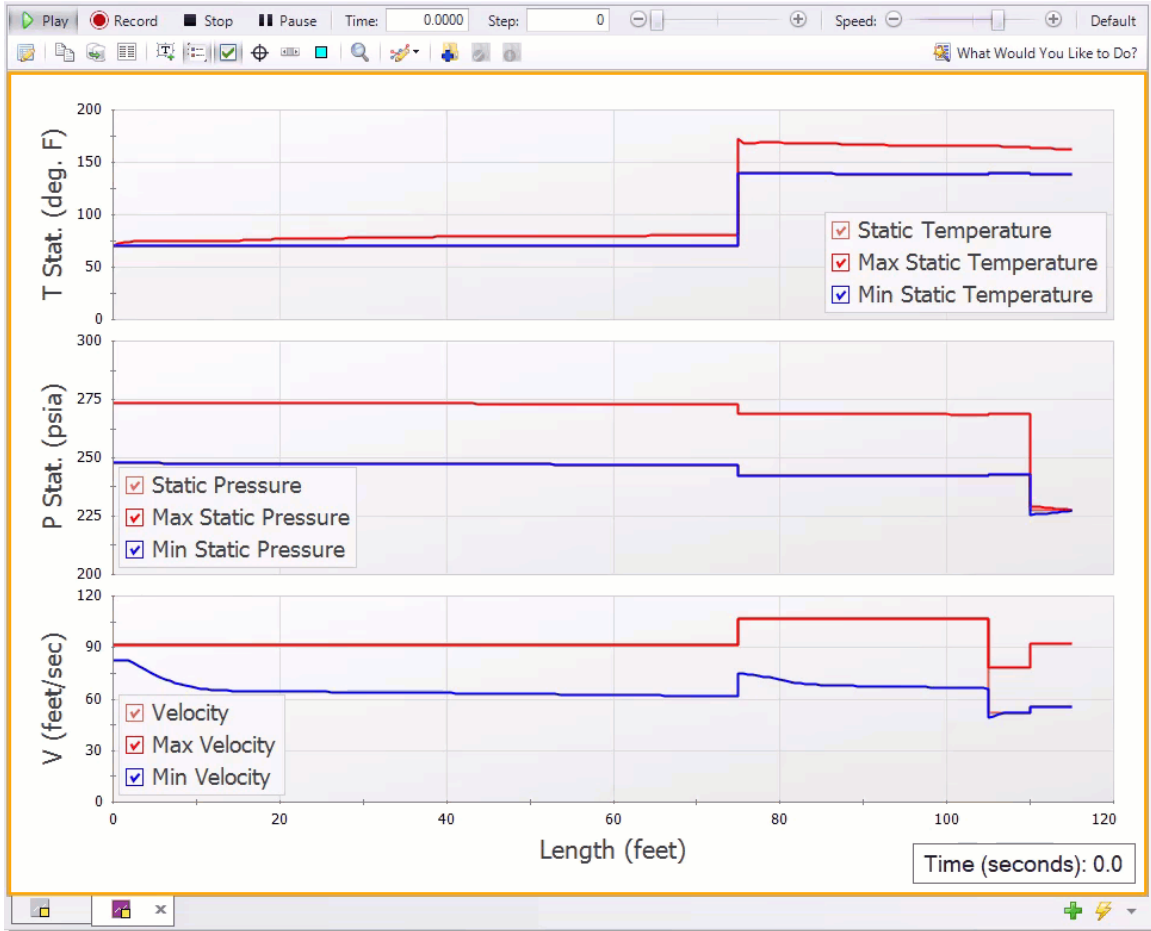

Figure 3.10   Flow to combustion basket #1 profile animation

> *To view the animated profile*
>
> *follow this link: https://bit.ly/3NbaokZ*

Step 11. 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 Steps 11 and 12 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.

Conclusion

The overall maximum temperature can be found in the General section of the Output window of each scenario. The difference between the three sectioning counts is summarized in Table 3.1. For these cases the maximum temperatures differ by 4.7 degrees.

Table 3.1   **Summary of the maximum Static Temperature for the three sectioning counts**

Scenario	Max Transient Temperature (deg. F)
8 Sections	173.2
4 Sections	170.2
2 Sections	168.5

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.

CHAPTER 4

High Pressure Steam - Fluid Property Models Example

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 that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, entering pipe and junction properties, and entering transient data. Refer to the Tank Blowdown Example in Chapter 2 for more information on these topics.

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*

Problem statement

In this problem, the inlet valves to two steam turbines in a high pressure steam system are closed simultaneously while the bypass line is valved open. Evaluate the maximum pressure at the turbine inlet. Additionally, quantify the mass flow rate through the bypass valve throughout the transient. Lastly, perform an analysis on whether two-phase flow occurs in the bypass line by conducting a sensitivity analysis on the fluid property model utilized.

Step 1. Start AFT xStream

Open AFT xStream. Click "Start Building Model".

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 4.1

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 4.2

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.
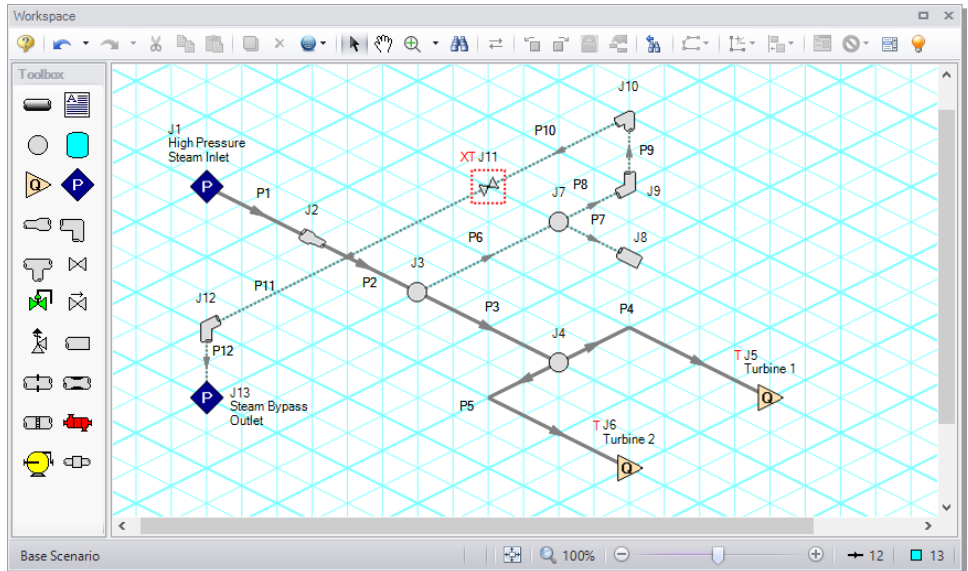

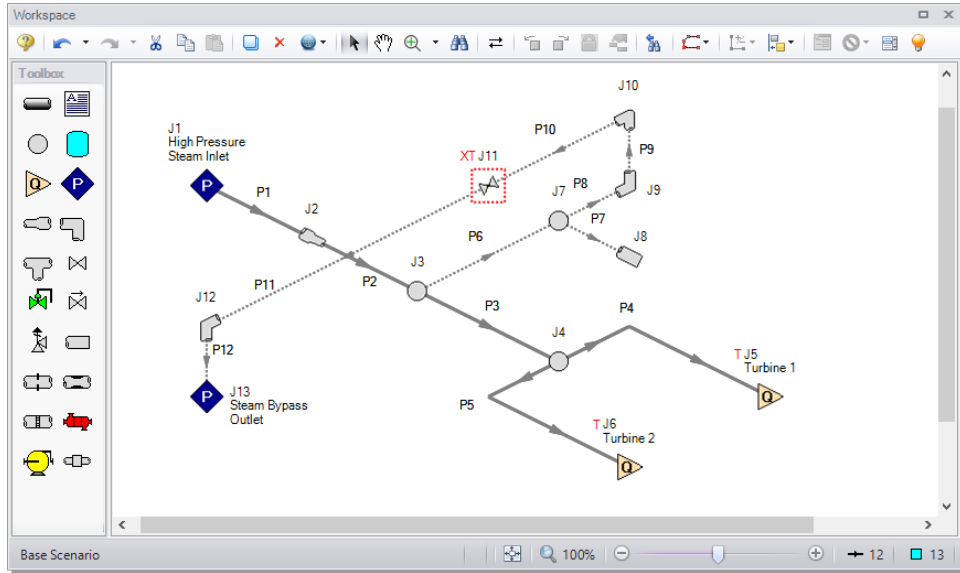**Figure 4.1   The isometric layout with the grid shown**

Figure 4.2   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 4.3). Use the Absolute Roughness of Steel (0.0018 in) as the Friction Model, along with the following data:

Pipe #	Inner Diameter (inches)	Length (feet)
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

Figure 4.3   User Specified Pipe Properties window

J1 - Assigned Pressure

1. Name = High Pressure Steam Inlet

2. Elevation = 0 feet

3. Static Pressure = 2425 psia

4. Static Temperature = 1060 deg. F

J2 - Area Change

1. Elevation = 0 feet

2. Type = "Conical Transition"

3. Angle (μ) = 45 Degrees

J3, J4, and J7 - Branch

Elevation = 0 feet

J5 and J6 - Assigned Flow

1. J5 Name = Turbine 1

2. J6 Name = Turbine 2

3. Elevation = 0 feet

4. Mass Flow Rate = 504000 lbm/hr

5. Static Temperature =1060 deg. F

6. On the Transient tab of both J5 and J6, enter the following data:

Time (seconds)	Mass Flow (lbm/hr)	Static Temperature (deg. F)
0	504000	1060
0.25	0	1060
5	0	1060

J8 - Dead End

Elevation = 0 feet

J9 - Bend

Elevation = 0 feet

Type = Standard Elbow (knee, threaded)

J10 and J12 - Bend

Elevation = 5 feet

Type = Standard Elbow (knee, threaded)

J11 - Valve

1.   Elevation = 5 feet

2.   Cv = 3000

3.   Xt = 0.7

4.   On the Optional tab, under Special Condition, select the radio button for Closed.

5.   On the Transient tab, enter the following data

Time (seconds)	Cv	Xt
0	0	0.7
2	3000	0.7
5	3000	0.7

J13 - Assigned Pressure

1.   Name = Steam Bypass Outlet

2.   Elevation = 0 feet

3.   Static Pressure = 435 psia

4.   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.

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

➢ **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 4.4. 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 4.5)

The graph should now match Figure 4.6. 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.
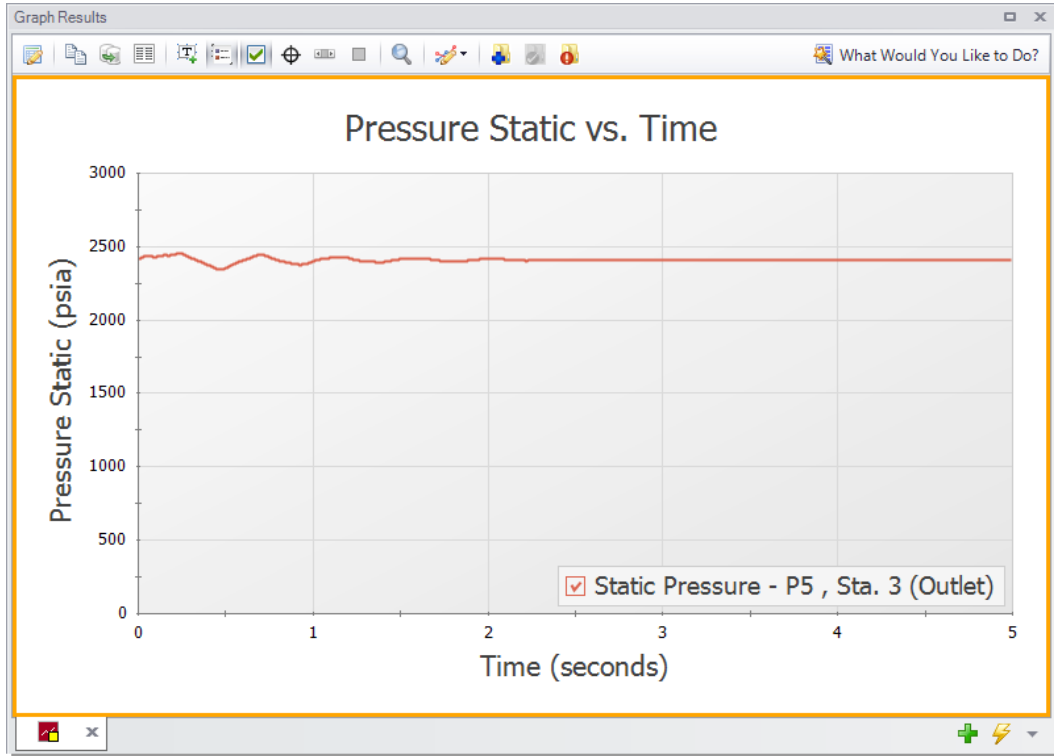
Figure 4.4   Static pressure at steam turbine 2 entrance

Figure 4.5   Axis Modification Window opened by right-clicking the y-axis
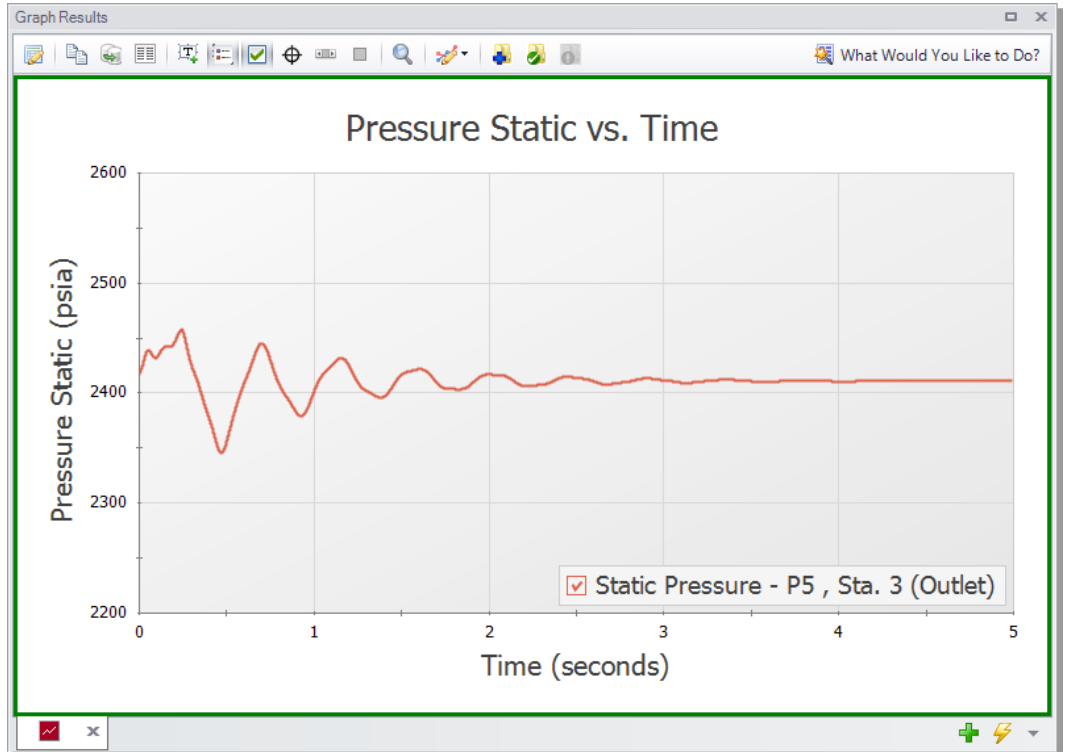
Figure 4.6   Static pressure at steam turbine 2 entrance with
modified y-axis

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

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 4.7 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 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.
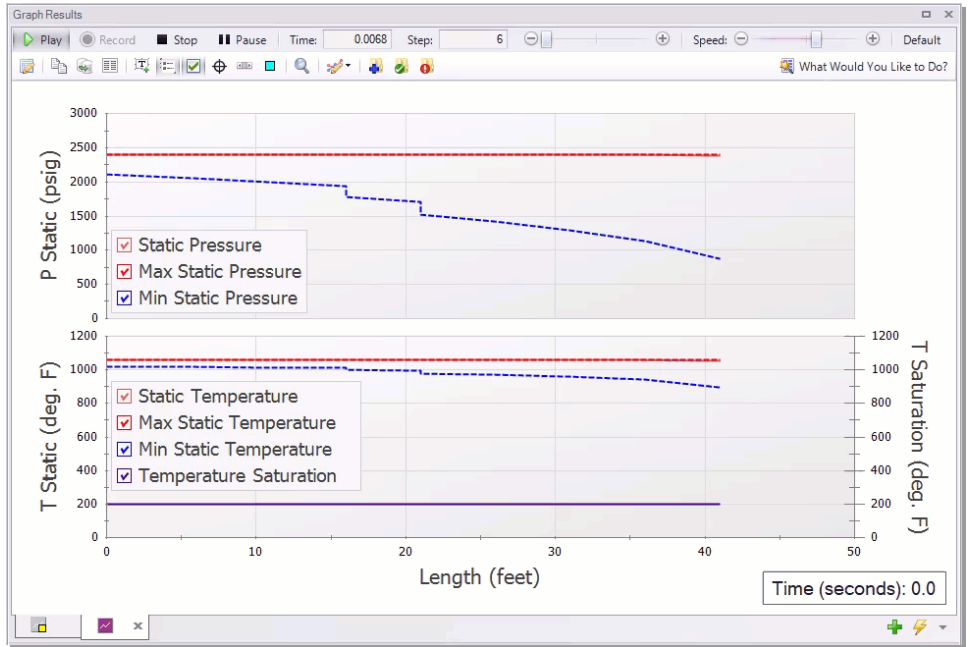
Figure 4.7   Bypass line animation profile

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

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 4.8, 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 4.8 has modified axes titles and its legend removed for clarity.
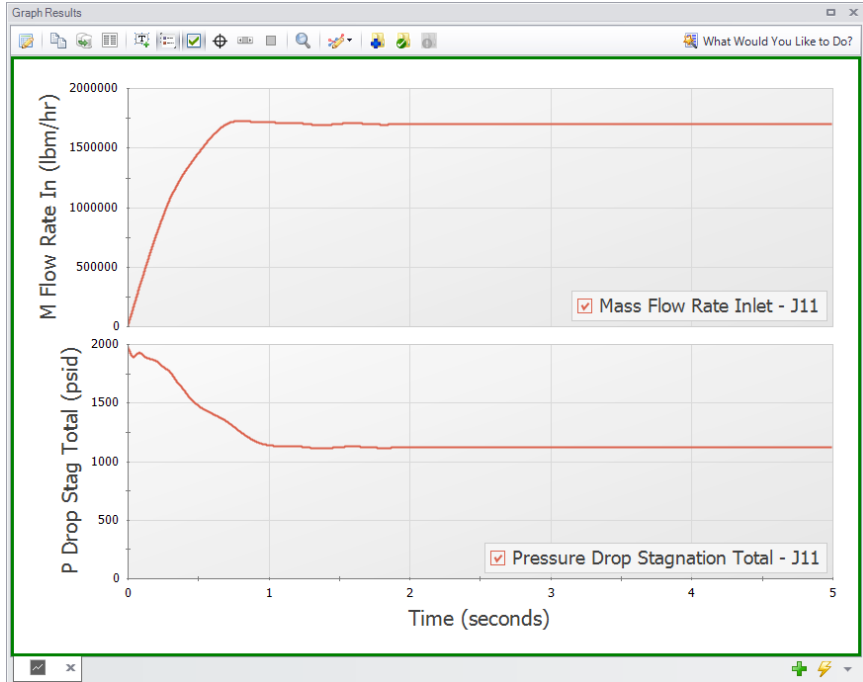
Figure 4.8   **Bypass valve mass flow rate and pressure drop transient**

Step 6. Evaluate the effects of changing fluid property models

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.
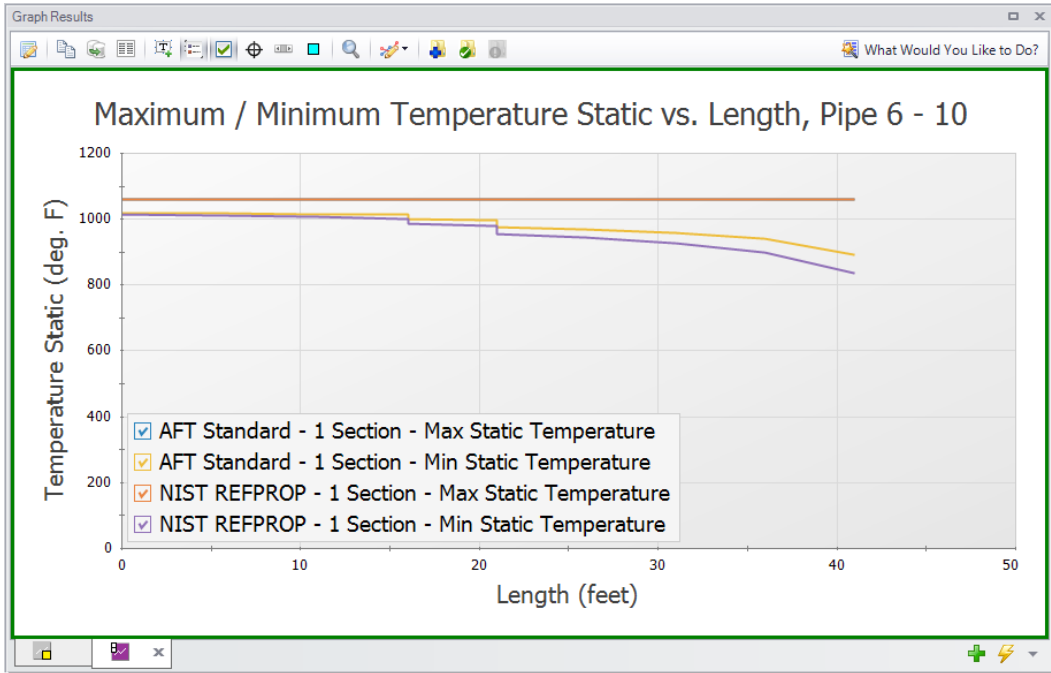
Figure 4.9   Bypass line profile obtained using NIST REFPROP

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 4.1 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 4.1 shows data for runs conducted on the computers at AFT as of publication. Additionally, Table 4.2 provides a library compatibility chart to assist with fluid property model selection.

Table 4.1   Comparison of High Pressure Steam System run times

High Pressure Steam System Run Times	
Fluid Property Model	**Run Time**
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 4.2   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			

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

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 that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, entering pipe and junction properties, and entering transient data. Refer to the Tank Blowdown Example in Chapter 2 for more information on these topics.

Model file

This example uses the following file, which is installed in the Examples folder as part of the AFT 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 Quick Start.xtr*

Problem statement

The inlet valves to two steam turbines are shut simultaneously in a high pressure steam system. As the design engineer, you are tasked with completing a piping stress analysis and need to capture the magnitude of the forces caused by the aforementioned steam transient. Additionally, determine how the valve closure time impacts the forces. Lastly, export the force data to CAESAR II® to check for compliance with standards.

Step 1. Start AFT xStream

Open AFT xStream. Click the Start Building Model button.

Step 2. Open model file

For this example, we will be starting from a pre-built model file for which setup been partially completed to save time. From either the Startup window or the File menu, browse to the model file name shown above. The Workspace should appear as shown in Figure 5.1.
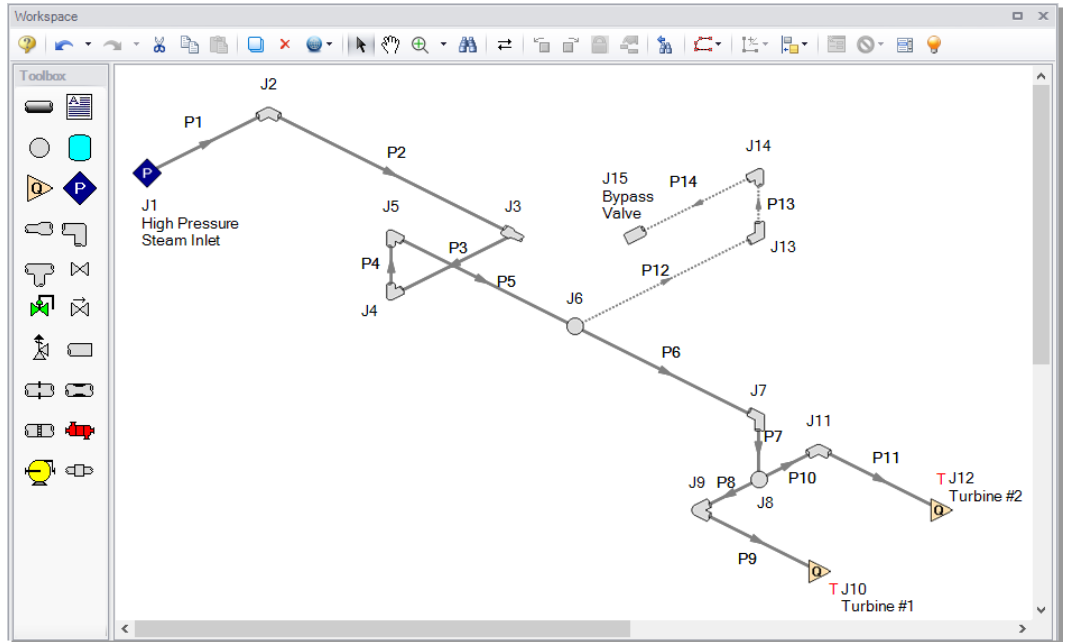
Figure 5.1   Model layout

Step 3. 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.

Step 4. 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 5. 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 5.2). Click Create. This will open the Force Definitions panel of the Analysis Setup window. The Force Definitions panel should appear as shown in Figure 5.3.

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

If a more robust view of the 3D orientation of the pipes is required, directional information can be entered using the optional Force Unit Vector columns. The Force Unit Vectors do not impact AFT xStream's calculations, but may be useful to export the output to piping stress analysis software as discussed in Step 11.

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.
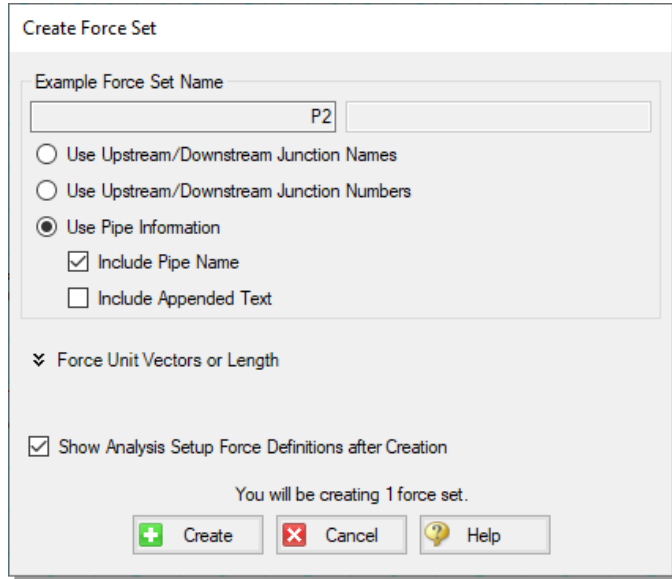
Figure 5.2   Adding force set from Workspace using the right-click menu
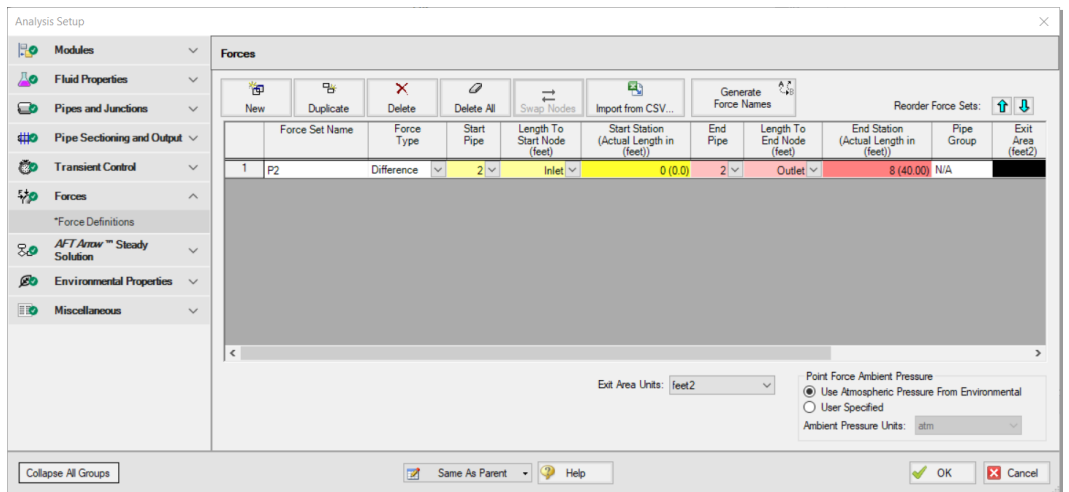
Figure 5.3   High Pressure Steam System Force input in the Analysis Setup window

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 5.4. Click OK to exit the Analysis Setup window.
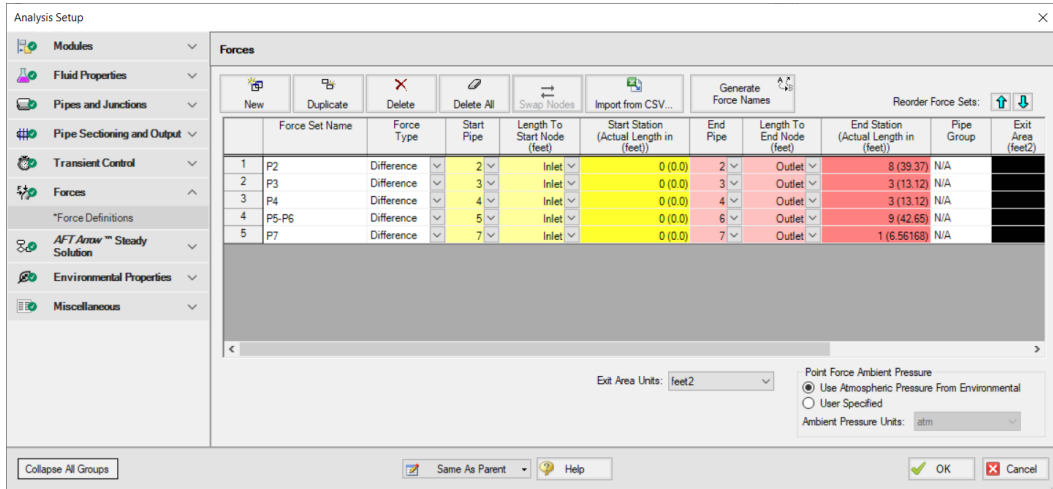
Figure 5.4   Completed Forces panel in the Analysis Setup window

Step 6. 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. 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 5.5.
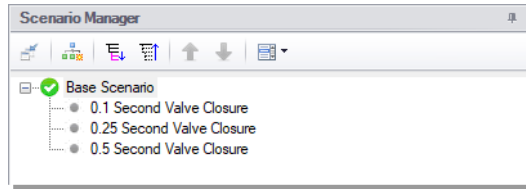
Figure 5.5   Completed Scenario Manager for High Pressure Steam - Forces

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:

Time (seconds)	Mass Flow (lbm/sec)	Static Temperature (deg. F)
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:

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

6. Enter the same Transient Data for Assigned Flow J12

Step 7. 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

3. In the Scenario Comparison - Selection window (Figure 5.6), 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 5.7). 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.
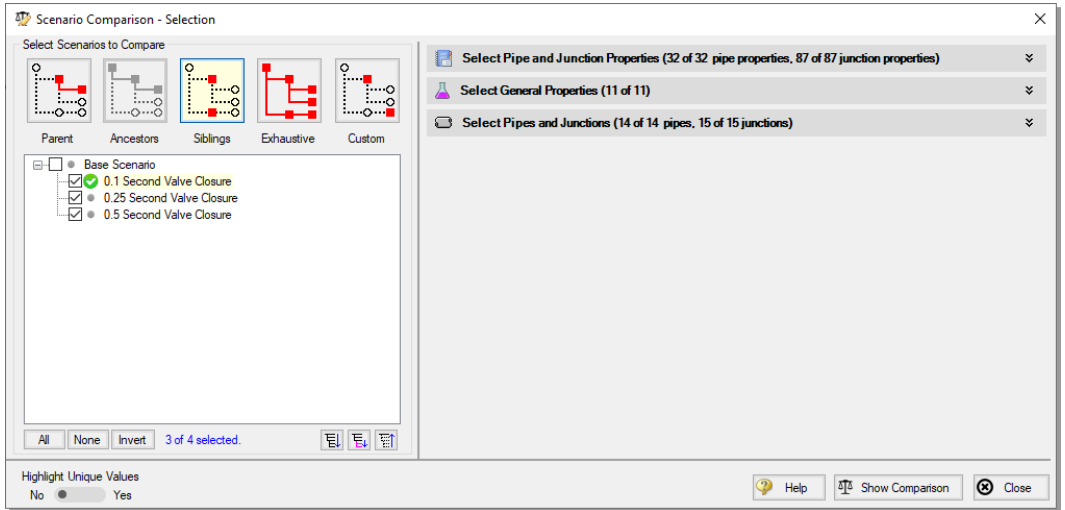
Figure 5.6   Scenario Comparison Tool - Selection window
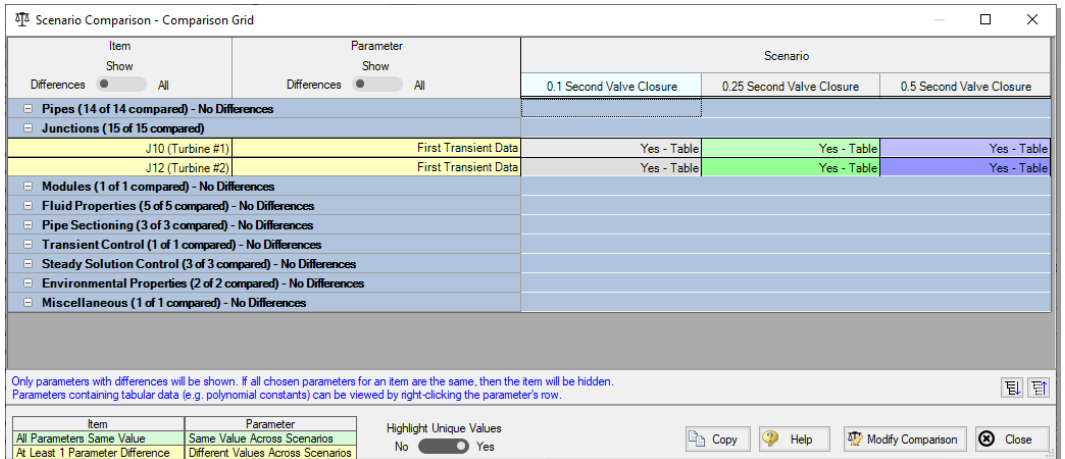
Figure 5.7   Scenario Comparison Grid of the 3 scenarios

Step 8. 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 5.4. 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 9. 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 5.8   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 5.9.

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 5.10.

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.
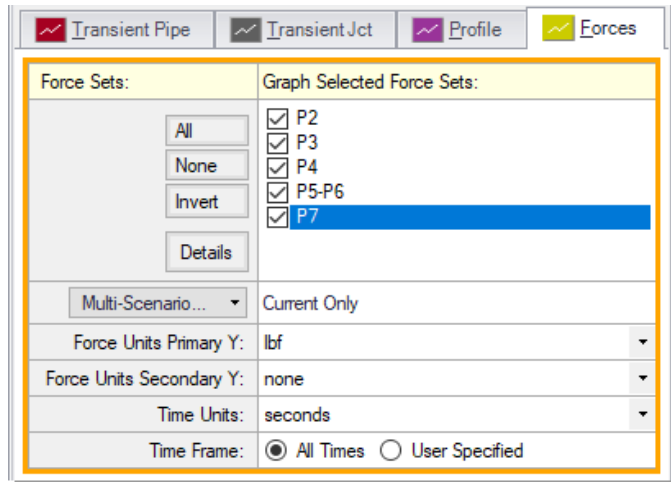
Figure 5.9   Graph parameters for force sets
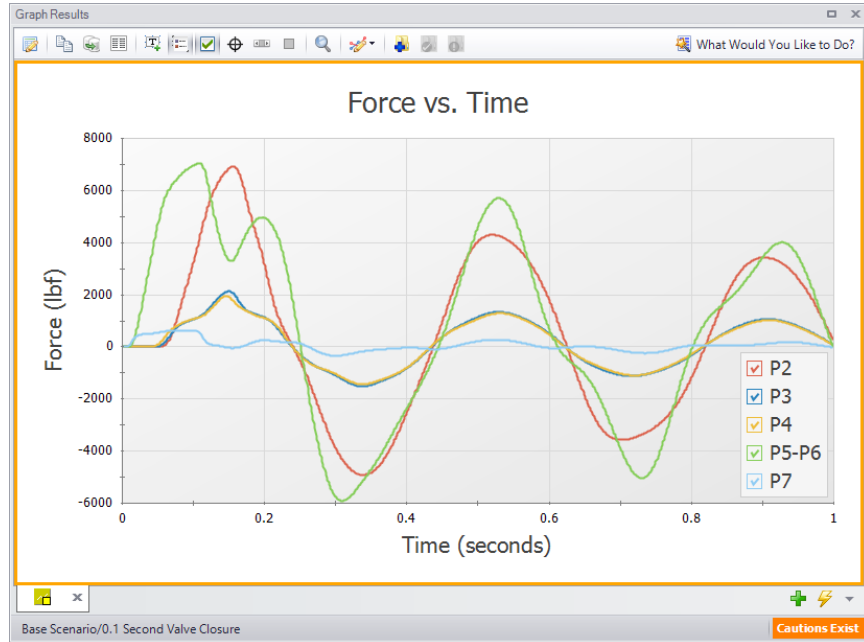
Figure 5.10 Graph for forces of the "0.1 Second Valve Closure" scenario

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 10. 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 5.11-14.

	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 5.11   Forces tab in the Output window for "0.25 Second Valve Closure" scenario

Figure 5.12   Graph for forces of the "0.25 Second Valve Closure" scenario

Figure 5.13   Forces tab in the Output window for the "0.5 Second Valve Closure" scenario
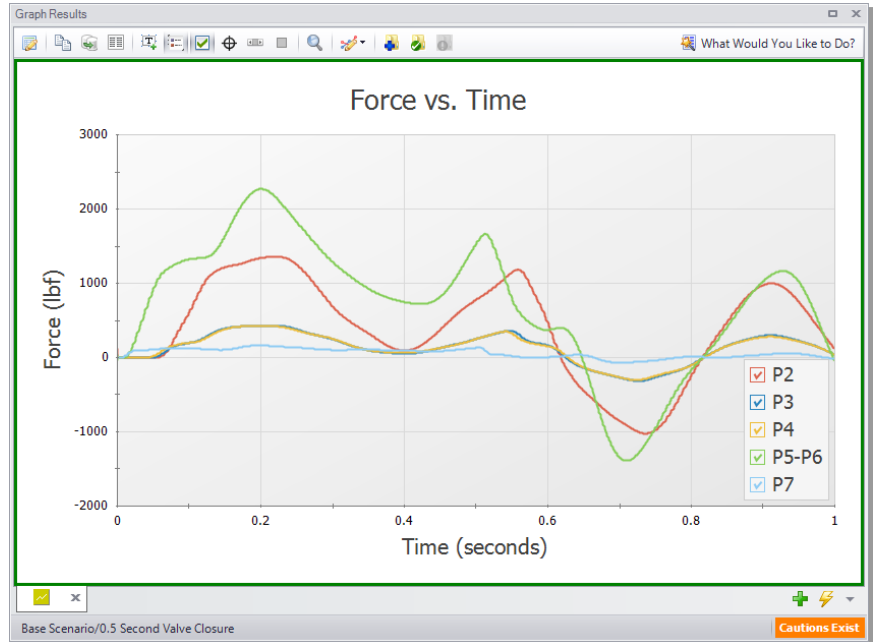
Figure 5.14 **Graph for forces of the "0.5 Second Valve Closure" scenario**

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 5.15-17 so that you may see the impact of changing these three parameters. Note that in Figure 5.16, ASME Steam, Chempak, and NIST REFPROP are superimposed on each other, indicating no meaningful difference for this case.
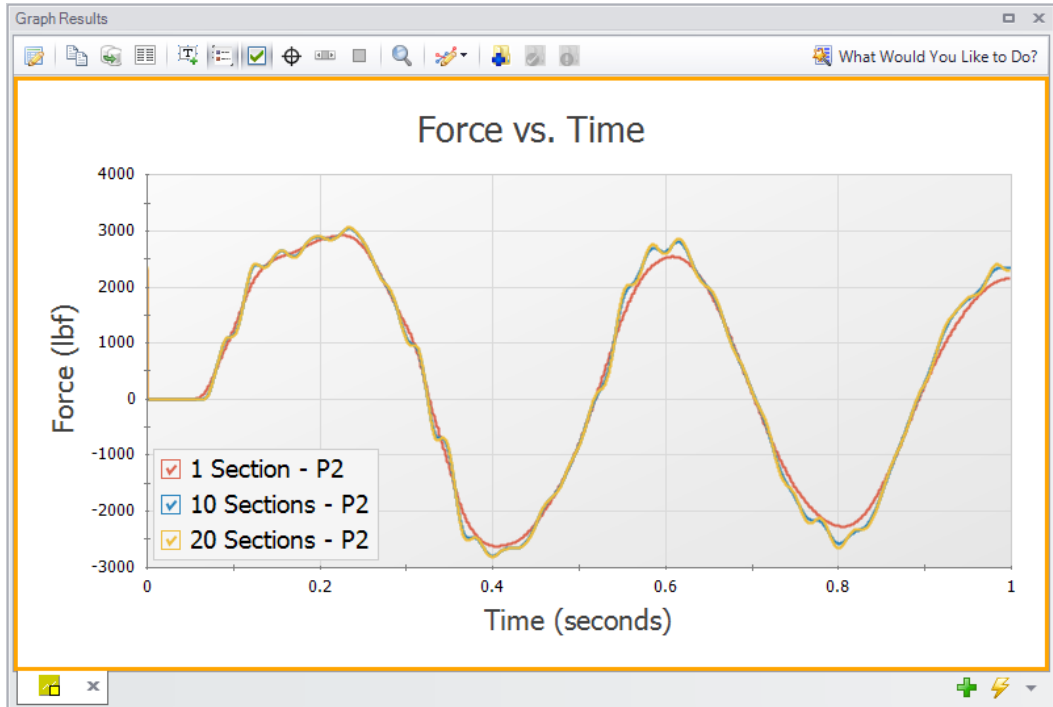
Figure 5.15 **Effect of sectioning on High Pressure Steam - Forces model. All runs use the AFT Standard fluid library and a Maximum Pipe Mach Number During Transient of 1.**
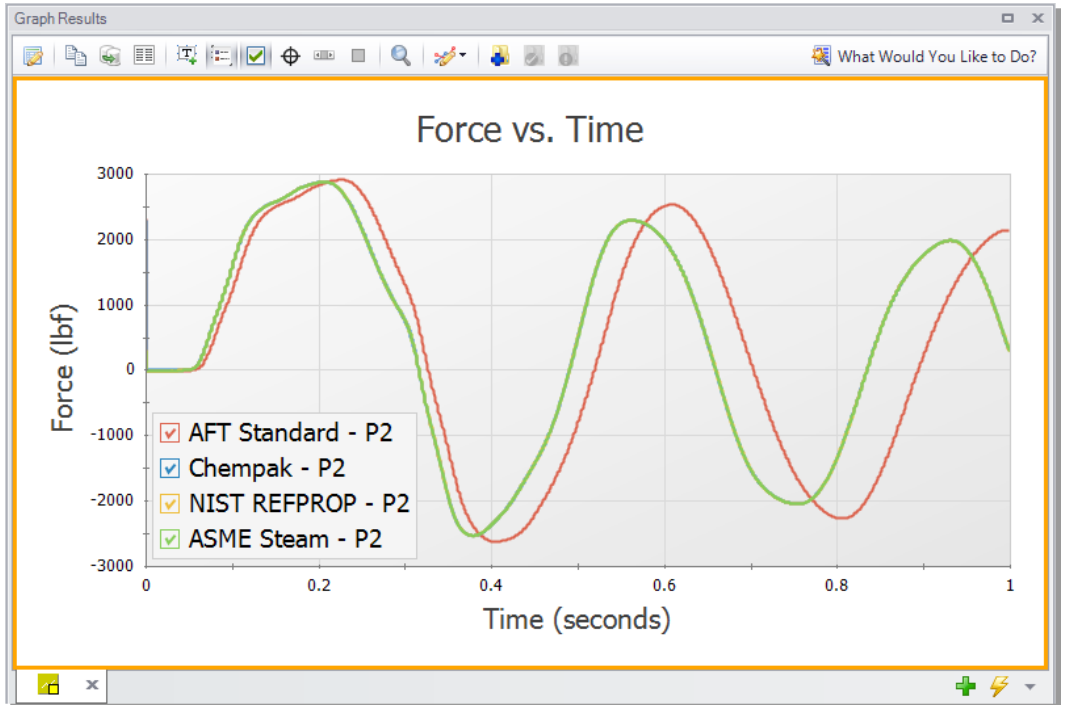
Figure 5.16   Effect of fluid property model on High Pressure Steam - Forces model. All runs use a Minimum Number of Sections Per Pipe of 1 and an Estimated Maximum Pipe Mach Number During Transient of 1.
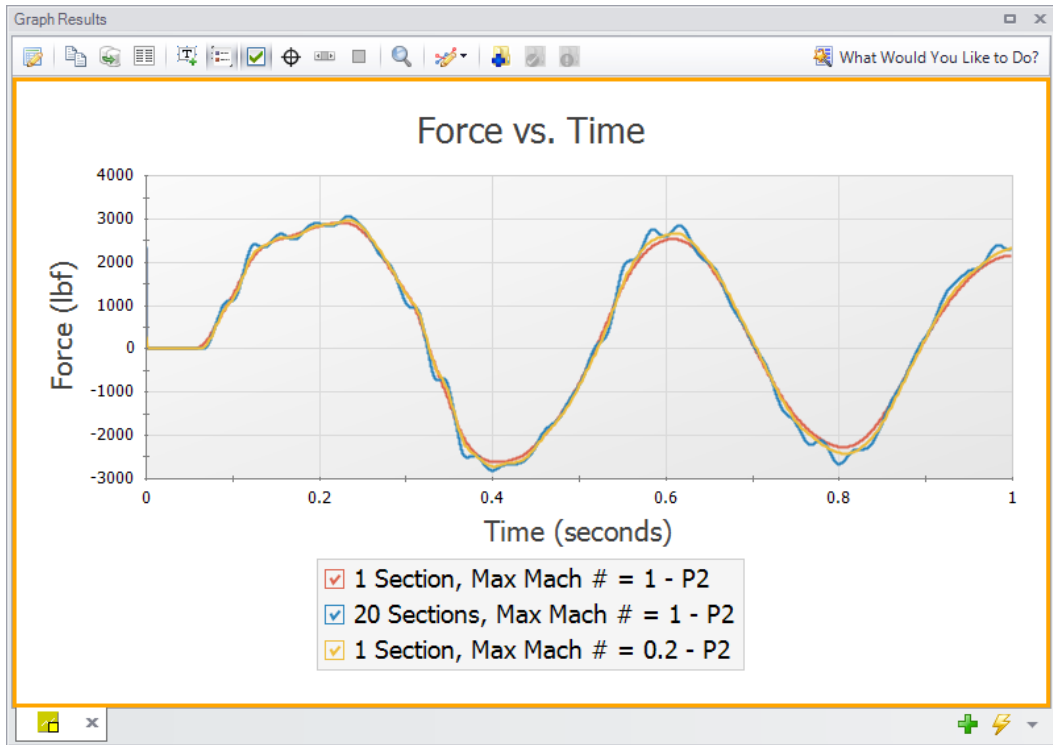
**Figure 5.17   Effect of Maximum Pipe Mach Number During
Transient on High Pressure Steam - Forces model. All
runs use the AFT Standard fluid library.**

Step 11. 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 5.18.
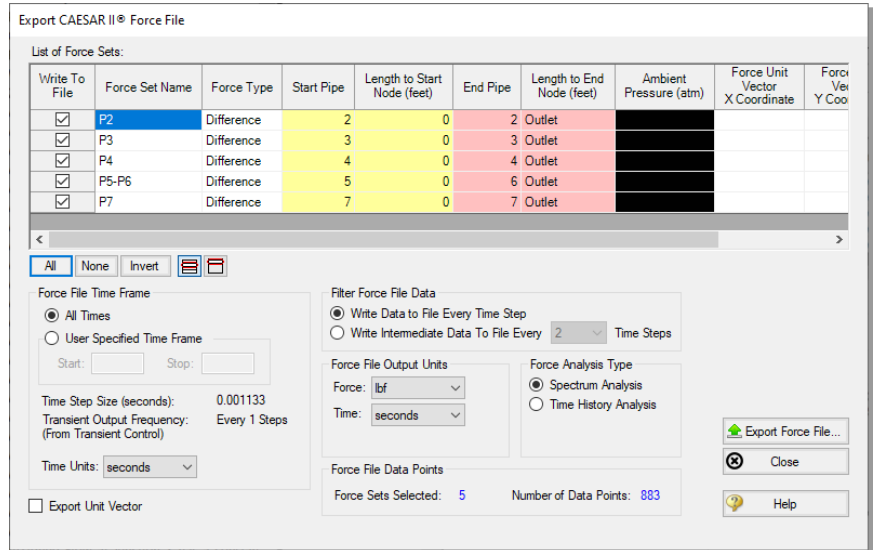
Figure 5.18   Export CAESAR II® Force File window

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.

AFT xStream Add-on Module - PFA Example

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 that the user has some familiarity with AFT xStream such as placing junctions, connecting pipes, entering pipe and junction properties, and entering transient data. Refer to the Tank Blowdown Example in Chapter 2 for more information on these topics.

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*

Problem statement

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

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 6.1.
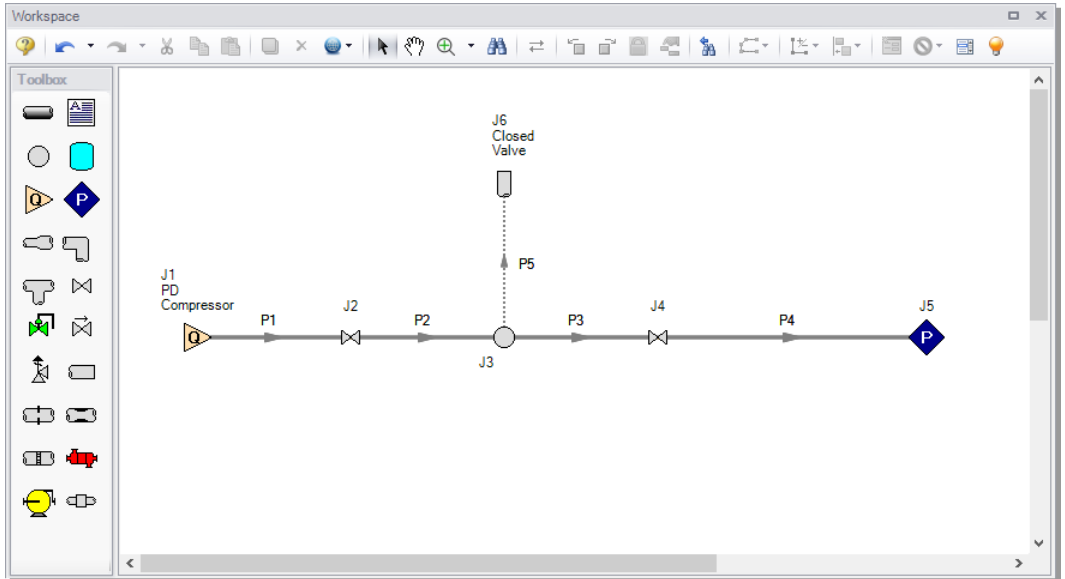
Figure 6.1   Layout of frequency analysis model

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

J1 - Assigned Flow

1.   Name = PD Compressor

2.   Elevation = 12 feet

3.   Mass Flow Rate = 100 lbm/s

4.   Temperature = 100 deg. F

J2 - Valve

1.   Elevation = 12 feet

2.   Cv = 446

3.   Xt = 0.8

J3 - Branch

Elevation = 12 feet

J4 - Valve

1.   Elevation = 0 feet

2.   Cv = 1570

3.   Xt = 0.6

J5 - Assigned Pressure

1.   Elevation = 0 feet

2.   Stagnation Pressure = 0 psig

3.   Temperature = 100 deg. F

J6 - Dead End

1.   Name = Closed Valve

2.   Elevation = 12 feet

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 6.2):

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.
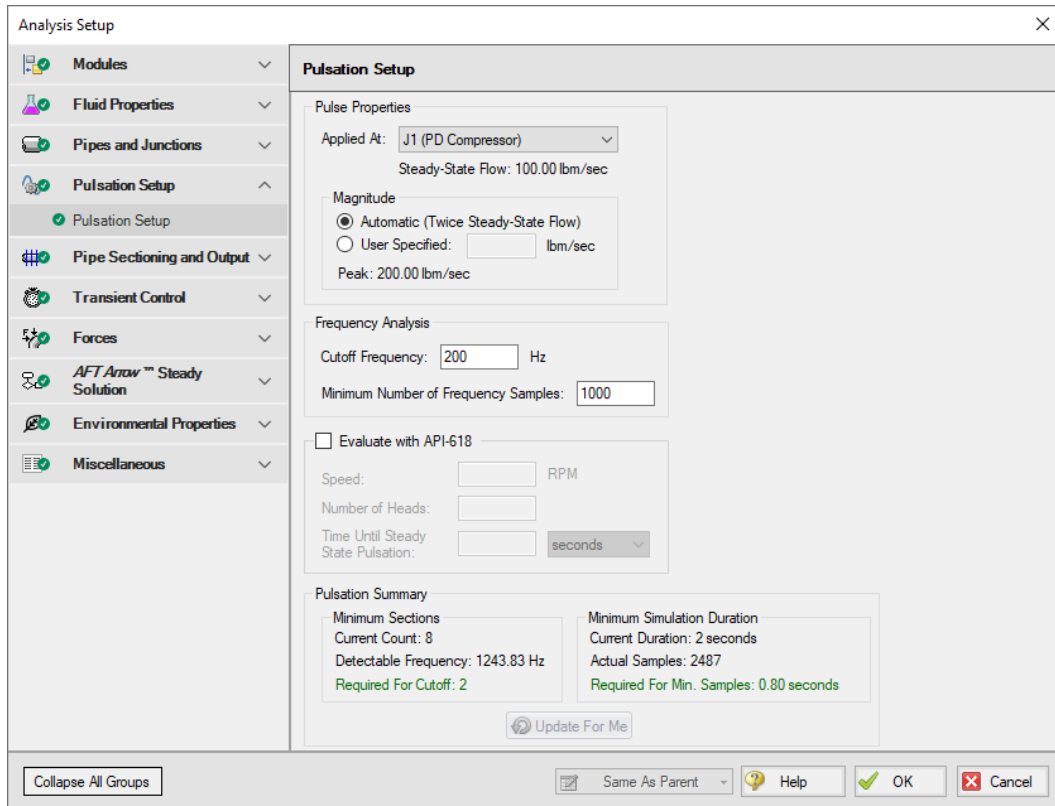
Figure 6.2   Pulse Setup panel in the Analysis Setup window with all pulsation settings defined

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 6.3. 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.
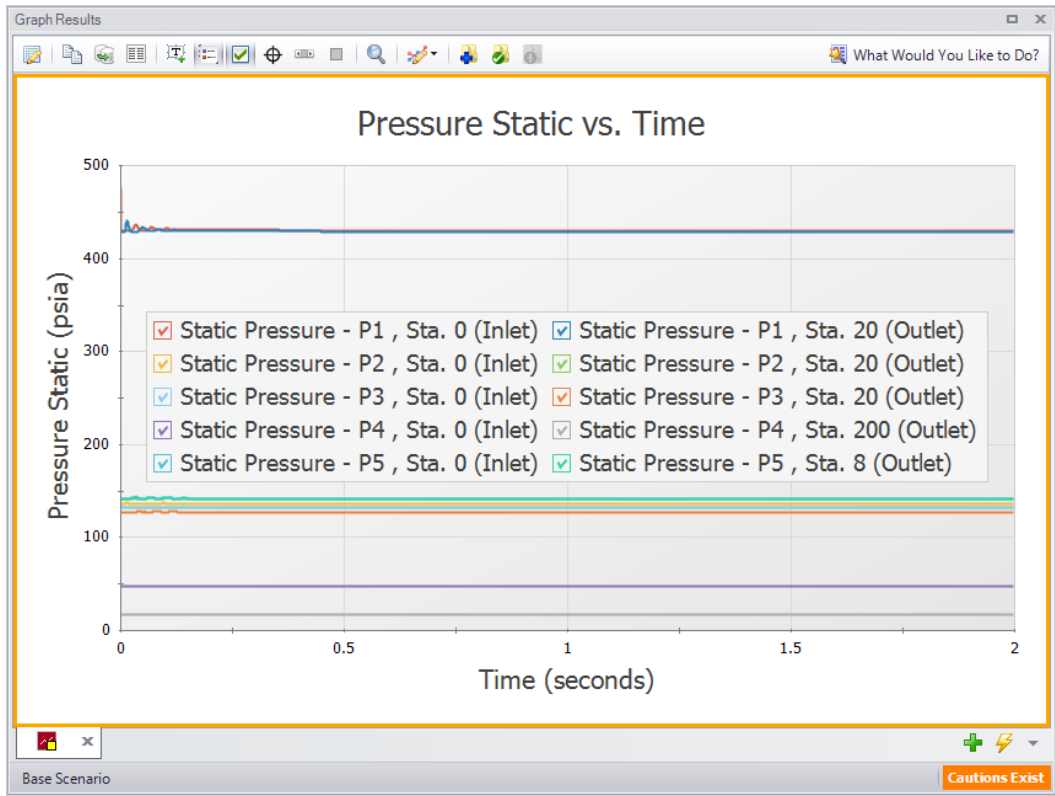
Figure 6.3   Pressure Static in each of the pipes for the frequency analysis

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 6.4)

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
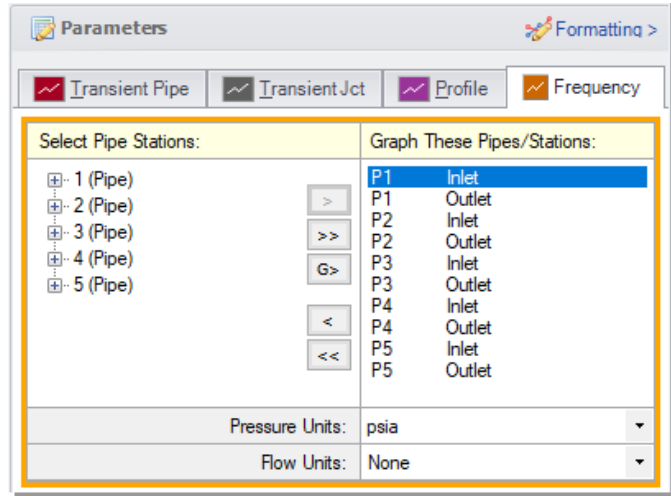
Figure 6.4   Graph Parameters section for the frequency analysis

Identify excitation frequencies to study

From the graph in Figure 6.5, 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.
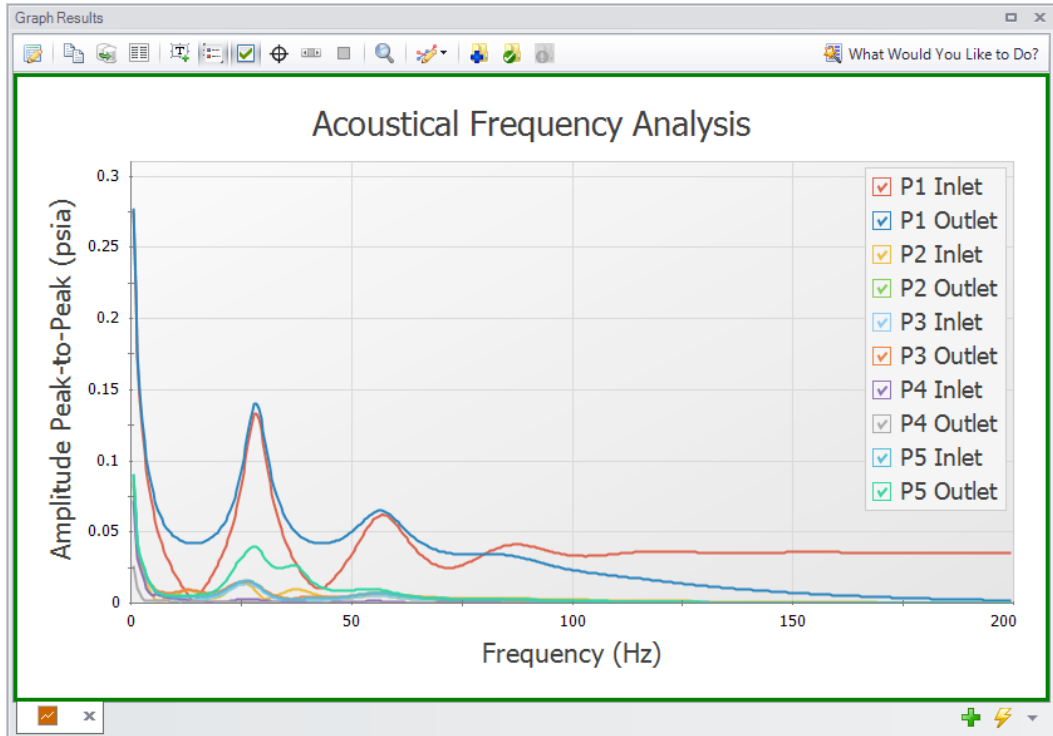
Figure 6.5   Acoustical frequency graph for the positive displacement compressor system

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 6.6).

8. Click OK to close the Properties window.
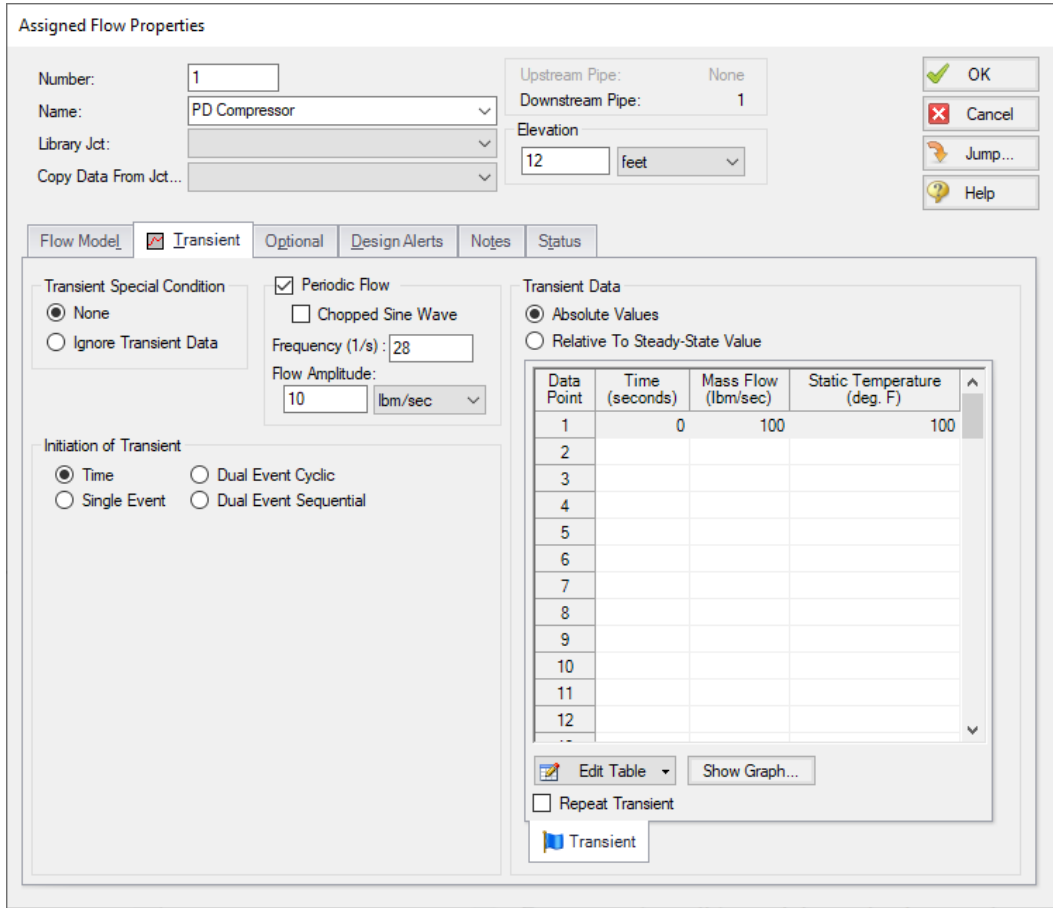
Figure 6.6 **Assigned Flow Properties window with Periodic Flow transient applied**

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 . 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.

In the Graph Results window, generate a graph of the static pressure following the same process used to create Figure 6.3. However, only include the pipe P1 inlet and outlet station. The resulting graph is shown in Figure 6.7. 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 6.7 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 6.8.

Note: The pulsation observed in this model represents a pure sine wave. Real systems might have a different forcing function.
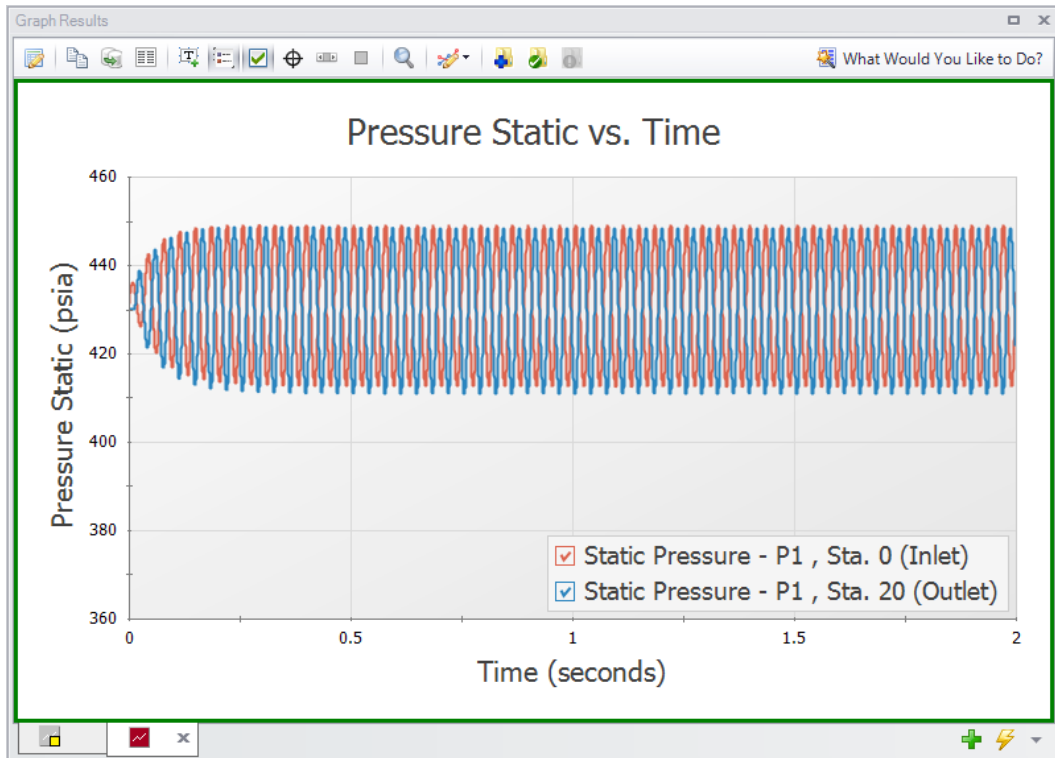
Figure 6.7   Pressure Static response at 29 Hz frequency
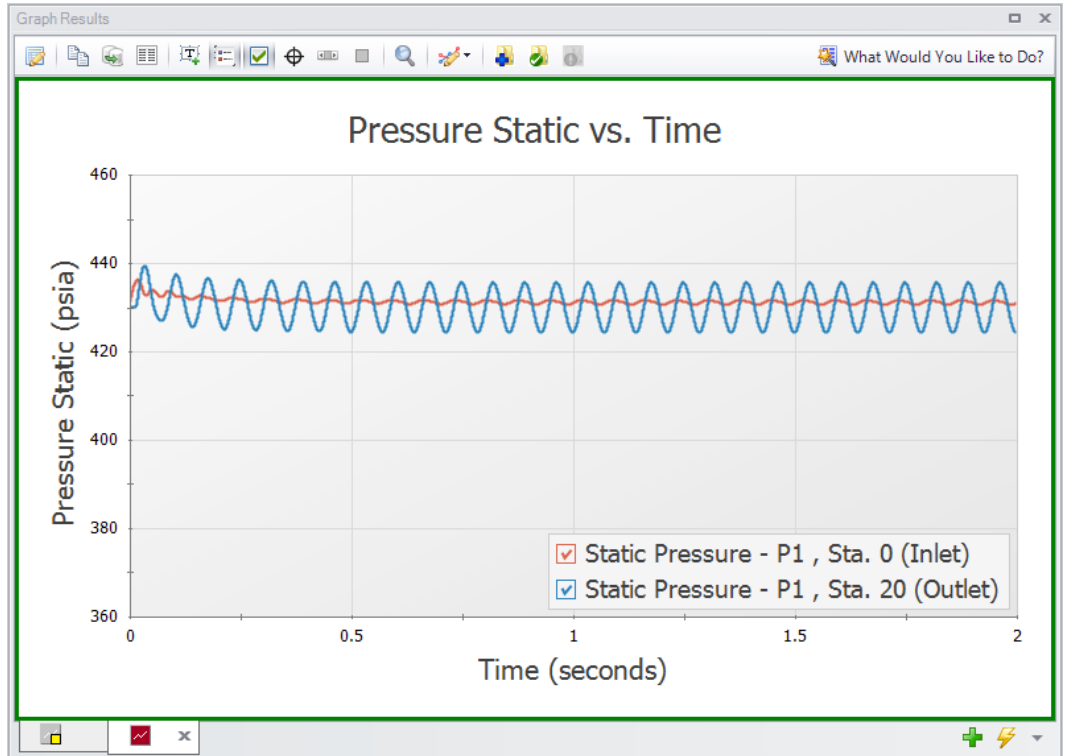
Figure 6.8   Pressure Static response at 14 Hz frequency

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.

Other AFT xStream Capabilities

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

Microsoft Excel® data integration

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

Integration with other software and data standards

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

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

Convective heat transfer

Besides the default adiabatic heat transfer model, pipes can be modeled using convective heat transfer, convective heat transfer with heat flux, or convective heat transfer with heat tracing. When any of the convective heat transfer models are selected conductive heat transfer in the fluid and pipe walls will be accounted for both radially and axially, well as accounting for convective heat transfer at the pipe wall surfaces.

Gas mixtures using NIST REFPROP and Chempak

NIST REFPROP and Chempak allow user-created gas mixtures using any of the pure fluid data provided in their databases. AFT xStream is restricted to non-reacting gas mixture calculations.

The ASME Steam Tables library

The ASME Steam Tables library offers users access to fluid properties obtained from the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam (ISPWS-IF97). This library is selected by choosing the option at the top of the Fluid panel in the Analysis Setup window, at which point it is automatically displayed in the Fluids in Current Model area.

Sonic choking

AFT xStream models sonic choking in complex pipe systems and accurately predicts the resulting choked flow rate during both the steady state and transient simulation. In addition, AFT xStream can accurately calculate multiple choking points in the system.

Positive displacement compressors

AFT xStream can model positive displacement compressors. Typically PD compressors are modeled as a known flowrate over time. Complicated periodic flowrates from multiple pistons and/or multiple out of phase compressors can also be modeled.

Flow and pressure control valves

AFT xStream can model flow and pressure control valves. AFT xStream accounts for control valves which lose and regain control during the transient.

Relief valves

The Relief Valve junction allows for a non-reclosing style relief device to be modeled. The loss through the relief valve can be represented using orifice sizes from API 526.

Repeat transient feature for periodic transient behavior

For elements which have periodic behavior (such as positive displacement compressors), one cycle of data can be entered and then specified to repeat once the cycle is finished.

Elevation changes

AFT xStream uses elevation differences between junctions to calculate gravity driven pressure changes. These elevation driven pressure changes can be especially important for high density gases with moderate elevation changes, pipe systems with significant elevation changes such as those found in overland natural gas pipelines, and gas piping for deep mining operations.

Fitting library

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

Design alerts

Design Alerts can be entered for pipes, then cross-plotted vs. system behavior. A common use is maximum allowed operating pressure.

Network libraries

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

Visual Report animations

The Visual Report window also provides the ability to animate the transient results as a qualitative tool to visualize the behavior in a network. One parameter can be animated using absolute values, values relative to the steady state, or a static map of all maximum/minimum transient values can be generated. The animation can also be recorded to a file.

Index

AFT xStream™ Add-on Module

PFA

Pulsation Frequency Analysis

Helps identify and avoid resonant frequencies in systems, especially in those caused by reciprocating compressors.

For Product Support:

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

AFT
Applied Flow Technology