

# ***AFT xStream***<sup>TM</sup>

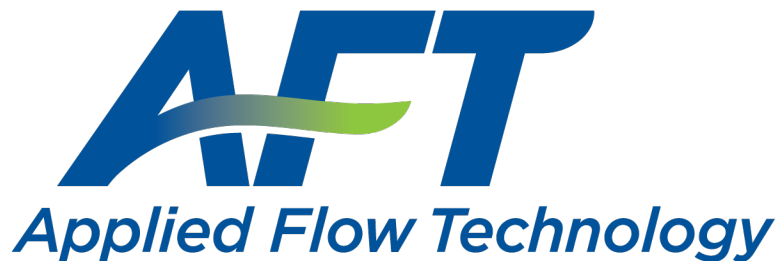
## **Help Guide with Examples**

English Units

**AFT xStream Version 2**

**Gas Transient Modeling in Piping Systems**

Published: January 09, 2024



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

# Contents

---

<b>Overview of AFT xStream</b> .....	<b>17</b>
New in Version 2 .....	18
Interface Improvements for xStream 2 .....	20
xStream Learning Resources .....	22
Engineering Assumptions .....	22
Startup Window .....	23
<b>Overview of Fluid Transients</b> .....	<b>25</b>
Wave Velocity and Communication Time .....	25
Pipe Sectioning - Introduction to Method of Characteristics .....	26
Minimizing Run Time .....	28
<b>Transient Forces</b> .....	<b>30</b>
Defining Forces .....	30
Force Set Types .....	33
Exporting Force Data .....	34
Force Theory .....	35
<b>Interpreting Transient Results</b> .....	<b>39</b>
Animating Graph Results .....	39
Animating Visual Report Results .....	43
Transient Result Accuracy .....	49
Transient Sensitivity Analysis Tutorial .....	51
<b>Fundamental Modeling Concepts</b> .....	<b>58</b>
Pipes and Junctions .....	58
Features for Modeling Irrecoverable Losses .....	59
Convention for Positive Flow Direction .....	59

---

Philosophy of Computer Modeling .....	60
Role of Pressure Junctions and How They Work .....	61
Open vs. Closed Systems .....	64
Curve Fitting .....	66
Stagnation vs. Static Pressure Boundaries .....	68
<b>Analysis Setup Overview .....</b>	<b>71</b>
Modules Group .....	72
Modules Panel .....	72
Fluid Properties Group .....	75
Fluid Panel .....	75
Pipes and Junctions Group .....	90
Pipes and Junctions Panel .....	90
Pulsation Setup Group .....	91
Pulse Setup Panel .....	91
Pipe Sectioning and Output Group .....	95
Pipe Sectioning and Output Panel .....	95
Transient Control Group .....	98
Simulation Mode/Duration Panel .....	98
Transient Solver Initialization Panel .....	98
Forces Group .....	100
Force Definitions Panel .....	101
AFT Arrow Steady Solution Group .....	104
Solution Method Panel .....	104
Tolerance Panel .....	131
Iterations Panel .....	134
Pressure and Temperature Limits Panel .....	137

---

---

Environmental Properties Group .....	137
Environmental Properties Panel .....	137
Miscellaneous Group .....	139
Advanced Parameters .....	139
Advanced Transient Parameters Panel .....	139
<b>Working with Pipe and Junction Objects .....</b>	<b>140</b>
Selecting and Finding Objects .....	140
Selecting Objects .....	140
Finding Workspace Objects .....	143
Groups .....	144
Pipe Drawing Modes .....	144
Snap to Grid .....	145
2D Freeform Grid .....	145
2D Orthogonal .....	146
Isometric .....	147
Global Edit .....	152
Global Pipe Editing .....	152
Global Junction Editing .....	153
Toolbox .....	154
Creating Pipes - the Pipe Drawing Tool .....	155
Joining or Splitting Pipes .....	157
Creating Junctions .....	158
Connecting Objects .....	158
Moving Objects .....	159
Editing Objects .....	160
Morphing Junctions .....	160

---



---

Defining Objects .....	161
Highlight Required Input .....	162
Inspecting Objects .....	162
Notes .....	163
Status .....	163
Workspace Pan .....	164
Moving Pipe and Junction Labels .....	164
Special Conditions .....	165
<b>Pipes .....</b>	<b>166</b>
Pipe Properties Window .....	166
Common Pipe Input Parameters .....	167
Pipe Model .....	168
Fittings & Losses .....	169
Heat Transfer in Pipes Overview .....	171
Convective Heat Transfer .....	173
Heat Tracing Flux Based On Wire Turns And Length.html .....	175
Steady State Heat Transfer In Pipes .....	179
Transient Heat Transfer in Pipes .....	185
Insulation for Pipes .....	188
Optional Input for Pipes .....	188
Add/Remove Pipe Segment .....	189
Reverse Direction .....	190
Pressure Drop in Pipes - Detailed Discussion .....	190
<b>Junctions .....</b>	<b>192</b>
Junction Window Overview .....	192
Two Window Formats .....	194

---

---

Common Junction Input Parameters .....	196
Optional Input .....	197
Specifying Losses .....	198
Conventions for Specifying Base Area .....	198
Reference Density .....	198
Equal Percentage Valves .....	200
Junction Loss Models .....	201
Discharge Coefficient Loss Model .....	201
K Factor Loss Model .....	202
Cv/Kv and XT Loss Model .....	205
Resistance Curve Loss Model .....	208
Defining Junction Transients .....	210
Junction Transient Data .....	210
Initiation of Transients .....	212
Multi-Condition Events .....	216
Area Change .....	219
Area Change Loss Model .....	220
Assigned Flow .....	221
Assigned Flow Transient Theory .....	222
Assigned Pressure .....	223
Assigned Pressure Transient Theory .....	224
Bend .....	225
Bend Loss Model .....	226
Branch .....	228
Branch Transient Theory .....	229
Check Valve .....	230

---

---

Compressor/Fan .....	232
Centrifugal Compressor .....	234
Compressor/Fan Configuration .....	235
Positive Displacement Compressor .....	239
Multiple Configurations .....	240
Affinity Laws .....	243
Transient - Speed vs. Time .....	245
Transient - Setpoint vs. Time .....	246
Control Valve .....	247
Control Valve Transient Theory .....	249
Dead End .....	250
Dead End Transient Theory .....	250
General Component .....	251
Heat Exchanger .....	252
Orifice .....	253
Orifice Loss Model .....	254
Relief Valve .....	256
Relief Valve Transient Theory .....	257
Screen .....	257
Screen Loss Model .....	258
Tank .....	259
Tank Transient Theory .....	261
Tee/Wye .....	262
Tee/Wye Transient Theory .....	262
Valve .....	263
Venturi .....	265

---

---

<b>Interface and User Options</b> .....	<b>267</b>
Menus and Toolbars .....	267
File Menu .....	268
Edit Menu .....	269
View Menu .....	271
Analysis Menu .....	272
Tools Menu .....	272
Library Menu .....	273
Arrange Menu .....	274
Window Menu .....	276
Help Menu .....	276
Toolbars .....	277
Print Preview .....	278
Primary Window Overview .....	279
Status Bar .....	280
Quick Access Panel .....	280
Model Status Light .....	288
Solution Progress Window .....	288
Workspace Window .....	288
Merging Models .....	289
Renumbering Workspace Objects .....	290
Scale/Flip Workspace .....	291
Magnifier Tool .....	291
Annotations .....	292
Extended Model Check .....	292
Keyboard Shortcuts for Workspace .....	293

---

---

Undefined Objects Window .....	295
Raw Data Changes .....	295
Duplicate Special .....	296
Model Data Window .....	296
Model Data Control Window .....	298
Output Window .....	300
Output Control Window .....	306
Output Parameters .....	310
Applied Standards .....	311
Graph Results Window .....	311
Quick Graph Formatting Window .....	316
Graph Types .....	317
Basic Graph Tutorial .....	320
Visual Report Window .....	331
Visual Report Control .....	332
User Options Overview .....	337
Unit System .....	337
Workspace .....	339
Parameters .....	342
Files .....	343
<b>File Import and Export .....</b>	<b>345</b>
Importing Piping Component Files .....	345
Importing GIS Shapefiles .....	348
Importing Caesar II Netral Files .....	350
Importing and Exporting Epanet Files .....	352
Importing and Exporting from Other AFT Products .....	354

---

---

Exporting to Excel .....	355
Importing Excel Data .....	359
<b>Design Alerts .....</b>	<b>364</b>
Design Alerts From Output .....	367
Design Alert Importing & Exporting .....	368
<b>Scenario Manager .....</b>	<b>370</b>
Creating, Organizing, and Editing Scenarios .....	371
Modifying Individual Scenarios .....	374
Scenario Inheritance .....	374
Running Models in Batch Mode .....	377
Open Model Using Last Scenario .....	378
Scenario Comparison Tool Overview .....	378
Accessing Scenario Comparison Tool .....	379
Modify Comparison .....	380
Scenario Comparison Grid .....	381
<b>Add-on Modules .....</b>	<b>384</b>
Introduction to the PFA Module .....	384
Defining Pulsation Settings .....	385
Interpreting Pulsation Results .....	389
PFA Module Glossary .....	389
<b>Libraries .....</b>	<b>391</b>
Library Manager .....	391
Library Browser .....	392
Junction Library .....	394
Fluid Library .....	394
Pipe Material Library .....	395

---

---

Insulation Library .....	397
Fittings & Losses Library .....	398
Network Library Overview .....	403
Benefits of Shared Libraries .....	403
Creating an enterprise-wide, network library system .....	404
Library Verification .....	407
Library Locations in General Preferences .....	408
<b>Theory and References .....</b>	<b>410</b>
Nomenclature .....	410
Review of Compressible Flow Theory .....	412
Transient Theory .....	429
Transient Flow Equations .....	429
Method of Characteristics .....	435
Compatibility Equations .....	441
Transient Sonic Choking .....	443
Compressible Flow Theory in Single Pipes .....	444
Extension of Single-Pipe Methods to Networks .....	444
Fundamental Equations .....	445
Continuity Equation at a Branch .....	446
Energy Equation at a Branch .....	446
Network Implementation .....	447
Verifying Network Solutions .....	448
Sonic Choking .....	449
Sonic Choking Detailed Description .....	450
Subsonic vs. Sonic Pressure Losses .....	454
Stagnation Properties .....	455

---

---

Pipe Material Library References .....	456
References .....	461
Chempak Technical Information - Detailed Discussion .....	463
Role of Pressure Junctions - Detailed Discussion (Long) .....	480
<b>Critical Warnings, Warnings, and Cautions .....</b>	<b>492</b>
Critical Warnings .....	492
Characteristic speed exceeded maximum determined from... .....	493
Global Energy Balance Disabled .....	493
Global Mass Balance Disabled .....	493
Junction X Had Reverse Flow Compressor Head Could Not Be Pre- dicted .....	494
Maximum transient pipe mach number X higher than estimate Y .....	494
Mach Number Exceeded 1 During Simulation .....	494
Results Do Not Represent Converged Solution .....	495
Required flow to supply the pressure control valve at junction X is above sonic flow .....	495
Setting relaxation lower than relative pressure/flow tolerance gives unre- liable results .....	495
Specified flow through/out of junction X is above sonic flow and cannot be supplied .....	496
Warnings .....	496
Automatic Sectioning Did Not Achieve Steady Conservation .....	497
Chempak Temperature/Pressure Outside Valid Range .....	497
Control Valve Open Percentage Outside Valid Range .....	497
Difference in Velocity Exceeds Specified Percent .....	498
Energy may not be balanced at Junction X .....	498
FCV could not control flow at Junction X .....	499
Flow may not be balanced at Junction X .....	499

---



---

Fluid is no longer in the vapor phase because the fluid temperature/pressure is...	499
Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy	500
Junction X Compressor/Fan added negative pressure	500
Junction X Exceeded End-of-Curve Flow Rate	500
Junction X Had Reverse Flow During Transient	500
Maximum system pressure exceeded enthalpy table's maximum pressure of X	501
Maximum Transient Pipe Temperature Higher Than Estimate	501
Maximum Transient Pipe Temperature Significantly Lower Than Estimate	501
MOC Steady Solution Did Not Reach Nominally Steady Before Transient	502
Physical flow area is less than the sonic area at junction X...	502
Pipe/Junction X property calculation exceeded maximum local iteration limit	503
Pipe/Junction X property calculation failed to continue towards convergence	503
Pipe X temperature is outside temperature limits in AFT Arrow Steady Solution Control	504
Pipe X Temperature Outside Valid Fluid Temperature Range	504
PRV could not control pressure at Junction X	504
PSV could not control pressure at Junction X	505
Temperature is below the saturation line for the following. . .	505
Transient choking occurred at Junction X. The set point could not be met.	505
Transient temperature was below fluid minimum temperature	506
Unique Solution Not Possible Relief Valve Cycles	506
Cautions	506

---

---

All transient junction behavior was disabled .....	507
Assigned Flow at Junction X has a constant flow rate during the transient... ..	507
Automatic Sectioning is enabled, but flow increases substantially during the transient .....	507
Check Valve closed at Junction X .....	508
Flow through Junction X is negative. Loss factors may be in error. ....	508
Junction X exceeded maximum event messages .....	508
Junction X is downstream of a choking point, stagnation/static properties cannot be calculated .....	508
Maximum transient pipe Mach number lower than estimate .....	509
Maximum transient pipe Mach number at least 0.2 higher than steady... ..	509
Pipe X length is shorter than the elevation change (between Junction Y and Z) .....	510
Transient data did not extend to Stop Time at Junction X .....	510
Transient temperature within 10 deg. F of minimum temperature .....	510
Transient Temperature within 10 deg. F of Saturation Temperature .....	510
<b>Troubleshooting .....</b>	<b>512</b>
Reducing Run Time .....	513
The model has both very long and very short pipes .....	513
The model has many pipes that can be combined .....	515
The model has an unusually short pipe .....	516
The simulation has a large number of time steps .....	517
Transient Solver Errors .....	518
General error occurred during the transient calculations .....	518
Junction iteration problem .....	519
Possible supersonic flow detected at the connection of Junction X and Pipe Y .....	519

---

---

User Specified AFT Arrow to Resolved allowable mismatch was exceeded .....	519
Diagnostics Window .....	520
Use the Model Data Window .....	521
Transfer Results to Initial Guesses .....	523
Poor Compressor Curve Fits .....	524
Troubleshooting Resistance Curves .....	524
Use the Output Window Sort Feature .....	525
Lower the Flow Rate Relaxation .....	526
Try Absolute Tolerance .....	526
Make Initial Flow Rate Guesses for Pipes .....	526
Change Boundary Conditions .....	527
Turn Off Parts of the Model .....	527
Break a Large Model Into Submodels .....	527
Use an AFT Standard Fluid .....	527
Run the MOC Steady Solution .....	528
Simplify Event Transients .....	528
Limit Properties to Saturation Conditions .....	528
Enable Transient Debug Files .....	529
<b>Technical Support .....</b>	<b>530</b>
Technical Support Options .....	530
Contacting AFT .....	531
Information on Software Installation .....	531
Chempak Database Installation .....	531
Updated Fluids Library .....	533
Special File Control Features .....	534
Converting Models from Earlier Versions .....	534

---

---

<b>Examples Overview</b> .....	<b>535</b>
Tank Blowdown Example (English Units) .....	536
Gas Turbine Fuel System Example .....	575
High Pressure Steam - Fluid Property Models Example .....	590
High Pressure Steam - Forces Example .....	606
Frequency Analysis - PFA Example .....	623
<b>Index</b> .....	<b>634</b>

## Overview of AFT xStream

Welcome to AFT xStream™ - a visual platform for modeling compressible fluid transients in pipe networks!

AFT xStream brings increased productivity to the pipe flow engineer by combining [traditional analysis](#) with an intuitive graphical user interface. By implementing a drag-and-drop method of pipe flow modeling, AFT xStream simplifies the pipe network modeling process.

AFT xStream can show both input data and analysis results in visual form, allowing rapid analysis of a model's validity. Identifying poor assumptions, catching typographical or modeling errors, and rerunning models are all accelerated because of AFT xStream's graphical environment.

Whether your pipe system model will be used to evaluate and improve an existing system or to design a new one, AFT xStream increases your productivity in the modeling process.

## What Can AFT xStream Do?

---

AFT xStream can be used to model a wide variety of compressible flow systems, including:

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Systems with [valve](#) transients
- Systems with [compressor](#) transients
- Systems with pressure or flow [control valve](#) transients
- Multiple design cases in a single model file ([Scenario Manager](#))

AFT xStream's steady-state solution engine is based on standard techniques used for many years in industry. A modified Newton-Raphson method is used to solve the fundamental equations of pipe flow that govern mass, momentum and energy balance. Solutions are obtained by iteration, and matrix methods optimized for speed are employed to obtain convergence. Once a steady-state solution is obtained, AFT xStream simulates the transient using the [Method of Characteristics](#), the most widely used method for solving fluid transient problems.

## Where to Start

---

The best place to start learning about AFT xStream is to work through our example problems found in the [xStream Learning Resources](#).

## Add-on Module Capabilities

---

AFT xStream has one add-on module that extends modeling capabilities, the Pulsation Frequency Analysis module. [Pulsation Frequency Analysis \(PFA\)](#) allows the engineer to predict resonant frequencies in the system and graphically show the frequency response of a system

---

## Who Can Use AFT xStream

---

AFT xStream assumes that the user possesses a good general knowledge of compressible flow theory and has at least a basic understanding of fluid transients. Even the most advanced and easy-to-use software package cannot make up for a lack of fundamental knowledge on the part of the user.

---

## Related Topics

---

- [xStream Learning Resources](#)
- [Primary Window Overview](#)
- [Overview of Fluid Transients](#)

---

## Related Blogs

---

[What Is Transient Analysis?](#)

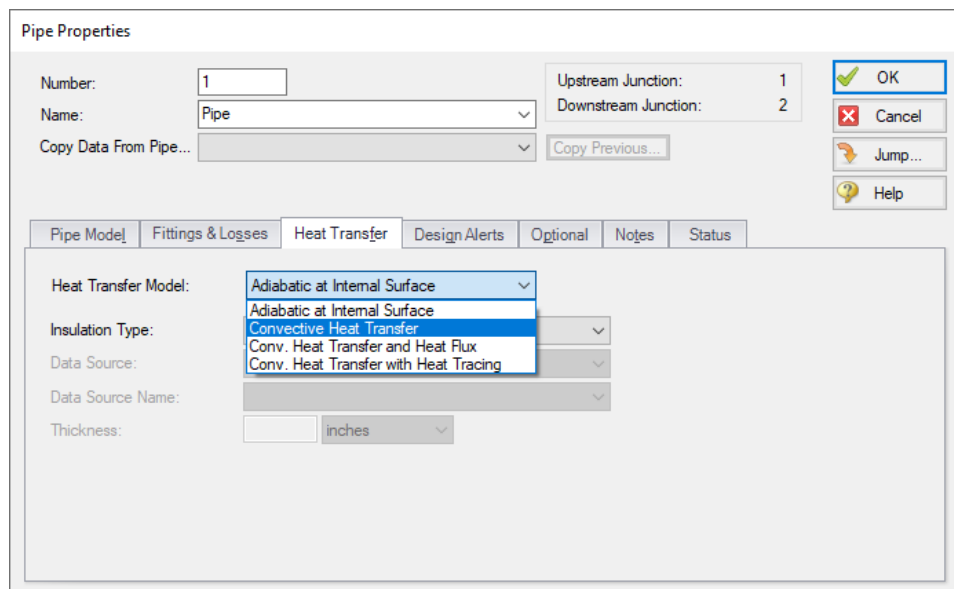
## New in Version 2

---

## Convective Heat Transfer

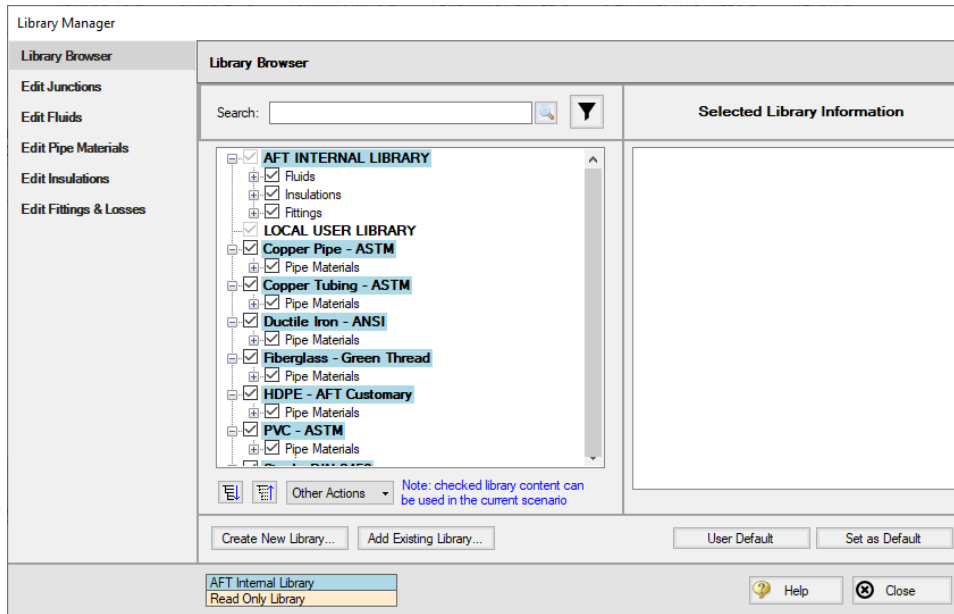
---

[Pipe heat transfer](#) calculation options for convective heat transfer are available for use. Radial and axial conduction are considered inside the pipe when a convective heat transfer model is used.



## Libraries

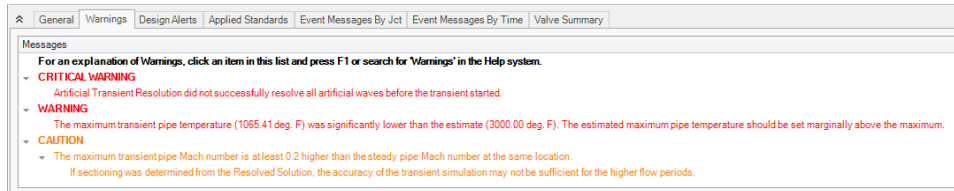
The database feature interface was completely overhauled to improve workability. The new system is called the [Library Manager](#) and significantly improves the ease and workability for users to browse and connect input information, now called libraries, into their model. For a list of differences between the Database and Libraries interfaces, see [Interface Improvements for xStream 2](#).



## Improved Organization of the Warnings in the Output

An improved display of the warnings and design alerts makes it easier to review and understand information in the following ways:

- [Critical Warnings, Warnings, and Cautions](#) are color-coded and grouped by type in collapsible sections
- [Design Alerts](#) are now shown on a separate tab in the General Section
- Double-clicking messages will take user to the related pipe/junction in the Workspace



## More Improvements

---

- **Pipes/Junctions**
  - Several new junctions have been added including the [Bend](#), [Screen](#), and [Venturi](#) junctions
  - Valve transients can be defined as open percent vs time
- **Workspace and Interface**
  - Pipes and Junctions can be set the Same as Parent directly from the Workspace
  - Custom Unit names can be defined in the [Unit System](#) section of the User Options window
  - Pipe/Junction notes can be searched in [Select Special](#)
  - Special Conditions can be directly changed from the toolbar for junctions that have multiple special condition options
- **Output and Reporting**
  - Show the number of warnings and design alerts in the [batch run](#) window
  - [Design alerts](#) are grouped together in the General Output section
  - New output parameters are available for the Relief Valve junction
  - API 618 analysis added for the [Pulsation Frequency Analysis](#) add-on module
- **Miscellaneous**
  - SRK and Peng-Robinson [equations of state](#) are now available for AFT Standard fluids
  - Saturation line data has been added for the AFT Standard fluids
  - List Junctions that have [Special Conditions](#) change in Solution Progress window
  - Silent Batch runs can be enabled to prevent notifications from appearing over other active windows

## Interface Improvements for xStream 2

There were many improvements made to the interface in xStream 2. These changes primarily involve consolidating inputs with the new Analysis Setup window, and updating the functionality of databases, which have been renamed as libraries.

### Analysis Setup Sources in xStream 1

---

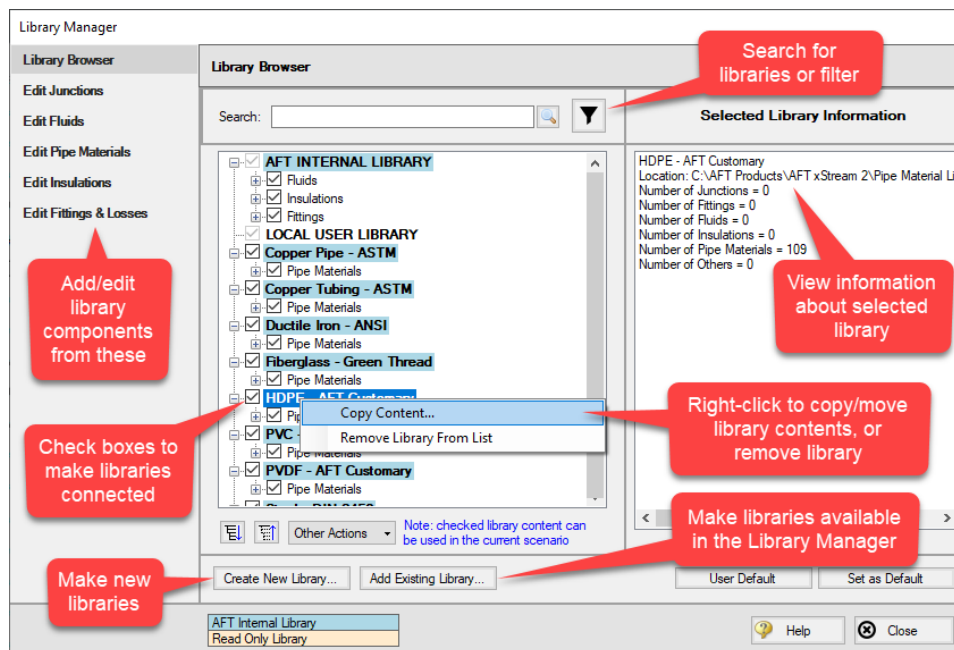
- **Simulation Mode:** The simulation mode settings were moved to the [Simulation Mode/Duration panel](#) in the [Transient Control group](#).
- **AFT Arrow Steady Solution Control:** The Steady Solution Control parameters have primarily moved to the [AFT Arrow Steady Solution Control group](#). The Special Controls have moved to the [Advanced Parameters panel](#) under the Miscellaneous group.
- **MOC Steady Solution:** The MOC Steady Solution panel was renamed to the [Transient Solver Initialization panel](#) and moved to the [Transient Control group](#).



- **Sectioning:** The Sectioning panel has been renamed to the [Pipe Sectioning and Output panel](#) in the Pipe Sectioning and Output group.
- **MOC Transient:** The Start/stop time inputs from this panel were moved to the [Simulation Mode/Duration panel](#). The options to save pipe stations were moved to the [Pipe Sectioning and Output panel](#).
- **List Undefined Objects:** The [undefined objects list](#) has been moved to the [Pipes and Junctions group](#). The undefined objects list can also still be viewed next to the Workspace by opening it from the View menu.
- **Add-on Modules Activation:** The activation window for add-on modules has moved to the [Modules panel](#)
- **Pulsation (PFA module only):** Settings have moved to the [Pulsation Setup group](#).

## Database Improvements

- Databases have been renamed as libraries throughout the interface to better reflect their functionality
- Connecting, creating, and transferring content between libraries can all be done from the [Library Browser](#), rather than switching between tabs in the Database Manager to do this
- Viewing the contents of libraries can also be done directly in the Library Browser, rather than opening a separate Browse Database Content window
- Windows to edit and add components to libraries can all be accessed directly from the Library Manager rather than having to navigate solely through the Library menu



## xStream Learning Resources

### Online Resources

---

Additional resources can be found in the [Learning Center](#) on our website. Resources found there include:

- **Training Seminar information**
- **Free Webinars**
- **Flow Expert Package information**
- **Case Studies**
- **Video Tutorials**
- **Tips & Tricks Blogs**
- **Quick Start Guides**

### Beginner: Tank Blowdown Problem

---

This is a simple example model which is recommended as a starting point for brand new AFT product users: [Beginner: Tank Blowdown Problem](#)

### Walk Through Examples

---

This links to a variety of example problems for beginning to advanced users which showcase different features and applications of AFT xStream: [Walk Through Examples](#)

### Module Walk Through Examples

---

An example using the PFA add-on module of AFT xStream to perform frequency analysis is provided: [Module Walk Through Examples](#)

## Engineering Assumptions

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

- Gas-phase flow only
  - AFT xStream is not meant for analysis of two-phase flow
- One dimensional flow
- No chemical reactions
- The pipes are rigid

- The Method of Characteristics (MOC) solution used here has several significant assumptions that impact accuracy - see the [Method of Characteristics](#) topic for more info
- **Additional assumptions exist for particular modeling components and features, discussed where relevant**

## Startup Window

The Startup Window is displayed when AFT xStream is opened. This window allows the user to quickly access files and resources, as well as define a basic starting point for a new model.

- **Open a Model**
  - **My Files** - AFT xStream keeps a list of recently opened model files. The most recent files can be opened with a single click here.
  - **Other Files** - Browse to any other model file, or open the built-in [Examples](#) folder.
- **Start a New Model**
  - **Activate Modules** - Any licensed [Module](#) can be activated immediately. Checking these boxes will search for an available license.
  - **Select Working Fluid**
    - **I Will Select Fluid Later** - No fluid will be defined, one will need to be defined in [Fluid Panel](#) before running the model.
    - **Select A Recently Used Fluid** - Applies a fluid used in a recent model to the current one.
  - **Modeling Preferences** - Displays various settings for the application, such as specifying the units to use or which [grid](#) to use. These options can be modified later. Click the **Modify** button to reveal the following settings, as shown in Figure 2. Click the **Remember My Preferences and Hide** button to update the modeling preferences for the model and to update the default settings for the application.
    - **Units** - US, Metric, or both sets of units with one as default can be displayed. Additionally, the displayed units can further be reduced to only show common units and those used by a certain industries.
    - **Language** - Select a language to display [Output](#) in.
    - **Interface** - Set up the grid to enable either [2D](#) or [isometric](#) drawing modes.
    - **Default Pipe Material** - Set the [pipe material](#) that is initially selected for new pipes.
- **Resources**
  - **User Setup** - Open the AFT website to view the latest AFT xStream release, or sign up for notifications of new releases via email.
  - **Help Files** - Open our detailed help file (you are here!) or the collection of worked Example problems.
  - **Video Tutorials** - Open the AFT website to our library of instructional videos.

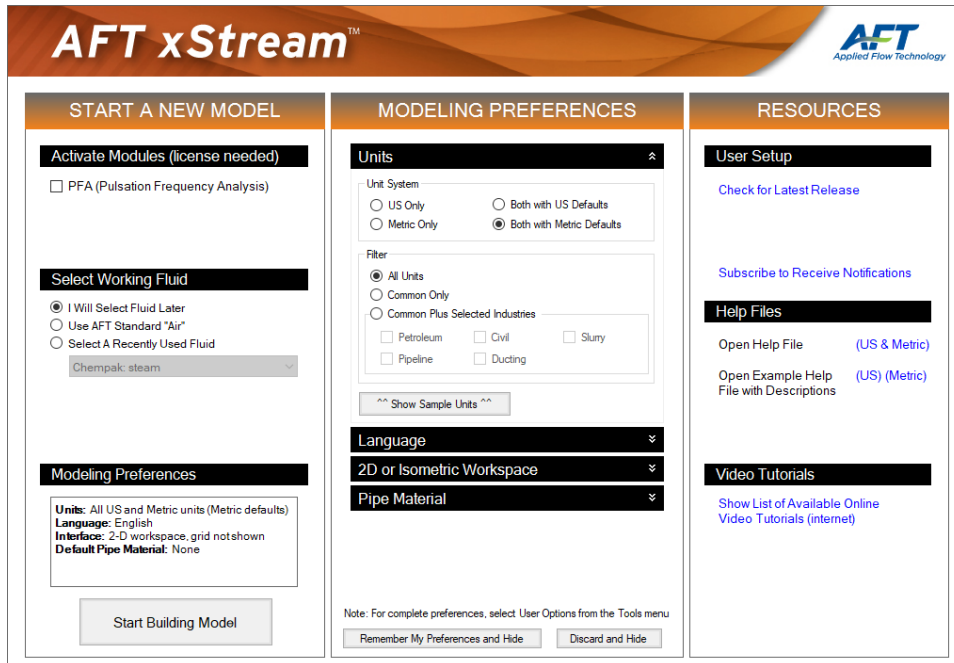


Figure 1: Startup Window in AFT xStream

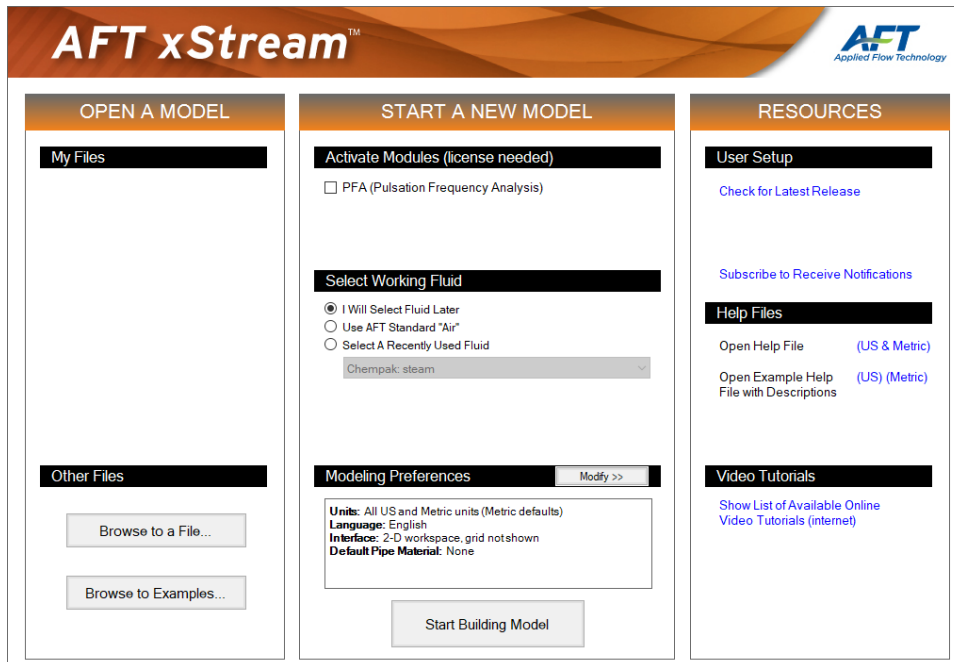


Figure 2: Startup Window showing Modeling Preferences

## Overview of Fluid Transients

A system is steady-state if its [properties](#) - pressure, velocity, etc - do not change with time. Any disturbance to a steady-state system will cause these properties to change - this change is a fluid transient. All fluid transients cause disturbances which propagate throughout the fluid as pressure, velocity and density waves.

Slow acting system changes - such as a tank pressure that changes over the course of a day - can be accurately approximated with a series of steady-state simulations. There are still rapidly-propagating transient pressures in the system, but their magnitude is generally small enough to be considered negligible.

Rapid changes can cause large transient pressures that propagate through the piping network. In extreme cases this causes catastrophic failure of piping and associated equipment. AFT xStream is focused on analyzing rapid changes to the state of a fluid piping system and the resulting transient behaviors via the [Method of Characteristics](#).

## Wave Velocity and Communication Time

### Wave Velocity

---

The wave velocity (sometimes referred to as characteristic speed) for gas transients is the velocity at which the pressure waves will propagate through the gas, as is calculated by the [Method of Characteristics](#). For gas flow the wave velocity is equivalent to the sonic velocity of the gas ( $a$ ) plus the velocity of the gas ( $V$ ). The sonic velocity and velocity will vary with pressure and density.

### Communication Time

---

An important consideration in fluid transient studies is how long it takes a wave to propagate through the system.

The time it takes for a wave to travel from one end of a system to the other is the distance ( $L$ ) divided by the wave velocity. Note that the wave velocity is not constant since the sonic velocity and velocity will vary with pressure and density. However, a minimum communication time can be estimated using the maximum sonic velocity (i.e. sonic velocity at the maximum temperature) and maximum velocity (i.e. sonic velocity), as is discussed in [Pipe Sectioning - Introduction to Method of Characteristics](#).

$$\Delta t_{end-to-end,min} = \frac{L}{2a_{T,max}}$$

When a transient event occurs at one end of a pipe, the wave travels at the wave velocity to the other end of the pipe. However, this is not sufficient time to analyze the problem. While the disturbance has

propagated through the entire system, the point of transient initiation has not yet seen the result of the reflected wave. For the wave to both travel to the end of the pipe and back requires double the above time. This time is known as the Pipe Period or Communication Time.

$$\textit{Minimum Communication Time} = \frac{L}{a_{T,max}}$$

---

## Related Blogs

[A Story About Fluid Communication](#)

### Pipe Sectioning - Introduction to Method of Characteristics

To run any transient simulation in AFT xStream, the pipes in the model must first be sectioned - required for the Method of Characteristics (MOC) solution approach utilized. A brief overview of the impact the Method of Characteristics has on the model is covered here - for a detailed discussion on the mathematics behind the method see: [Method of Characteristics](#).

---

## What is Sectioning?

The Method of Characteristics requires the entire system to be solved every time step. To accomplish this, every section must use a common time step. If the time steps were allowed to vary throughout the model, some portions of the system would be solved "ahead" of others, resulting in a time-distorted solution.

Furthermore, the Method of Characteristics requires that any information propagated from a given calculation point arrives at the neighboring calculation points after one time step. The method cannot allow information to be "lost" between time steps or calculation stations - the distance between calculation points is locked to the duration of a time step.

The time  $\Delta t$  it takes to travel from one point to another is determined by the linear distance  $\Delta x$ , and [wave velocity](#)  $a + V$  between those points:

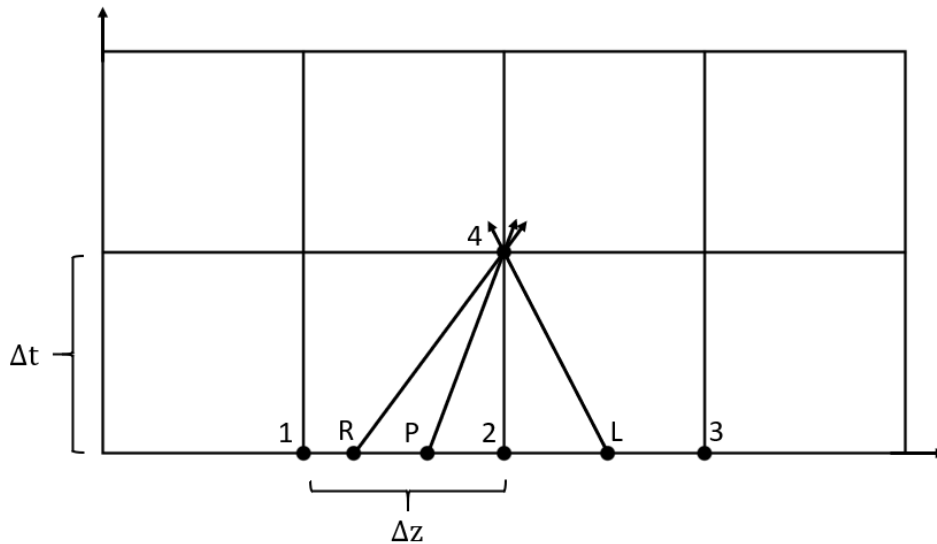
$$\Delta t = \frac{\Delta x}{a + V}$$

For systems with liquid flow the velocity is relatively small and can be neglected, which results in a constant characteristic line slope:

$$\Delta t = \frac{\Delta x}{a}$$

However, for gas flow the velocity cannot be neglected, which means that the characteristic line slope will change throughout the transient. This complicates the choice of a characteristic grid size, since the AFT xStream MOC solution requires a fixed grid. For a fixed time step, the distance the wave travels will vary based on the wave velocity at that time step. Interpolation must be performed between the known grid points to determine the unknown pressure, velocity and temperature at time step  $t$  in order to calculate the pressure, velocity, and density at time step  $t + \Delta t$  at the grid intersection.

For example, consider Figure 1 below where the parameters at point 4 are being determined. At time step  $t$ , points 1, 2, and 3 are known from the previous MOC calculation. However, based on the wave velocity at time step  $t$  the characteristic lines will be unlikely to fall on the known grid points, so interpolation must be performed. Ideally the points R, P, and L will fall between points 1 and 3 on the grid for a stable solution.



**Figure 1: Characteristic grid. Point 4 sees effects from points R, P, and L after some time  $dt$ , which has equal magnitude of  $dx/a$ . Points R, P, and L must be interpolated from know points 1, 2, and 3**

To ensure that grid points 1 and 3 are not exceeded the minimum time step can be approximated using the maximum value of  $a + V$ . Recall that the maximum velocity for gas flow in a pipe is limited to the sonic velocity, and that the maximum sonic velocity will occur at the maximum temperature ( $T,max$ ). Therefore, the minimum time step can be calculated as follows.

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

**Note:** The maximum temperature during the transient will likely be unknown when the model is first run. It is best practice to choose a conservatively large temperature when first running the model. The maximum temperature can be decreased for subsequent runs to increase the accuracy of the results as necessary.

---

Note that the multiplier of 2 for the maximum sonic velocity, known as the courant factor, was derived assuming that the velocity reaches the sonic velocity (Mach number reaches 1). If the Mach number does not reach 1 during the transient simulation, then using two times the maximum sonic velocity will be an overestimate for the maximum velocity. This large maximum velocity allows for a stable solution, but also reduces the accuracy of the solution since points R and L are farther away from the grid points 1 and 3.

For simulations where the Mach number in the pipes remains significantly below one for the whole duration the accuracy can be increased by decreasing the courant factor, which should be equivalent to one plus the maximum pipe Mach number. Calculating the courant factor using a mach number less than the maximum pipe Mach number will result in an unstable solution and may prevent the MOC from converging.

**Note:** In xStream the courant factor is adjusted based on the Estimated Maximum Pipe Mach Number During Transient defined in the [Sectioning panel](#) of Analysis Setup. By default the estimated maximum pipe mach number is fixed at 1, and can only be changed if advanced options are enabled.

---

## Sectioning Effect on Accuracy

---

One of the largest sources of error in xStream results from the interpolations performed to determine the properties at points R, P, and L. To increase the accuracy of calculations the width between the grid points,  $\Delta x$ , can be reduced by increasing the [minimum number of sections](#). For a detailed discussion on determining result accuracy, see the [Transient Results Accuracy](#) topic.

## Sectioning Effect on Run Time

---

As is discussed above, the accuracy of the results in xStream will be closely tied to the number of calculation sections. However, the more calculation stations there are, the longer the required run time will be. xStream simulation times may require hours or days to run, depending on the size of the model and the computational speed of the computer. For information on reducing run times see the [Minimizing Run Time](#) topic.

## Related Topics

---

Method of Characteristics..... 435

### Minimizing Run Time

Models built in AFT xStream can experience long run times, sometimes requiring hours or day to complete. Ultimately, the best method to reduce model run times is to work on a computer with a more

---



powerful processor and with higher read/write speeds.

There are modeling methods which can be used to decrease the run time, though these methods will typically result in some decrease in result accuracy. When using any of the following methods the engineer should always perform a sensitivity analysis to determine what impact these changes have on the accuracy of results.

- Include losses from static components in the pipe [Fittings & Losses](#) tab. This can be done using the feature to [combine pipes](#).
- Artificially increase the length of the shortest pipe in the model. This will decrease the number of sections used in longer pipes in the model, therefore decreasing run time.
- Set the Minimum Number of Sections Per Pipe to 1 in the [Sectioning Panel](#).
- Use the [AFT Standard fluid library](#) for fluid properties. Results obtained using the AFT Standard library should always be compared to results using one of the higher accuracy fluid libraries.

### Related Topics

---

Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51
Reducing Run Time.....	513

## Transient Forces

This section focuses on imbalanced force calculations in xStream. Topics included discuss how xStream calculates forces, what types of force sets can be defined, and how to configure force calculations in xStream.

### Contents

---

<a href="#">Defining Forces</a> .....	30
<a href="#">Force Set Types</a> .....	33
<a href="#">Exporting Force Data</a> .....	34
<a href="#">Force Theory</a> .....	35

## Defining Forces

AFT xStream includes the ability to calculate dynamic, unbalanced pneumatic forces. These unbalanced forces occur when a transient pressure wave passes through a portion of the system.

---

**Note:** As AFT xStream is only a hydraulic solver, only hydraulic forces are accounted for. AFT xStream is not a pipe stress analyzer and as such does not have the ability to directly model piping supports or similar objects. Hydraulic force files can be generated for [import into software](#) designed for force analysis.

---

### Defining Force Sets

---

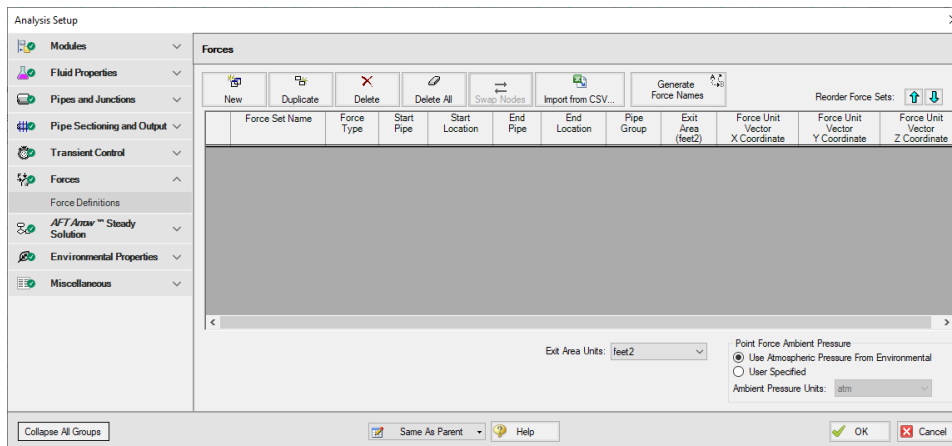
In order to process force data, the locations where the forces are to be calculated must be defined. Force Sets can be defined either from the Workspace or from the Forces panel of the [Analysis Setup window](#).

---

**Note:** Force calculation accuracy in AFT xStream is dependent not only on accurately defining the force sets, but can also be highly dependent on the [Sectioning](#) and [Fluid panel](#) settings. This is discussed further in the [High Pressure Steam - Forces example](#).

---

#### Defining from Analysis Setup



**Figure 1: Force set data is defined on the Force Definitions panel in the Analysis Setup window**

- **New** - Add a new force set to the table
- **Duplicate** - Copy an existing force set
- **Delete/Delete All** - Remove force sets
- **Swap Nodes** - The calculation will be the same, but the sign of the force will be reversed. This may be important for proper interpretation of results or for exporting data to other applications.
- **Import from CSV** - If force sets are defined in another application or by hand they can easily be imported with a comma separated value file. Click the Import from CSV button for an explanation on the required format.
- **Generate Force Names** - Automatically changes the names for the selected force sets to "upstream junction name - downstream junction name" using the names from the connected junctions on the workspace.
- **Point Force Ambient Pressure** - If using a Point force set, the ambient pressure is required to calculate the force balance. This defaults to the [Environmental Properties](#) value, but can be changed here.

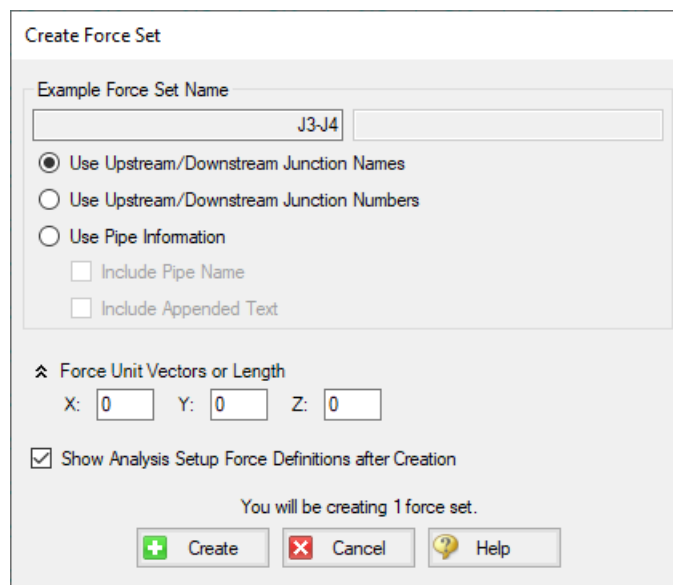
Each individual force set has a number of required parameters:

- **Apply** - Whether to include the force set calculation. This allows disabling force sets without deleting them.
- **Force Type** - There are several force set types available. They are described in detail in [Force Set Types](#).
- **Force Set Name** - A unique name for the force set.
- **Start Pipe** - Where the first node is located.
- **Start Location** - Force Sets in xStream must have their endpoints at either the inlet or outlet of a pipe.
- **End Pipe** - The second (for difference types) node. Note that the selection must be valid for the specified force type. See [Force Set Types](#).

- **End Location** - Force Sets in xStream must have their endpoints at either the inlet or outlet of a pipe.
- **Pipe Group** - For force sets across 3+ pipes, a group is required and is selected here.
- **Exit Area** - For Exit type forces, an exit area is required.
- **Force Unit Vector Coordinates** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. xStream is still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the [Export Force Data](#) tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

### Defining from the Workspace

Force Sets for individual pipes can be defined directly from the workspace by right-clicking on a selected pipe. Choose the Create Force Set With Selected Pipe... option to launch the Create Force Set window, shown below in Figure 2. If multiple pipes are selected, an individual Difference Force Set will be created for each pipe applying the specified settings for each.



**Figure 2: Create Force Set window in the Workspace**

Each force has several required and optional parameters:

- **Pipe Name** - If the pipe has a name entered other than 'Pipe', the user can choose whether to include that pipe name or not in the Force Set name.
- **Force Set Name** - Users can add a name for the Force Set to better organize and manage Force Sets in the model.
- **Force Unit Vectors or Length** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. xStream is still only a one-dimensional fluid modeling

software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the [Export Force Data](#) tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

- **Show Analysis Setup Force Panel after Creation** - If checked the user will be brought to the Force panel of the Analysis Setup window to view all Force Set information defined for that scenario.

---

## Related Topics

Force Definitions Panel.....	101
------------------------------	-----

---

## Related Examples

High Pressure Steam - Forces Example .....	606
--	-----

## Force Set Types

There are several types of forces that can be [defined](#) for analysis with AFT xStream:

- **Point** - Point forces are simply the difference in pressure between the fluid and ambient multiplied by the pipe area. This can be useful for places where the fluid is leaving the system, such as a relief valve or exit valve.
- **Difference** - Difference forces are defined as the unbalanced force between two nodes, or [computation stations](#) along two sequential pipes. An example of this is the pneumatic loading between two elbows along the length of a pipe. See Figure 1 below.
- **Difference Exit** - Similar to the difference force set in that two nodes are required. However, the user defined exit area will be used for calculating the force across the section. This is useful for modeling forces on something such as a nozzle.
- **Difference (3+ Pipes)** - The same as difference forces, but spans across more than two pipes. A pipe [group](#) must be defined that contains the pipes between the two locations. The Difference (3+ Pipes) force set type must still include only sequential pipes.
- **Difference Exit (3+ Pipes)** - The same as difference exit, but spans across more than two pipes, requiring a group.

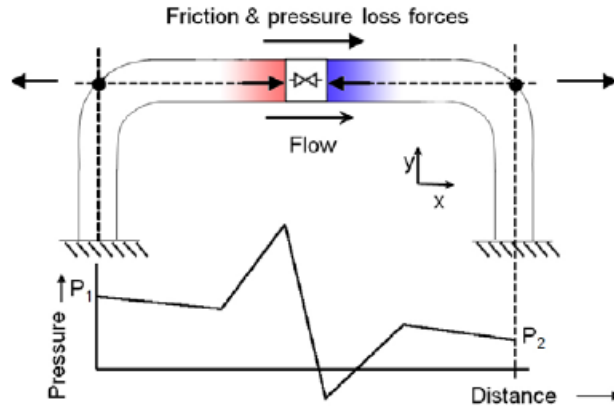


Figure 1: Difference force type schematic. For Difference Exit, the area used at End Node is user defined

## Related Topics

Force Definitions Panel.....101

## Related Examples

High Pressure Steam - Forces Example.....606

## Exporting Force Data

The transient force data can be exported by selecting the Export Force File from the File menu while viewing the Output window.

CAESAR II, TRIFLEX, ROHR2, and AutoPIPE Force Files can be selected.

The time frame of interest, quantity of data points, force balance components, and units can be customized. Note that the units for CAESAR II files may only be set to lbf or N as these are the only units accepted by CAESAR II.

**Note:** Force Set unit vectors can only be exported to CAESAR II (v12 and higher) and AutoPIPE. To export any Force Sets to AutoPIPE, users must have defined force unit vectors for each Force Set.

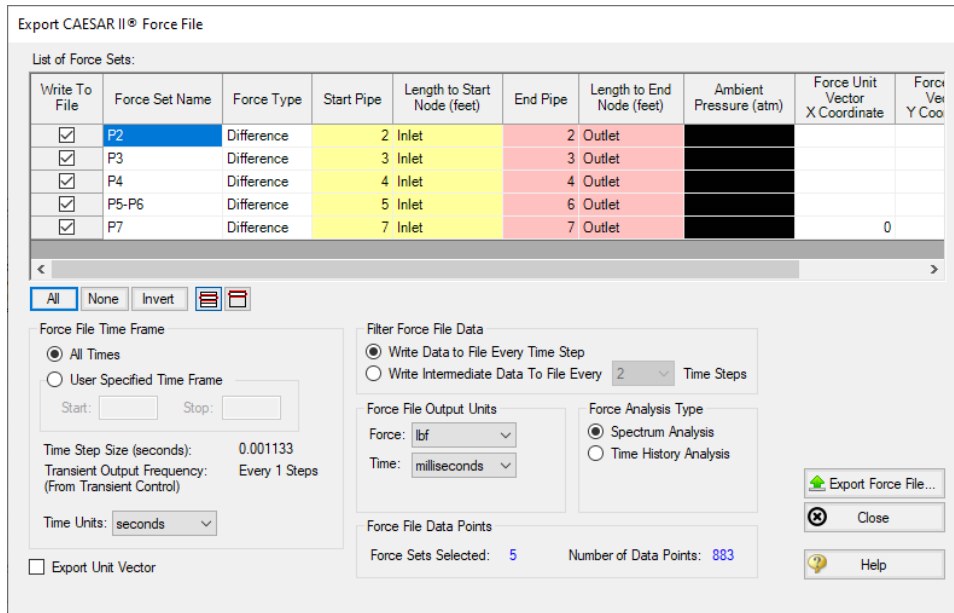


Figure 1: Exporting force results to a Caesar II Force File

**Related Topics**

Force Definitions Panel.....101

**Related Examples**

High Pressure Steam - Forces Example .....606

**Force Theory**

The force solution methods in AFT xStream are intended to calculate the unbalanced pneumatic loading on a series of axial aligned pipes rather than the total forces on those same pipes. Thus, most systems will see steady-state forces calculated as zero. However, systems such as an exit nozzle will see a non-zero steady state force, since there is a hydraulic force imbalance across that valve even at a steady flow rate.

It is important to keep in mind that AFT xStream is not a tool built for the analysis of pipe stress. In real systems, the configuration and location of supports, the deflection of pipes and components, and other effects of dynamic loading should be considered. All piping and components in AFT xStream are considered to be perfectly fixed in time and space, unless directed to change in a specified way, such as a valve closure. The forces that can be calculated by a hydraulic solver such as AFT xStream are only unbalanced hydraulic forces - not forces or stresses in pipe walls or supports.

Considering the above, the calculations for forces proceed under the following assumptions:

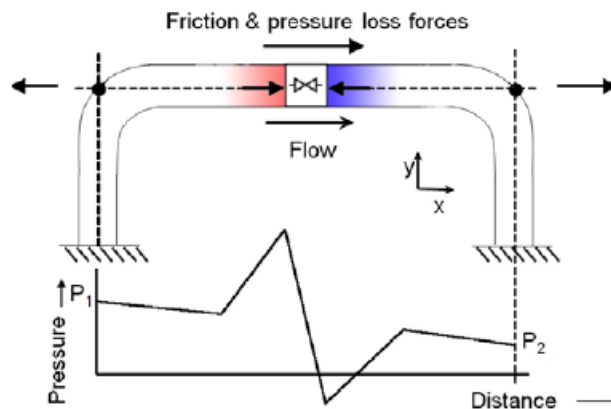
- All piping and components are assumed to be completely rigid. No deflections or deformations are considered.
- The force due to fluid and piping weight is neglected.
- The volume of any junctions (bends, valves, etc.) is considered negligible. This is consistent with how junctions are modeled throughout AFT xStream.
- All Difference Force sets comprise a pipe or set of pipes that are axially aligned between the beginning or ending nodes.
- Both beginning and ending nodes for a Difference force set type are assumed to be 90 degree elbows.
- The total force acting on the piping is assumed to be equal and opposite to the forces acting on the fluid.

**Note:** Forces calculated in xStream are always referenced as defined by the force set inlet and outlet. The reference direction for each pipe does not affect the forces calculated.

A complementary resource to this discussion is an AFT whitepaper titled *Evaluating Dynamic Loads in Piping Systems Caused by Waterhammer*, available [on our website](#).

### Difference Force Set Calculations

A Difference Force Set effectively represents the force between two elbows along a pipe. An example case for a difference force set can be considered for a stretch of pipe at constant elevation with a valve in the middle, as depicted in Figure 1 below.



**Figure 1: Simplified force diagram for fluid flow in a pipe**

With flow through such a pipe, multiple forces act on the fluid, including friction, pressure, gravity (which AFT xStream neglects for force set calculations), normal forces from the pipe to the fluid, and forces due to changes in fluid momentum as flow is redirected through the elbows. Only forces which are axially aligned with the pipe will be considered.



Newton's second law relates these forces to the acceleration of the fluid, using the equation  $F=m*a$ . xStream uses this equation to determine the overall reaction force from the change in fluid velocity at each time step. Equation 1 below shows how xStream determines the force from the fluid acceleration.

$$F_x = \sum_0^N \frac{\partial \bar{m}}{\partial t} \delta x \quad (1)$$

Using Equation 1 above, xStream takes the change in mass flow rate between the previous time step and the current time step to determine the change in velocity (and thus acceleration) of the fluid in each section to give the unbalanced force in each section. The total force for each pipe is then given by the sum of forces for all sections in a pipe.

In the steady state system, the total unbalanced hydraulic forces sum to zero, as the fluid has no acceleration, and thus no net forces acting on it.

### Point Force Set Calculations

---

Point and Exit Force Sets in xStream represent the force caused by the pressure differential between the fluid and the ambient conditions. The Point force uses Equation 2 below. Note that this force is assumed to be applied over the flow area of the pipe. This assumption may not always be valid. For example, a blind flange would see the entire Point force, but a smaller tap on the side wall of a pipe would not.

$$F_{point} = (P - P_{amb}) A_{flow} \quad (2)$$

### Exit Force Set Calculations

---

The Exit force captures the same fluid acceleration as the Difference Force Set, but also captures the force due to fluid exiting the system at the end of the force set. To do so, xStream accounts for the momentum change through the exit, as shown in Equation 3 below. Here,  $N$  is the final pipe section of the force set, connected to the exit.

$$F_x = \sum_0^N \frac{\partial \bar{m}}{\partial t} \delta x + \dot{m}_N V_N \left( \frac{A_N}{A_{exit}} \right) \quad (3)$$

**Note:** Users of AFT Impulse may be familiar with the component-based approach Impulse uses to determine the total unbalanced hydraulic forces acting on a system. That approach has an implied assumption of constant density. Since the assumption of constant density in xStream does not hold true, xStream uses the approaches described above instead. If the approach implemented in xStream is used on an incompressible fluid in Impulse, the resulting forces are the same as those calculated using the component-based approach in Impulse.

---

### Related Topics

---

Force Definitions Panel.....	101
------------------------------	-----

### Related Examples

---

High Pressure Steam - Forces Example .....	606
--	-----

## Interpreting Transient Results

This section discusses how to begin understanding complex transient results. Topics included discuss how to use the output animation features in the Graph Results and Visual Report windows, and how to interpret transient results accuracy.

### Contents

---

<a href="#">Animating Graph Results</a> .....	39
<a href="#">Animating Visual Report Results</a> .....	43
<a href="#">Transient Result Accuracy</a> .....	49
<a href="#">Transient Sensitivity Analysis Tutorial</a> .....	51

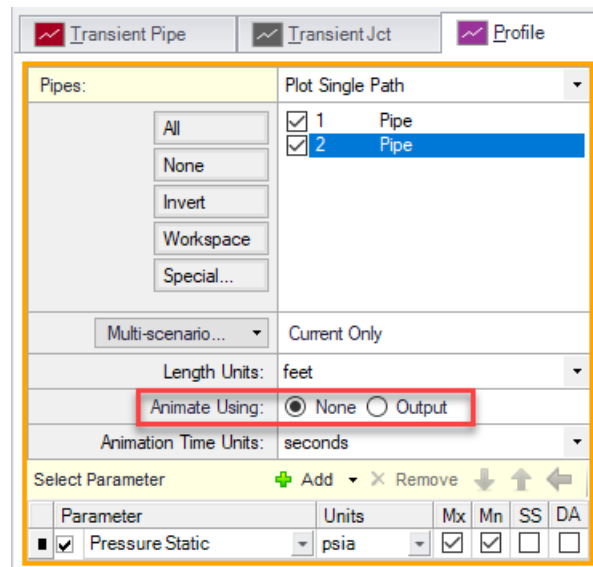
## Animating Graph Results

Animating the results of a transient simulation is a powerful way to understand how a system is behaving. Plots of fluid parameters at a given location over time are valuable and easy to interpret, but they cannot directly convey the most important aspect of a fluid transient event: how the pressure and flow waves propagate through a system and interact with one another. Animating the results of a transient simulation allows these effects to be observed directly. Animations can also be created for [Multiple Scenarios](#).

### Creating an Animated Graph

---

Since animations display information along a flow path, only Profile type graphs may be animated.

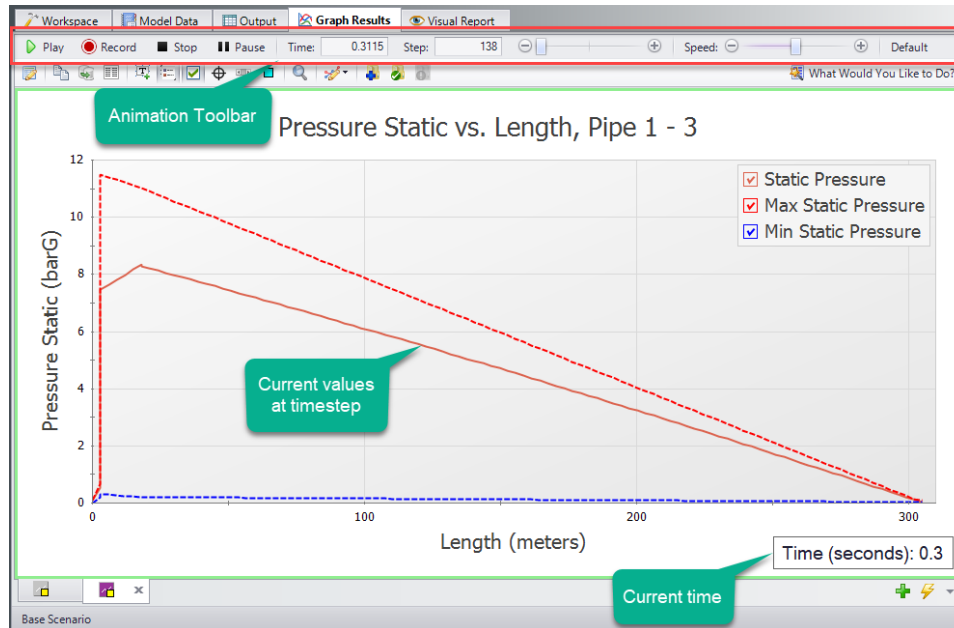


**Figure 1: Options for animation under Profile type graphs in the Graph Results window**

Profile graphs are animated using the output saved for each computation station in the model. By default, output is saved for all stations in an xStream model. Profile graphs cannot be animated if the Inlet/Outlet stations for each pipe are the only computation stations where output is saved. To allow animation, the user must have the "Pipe Stations To Include In Transient Output File" set to All on the [Pipe Sectioning and Output panel](#).

### Using an Animated Graph

Once the animated graph is created, it will look much like a normal Profile graph, but a few components have been added.



**Figure 2: Components added to the Graph Results window when animating a Profile graph**

The animation toolbar has several controls:

- **Play** - Start the animation from the current time.
- **Record** - Begin playing the animation from the current step, and when the playback is stopped, prompt the user to save an .mp4 video recording of the animation to file.
- **Stop** - Halt the animation and reset the time to zero.
- **Pause** - Pause the animation at the current time.
- **Time** - The actual simulation time of the currently displayed data.
- **Step** - The simulation step of the currently displayed data.
- **Time/Step slider** - Seek for a certain point in the animation.
- **Speed slider** - Change the speed of the playback.
- **Default** - Reset the speed to default.

By default, the graph will display the current values for the time step as well as the maximum and minimum profiles. These can be hidden or displayed in the graph Parameter selection area, like any other Profile graph. The Steady-State solution can be displayed, as well as applicable Design Alerts.

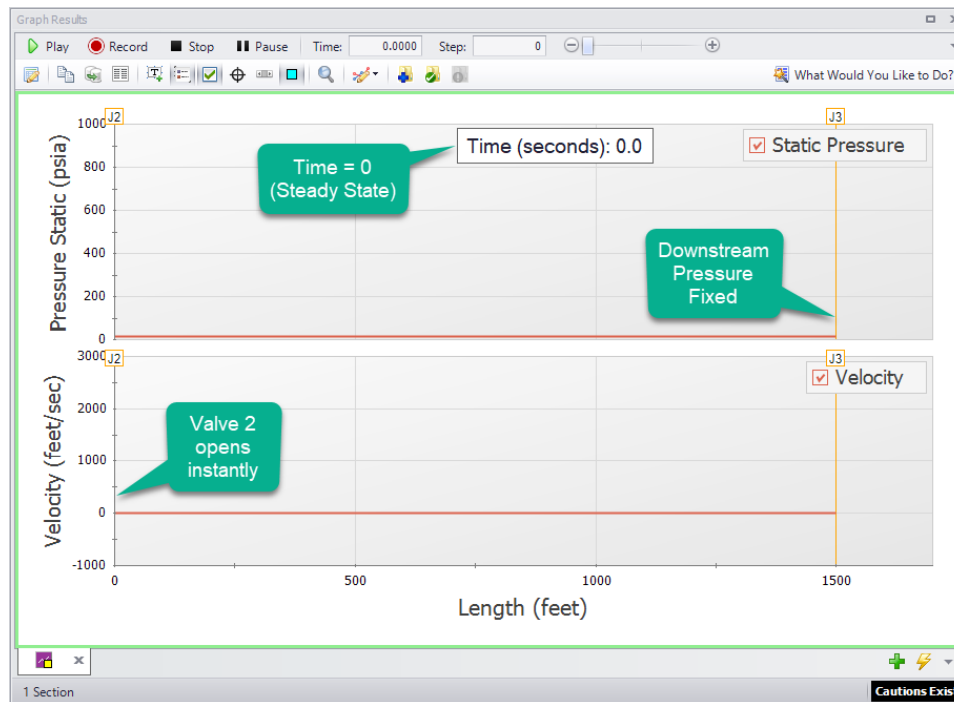
The Current Time box can be moved anywhere within the Graph area.

### Using Animations to Interpret Behavior

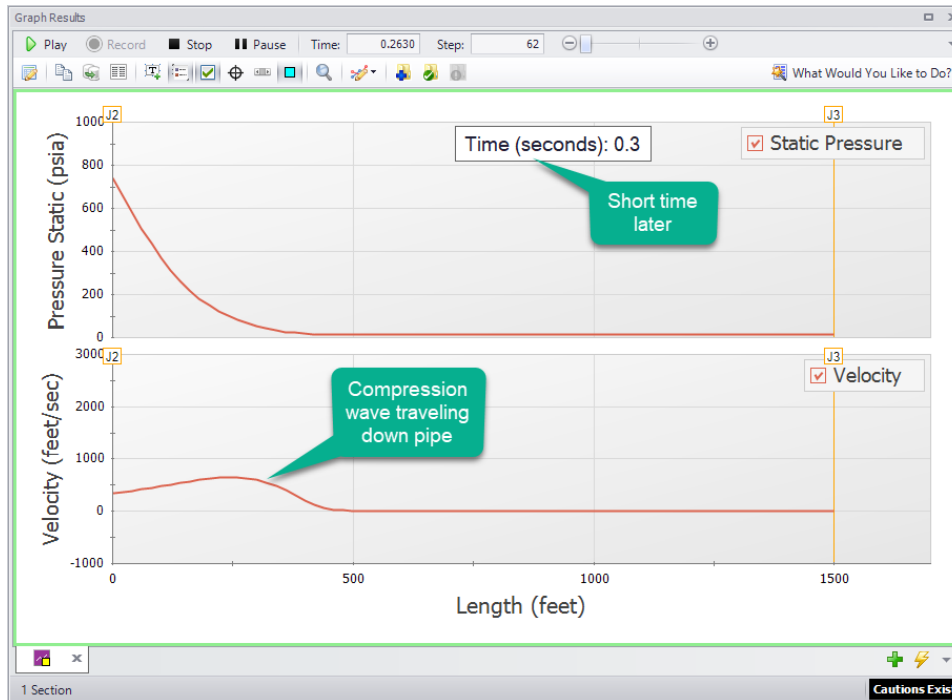
There are many ways that animations are helpful in interpreting behavior. Below, we consider the case of an instantaneous valve opening.

## Interpreting Transient Results

Like normal Profile graphs, multiple parameters can be displayed and animated as a single Graph List Item. The images below show the relationship between pressure and flow (velocity) for this case at the beginning and partway through the animation.



**Figure 3: Animation showing steady-state (time step zero) results**



**Figure 4:** The movement of the wave within the pipe can be directly visualized

### Animating Visual Report Results

Animating the results of a transient run is an extremely powerful way to understand how a complex system is behaving. While transient plots of fluid parameters over time at a given location are valuable and easy to interpret, they do not directly convey the most important aspect of a fluid transient event - how the pressure and flow waves propagate through a system and interact with one another. Animating the results of a transient simulation allows you to observe these effects directly.

Unlike the Graph Results Animation feature, Visual Report Animation provides the added benefit of being able to observe how waves propagate through branching paths and similar situations which are not restricted to one continuous flow path.

### Saving Output for Animation

The Visual Report Animation feature works by referencing the output for each computational pipe section in the animated pipes. In order to accomplish this, data for each pipe station being animated must be saved to the output file. xStream will save the data for all pipe stations by default. The pipe station output settings can be changed on the [Pipe Sectioning and Output panel](#) if needed.

### Creating an Animation

---

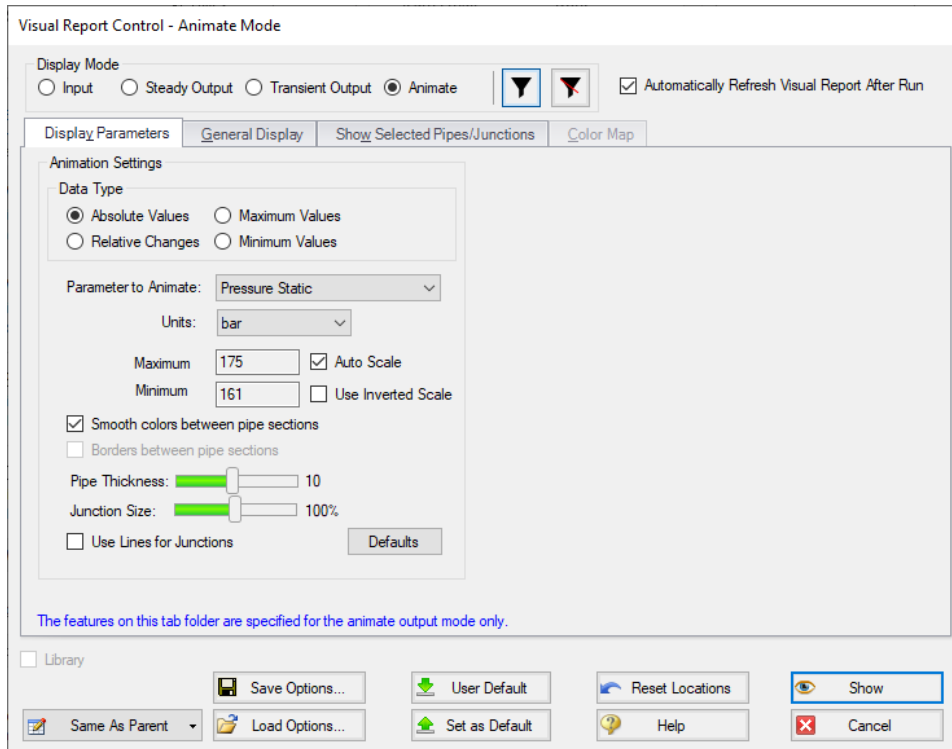
To enable animation in the Visual Report, the Display Mode will need to be set to "Animate" in the [Visual Report Control](#) window.

When Animate is selected, the following Animation Settings become available.

- **Data Type**
  - **Absolute Values** - This option directly uses the transient values for the pipes being animated. xStream will then automatically generate evenly spaced values in between the absolute minimum and maximum which will be displayed on the color scale.
  - **Relative Changes** - This option displays the changes in the transient results relative to the steady state values in order to animate the delta throughout the run. xStream will determine the maximum and minimum changes from the steady state, then automatically generate evenly spaced intervals between these values to display on the color scale.
  - **Maximum Values** - This option does not animate the data, but instead colors the pipes based on the maximum values during the transient for each of the pipe stations.
  - **Minimum Values** - This option does not animate the data, but instead colors the pipes based on the minimum values during the transient for each of the pipe stations.
- **Parameter to Animate** - The parameter and units for the animation can be selected
- **Auto Scale** - If this box is checked, xStream will automatically choose the maximum and minimum values for the selected parameter to assign the color scale for animation. This box is checked by default.
- **Maximum** - The maximum value for the color scale can be changed here if the Auto Scale box is unchecked
- **Minimum** - The minimum value for the color scale can be changed here if the Auto Scale box is unchecked
- **Smooth colors between pipe sections** - This choice sets the color gradient on the Visual Report to be displayed as a smooth transition, rather than an abrupt transition in color.
- **Borders between pipe sections** - This choice displays lines on the pipes to explicitly delineate the sections. This can only be enabled if the smooth colors option is disabled.
- **Pipe Thickness** - This option adjusts the thickness of the colored animation pipes. The thickness of the original pipes can be adjusted in the [Pipe Properties window](#) on the Optional tab.
- **Junction Size** - This option allows the size of the junctions in the Visual Report window to be adjusted for the animated model. The junctions can also be set to 0% to automatically hide all junctions. This sizing will only be applied on the Visual Report window.
- **Use Lines for Junctions** - This option allows the junctions to be represented as bold lines instead of symbols as an alternate method of displaying the junction locations.
- **Defaults** - This button allows the user to reset the Visual Report Animation settings to the defaults.

It should be noted that once the animation settings have been changed for a scenario, those settings will be retained even if the scenario is re-run. Each scenario can retain different Visual Report Settings if desired; however, there is no way to save multiple Visual Report configurations within the same scenario.





**Figure 1: Options for the Animate display mode in Visual Report Control**

## Viewing Animation

Once the Visual Report Control Settings have been configured and the visual report has been shown, several animation-specific components will appear, including the animation toolbar and Visual Report Animation tab, as shown below.

## Interpreting Transient Results

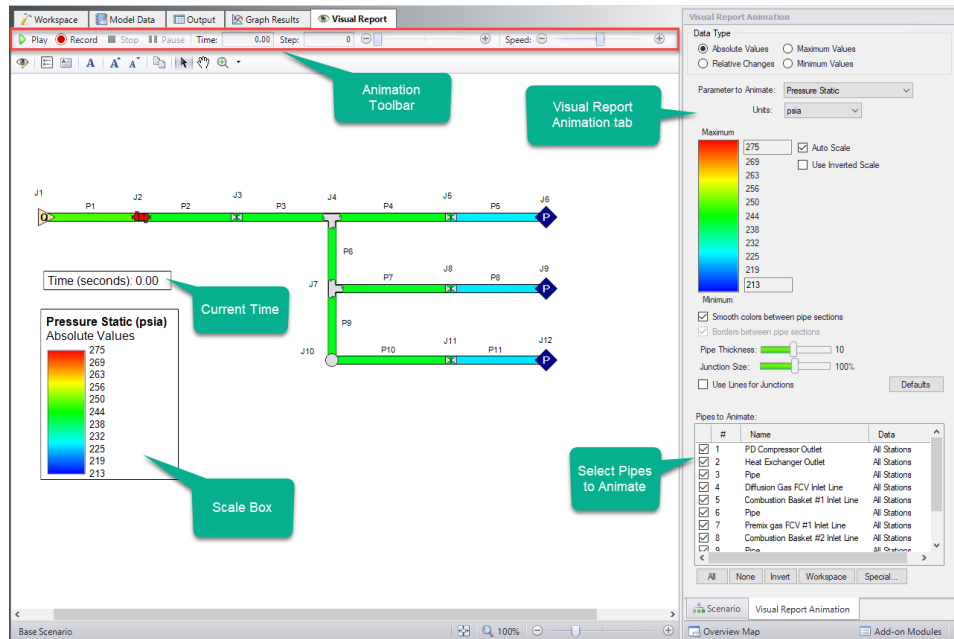


Figure 2: Components added to the Visual Report window when using the animation display mode

The Visual Report Animation tab in the Quick Access Panel provides quick access to the Animation Settings from the Visual Report Control window, as well as allowing the user to select which pipes will be enabled for animation. When disabled, the pipe will simply be shown using the workspace settings. Pipes that are not being animated can be hidden in the [Visual Report Control](#) window on the Show Selected Pipes/Junctions tab.

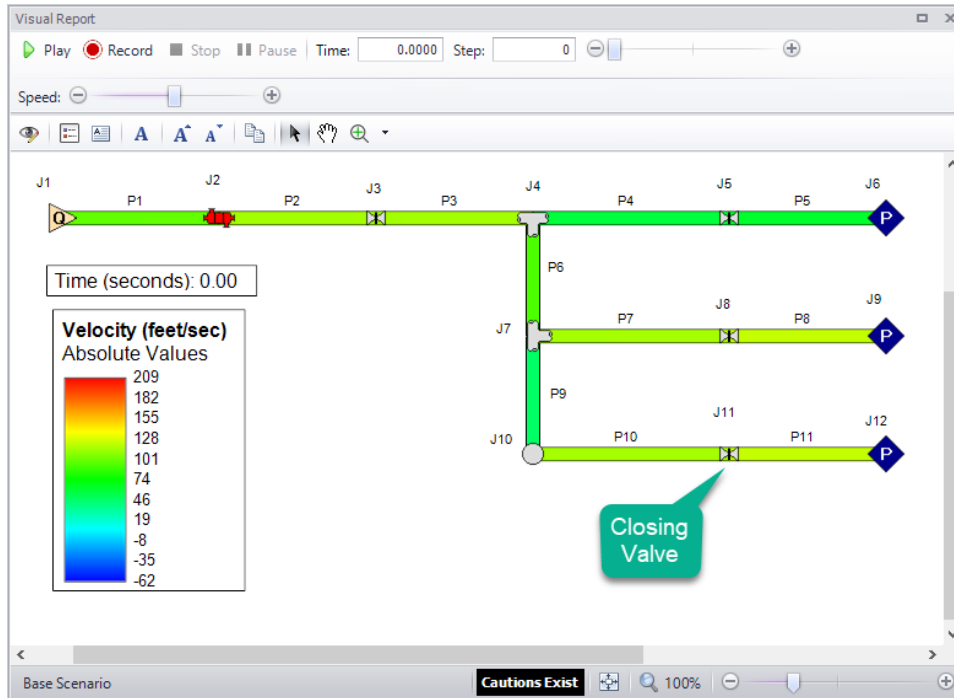
The animation toolbar has several controls:

- **Play** - Start the animation from the current time.
- **Record** - Begin playing the animation from the current step, and when the playback is stopped, prompt the user to save an .mp4 video recording of the animation to file.
- **Stop** - Halt the animation and reset the time to zero.
- **Pause** - Pause the animation at the current time.
- **Time** - The actual simulation time of the currently displayed data.
- **Step** - The simulation step of the currently displayed data.
- **Time/Step slider** - Seek for a certain point in the animation.
- **Speed slider** - Change the speed of the playback.
- **Default** - Reset the speed to default.

The Current Time box and Scale can be moved anywhere within the Graph area.

## Using Animations to Interpret Behavior

There are many ways that animations are helpful in interpreting behavior. Below, we consider the simple case of a valve closure.



**Figure 3: Animation showing steady-state (time step zero) results**

## Interpreting Transient Results

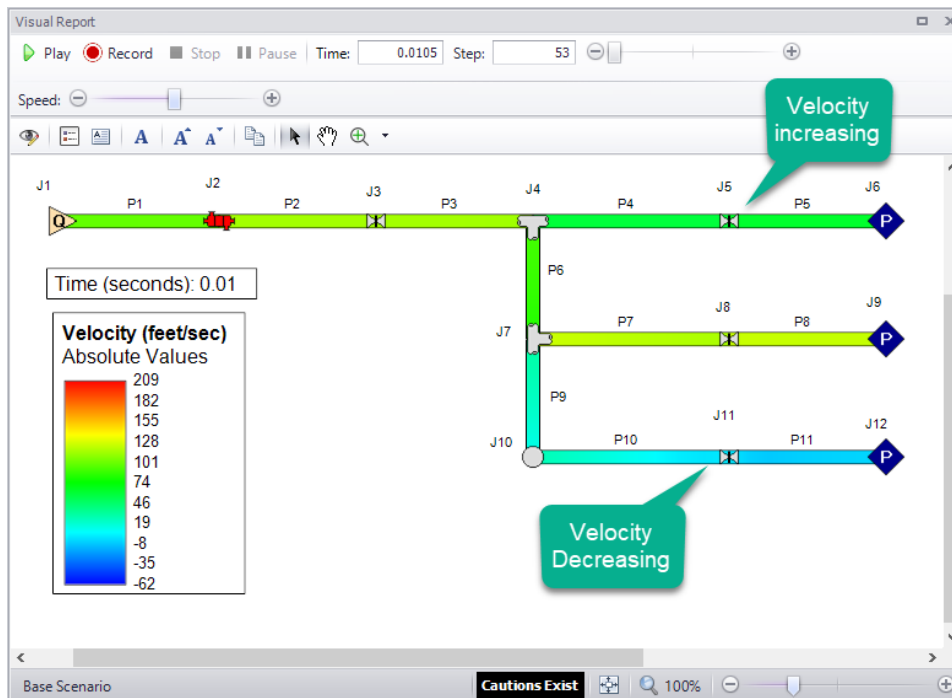


Figure 4: Velocity slows at the valve as the valve rapidly closes, and increases elsewhere

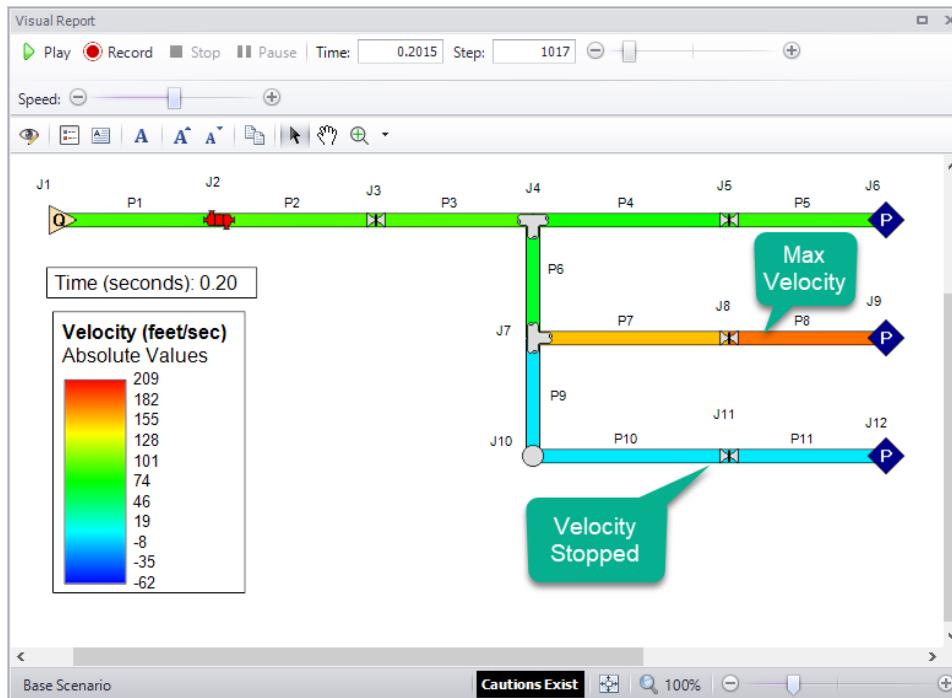


Figure 5: The movement of the wave within the pipe can be directly visualized

### Transient Result Accuracy

Modeling transient, compressible flow in a pipe network is an inherently imprecise endeavor. Individually modeling transient flow, compressible flow, or transient compressible flow in a single pipe can be done to a high degree of precision, but the practical application of all three factors forces limiting assumptions to be made. AFT xStream is no exception. The methods it employs start from three fundamental equations: conservation of mass, momentum, and energy. However, solving those equations on a practical scale requires several limiting assumptions that introduce some amount of error into the results. The derivation process and these assumptions are discussed in detail in the [Transient Theory](#) section.

As the user analyzes and interprets their transient results, it is important they keep in mind how the assumptions behind the [MOC solution](#) introduce error, how to gauge that error, and how to reduce that error.

### Sources of Error

---

The MOC solution process includes multiple assumptions, with two primary assumptions being made. First, flow parameters are assumed to vary linearly between computation stations, allowing linear interpolation to be performed. Second, fluid properties are assumed to be constant while integrating along characteristic lines, allowing those integrals to be solved directly. In effect, this means that fluid properties are only updated once per time step, with their change over time assumed to be linear.

The nature of the MOC solution forces both assumptions to be made. The MOC solution requires that the time step at each location in the system be uniform, meaning the section length must also be held constant. In an incompressible fluid system such as an AFT Impulse model, these assumptions introduce a minimal amount of error. However, in a compressible fluid system such as an AFT xStream model, these assumptions introduce a potentially significant amount of error as the compressible nature of a gas means flow parameters and fluid properties are changing continuously in time and space.

Users of AFT Arrow may be familiar with how Arrow captures the continuously changing nature of compressible flow. Arrow calculates results at intervals along a pipe which are adjusted based on the change in Mach number, thus the compressible flow effects along a whole pipe are accurately captured even though properties between computation stations are assumed to change linearly.

xStream however, does not have the ability to adjust the locations at which properties are calculated to be specific for an individual pipe. [Section lengths](#) must be held constant across all pipes in a system. Adjusting the location of computation stations along a single pipe would require the time step to change across the simulation, which in turn could cause time to run faster in some parts of the system than in others.

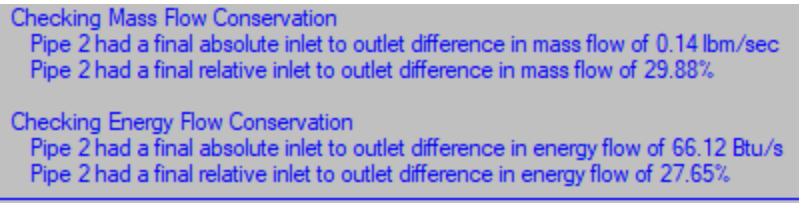
### Gauging Error

---

These assumptions have the major implication that mass and energy may not be perfectly conserved along the length of a pipe. The error introduced by these assumptions increases as the compressible flow effects become more pronounced. At low Mach numbers, mass and energy can be almost perfectly conserved, while near the sonic velocity, potentially significant amounts of mass and energy may escape the system.

Users can gauge this error in their system via two means. The first is more quantitative and is performed for the user by xStream. Prior to implementing the MOC transient solver, xStream solves the system once using the same computational methods as used by Arrow, and once by running the MOC solution without any transient events to determine the [MOC Steady Solution](#).

The mass and energy loss along a pipe in the MOC Steady Solution can then be quantified. The mass and energy flow out of each pipe are subtracted from the mass and energy flow into each pipe. In a nominally steady-state system, the difference should be zero. However, the assumptions behind the MOC solution mean this difference may not be zero. The steady-state error is reported in the [Solution Progress window](#), and the simulation process will be halted if the steady-state error exceeds the allowable set points. Figure 1 below, shows what that reported error and halted solution looks like. It is important to note that this quantitative approach to gauging error is only valid for a steady-state system. A transient system will inherently see differences in inlet and outlet mass and energy flows as waves travel along each pipe.



```
Checking Mass Flow Conservation
Pipe 2 had a final absolute inlet to outlet difference in mass flow of 0.14 lbm/sec
Pipe 2 had a final relative inlet to outlet difference in mass flow of 29.88%

Checking Energy Flow Conservation
Pipe 2 had a final absolute inlet to outlet difference in energy flow of 66.12 Btu/s
Pipe 2 had a final relative inlet to outlet difference in energy flow of 27.65%
```

**Figure 1: Solution Progress window showing an example model with high mass and energy conservation error**

The second approach users can employ to gauge the error in their systems is more qualitative and must be performed by the user. Flow parameters and fluid properties change more rapidly in time and space at higher Mach numbers than they do at lower Mach numbers. Thus, in a system which sees high velocity or sonic choking either in steady state or during the transient simulation, users can expect to see relatively large mass and energy losses compared to systems with low velocities throughout. While users cannot directly quantify those losses, they can expect to be more confident in the results determined by xStream at lower velocities.

## Reducing Error

---

The best way to minimize the error associated with the assumptions by the MOC solution process is to increase the number of sections in each pipe, which also decreases the time step in the simulation. This approach is similar to how Arrow captures compressible flow effects in steady-state systems. By calculating system parameters more frequently, the assumptions the MOC solution process makes are violated to a lesser degree.

In the Sectioning panel of the Analysis Setup window, users define the minimum number of sections a pipe can have. Setting a minimum section number of 5 means the shortest pipe must have at least 5 sections. Increasing that number forces xStream to use smaller section lengths throughout the entire system.

## Related Topics

---

Transient Flow Equations.....	429
Method of Characteristics.....	435
Transient Solver Initialization Panel.....	98
Minimizing Run Time.....	28
Transient Sensitivity Analysis Tutorial.....	51
Reducing Run Time.....	513

## Transient Sensitivity Analysis Tutorial

AFT xStream requires a large number of inputs to be defined in order to run a transient simulation. Many of those inputs directly affect the transient simulation results, and it is important for users to carefully evaluate that effect. This tutorial will focus specifically on three parameters: the [Minimum Number of Sections per Pipe](#) defined in the Sectioning Panel, the [Fluid Property](#) model used, and the estimated [Maximum Mach Number in Pipe](#) value.

We will use the model shown below in Figure 1 of a system with an abrupt stoppage of flow at a Turbine Shutoff Valve (TSV) to indicate how a user might analyze results and refine these inputs over subsequent runs. To help quantify how these parameters impact the transient simulation results, we will look at [transient forces](#) in the system, using a Difference Force Set applied to each pipe. Please note that this tutorial is not meant to be an exact step-by-step procedure to follow, but is instead meant to be a general guideline to help the user better understand how to evaluate xStream results.

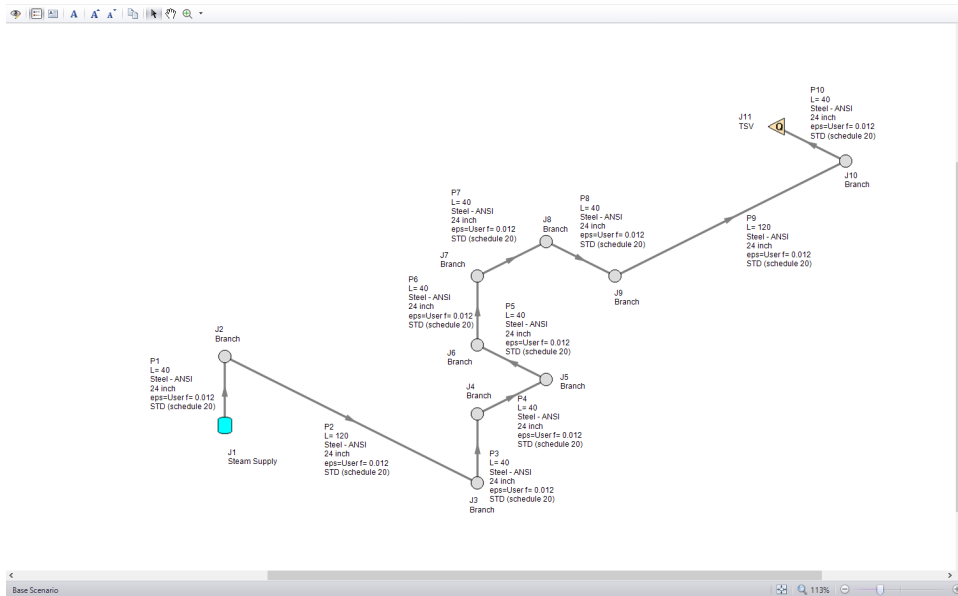


Figure 1: Layout of Tutorial Example

### Baseline Results

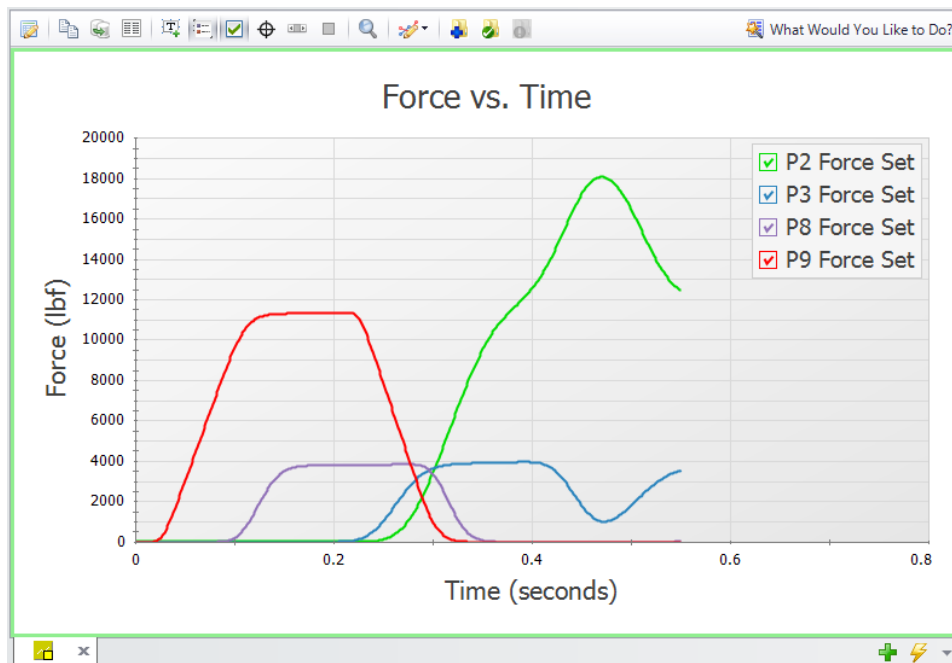
We must start our sensitivity analysis by determining some baseline results for the system. The first run of the model uses standard settings for all inputs: AFT Standard Steam using the Three-Parameter method, the default Maximum Pipe Mach Number During Transient of 1, and a Minimum Number of Sections per Pipe of 10. Force results from this run for Pipes 2,3,8, and 9 are shown below in Figure 2.

Pipes P2 and P9 see the largest transient forces in this model, mostly because P2 and P9 are 120 ft long with the other pipes in the system are 40 ft long. Transient forces are caused by the presence of transient waves with some pressure differential across them traveling through the system. A longer pipe which is long enough to contain the entire wavefront will see a larger pressure differential across it than a shorter pipe which only ever sees a portion of the wavefront. Thus, longer pipes tend to see larger transient forces, with the maximum transient force seen when the entire wavefront is shorter than the pipe length.

We can also see that P2 experiences a significantly higher maximum transient force than P9 even though they have the same length. This difference is due to wave coalescence (or wave steepening). A wave moving through a gas raises the local temperature, in turn raising the local sonic velocity. The temperature and sonic velocity behind the wavefront is thus higher than that ahead of the wavefront, meaning the back of the wavefront can have a higher wave velocity ( $V+a$ ) than the front of the wavefront. As the wave travels, the back of the wavefront will catch up to the front of the wavefront, steepening the wave until a shock wave is formed.

In this example, P9 is closer to the transient event at J11, and the wavefront is still relatively dispersed. However, the wavefront has steepened by the time it reaches P2, meaning more of the wavefront can fit in P2 and a larger transient force is seen.

This tutorial will focus on the transient force results from P2, as they are the largest magnitude forces in the model.



**Figure 2: Force Results for AFT Standard, 10 Minimum Sections, Standard Maximum Mach Number in Pipe**

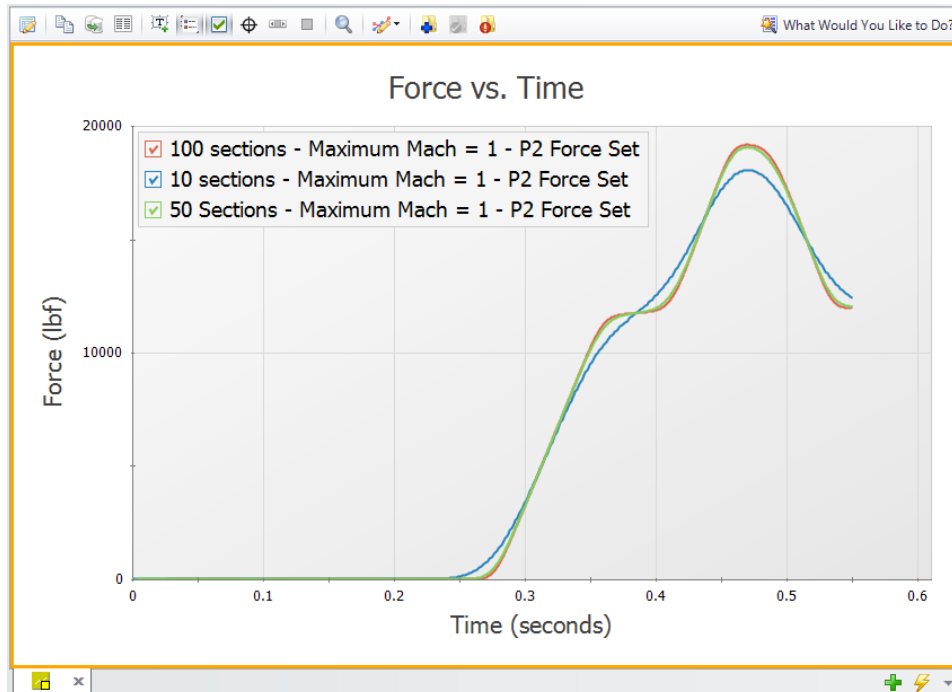
### **Minimum Number of Sections Per Pipe**

The first parameter for our sensitivity analysis is the Minimum Number of Sections Per Pipe. The baseline scenario used a minimum of 10 sections, and we will also run scenarios with a minimum of 50 and 100 sections. All other inputs are held constant, and results for the P2 Force Set are shown in Figure 3 below.



Comparing the 10 section and 50 section scenarios, the 50 section scenario gives a force that is 1,013 lbf higher than the 10 section run (or approximately a 5.6% difference). This difference indicates that using a minimum of 10 sections was too low, since using a finer grid has a noticeable effect. However, the 50 section and 100 section scenarios have minimal differences in the force results. This similarity indicates that the [error inherent](#) to applying the [Method of Characteristics](#) (MOC) to a compressible fluid may have been reduced when using a minimum of 50 sections, and using a finer grid will not reduce that error further.

We now know that 50 sections is a good point to balance accuracy with run-time concerns.



**Figure 3: Effect of Sectioning on Forces**

### **Fluid Property Model**

The next parameter for our sensitivity analysis is the Fluid Property Model. The baseline scenario used AFT Standard Steam with the Three-Parameter equation of state, and we will also run scenarios using AFT Standard Steam with the Redlich-Kwong equation of state, Chempak Steam, NIST REFPROP Steam, and ASME Steam. All other inputs are held constant, and results for the P2 Force Set are shown in Figure 4 below.

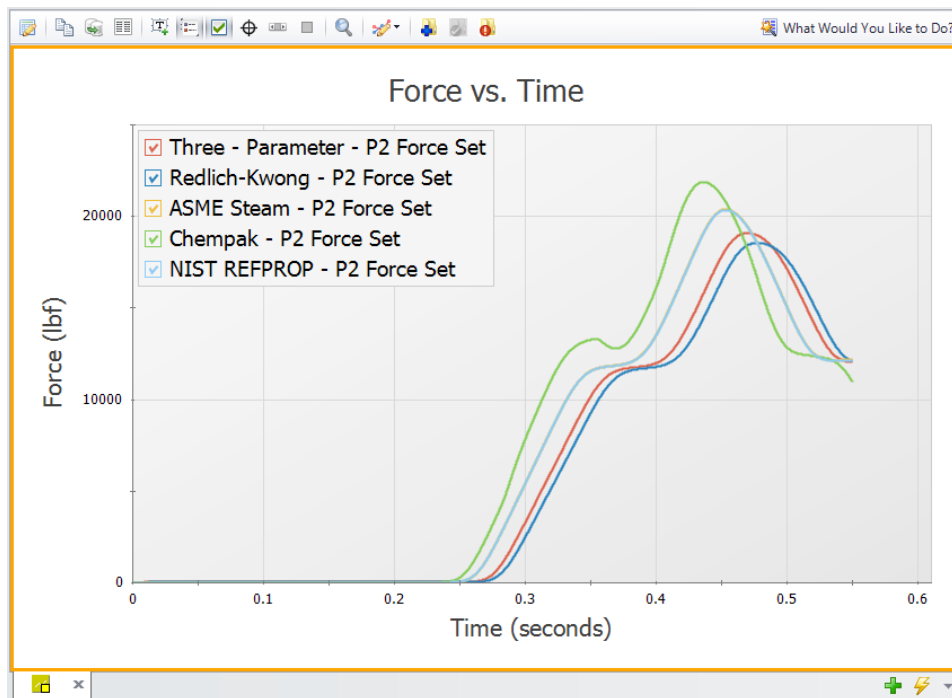
The results shown in Figure 4 below show that the Fluid Property Model can have a substantial impact on results. The largest difference in results is between the Chempak and Redlich-Kwong results, with an absolute difference of 3,327lbf, or a relative difference of 18%.

The observed difference in transient force magnitude for various Fluid Property models forces us to evaluate which model is best suited for this system. The Redlich-Kwong and Three-Parameter equations of state were developed for light, non-polar molecules - hydrocarbons in particular. Water (or steam, in xStream) is a relatively light molecule which is highly polar, meaning the accuracy of those Fluid Property models may be reduced when evaluating the steam in this system. Chempak has an increased accuracy

from the AFT Standard Steam models, and gives force results which are substantially higher. It is difficult to say how reliable the Chempak results are until we compare them against other models.

Finally, the scenarios using NIST REFPROP Steam and ASME Steam give effectively identical results, and the source for each model (NIST and ASME, respectively) is highly reliable. We can thus say that these results are the most reliable results, and they should be used for subsequent runs of our model. The only distinguishing factor between these two Fluid Property Models is the model run time, where the NIST REFPROP model will run significantly faster than the ASME Steam model. See the [Fluid Property Model example](#) for more information on this topic.

To learn more about the strengths and weaknesses of each fluid property library, visit the [Fluid panel](#) topic. Note that while the NIST REFPROP and ASME Steam models may offer improved accuracy, the AFT Standard fluid models run significantly faster. Users should always use the AFT Standard fluid models for initial runs as the xStream model is gradually refined.



**Figure 4: Effect of library on Forces (minimum 50 sections used in each pipe, ASME Steam has indistinguishable differences from NIST REFPROP and thus cannot be seen in this figure)**

### **Effect of Maximum Pipe Mach Number During Transient**

The last parameter for our sensitivity analysis is the Maximum Pipe Mach Number During Transient, defined in the [Pipe Sectioning and Output panel](#). The MOC grid size is determined by the wave velocity ( $V+a$ ), using the maximum estimated sonic velocity. This maximum sonic velocity is in turn a function of the Estimated Maximum Pipe Mach Number During Transient and the Estimated Maximum Pipe Temperature During Transient input parameters. Refining the estimated maximum sonic velocity can improve the accuracy of our results by reducing the amount of interpolation necessary while performing the MOC calculations.

## Interpreting Transient Results

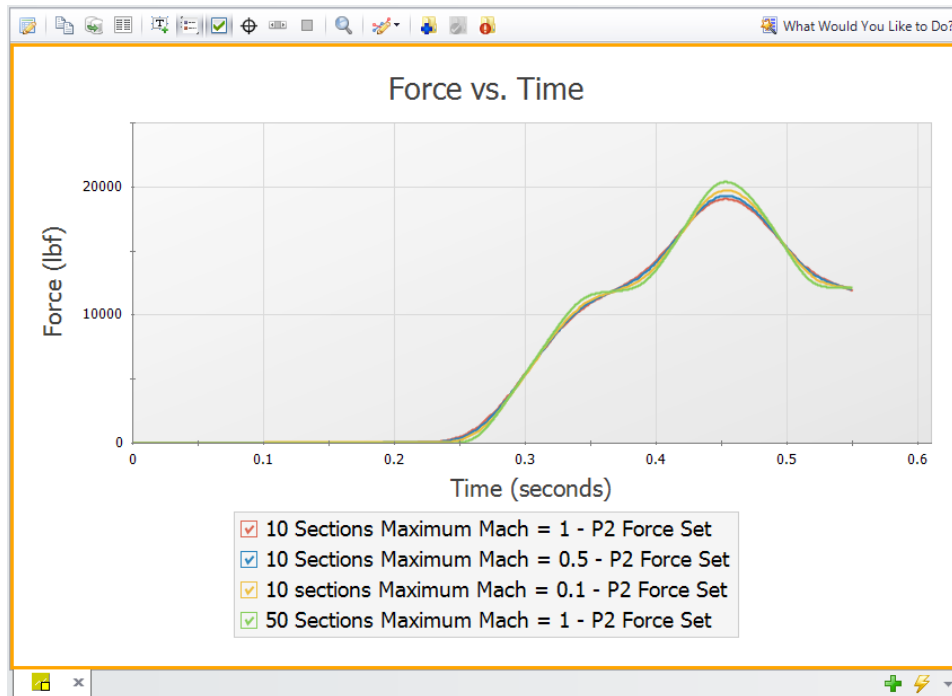
The results for our model when using a minimum of 10 sections and ASME Steam show a Maximum Mach Number of 0.06, seen in Figure 5 below. Reducing the Estimated Maximum Pipe Mach Number During Transient from the default value of 1 to a value closer to 0.06 may improve our results.

Figure 6 below compares transient force results with Estimated Max Mach Numbers of 1, 0.5, and 0.1. A scenario with 50 sections is included as a reference for what the expected results are, based on our previous analysis of how sectioning affects results. We can see that better estimates of the Max Mach Number for the transient simulation bring the results with 10 sections closer towards the 50 section reference results. However, better estimates for the Max Mach Number are unable to make the 10 section results match the 50 section results, indicating that the user should use the Estimated Max Mach Number parameter to compliment adding additional sections rather than to replace adding additional sections.

**Caution:** Never define the Estimated Maximum Pipe Mach Number During Transient lower than the value observed in the system! An estimate lower than the actual value will force the MOC calculations to extrapolate, which can add significant instability and uncertainty to the results. Output which shows that the specified maximum Mach number was exceeded should be disregarded, and a better Estimated maximum Pipe Mach Number During Transient should be defined.

Pipes		Transient Output			Transient Max/Min			Forces					
Pipe	in. 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	Max. Mach Number	Max. Mach Number Time (seconds)	Max. Mach Number Station	Min. Mach Number	Min. Mach Number Time (seconds)	Min. Mach Number Station
1	0	544.4	0.4761	10	539.4	0.1381	10	0.05994	0.1731	10	-5.067E-02	0.5498	0
2	0	554.5	0.4550	30	539.2	0.05157	30	0.05999	0.09210	30	-4.670E-02	0.5498	0
3	0	555.2	0.4513	10	539.1	0.02855	10	0.06002	0.06355	10	-2.290E-02	0.5498	0
4	0	555.4	0.4596	10	539.0	6.447E-03	10	0.06004	0.03500	10	-1.556E-02	0.5498	0
5	0	555.5	0.4835	10	538.9	0.000	10	0.06005	0.01566	10	-8.955E-03	0.5498	0
6	0	555.6	0.5065	10	538.8	0.000	10	0.06007	0.03408	10	-3.763E-03	0.5498	0
7	0	555.7	0.5296	10	538.8	0.000	10	0.06008	0.06078	10	-6.530E-04	0.5498	0
8	0	555.8	0.5498	10	538.7	0.000	10	0.06009	0.06999	10	5.430E-04	0.5498	0
9	30	556.0	0.5498	30	538.4	0.000	30	0.06014	0.000	30	1.893E-04	0.5498	30
10	10	556.2	0.5498	10	538.4	0.000	10	0.06016	0.000	10	-1.778E-16	0.2109	10

**Figure 5: Maximum Mach Number obtained from 10 Sections, ASME Steam, Maximum Mach Number of 1**



**Figure 6: Effect of Maximum Pipe Mach Number During Transient on Forces**

### **Comparing Initial and Optimized Inputs**

Our sensitivity analysis on Minimum Number of Sections Per Pipe, Fluid Property Model, and Estimated Maximum Mach Number in Pipe showed what inputs to use for this model. Figure 7 below compares the initial results (10 minimum sections, AFT Standard Steam with the Three-Parameter equation of state, and a Max Mach Number of 1) against the final results (50 minimum sections, ASME Steam, and a Max Mach Number of 0.1). These transient force results show a different timing of the transient waves, a different shape of the wavefront, and a difference in the maximum force magnitude of 2,512lbf.

These differences help show the importance of gradually refining the model input parameters. Initial runs should be performed with default or more simple options which have faster run times as the user starts to understand the system. Then, the user should refine the inputs for more advanced parameters to reduce the uncertainty in their results.

Note that some systems may be more sensitive or less sensitive to the three parameters considered in this tutorial. The user should follow these steps on their own system and use their engineering judgment to determine the best model inputs.

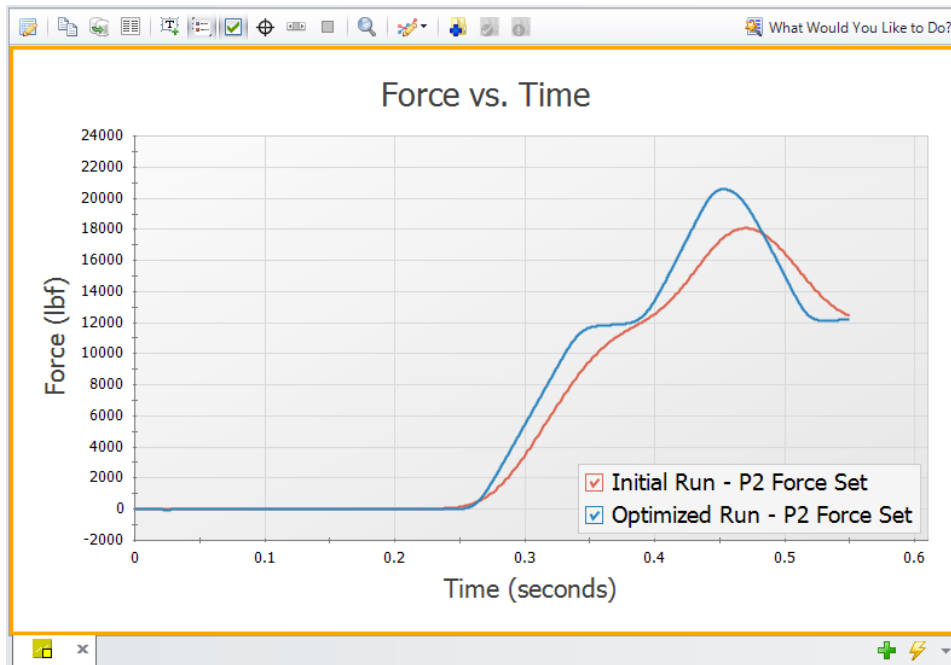


Figure 7: Initial Run vs Modified Run

**Related Topics**

Transient Forces.....	30
Force Definitions Panel.....	101
Fluid Panel.....	75
Equation of State Models.....	82
Pipe Sectioning - Introduction to Method of Characteristics.....	26
Pipe Sectioning and Output Group.....	95
Force Theory.....	35
Method of Characteristics.....	435
Minimizing Run Time.....	28
Transient Result Accuracy.....	49
Reducing Run Time.....	513

**Related Examples**

High Pressure Steam - Forces Example.....	606
---	-----

## Fundamental Modeling Concepts

This section focuses on the critical concepts and theories that comprise the model building process in AFT xStream. Topics include discussions on the role of pipe and junction objects, descriptions of how boundary conditions and flow direction are specified, and instructions on how to model closed systems.

### Contents

---

<a href="#">Pipes and Junctions</a> .....	58
<a href="#">Features for Modeling Irrecoverable Losses</a> .....	59
<a href="#">Convention for Positive Flow Direction</a> .....	59
<a href="#">Philosophy of Computer Modeling</a> .....	60
<a href="#">Role of Pressure Junctions and How They Work</a> .....	61
<a href="#">Open vs. Closed Systems</a> .....	64
<a href="#">Curve Fitting</a> .....	66
<a href="#">Stagnation vs. Static Pressure Boundaries</a> .....	68

## Pipes and Junctions

AFT xStream employs two fundamental pipe system constructs: pipes and junctions.

[Pipes](#) are conduits for one-dimensional gas flow. [Pressure drop due to friction](#) occurs in pipes.

[Junctions](#) are connector points for pipes and are elements at which flow balances are made and where transients are initiated. Some junction types can only connect to one pipe; others can connect with up to twenty-five.

In addition to balancing flow, junctions also influence the flow or pressure behavior of the system. For example, an infinite [tank junction](#) applies a constant pressure at a location, and the flow there is free to adjust in whatever manner is consistent with the governing equations. In contrast, an [assigned flow junction](#) applies a known flow rate at its location, allowing the pressure to adjust to that level dictated by the governing equations. The various junction types allow you to specify special kinds of [transient behavior](#), [irrecoverable pressure losses](#), or fluid behavior.

Junctions communicate with each other through the pipes connecting them. Each pipe must be connected to two junctions. There are no exceptions to this rule.

A pipe differs from a junction in that it has a [reference positive flow direction](#). To say a pipe has a flow rate of 1 lbm/sec is meaningless unless the flow direction is specified.

In general, you do not need to specify the actual flow direction in a pipe, because AFT xStream sorts out the true physical flow directions of the system you define. For example, if you specify the reference positive flow direction from left to right, and the solution is in fact the opposite direction, then that information will be presented in the output as a negative flow rate. Inlet and outlet conditions correspond to the reference positive flow direction, not the actual flow direction. However, for easier analysis of results, it can be helpful to ensure all pipes are configured so that they have positive flow in the steady state system.

It is important to specify the proper flow direction for pipes which connect to junctions such as [compressors/fans](#) or [control valves](#). AFT xStream uses the flow direction of the pipes to determine in which direction to increase or reduce pressure.

### Features for Modeling Irrecoverable Losses

AFT xStream works almost exclusively with the concept of the loss factor, also known as K factor or resistance factor. In some Properties windows, other popular methods for specifying loss information are provided where they are consistent with the loss factor method. For example, you will find Cv (flow coefficients) for valves. No provision has been made for the equivalent length approach to solving pressure drop problems. See the [Junction Loss Models](#) section for more information.

### Related Topics

---

Conventions for Specifying Base Area .....	198
Pressure Drop in Pipes - Detailed Discussion .....	190

### Convention for Positive Flow Direction

#### Reference Positive Flow Direction

---

Next to each pipe's ID number is an arrow that indicates the reference positive flow direction for the pipe. AFT xStream assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing [Reverse Direction](#) from the Edit menu or selecting the reverse direction button on the Toolbar.

The reference positive flow direction is used for reference purposes only and need not be the actual flow direction. If the reference positive direction is the opposite of that obtained by the Solver, the output will show the flow rate as a negative number.

#### Convention for Flow Entering and Exiting

---

Many of the junction types permit fluid flow to come into or pass out of the system in a prescribed manner. The most obvious of these is the [tank junction](#). Tanks maintain a constant pressure at a node, and the flow adjusts according to the governing equations. While in principle the gas is stored or removed from the tank, AFT xStream considers the gas as coming into or going out of the system. The stored tank fluid is not part of the system.

The convention for defining inward or outward flow for the system or any AFT xStream object is that flow in is positive (that is, the object gains the flow) and flow out is negative (the object loses the flow).

### Philosophy of Computer Modeling

Some might say that a "good" computer model is one that accurately predicts the response of a system; but this is only partly true. The practicing engineer generally must weigh the accuracy of the model against the risk, schedule, and/or budget involved. Through experience an engineer learns where to focus resources and when an answer is good enough to base important decisions on.

Although AFT xStream is designed to speed up the modeling process, it should never be used as a "black box". Your model input and output should always be carefully reviewed.

To make the most effective use of AFT xStream, you should follow these guidelines:

- Be aware of what AFT xStream can and cannot do and the [assumptions](#) it makes.
- Learn and use the tools in AFT xStream that are appropriate for the specific analysis you want to perform. Detailed and accurate answers demand detailed and accurate input. Quick and dirty answers require less detailed input.
- Learn and use the tools in AFT xStream that allow you to process information in the formats that mean the most to you. Because pipe flow applications vary so much within different industries, AFT xStream is customizable for the way you want to do your pipe flow work. Understanding the tools will help you gain greater conceptual control over your pipe flow analysis and reduce modeling errors.
- Check your input data. Use the features available in AFT xStream to review your results for input errors. Pay attention to the information in the output report. Don't neglect warnings in the output. Perform hand analysis checks of results.
- Bound your problem. Instead of trying to calculate the exact resistance in a network, or the right loss factor for an orifice, use your judgment, guided by data, on how much variation might occur. By running your model at the upper and lower limits, you can obtain answers, for example, on the maximum and minimum possible pressure drops in the system. If the results at the maximum and minimum are acceptable, your analysis is probably finished. If they are not acceptable, you can further refine the input model.
- Recognize whether your problem is one of maximization or minimization. That is, recognize whether the design goal is to minimize the flow rate, for example, or maximize the flow rate. Then make your assumptions conservative enough to provide high confidence that your design requirements are being satisfied.

### Related Topics

---

Overview of AFT xStream .....	17
xStream Learning Resources .....	22
Primary Window Overview .....	279
Overview of Fluid Transients .....	25



## Related Examples

Tank Blowdown Example (English Units)..... 536

## Role of Pressure Junctions and How They Work

### Role of Pressure Junctions

In modeling a generalized pipe network, it is possible to construct models that do not have a unique solution. A common occurrence of this is when a model contains one or more sections which are completely bounded by known flow rates. The simplest example is the system shown in Figure 1. In AFT xStream terms, the system has two [assigned flow junctions](#).



**Figure 1: Model with two assigned flows. This model does not have a unique solution**

Obviously, the flow in the pipe is known. But what is the pressure at the inlet? At the outlet? It cannot be determined because there is no *reference pressure*. The reference pressure is that pressure from which other pressures in the system are derived. There can be more than one reference pressure, but there always has to be at least one.

The model in Figure 1 can be built with AFT xStream and if you try and run this model, AFT xStream will inform you that it cannot run it because of the lack of a reference pressure (Figure 2). In AFT xStream there are several junctions that can act as a reference pressure: the tank, assigned pressure, valve (with the exit valve option), and pressure control valves.

When you try and run a model that has one or more sections completely bounded by known flows (or Flow Control Valves or Compressors/fans operating in assigned flow mode), the Need Reference Pressure window (shown below) is displayed. This window informs you which junctions bound the sections where a reference pressure is needed to obtain a unique solution.

In multi-pipe systems there are a host of other possibilities that present themselves. All other configuration possibilities which lack a reference pressure ultimately boil down to the same problem that exists in Figure 1.

See [Role of Pressure Junctions - Detailed Discussion \(Long\)](#) for more information.

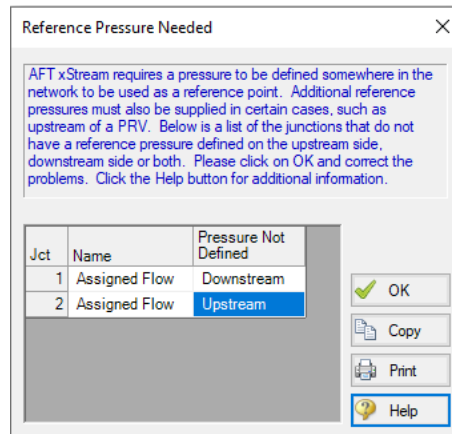


Figure 2: AFT xStream message when you try to run a model without sufficient reference pressures

## How Pressure Junctions Work

Pressure junctions (e.g., [Tanks](#) or [Assigned Pressures](#)) in AFT xStream are an infinite source or sink of fluid. This means that they can draw or discharge as much fluid as is necessary to maintain the specified pressure. Some engineers have difficulty grasping this concept, and misuse pressure junctions in AFT xStream. This section offers a physical example of how pressure junctions work to clarify for those having difficulty with the concept.

Consider the physical system shown in Figure 1. This system compresses air from a low pressure supply tank to a high pressure discharge tank.

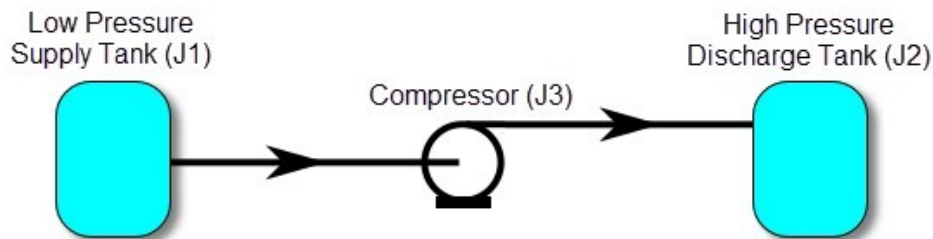
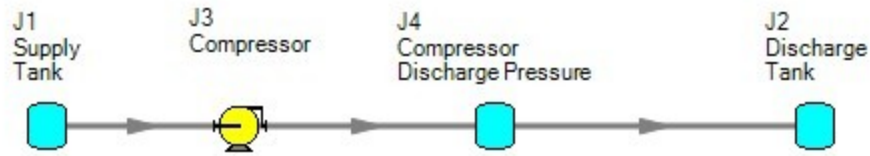
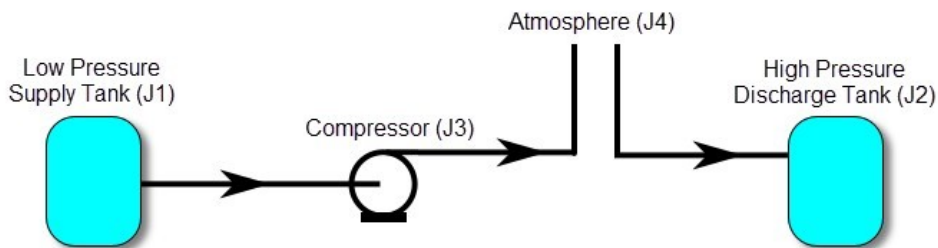


Figure 1: Physical compressor system to transfer air from low pressure tank (J1) to high pressure discharge tank (J2)

An engineer is tasked with sizing the compressor for this system. The engineer has an idea of the discharge pressure needed for the compressor, and builds an AFT xStream model as shown in Figure 2. The Figure 2 model is attempting to represent the physical system in Figure 1, but it in fact represents the physical system in Figure 3.



**Figure 2: Model to size compressor attempting to represent system in Figure 1, but actually represents the system in Figure 3**

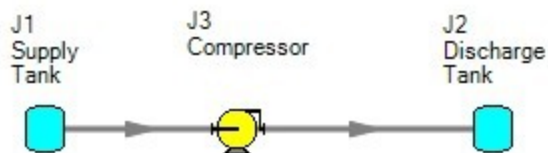


**Figure 3: Physical system represented by model in Figure 2**

Why? Because the tank junction (J4), which is an infinite source of fluid, has been located between the compressor (J3) and the discharge tank (J2). The J4 tank isolates the supply tank and compressor (J1 and J3) from the discharge tank (J2). The J4 tank is very much like placing the atmosphere between the supply and discharge piping.

If the engineer changes the pressure in J4, it is like changing the atmospheric pressure. It will change the flow rate in the pipes, but no matter how high the pressure of J4, it does not change the fact that the compression system is isolated from the discharge tank. The flow rate in the piping from J4 to J2 is entirely dependent on the difference in pressures between J2 and J4, and is not influenced by the J3 compressor in any way.

How then should the physical system in Figure 1 be modeled? It should be modeled as shown in Figure 4.



**Figure 4: Proper model of physical system shown in Figure 1**

If the goal is to size the compressor at J3, the model in Figure 4 can be used with the compressor modeled as a fixed flow compressor.

## Related Topics

Role of Pressure Junctions - Detailed Discussion (Long).....	480
Assigned Pressure.....	223
Tank.....	259

### Related Blogs

---

[Reference Pressure Needed: Where It Comes from and How to Address it](#)

## Open vs. Closed Systems

### Model a Closed Loop System

---

A closed circulating system can easily be modeled with AFT xStream by locating a [Tank junction](#) anywhere in the system and setting the pressure to the known pressure at that point. Inflow and outflow pipes will then need to be connected. The Tank junction merely acts as a reference pressure against which all other pressures in the system are compared. Because there are no other inlets or outlets to the system, then whatever is delivered by the Tank junction to the system will also be received from the other pipes, thus achieving a net zero outflow rate for the system.

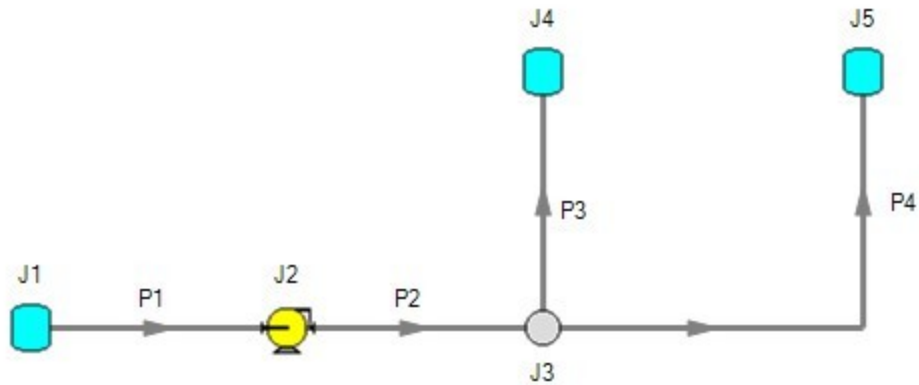
### Balancing Mass

---

In order to model a closed system, only one pressure junction is used in the model. Typically this would be a Tank junction. Pressure type junctions are an infinite source of fluid and do not balance flow. How then can one be used to flow in a closed system?

To answer this question, it is worth considering how AFT xStream views a closed system model. AFT xStream does not directly model closed systems, and in fact does not even realize a closed system is being modeled.

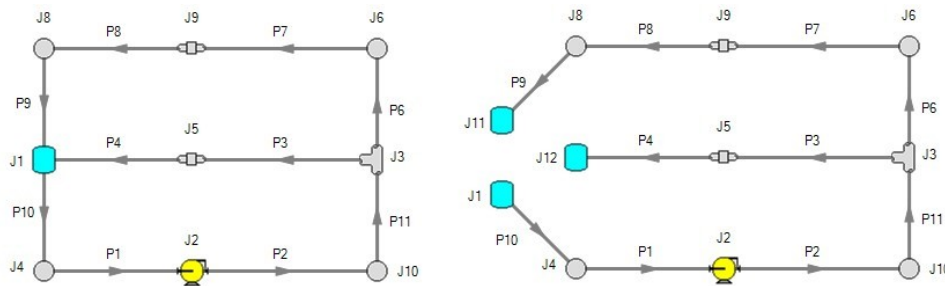
Consider the system shown in Figure 1. This is an open system. Fluid is taken from J1 and delivered to J4 and J5. Because the AFT Arrow Steady solution engine in xStream solves for a mass balance in the system, all flow out of J1 must be delivered to J4 and J5. Because the flow is steady-state, no fluid can be stored in the system; what goes in must go out.



**Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5**

Now consider the systems in Figure 2. The first system appears to be closed, while the second appears to be open. If the same boundary condition (i.e., pressure, temperature and surface elevation) is used for J1, J11 and J12 in the second system, to AFT xStream it will appear as an identical system to the first system. The reason is that AFT xStream takes the first system and applies the J1 tank pressure as a boundary condition to pipes P4, P9 and P10. The second system uses three tanks to apply boundary conditions to P4, P9 and P10. But if the tanks all have the same pressure, temperature and elevation, the boundary conditions are the same as J1 in the first system. Thus the same boundary condition is used for P4, P9 and P10 in both models, and they appear identical to AFT xStream.

But how is the flow balanced at J1 in the first system? Looking at the first system, one sees that to obtain a system mass balance, whatever flows into P10 must come back through P4 and P9. Because there is overall system mass balance by the solver, it will give the appearance of a balanced flow at the pressure junction J1. If there is only one boundary (i.e., junction) where flow can enter or leave the pipe system, then no flow will enter or leave because there isn't anywhere for it to go. Thus the net flow rate will be zero at J1 (i.e., it will be balanced). But recognize that AFT xStream is not applying a mass balance to J1 directly. It is merely the result of an overall system mass balance.



**Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two system will appear identical to AFT xStream**

## Balancing Energy

Consider the system shown in Figure 1. We know that the flows are balanced at J1, but how can the energy be balanced? For example, assume the user sets an initial temperature of 100° F at J1. This temperature will be the inlet pipe stagnation temperature for all pipes that flow out of J1 in the steady state. In this case, the 100° F will apply only to pipe P10.

The pipes flowing into the tank (P4 and P9) will have their own temperatures that are obtained by balancing energy along their individual flow paths. This will include heat exchanger input and heat transfer to or from pipes, as well as any heat of compression effects in the compressor.

The only way to obtain an overall system energy balance is for the J1 tank temperature to adjust to the mixture temperature (mixture stagnation enthalpy, to be more precise) of all inflowing pipes. This can be enforced using the Finite tank option with an Adiabatic Thermodynamic Process. There is a unique “mixture temperature” (or mixture stagnation enthalpy) that will yield an energy balance at J1. This will be the temperature/enthalpy from [Equation 49](#).

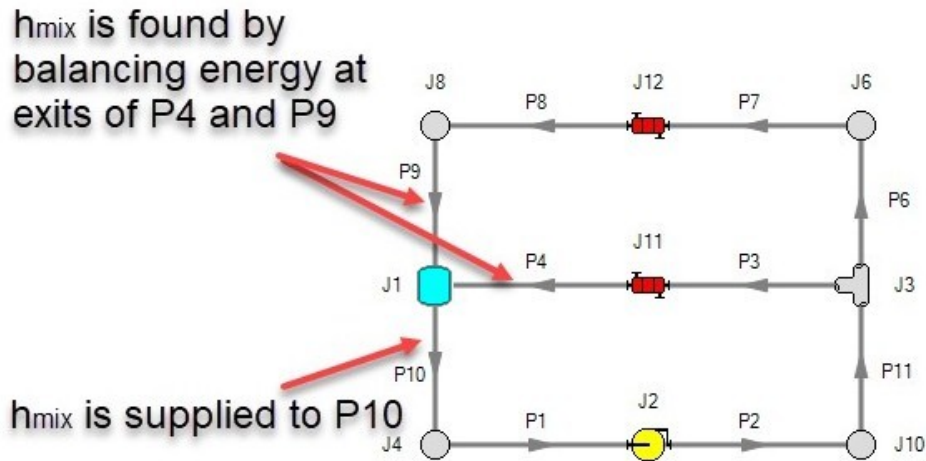


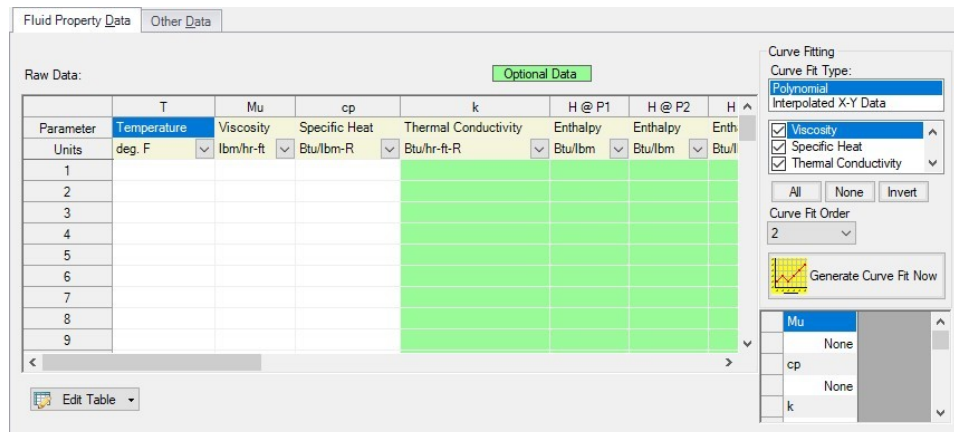
Figure 1: Example of balancing energy in a closed system

## Curve Fitting

There are many places in AFT xStream where the user is asked to define how a property changes in relation to another. Some examples are:

- [Compressor/Fan Curves](#)
- [Junction Resistance Curves](#)
- [Custom Fluids](#)
- [Pipe Physical Properties](#)

These properties are entered in a table format, relating one particular property to several others. For example, see the below definition of a custom fluid:



**Figure 1: Custom Fluid Definition**

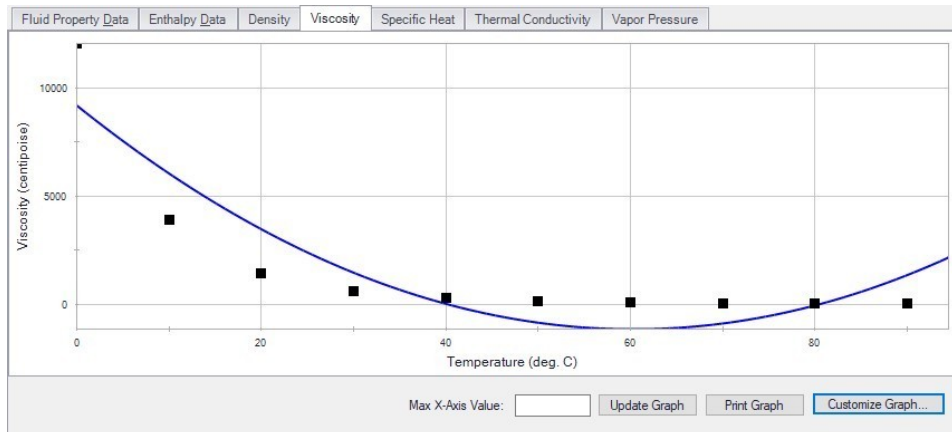
There are several parameters that control the behavior of the fluid and they are entered as they relate to temperature.

However, it is important to recognize that these values in particular are not directly used in the solution. Instead, the data is used to create a best-fit curve. This curve is then used in the solution process. This is an important step to take because there must be a continuous relationship between the independent variable (e.g. temperature) and the dependent variable (e.g. density). Even if the final solution was close to one of the defined points, the solver needs continuous data to proceed.

There are some options for how the data is curve fit. First, there are two Curve Fit Types:

- **Polynomial** - The data is best fit to a polynomial of order 0,1,2,3, or 4. Note that a 0th order polynomial is a constant value, and a 1st order is a linear fit.
- **Interpolated X-Y Data** - The data has a linear interpolation drawn between every data point.

One of the critical items to note with any curve fit is that you may not be operating close to one of your defined points if the data does not conform well to a polynomial relationship. For example, the below fluid has a rapidly decaying viscosity that does not follow a polynomial relationship. Fitting the data to the standard 2nd order polynomial results in the following:



**Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data**

The viscosity relationship in the model will use the blue curve, and not the user-defined black points. Depending on the operating range of the system, this may significantly impact the results! For this data in particular, using the Interpolated X-Y Data option may be a better choice.

After data has been curve fit, the constants for the resulting curve are displayed in the lower right of the associated window. For example, in Figure 1 above there are a, b, and c constants shown for the Density (Rho) curve fit. These values use a base unit, which is not necessarily the unit the user has defined for that parameter.

## Stagnation vs. Static Pressure Boundaries

With two exceptions (to be discussed), all pressure-type boundary conditions in AFT xStream are stagnation. This works very well for things such as large storage tanks or atmospheric boundaries, where the volume associated with a pressure is large and will not change (significantly) with time. These boundaries have no velocity associated with them, and using stagnation pressure is thus appropriate. These boundary conditions are most clearly rendered in AFT xStream by use of a Tank junction.

For simplicity, we will discuss this in terms of an incompressible flow example. Consider the tank and connected pipe in Figure 1. When the fluid in a tank flows into connecting pipes, the static pressure immediately drops due to the increase in velocity.

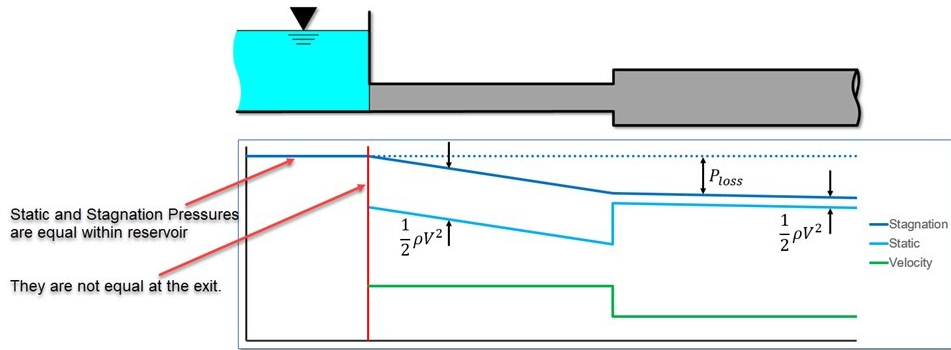
It is tempting to say that because there is no velocity in the tank, then it does not matter whether the tank boundary pressure is considered as a static or stagnation pressure because they are equal. This is a misconception. The pressure boundary condition in an AFT xStream model is applied at the exit of the junction, not within the junction itself. Inside the tank, the static and stagnation pressures are the same, but they are not the same at the exit - they differ by the amount of the dynamic pressure.

---

**Note:** An incompressible liquid flow example is given because it is easier to discuss, but the principles apply similarly to compressible flow.

---





**Figure 1: Pressure profile along horizontal pipe shown in Figure 1 (incompressible flow)**

The appropriateness of either stagnation or static pressure as a boundary condition depends on whether there is a velocity associated with the pressure that is specified. Is the pressure in a vessel or large (lower velocity) header, or is it in a pipe somewhere in the middle of a system? If the first, stagnation is more appropriate; if the second, static.

Pressure boundary conditions with no velocity are more common across all piping industries, and are usually appropriate. However, this is not always the case.

### When to use Static Pressure

Now let's consider the application of a true static pressure boundary condition. A static boundary condition has a unique velocity associated with it. The Bernoulli equation, which in this integrated form applies only to incompressible flow, is given by:

$$P_1 + \frac{1}{2}\rho V_1^2 + \rho g z_1 = P_2 + \frac{1}{2}\rho V_2^2 + \rho g z_2 + \Delta P_f$$

From this equation it can be seen that application of the static pressure is not sufficient to specify location 1. The velocity and elevation is also required. Note that though the equation above is only applicable for incompressible flow, the general concept is applicable for compressible flow.

In AFT xStream, elevation data is entered for junctions. The pipes adopt the elevation of the junction to which they are connected.

But where does the velocity information in the Bernoulli equation come from? One might respond that it can be obtained from the flow rate. But this raises another question: what if the flow rate isn't known? Put another way, what if the flow rate is what we are solving for?

Because the stagnation pressure combines the static pressure with the velocity information (in the form of Mach number, see [Stagnation Pressure](#) definition), when the user supplies a stagnation pressure boundary condition AFT xStream can avoid the need for an associated velocity condition at location 1. A stagnation pressure boundary can thus be supplied to multiple pipes that might have different velocities. That is why the same Tank junction can connect with multiple pipes.

However, if the user specifies a static pressure as the boundary condition, a unique velocity (or Mach number) must be supplied. Thus it is only possible to connect a static pressure to a single pipe with a unique velocity.

---

**Important:** A static pressure boundary has a unique velocity (and Mach number) associated with it, and can thus connect to only one pipe.

---

Where would one find a static pressure boundary? The best example is when the boundary condition is inside a pipe. A pipe system model can start and end anywhere it is convenient for the user. It may be convenient to not start the model at the physical boundary (such as a tank) but at a particular location in the pipe system. This could be, for example, at the location of a pressure measurement. Or it could be at the boundary of the pipe system for which your company is responsible, with another company responsible for what is on the other side of that boundary.

If one needs to model such a situation, the Assigned Pressure junction allows one to model either a static or stagnation pressure. The default stagnation pressure allows connection of up to 25 pipes. If static pressure is chosen, only one connecting pipe is allowed.

### Static Pressure at Pressure Control Valves

---

Another example of static pressure is at pressure control valves. For [pressure control valves](#) (i.e., PRV's and PSV's) the default control pressure is static pressure. The reason is that the measured pressure that provides feedback to the controller will typically be a static pressure measurement. You have the option of modeling pressure control valves as either static or stagnation pressure.

### Related Topics

---

Review of Compressible Flow Theory.....	412
Stagnation Properties.....	455

## Analysis Setup Overview

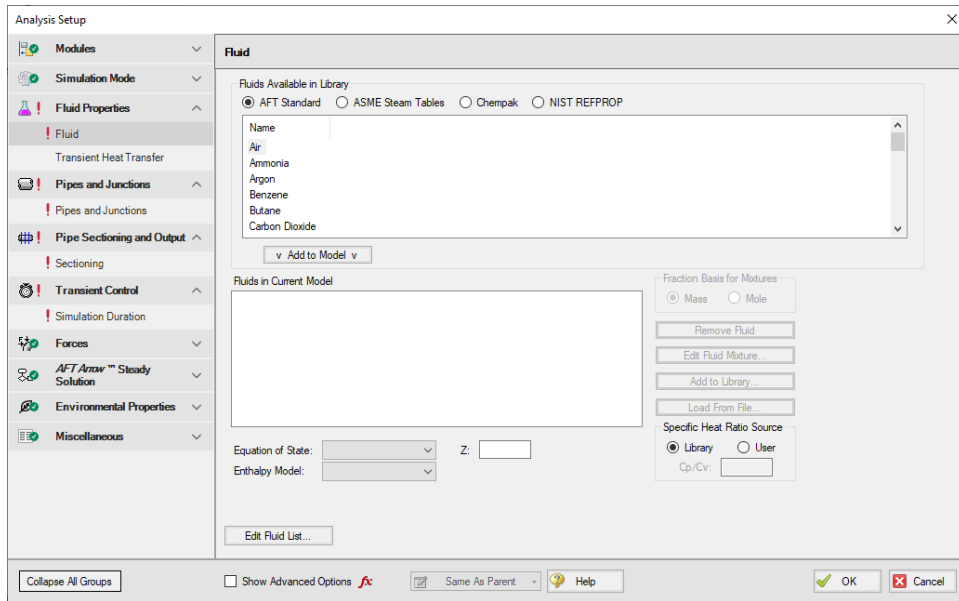
The Analysis Setup window contains all of the settings which the user must define to run the model, including fluid properties, steady state and transient solution controls, pipe sectioning settings, and other related options. The Analysis Setup window can be launched from either the toolbar or the Analysis menu.

The controls in Analysis Setup are divided into different *groups* in the navigation pane on the left of the window (Figure 1). Each group has one or multiple *items*. Clicking on an item opens the *panel* for that item.

A group is not fully defined until each item in that group is defined. Each group will have either a green check mark, indicating that all required items have been defined, or a red exclamation point, indicating that some or all of the required inputs are missing. Note that some groups will have a green checkmark by default, indicating that the required inputs have default settings already applied, or all inputs for that group are optional. Groups that are undefined will change to a green check mark when the required inputs for that group are entered, and the user navigates to a different panel. Note that none of the changes will be applied to the model until OK is selected. Clicking Cancel will discard all changes made in the window.

The groups in Analysis Setup are

- [Modules Group](#)
- [Fluid Properties Group](#)
- [Pipes and Junctions Group](#)
- [Pulsation Setup Group](#)
- [Pipe Sectioning and Output Group](#)
- [Transient Control Group](#)
- [Forces Group](#)
- [AFT Arrow™ Steady Solution Group](#)
- [Environmental Properties Group](#)
- [Miscellaneous Group](#)



**Figure 1: Default state of the Analysis Setup window in a new blank model**

## Modules Group

AFT xStream has one optional add-on module which extends its modeling capabilities into other areas. The modules allow pulsation frequency modeling. The Modules panel in the Modules group in Analysis Setup allows you to activate and enable the add-on modules. An additional license to the base AFT xStream license is required to use a module. These modules include:

- [PFA \(Pulsation Frequency Analysis\)](#) - Helps identify and avoid resonant frequencies in systems, especially in those caused by positive displacement compressors.

## Modules Panel

The Modules Panel allows users to activate and enable the optional add-on modules and/or cost calculations.

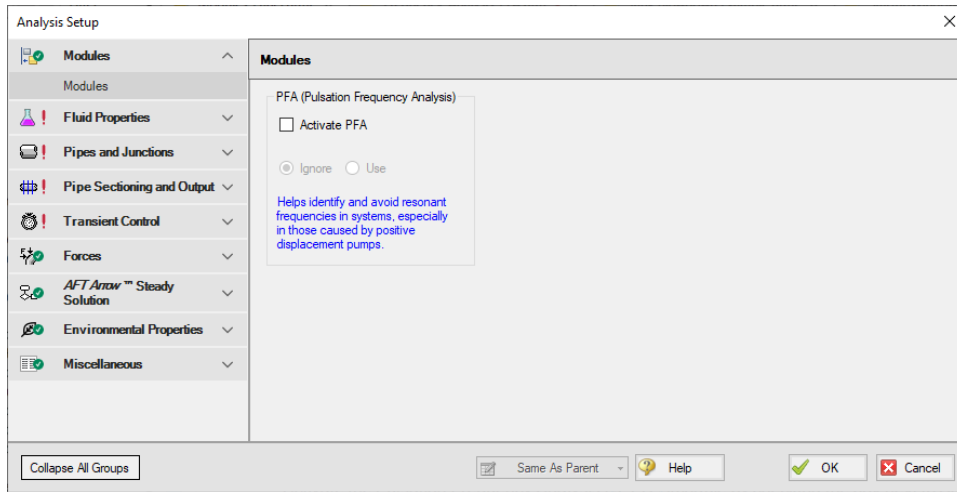
*Activating* a module checks out a license seat for the module.

*Enabling* a module turns on all functionality for the module. By default activating a module will also automatically enable the module for use. Disabling a module keeps the license seat checked out but the module calculations are not applied to the model. Disabling a module may be desired for various reasons, as is discussed further for each module below.

---

**Note:** A module specific license is required to activate the PFA add-on module.

---



**Figure 1: The default state of the Modules panel**

## PFA (Pulsation Frequency Analysis)

Activate the Pulsation Frequency Analysis (PFA) module by checking the box next to Activate PFA. Once activated, enable it by selecting Use or disable it by selecting Ignore. When the PFA module is activated, the Pulsation Setup group appears in the Analysis Setup. The Pulsation Setup group has one item, Pulse Setup.

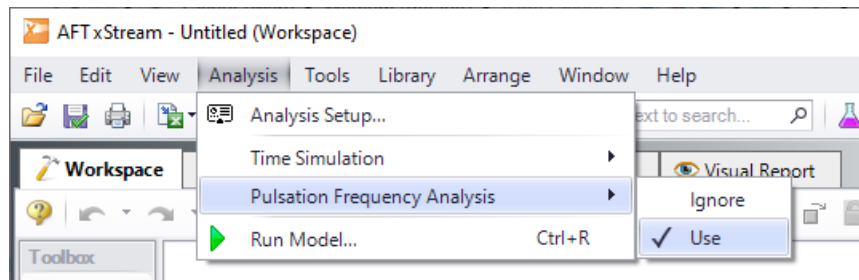
The PFA module can be enabled before or after a model is built. Pre-existing models built with the base AFT xStream application can have the PFA module activated.

Setting the PFA module to Ignore causes the PFA module to function like the base AFT xStream application. One difference is that users are still able to enter pulsation data and it be retained in the model. If a model with PFA input is opened in the base AFT xStream application that cannot access a license to PFA, this data will be lost. Table 1 relates the differences between using AFT xStream without the PFA module activated, using it with the PFA module activated but disabled (Ignore mode), and using it with the PFA module activated and enabled.

The PFA module can also be toggled between Use and Ignore from the Analysis menu in the Pulsation Frequency Analysis option as shown in Figure 2. Use this feature to quickly toggle the PFA module between Use and Ignore without opening the Modules panel in Analysis Setup.

**Table 1: AFT xStream feature accessibility based on PFA activation and enablement**

Feature	PFA Not Activated	PFA Activated	
		PFA Disabled	PFA Enabled
Pulsation Setup group in Analysis Setup	Not Visible	Visible	Visible
Pulsation Frequency Analysis in Analysis menu	Not Visible	Visible	Visible
Pulsation Setup in Analysis menu	Not Visible	Visible	Visible
Steady-State Pulsation Tolerance on Tolerance panel	Not Visible	Not Visible	Visible
Frequency tab in Graph Control, Graph Results window	Not Visible	Not Visible	Visible



**Figure 2: Select Use from the Pulsation Frequency Analysis in the Analysis menu to perform pulsation analysis**

## Fluid Properties Group

The Fluid Properties group in [Analysis Setup](#) gives you control over the fluid properties that influence your pipe system analysis. There is one item in the Fluid Properties Group, the [Fluid panel](#).

## Related Topics

---

References.....	461
-----------------	-----

## Related Blogs

---

[Get More Flexible With Your Fluids](#)

## Fluid Panel

The Fluid panel in the [Analysis Setup window](#) gives you control over the fluid properties that influence your pipe system analysis. A default fluid is not selected - one must be defined to run any simulation.

The first selection that must be made is what Fluids library to use.

The available [Fluid libraries](#) are:

- **AFT Standard** - An included library that offers 30 common fluids to the user. Saturation line data was added for AFT Standard fluids in xStream 2.
- **ASME Steam/Water**- Data from the 1997 ASME Steam Tables.
- **NIST REFPROP** - Data [compiled by](#) the National Institute of Standards and Technology (NIST) using thermophysical property models for a wide variety of fluids. User defined mixtures of these fluids can be defined. For more info see the following sources: [REFPROP FAQ](#), [NIST Chemistry WebBook](#), and [NIST Chemical Name Search](#).
- **Chempak Fluid** - A large library of fluids with detailed property information. User defined mixtures of these fluids can be defined. See the [Chempak Detailed Discussion](#) for more information.

After selecting a library, the Fluids Available in Library list is populated. One of the fluids can be selected and used with the Add To Model button.

When determining which fluid package is most appropriate, it is recommended that you consult the table below to ensure your selection suits your needs.

**Table 1: Compatibility Chart for Fluid library Selection**

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

\* Evaluating the complexity of a system is most easily done in early runs of the model. It is suggested that all early runs should be run with an AFT Standard fluid (even if the real system fluid is a mixture), and the run-time recorded. This runtime should inform your decision on which fluid library is needed to run the more polished runs when high accuracy results are needed. Run times can vary significantly between the 4 libraries, so make sure you evaluate your time constraints and accuracy needs when weighing this decision. Factors that can impact the run time include the ratio in the length of the smallest pipe to the largest pipe, presence of mixtures, usage of Junctions containing graphs, and the presence of sonic choking.

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

### AFT Standard Fluid Library

The AFT Standard fluid library offers multiple fluids for easy use in AFT xStream models. Density and enthalpy can be evaluated for these fluids using any of six equation of state models and three enthalpy models to account for both temperature and pressure dependence.

Transport properties for the AFT Standard fluids (specific heat, dynamic viscosity and thermal conductivity) and Saturation Pressure are assumed to be functions of temperature only using polynomial curve fits. This assumption is generally reasonable below the fluid's critical point, although systems at very high pressure usually start to show some pressure dependence.

To add a fluid into your model, select it from the Fluids Available in Library list and click the Add to Model button. The fluid will appear in the Fluids in Current Model area. Once the fluid is added, specify the temperature in the Fluid Properties area on the right to calculate values of density, dynamic viscosity, and vapor pressure, as shown in Figure 1.



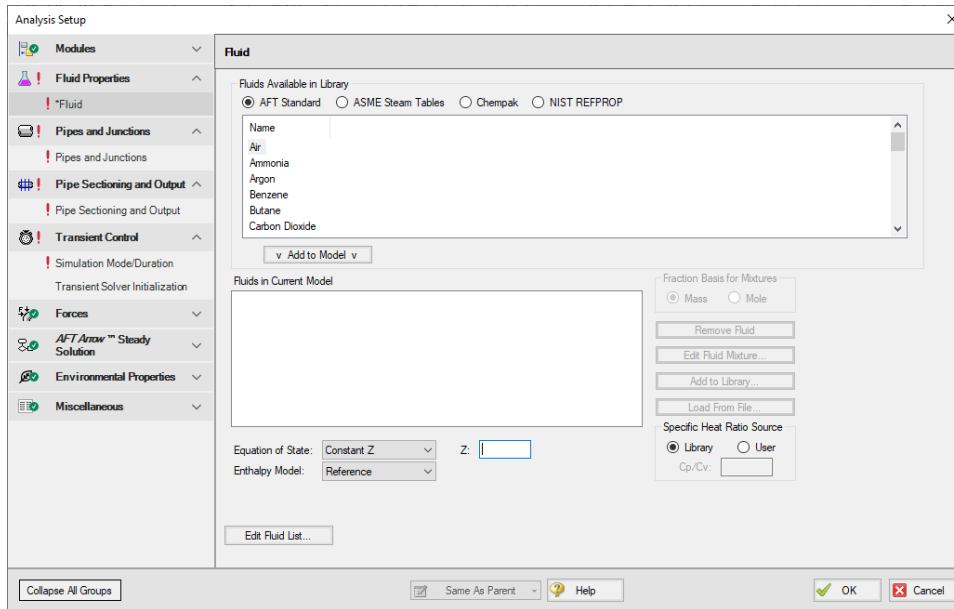


Figure 1: The Fluid panel in the Analysis Setup window with AFT Standard library selected

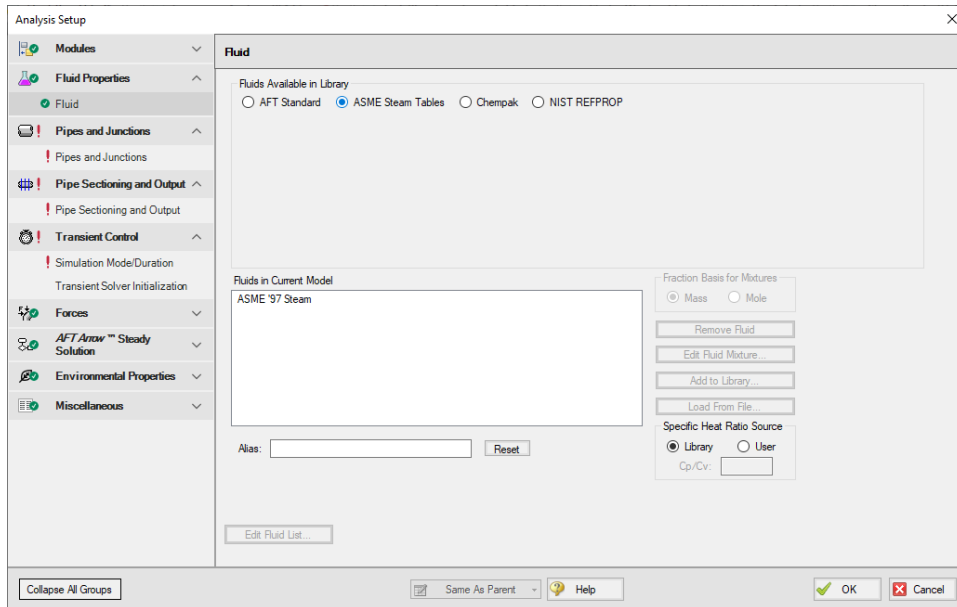
## ASME Steam Tables

The ASME Steam Tables Library offers users access to water and steam properties obtained from the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties Steam (see [ASME Press in References](#)).

After selecting ASME Steam/Water, "ASME '97 Steam" is automatically added to in the Fluids in Current Model area as shown in Figure 2.

Detailed fluid properties information from SteamCalc can be calculated using the AFT SteamCalc Viewer, which can be accessed from Tools > Fluid Property Viewers > SteamCalc. The SteamCalc Viewer will be launched as a separate application, which can then be used to calculate fluid properties at various conditions.

For more information on AFT SteamCalc Viewer see the [SteamCalc Help file](#).



**Figure 2: Fluid Panel in Analysis Setup with ASME Steam library selected**

## Chempak Library

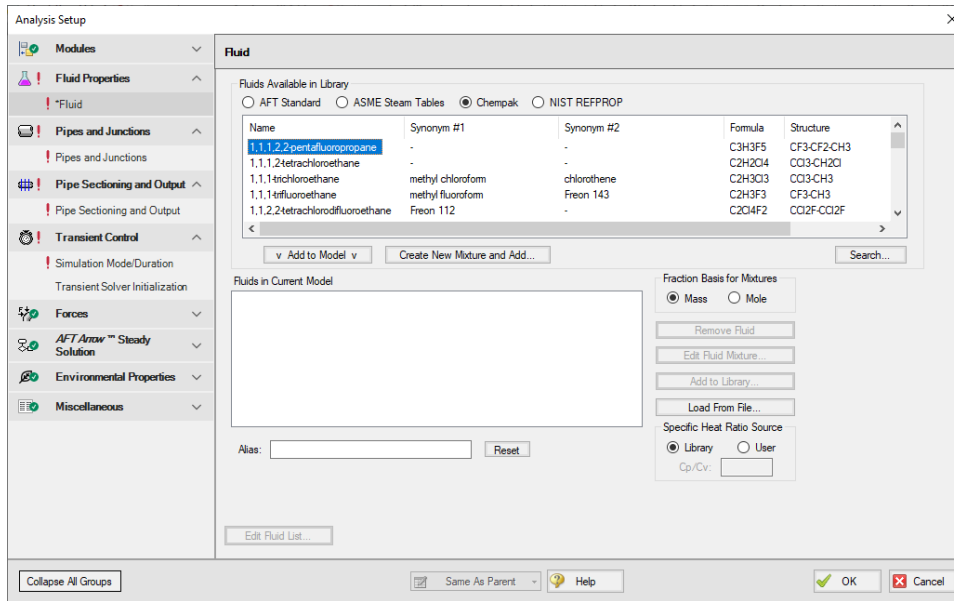
The optional add-on Chempak Data offers approximately 700 fluids to the user. In addition, it offers non-reacting mixture calculations.

A Chempak fluid is added the same way as an AFT Standard fluid is added. To add a Chempak fluid, select Chempak Fluid from the Fluid panel in the Analysis Setup window. From here, select the appropriate fluid from the library and click the Add to Model button. After a fluid is added to the model, it will appear in the lower list under Fluids in Current Model.

Information on the theory and calculation methods used in Chempak is given in the [Chempak technical information](#) topic.

Detailed fluid properties information from Chempak can be calculated using the AFT Chempak Viewer, which can be accessed from Tools > Fluid Property Viewers > Chempak. The Chempak Viewer will be launched as a separate application, which can then be used to calculate fluid properties at various conditions. A Chempak Data license is required to use the Chempak viewer.

For more information on AFT Chempak Viewer see the [Chempak Viewer Help File](#).



**Figure 3: Fluid Panel in Analysis Setup with Chempak library selected**

## NIST Refprop Library

The NIST REFPROP Library offers users access to fluid properties from Version 10 of the NIST Reference Fluid Thermodynamic and Transport Properties Library.

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model, as shown in Figure 3.

Fluids from the NIST REFPROP library support [fluid mixtures](#).

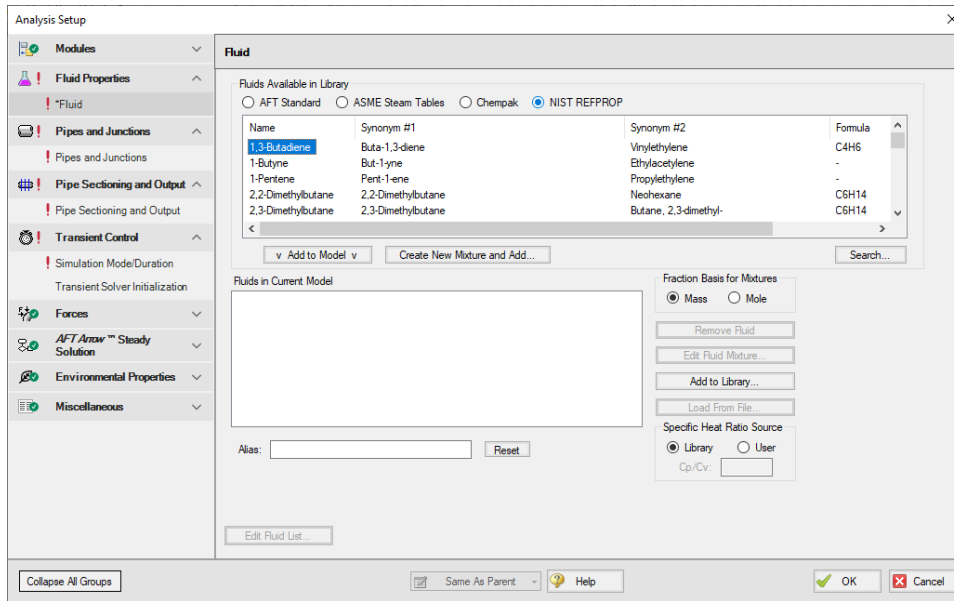


Figure 3: Fluid Panel in Analysis Setup with NIST Refprop selected

## Related Examples

High Pressure Steam - Fluid Property Models Example ..... 590

## Related Topics

References ..... 461  
 Use an AFT Standard Fluid ..... 527

## Related Blogs

[Get the Most Out of AFT's Fluid Property Viewers](#)

[Get More Flexible With Your Fluids](#)

## Creating Mixtures

Users have the option of creating a mixture of fluids with NIST REFPROP and Chempak.

To create a mixture, first select the Chempak or NIST REFPROP Fluid Libraries in the [Fluid panel](#). Once this has been selected, the option to Create New Mixture and Add... button becomes available. Select this button, then locate and highlight the first fluid in the mixture you wish to specify. Enter the mass or mole fraction of the fluid in the mixture, and click Add Fluid to Mixture. Repeat this process for all fluids in the mixture as shown in Figure 1.

**Note:** The fraction basis of all mixtures is either mass or mole, and it is specified on the Fluid panel itself. If you would like to change the fraction basis for the mixture (from mass to mole or from mole to mass), click on the desired fraction basis located on the right of the Fluid panel. Note that this option is only available after the model fluid is set as a Mixture.

To add a mixture to the Fluid panel, the Total Fraction for the table must equal 100%. To help achieve this, the Set as Remainder button can be used to add the remaining percentage to the selected component in the table so that the fractions in the table will add up to 100%. Alternatively, the Ratio Fractions to Total button will scale up the fractions in the table so that they will add to 100%.

Components can be deleted from the table by using the Remove Component or Clear All Components buttons.

Mixture data can be exported to a file which can then be imported into other AFT applications, or shared with other AFT application users by using the Export Mixture to File... and Import Mixture from File... buttons in this window.

**Note:** The NIST REFPROP fluid library is not able to calculate the mixing parameters for most mixtures containing steam.

The screenshot shows the 'Create Mixture' dialog box. At the top, there is a section 'Fluids Available In Library' with a table listing various fluids and their synonyms. Below this is a 'New Mixture Name' field containing 'New Mixture 1'. The 'Composition' section contains a table with two columns: 'Components in Mixture' and 'Mass Fraction (%)'. The table has 8 rows, with the first two rows populated: Methane (80%) and Ethane (20%). To the right of the table are several buttons: 'Set as Remainder', 'Ratio Fractions to Total', 'Remove Component', 'Clear All Components', 'Import Mixture from File...', and 'Export Mixture to File...'. At the bottom, there is a 'Total Fraction' field set to 100%. The dialog also includes 'OK', 'Cancel', and 'Help' buttons at the very bottom.

Name	Synonym #1	Synonym #2
Ethane	Ethane	R-170
Ethanol	Ethyl alcohol	Methyl carbinol
Ethylbenzene	Phenylethane	Benzene, ethyl-
Ethylene	Ethene	R-1150
Ethylene glycol	1,2-Ethandiol	Glycol alcohol
Ethylene oxide	Ethylene oxide	Oxirane

	Components in Mixture	Mass Fraction (%)
1	Methane	80
2	Ethane	20
3		
4		
5		
6		
7		
8		

**Figure 1:** The Create Mixture window (opened from the Fluid panel) allows you to create predefined mixtures for the model.

## Related Blogs

---

[Get More Flexible With Your Fluids](#)

### Equation of State Models

AFT xStream uses the Ideal Gas model with a corrected compressibility factor,  $Z$ , to model AFT Standard fluids. The  $Z$ -factor can be calculated by numerous correlations. xStream offers 6 options: Constant  $Z$ , Ideal Gas, Peng-Robinson, Redlich-Kwong, Soave Redlich-Kwong, and Three-Parameter.

The Ideal Gas option leaves the  $Z$ -factor defined at 1, meaning the fluid is modeled as an Ideal Gas. The Constant  $Z$  option allows the user to specify a value for  $Z$  which xStream then uses for all property calculations. The other Equation of State options are discussed in turn below.

### Equations of State Nomenclature

---

There are a few characterizing fluid properties that are common to all of the Equations of state below.

The **reduced temperature** ( $T_r$ ) and **reduced pressure** ( $P_r$ ) are the ratios of the thermodynamic properties to the critical properties. For example,

$$P_r = \frac{P}{P_c}$$

is the definition of the reduced pressure. The reduced temperature could be written similarly.

$\omega$  is the **acentric factor**, for which values can be found in chemical engineering data books. It is a correction factor that can be set to zero if you cannot find data for the fluid. However, accuracy will be reduced.

### Redlich-Kwong

---

The Redlich-Kwong Equation of State is given by the following cubic equation.

$$P = \frac{RT}{V - b} - \frac{a}{T^{0.5} V(V + b)}$$

where

$$a = 0.42748 \frac{R^2 T_c^{2.5}}{P_c}$$

$$b = 0.08664 \frac{R T_c}{P_c}$$

And  $P_c$  and  $T_c$  are the critical pressure and temperature, respectively.  $V$  is the molar volume.

An alternative form of the Redlich-Kwong equation can be obtained by multiplying the Redlich-Kwong equation by  $V/RT$ .

$$Z = \frac{1}{1-h} - \frac{a}{bRT^{1.5}} \left( \frac{h}{1+h} \right)$$

where

$$h \equiv \frac{b}{V} = \frac{b}{ZRT/P} = \frac{bP}{ZRT}$$

Elimination of the "a" and "b" terms gives

$$Z = \frac{1}{1-h} - \frac{4.9340}{T_r^{1.5}} \left( \frac{h}{1+h} \right)$$

$$h = \frac{0.08664 P_r}{Z T_r}$$

where  $T_r$  and  $P_r$  are the [reduced temperature](#) and [reduced pressure](#).

---

### Soave-Redlich-Kwong (SRK)

---

Soave proposed a modified version of the Redlich-Kwong method as follows

---

$$P = \frac{RT}{V - b} - \frac{a(T)}{V(V + b)}$$

where

$$a(T) = 0.42747 \frac{R^2 T_c^2}{P_c} [1 + m (1 - T_r^{0.5})]^2$$

$$m = 0.480 + 1.574\omega - 0.176\omega^2$$

where  $T_r$  is the reduced temperature, and  $\omega$  is the [acentric factor](#).

The SRK equation can also be written as in terms of Z as

$$Z^3 - (1 - B) Z^2 + (A - 3B^2 - 2B) Z - (AB - B^2 - B^3) = 0$$

where

$$A = \frac{aP}{R^2 T^2}$$

$$B = \frac{bP}{RT}$$

The largest root of Z for the cubic equation is the compressibility factor for the vapor form of the fluid.

## Peng-Robinson

---

The Peng-Robinson Equation of State uses a similar form to the Redlich-Kwong equation of state cubic equation. The Peng-Robinson cubic equation is given as follows



$$P = \frac{RT}{V - b} - \frac{a\alpha}{V(V + b) + b(V - b)}$$

where

$$a = 0.45724 \frac{R^2 T_c^2}{P_c}$$

$$b = 0.07780 \frac{RT_c}{P_c}$$

$$\alpha = \left[ 1 + (0.37464 + 1.54226\omega - 0.26992\omega^2) (1 - T_r^{0.5}) \right]^2$$

And  $P_c$  and  $T_c$  are the critical pressure and temperature, respectively,  $V$  is the molar volume,  $T_r$  is the [reduced temperature](#) and  $\omega$  is the [acentric factor](#).

The above equation for pressure can be rewritten as

$$Z^3 - (1 - B) Z^2 + (A - 3B^2 - 2B) Z - (AB - B^2 - B^3) = 0$$

where

$$A = \frac{aP}{R^2 T^2}$$

$$B = \frac{bP}{RT}$$

The largest root of  $Z$  for the above equation is the compressibility factor of the vapor form of the fluid.

---

### Three-parameter

---

xStream implements a Three-Parameter Equation of State derived by Pitzer for the second coefficient in the Virial Equation of State. The Three-Parameter model as used by xStream is taken from page 102 of [Smith, van Ness, and Abbott](#) and shown below.

$$Z = 1 + B_0 \frac{P_r}{T_r} + \omega B_1 \frac{P_r}{T_r}$$

where Z is the compressibility factor, and Pr and Tr are the [reduced pressure and temperature](#), respectively.

The constants B0 and B1 are given by the following

$$B_0 = 0.083 - \frac{0.422}{T_r^{1.6}}$$

$$B_1 = 0.139 - \frac{0.172}{T_r^{4.2}}$$

where  $\omega$  is the [acentric factor](#).

### Related Topics

---

Nomenclature.....	410
References.....	461

### Enthalpy Models

For superheated gases, enthalpy is a function of both temperature and pressure:

$$h = f(P, T)$$

However, enthalpy is more strongly a function of temperature, and thus you will frequently see the approximation as follows:

$$h_1 - h_2 = c_p (T_1 - T_2)$$

where the specific heat at constant pressure is used as a proportionality constant. For processes that are not at constant pressure, the above approximation starts to lose accuracy.

AFT xStream offers three ways to calculate the enthalpy for the AFT Standard Fluid Library. Below is a description of each.

## Reference Enthalpy

---

The Reference Enthalpy Method uses the following equation to calculate enthalpy

$$h = h_{ref} + C_{p,avg} (T - T_{ref})$$

The reference enthalpy and temperature is entered in the fluid library, and should be for a state that is as near as possible to the operating conditions of your model. As seen from the equation, the pressure dependence of enthalpy is ignored. In many cases, this assumption is acceptable.

## Generalized Enthalpy

---

The Generalized Enthalpy Method is an extension of the Reference Enthalpy Method. The Generalized Enthalpy, similar to the [Three-Parameter Method](#), uses critical property data to calculate a Residual Enthalpy. The method is shown below.

$$h = h_{ref} + C_{p,avg} (T - T_{ref}) + h_{residual}$$

where:

$$h_{residual} = RT_c P_r \left[ B_0 - T_r \frac{dB_0}{dT_r} + \omega \left( B_1 - T_r \frac{dB_1}{dT_r} \right) \right]$$

B0 and B1 are as given previously, and:

$$\frac{dB_0}{dT_r} = \frac{0.675}{T_r^{2.6}}$$

$$\frac{dB_1}{dT_r} = \frac{0.722}{T_r^{5.2}}$$

### Table Lookup Enthalpy

The third method of obtaining enthalpy data is by using AFT xStream's two-dimensional table. In this method, a sequence of polynomial curve fits as a function of temperature are used. Each step in the sequence is for a different pressure.

$$P_1 = 25 \text{ psia}$$

$$h_1 = a_1 + b_1 T + c_1 T^2 + d_1 T^3 + e_1 T^4$$

$$P_2 = 50 \text{ psia}$$

$$h_2 = a_2 + b_2 T + c_2 T^2 + d_2 T^3 + e_2 T^4$$

With this method, enthalpies would be obtained by interpolating the polynomials between 25 and 50 psia at a given temperature. Up to 10 pressure table entries can be used. If the pressure is outside the table range, the enthalpy is extrapolated.

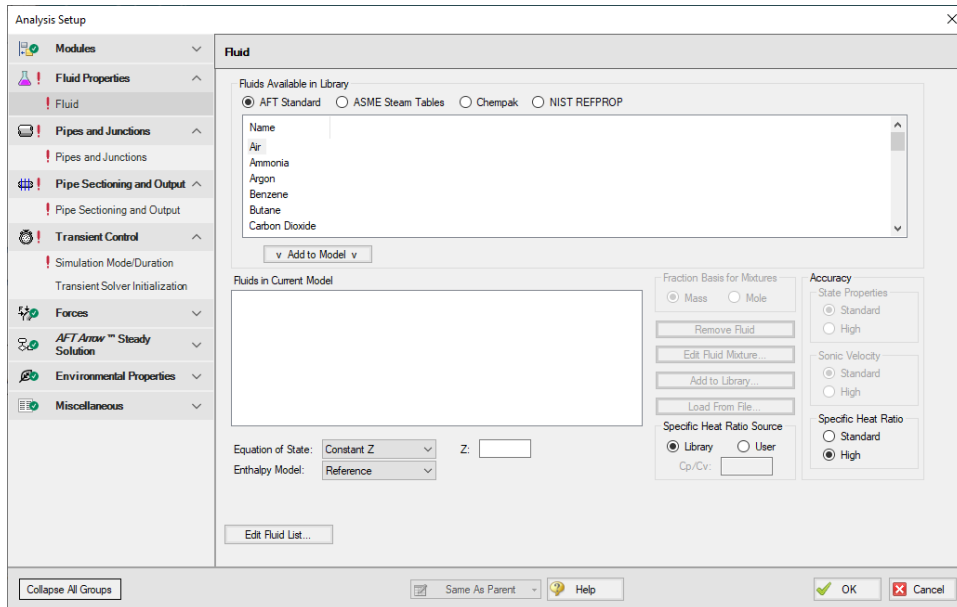
### Related Topics

Nomenclature .....410

### Fluid Accuracy Settings

Several advanced settings are available to fine-tune the accuracy of the fluid properties for the xStream simulation. These options can be enabled from Tools -> [User Options](#) by going to the Show options under General (global).

Once enabled the Fluid Accuracy settings are located on the [Fluid panel](#) below the Fluids Available in Library area (Figure 1).



**Figure 1: Fluid panel in Analysis Setup with the Fluid Accuracy Settings enabled**

## Specific Heat Ratio

There is a choice of standard or high accuracy for the specific heat ratio. The standard accuracy uses [Equation 80](#) for  $\gamma$ . The high accuracy uses [Equation 81](#). The standard accuracy will usually be suitable, but for completeness you may want to use the high accuracy in your final runs and see if the results change.

The high accuracy option should not have a significant affect on run time (at most 10%). However, it can result in convergence problems.

This option is applicable to both the AFT Standard as well as Chempak fluids.

## State Properties

The State Properties option influences the number of elements used in curve fits for calculating property data. You will usually want to start with standard accuracy, and use high accuracy for final runs, depending on the importance and run time for the model. The high accuracy option will increase run time by about a factor of two.

When performing interpolations for properties, a 4 point or 16 point interpolation scheme can be used. The 4 point scheme implemented in AFT xStream is the Standard Accuracy option. The 16 point scheme is the high accuracy option. Use of the High accuracy option will roughly double the run times for models.

This option is applicable to only to Chempak fluids.

## Sonic Velocity

The standard accuracy sonic velocity option will usually be acceptable. The standard option uses [Equation 87](#) for all sonic velocity calculations. The high accuracy option uses [Equation 84](#), calculating the derivative using a numerical difference (also called perturbation derivative) along the isentropes. This is time consuming, and can cause models to run ten times as long. You should not use this option unless you have reason to believe Equation 87 has unacceptable accuracy as evidenced from the output values of sonic velocity.

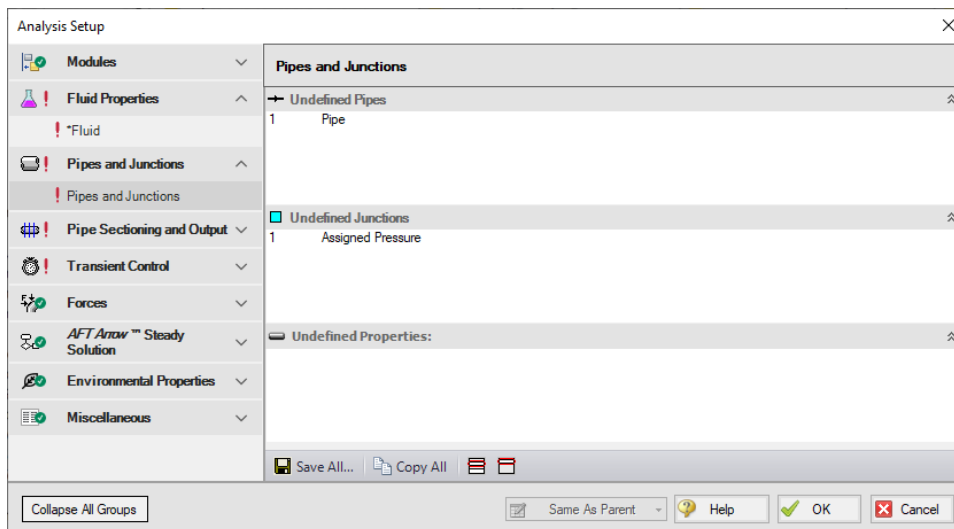
This option is applicable to only to Chempak fluids.

## Pipes and Junctions Group

The Pipes and Junctions Group in [Analysis Setup](#) has one item: the [Pipes and Junctions panel](#). When the Pipes and Junctions group has a green checkmark next to its name, all pipes and junctions are fully defined. When a red exclamation point is next to its name, there is at least one pipe or junction that is undefined. The model cannot run until all pipes and junctions are defined.

## Pipes and Junctions Panel

The Pipes and Junctions panel within Analysis Setup displays the status of pipes and junctions within the model.

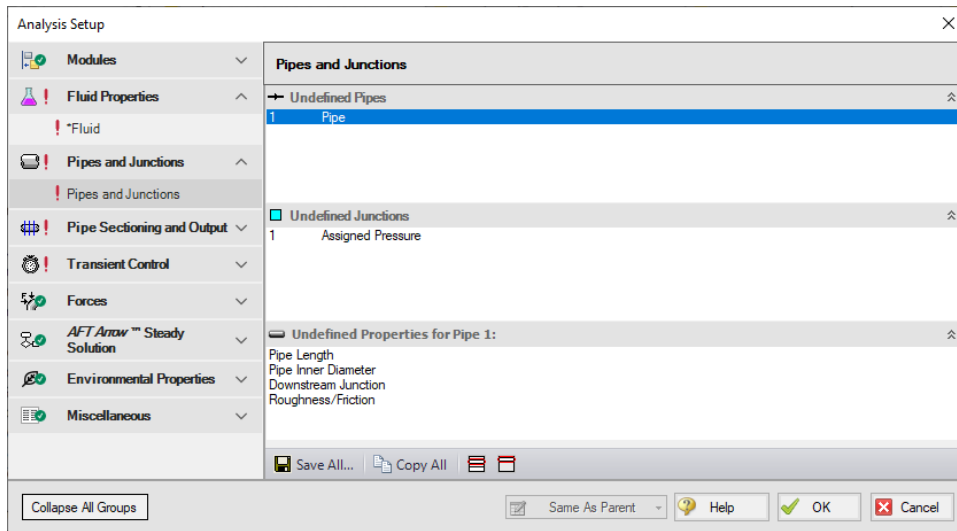


**Figure 1: Default state of the Pipes and Junctions panel**

This panel is made up of three sections:

- Undefined Pipes
- Undefined Junctions
- Undefined Properties

The Undefined Pipes and Undefined Junctions sections displays the number and name of all pipes and junctions, respectively, that are not fully defined. If all pipes and junctions in the model are fully defined, the messages "All Pipes Are Defined" and "All Junctions Are Defined" will be displayed. Clicking on the undefined pipe or junction will display the undefined properties in the list below, as shown in Figure 2. All undefined data for all objects can be saved to the file or copied to the clipboard. Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.



**Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed**

## Pulsation Setup Group

([PFA module only](#)) The Pulsation Setup group in [Analysis Setup](#) allows you to define the pulsation analysis parameters.

The Pulsation Setup group has one panel, the Pulse Setup panel. The Pulse Setup panel allows the user to define properties of the pulse including the source, cutoff frequency, low-pass filtering, and magnitude.

## Pulse Setup Panel

The Pulse Setup panel (Figure 1) allows you to specify required input for the [PFA module](#), including the junction at which the pulse will be applied, as well as the pulse properties. This panel will not be visible unless the PFA module has been activated from the [Modules panel](#).

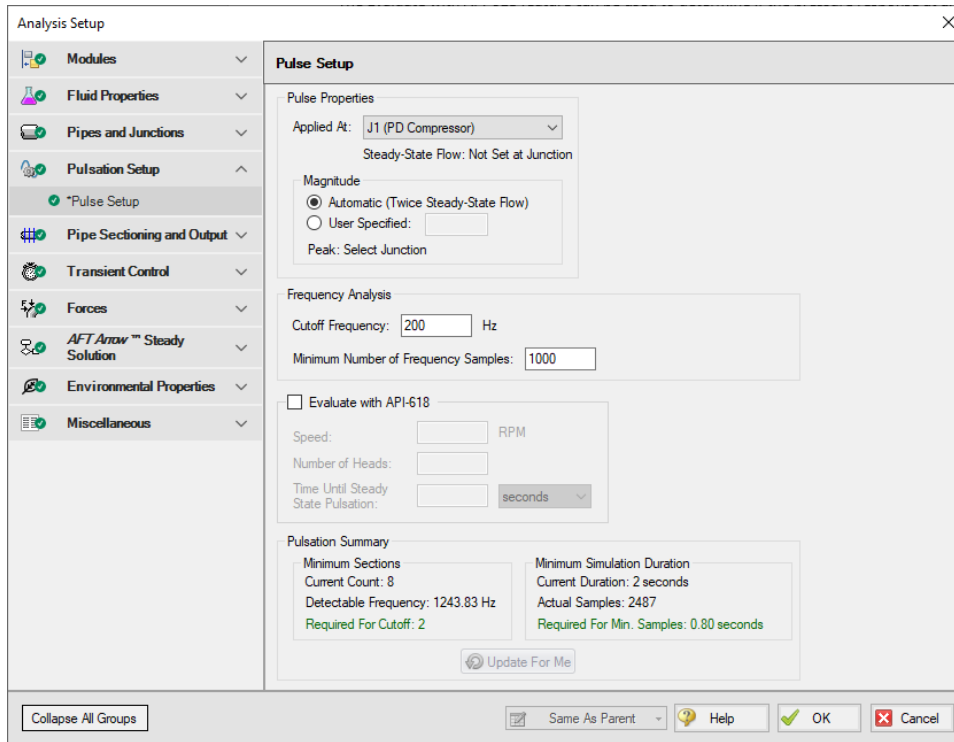


Figure 1: Pulse Setup panel in the Analysis Setup window

## Pulse Properties

At the top of the window the Pulse Properties can be specified, including the junction that will act as the source of the pulse and the magnitude of the pulse. By default, the magnitude of the pulse is twice the steady-state flow, but you have the option to define this magnitude as “User Specified”. This pulse will be assumed to last for two time steps and then return to steady-state flow. The pulse can only be applied to an assigned flow junction.

## Cutoff Frequency

The cutoff frequency in xStream is the maximum frequency for displayed output. The defined cutoff frequency must be less than or equal to the detectable frequency for the analysis, which is calculated using the equation for maximum frequency as follows

$$Max\ Frequency = \frac{1}{2\ dt}$$

where dt is the time step. The time step for the model is not known until run-time; However, the time step can be estimated based on the inputs on the [Pipe Sectioning panel](#). The Pulsation Summary at the



bottom of the Pulse Setup panel shows the Detectable Frequency based on the current settings on the Pipe Sectioning panel, as well as the minimum number of sections required to capture the specified cutoff frequency. If the current Cutoff Frequency and sectioning are not valid for the analysis, then either the Cutoff Frequency should be decreased, or the minimum number of sections should be increased.

When the Analysis is run it is possible to account for all frequencies up to the Detectable Frequency calculated based on the time step. However, the results in the output will only display frequencies up to the user defined Cutoff Frequency. Therefore it is important to define a large enough Cutoff Frequency to be able to analyze all relevant frequency responses.

---

### Minimum Number of Frequency Samples

---

The minimum number of frequency samples used to generate the forcing function is specified in the Frequency Analysis section of the Pulse Setup panel. The default of 1000 should be sufficient for most cases. The Minimum Number of Frequency Samples will impact the required simulation duration - a larger number of samples will require a longer simulation duration. The required simulation duration can be calculated by dividing the minimum number of frequency samples by the detectable frequency. If the current simulation duration is not long enough to accommodate the minimum number of frequency samples, then the simulation duration should be lengthened on the [Simulation Mode/Duration panel](#), or the minimum number of frequency samples should be decreased.

---

### Evaluate with API-618

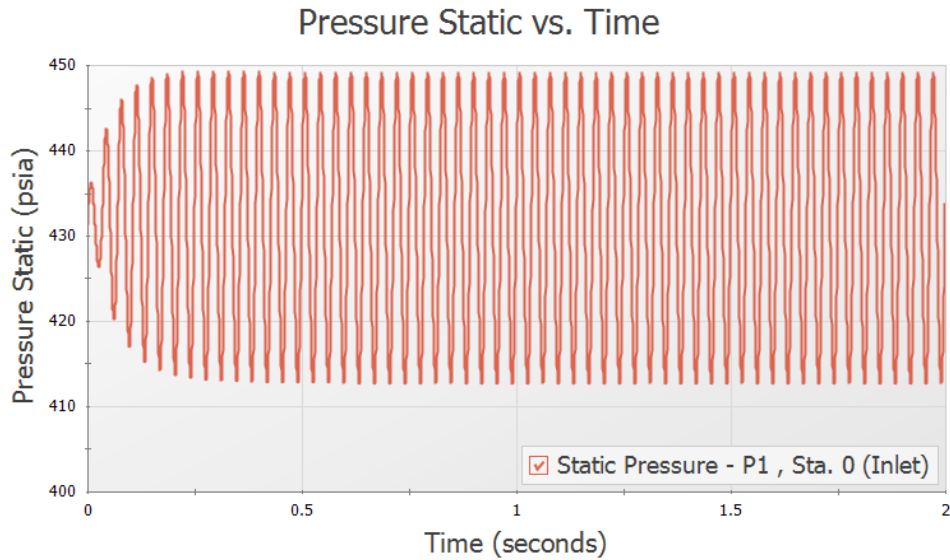
---

The Evaluate with API-618 feature can be used to determine if the pressure response at any of the acoustic natural frequencies of the system exceeds the limits defined in API-618 for reciprocating compressors. To evaluate API-618 the model should be first be run with PFA enabled and the API-618 feature disabled to determine the acoustic frequencies. Once the excitation frequencies are known, child scenarios can be created by the user to evaluate the pressure results when the compressor operates at the excitation frequencies, as is demonstrated in the last step of the [Frequency Analysis Example](#). The API-618 evaluation should be enabled in the excitation frequency child scenarios to check if the pressure results exceed the limits in API-618.

The required inputs to check API-618 are as follows:

- **Speed** - The speed the reciprocating compressor is operating at.
- **Number of Heads** - The number of pistons in the reciprocating compressor.
- **Time Until Steady State Pulsation** - The time required for the dissipation of artificial transients caused by the introduction of a flow rate wave to the system. This time should be determined by running the scenario with the "Time Until Steady State Pulsation" set to 0, then plotting the pressure results at the outlet of the compressor. The Time Until Steady State Pulsation should be set as the time at which the pressure waves have consistent amplitudes. For example, in Figure 2 below the waves begin to reach constant amplitude at about 0.25 seconds. It is best to choose a slightly longer time interval than the estimated time interval for increased accuracy when checking API-618.

Note the Time Until Steady State Pulsation input only changes which pressure results are included in the API-618 evaluation, the transient results are not impacted by this time.



**Figure 2: Example pressure results at the outlet of the reciprocating compressor with an oscillating flow transient specified.**

The equations for the API 618 peak to peak pressure level limits are as follows ([American Petroleum Institute 1995](#), pg 40):

$$P_1 (\%) = \left( \frac{300}{(P_L ID f)^{0.5}} \right) \quad \text{US Units}$$

$$P_1 (\%) = \left( \frac{397.1}{(P_L ID f)^{0.5}} \right) \quad \text{SI Units}$$

where  $P_1(\%)$  is the maximum allowable peak to peak pressure level of individual pulsation components at fundamental and harmonic frequencies, as a percentage of absolute line pressure,  $P_L$  is the average absolute line pressure in psi (bar for SI),  $ID$  is the inside diameter of the pipe in inches (mm), and  $f$  is the pulsation frequency in hertz.

## Pulsation Summary

The Pulsation Summary section at the bottom of the Pulse Setup panel provides information on the current transient settings in the model to check that the sectioning and simulation duration are sufficient based on the user-specified Cutoff Frequency and Minimum Number of Frequency Samples. Therefore the Pulsation Summary will only be able to display results if the [Pipe Sectioning panel](#) and [Simulation Mode/Duration panel](#) have already been fully defined.

The Update For Me button under the Pulsation Summary can be used to update the Minimum Number of Sections and/or the Stop Time for the simulation if they are not sufficient based on the user-specified Cutoff Frequency and Minimum Number of Frequency Samples.

A short description of each of the Pulsation Summary items is given here.

- Minimum Sections
  - **Current Count** - Displays the "Minimum Number of Sections Per Pipe" defined on the Pipe Sectioning panel
  - **Detectable Frequency** - Displays the Max Frequency calculated as discussed above in the Cutoff Frequency section
  - **Required for Cutoff** - Displays the required Minimum Number of Sections Per Pipe which allow the Detectable Frequency to be greater than or equal to the Cutoff Frequency
- Minimum Simulation Duration
  - **Current Duration** - Displays the current simulation duration defined on the Simulation Mode/Duration panel (Stop Time minus Start Time)
  - **Actual Samples** - Displays the Actual Samples used for the frequency analysis, calculated by multiplying the Current Duration by the Detectable Frequency
  - **Required For Min. Samples** - Displays the minimum required simulation duration based on the user-specified Minimum Number of Frequency Samples and the calculated Detectable Frequency

### Pipe Sectioning and Output Group

The Pipe Sectioning and Output Group in [Analysis Setup](#) has one item: the [Pipe Sectioning and Output panel](#). The Sectioning panel contains settings related to setting up the [Method of Characteristics](#) grid used for the transient solution method.

### Related Topics

---

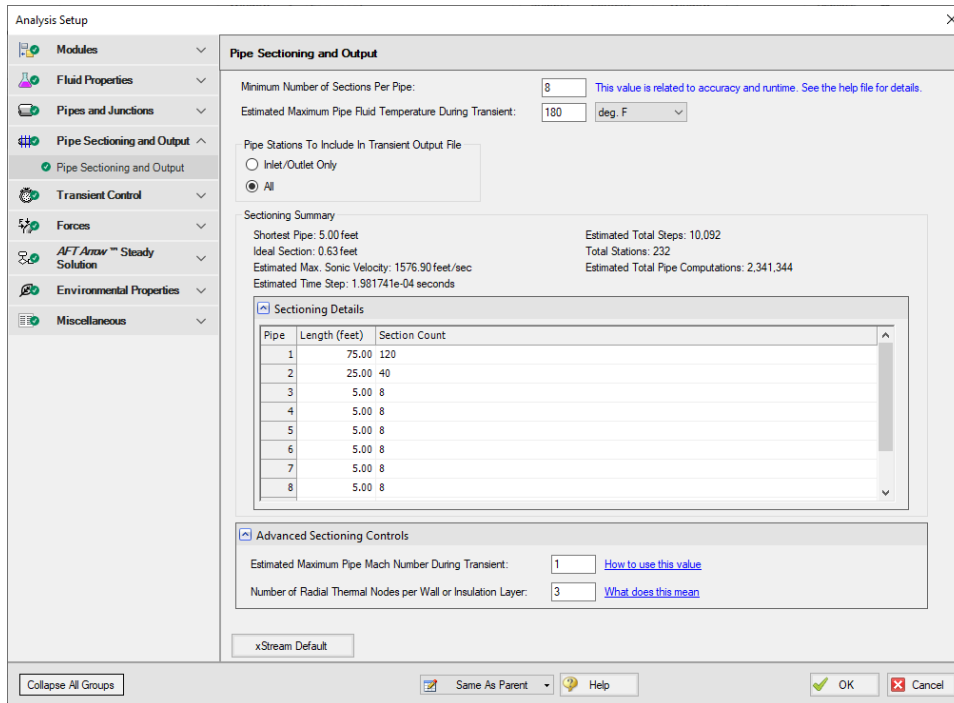
Method of Characteristics.....435

### Pipe Sectioning and Output Panel

Before running a transient simulation, there are several pipe sectioning options that must be defined to section the pipes as described in [Pipe Sectioning - Introduction to Method of Characteristics](#).

The user is required to specify the Sectioning item when the Transient [Simulation Mode](#) is selected. How the pipes are sectioned is controlled in the Pipe Sectioning and Output panel of Analysis Setup.

When the system has been defined, the Pipe Sectioning and Output panel will also show a Sectioning Summary which contains details of the sectioning including the estimated maximum sonic velocity, the estimated time step, and the estimated total steps. These numbers are only estimated values, as the actual values are determined after xStream gets results from the AFT Arrow Steady Solution. However, these estimates can help the user understand how many sections the system will have, and thus how long a run might take.



**Figure 1: The Sectioning panel in the Analysis Setup window**

## Sectioning Options

- Minimum Number of Sections Per Pipe** - Defines the minimum number of calculation sections that will be used in each pipe in the model. Some pipes may use more than the minimum number of sections depending on the time step and maximum sonic velocity in the pipe. The ideal number of sections will vary based on the relative length of the shortest pipe compared to other pipes in the model, and the speed of transients in the model.
- Estimated Maximum Pipe Fluid Temperature During Transient** - Temperature used to determine the maximum sonic velocity for sectioning calculations. If the actual maximum temperature is not known, a higher temperature than expected should be used.

## Transient Output Options

The **Pipe Stations To Include In Transient Output File** option controls how much data is saved to the file. By default all time steps for all pipe stations will be saved to file. The user may also set only the pipe inlet/outlet station data to be saved to file. Saving only the inlet and outlet stations to file will decrease the run time for the model, but also means that the user will not be able to plot the results at interior calculation stations for the pipes.

## Numbering Convention for Pipe Sectioning

Once a pipe is broken into sections, computation takes place where the sections join together. These are called pipe stations. The total number of stations in a pipe is equal to the number of sections plus 1. The station at the beginning of the pipe is numbered zero.

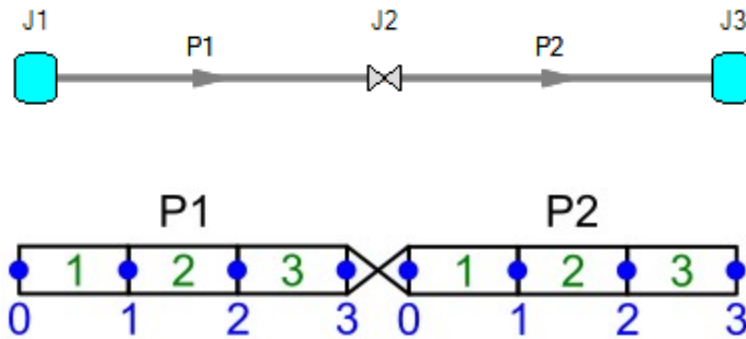


Figure 2: Computing Stations (blue) exist at the boundary of every section (green)

To observe parameters at the inlet or outlet of a junction, view the station adjoining the junction. In the above example, the inlet to the valve is P1, Station 3 and the outlet is P2, Station 0.

## Advanced Sectioning Controls

- Estimated Maximum Pipe Mach Number During Transient** - Adjusts the MOC grid time step. An input value of 1 will ensure stability. For simulations with lower maximum Mach Numbers, reducing this input value can improve accuracy and runtime. If Mach Numbers higher than the Estimated Maximum Pipe Mach Number are encountered during simulation, the results will not be valid, and the simulation can become unstable or fail to converge. This can only be edited if advanced options are enabled.
- Number of Radial Thermal Nodes per Wall or Insulation Layer** - Defines the number of calculation nodes that will be used in the pipe wall and in each of the insulation layers when calculating radial conduction. The default setting of 3 calculation nodes per layer should be sufficient for most applications; However, changing the number of radial nodes may improve calculation stability if the heat rate changes rapidly across the pipe wall and insulation. See the topic on [Transient Heat Transfer](#) for more information on how the transient heat transfer calculations are performed.

## Related Examples

Gas Turbine Fuel System Example .....575

## Transient Control Group

The Transient Control Group in [Analysis Setup](#) has two items: the [Simulation Mode/Duration panel](#) and the [Transient Solver Initialization panel](#). The Simulation Duration panel contains settings to define if the model will be run in steady-state or transient and when the simulation start and stop times are. The Transient Solver Initialization panel has settings related to eliminating artificial transients.

## Simulation Mode/Duration Panel

The Simulation Mode/Duration panel allows the user to define settings that will affect the transient results.

The Time Simulation options control what calculations will be performed by xStream. There are two time simulation options:

- **AFT Arrow Steady** - This simulation mode runs the network solution methodology found in AFT Arrow to give steady-state results for the model.
- **Transient** - This simulation mode runs the MOC solution methodology for the transient simulation as defined by the user. This option is the default selection.

The **Start Time** and **Stop Time** refer to the beginning and end times for the simulation. The units can be specified using the drop-down box.

## Transient Solver Initialization Panel

### Artificial Transient Resolution

---

AFT xStream employs two separate solvers in its solution method: the [AFT Arrow Steady](#) Solver is used to calculate the steady state solution to initialize the transient solution, and the [Transient](#) Solver is used to run the [Method of Characteristics \(MOC\)](#) transient solution. The AFT Arrow Steady solver and the Transient solver use different sectioning methods, which causes small discrepancies between the converged Arrow solution and the first time step of the transient solution. If xStream were to proceed directly from the AFT Arrow Steady solution to the Transient simulation, artificial transient waves would be introduced into the system as is shown in Figure 1. Checking the option to **Delay start of transient simulation until artificial transients are resolved** ensures that the artificial transient waves are allowed to die out before the transient simulation begins, as is depicted in Figure 2. It is strongly recommended to allow the artificial transients to resolve before the transient simulation begins for higher result accuracy.

Once xStream determines the artificial transients have been resolved, xStream checks the resolved solution against the AFT Arrow Steady solution. If the velocity results have a larger relative velocity change than the specified **Allowed Mismatch Between AFT Arrow Steady and Resolved Solutions** in the Transient Solver Initialization panel, a warning is given. By default "Do not proceed to the transient simulation..." is enabled to stop the transient run if the mismatch exceeds the allowed mismatch. The user can disable the option that prevents the transient solution from proceeding; however, note that large changes between the AFT Arrow Steady solution and Resolved Solution can introduce error into the transient results.

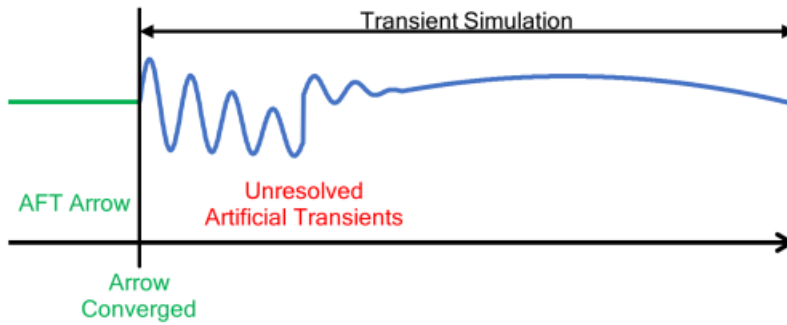


Figure 1: Illustration of the Transient Solver Initialization process without artificial transient resolution

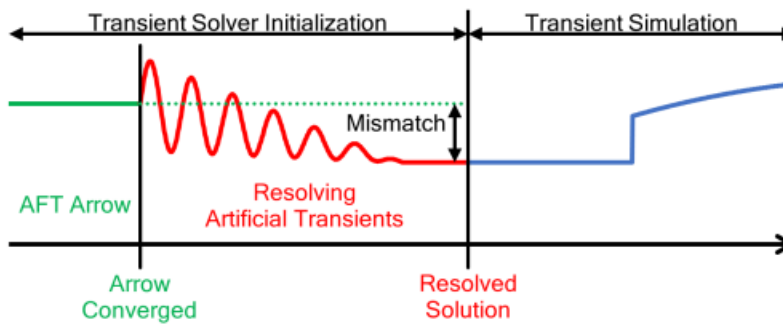


Figure 2: Illustration of the Transient Solver Initialization process with artificial transient resolution

The artificial transients are resolved by running the Method of Characteristics in a series of **Blocks**. Each Block consists of 100 time steps. After each Block is run xStream compares the velocity results at the beginning and end of the block for every calculation station. If the velocity change from the beginning to the end of the block at any point is larger than the allowable tolerance defined in the Transient Solver Initialization panel, then the process is repeated until the **Maximum Number of Blocks** is reached, or the velocity tolerance is met for two consecutive blocks.

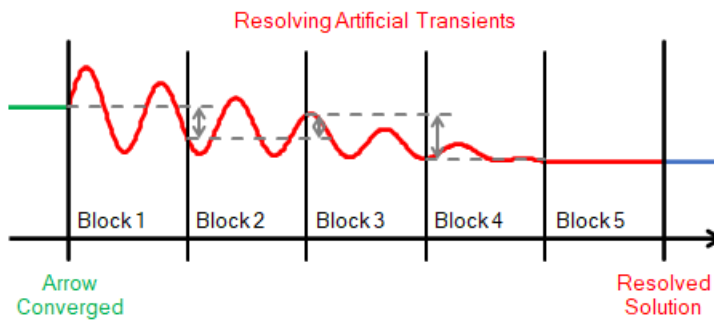


Figure 3: Example of blocks used during artificial transient resolution period

## Advanced Settings

The advanced settings should generally be used for troubleshooting purposes only. There are a number of settings that can be customized.

- **Disable transient junction behavior for the entire simulation** - This option sets all junction transient events to be ignored, and fixes the pressure/temperature at all finite tanks for the duration of the transient. By disabling the junction transient behavior in this way the user can directly view the artificial transients in the model for troubleshooting purposes.
- **Bypass AFT Arrow Steady and start the transient with fixed values for all stations** - This option uses arbitrary guesses to initialize the period of resolving artificial transients instead of using the AFT Arrow Steady solution. The resolved solution should still be the same regardless of the whether the AFT Arrow Steady solution is used or just initial guesses; however, the artificial transient resolution will likely take more time when fixed guesses are used.
- **Allowed mismatch between AFT Arrow Steady and resolved solution** - The allowed velocity mismatch criteria can be modified here. Additional pressure and temperature criteria can also be applied by checking the corresponding boxes.
- **A series of time steps (a block) is considered resolved when every station meets these tolerances from beginning to end of the block** - The tolerances in this section are used to determine if the artificial transients are resolved by comparing the tolerance values to the velocity change at the beginning and end of each block as described in the Artificial Transient Resolution section above. Additional pressure and temperature criteria can also be applied by checking the corresponding boxes.
- **Iteration Limits**
  - **Maximum number of blocks** - The maximum number of blocks that xStream will run to attempt to resolve the artificial transients.
  - **Time steps per block** - The number of time steps that will be run in each block. If there are too few time steps the blocks may falsely converge.
  - **Successive blocks required** - The number of successive blocks that must meet the tolerances for the artificial transients to be considered resolved.

## Related Topics

Transient Result Accuracy.....	49
Method of Characteristics.....	435

## Forces Group

The Forces group in [Analysis Setup](#) has one item: the [Forces panel](#). By default, the Forces group is fully defined and will have a green checkmark next to its name.

## Related Topics

Transient Forces.....	30
-----------------------	----



## Related Examples

High Pressure Steam - Forces Example ..... 606

## Force Definitions Panel

AFT xStream includes the ability to calculate dynamic, unbalanced pneumatic forces. These unbalanced forces occur when a transient pressure wave passes through a portion of the system.

**Note:** As AFT xStream is only a hydraulic solver, only hydraulic forces are accounted for. AFT xStream is not a pipe stress analyzer and as such does not have the ability to directly model piping supports or similar objects. Hydraulic force files can be generated for [import into software](#) designed for force analysis.

## Defining Force Sets

In order to process force data, the locations where the forces are to be calculated must be defined. Force Sets can be defined either from the Workspace or from the Forces panel of the [Analysis Setup window](#).

**Note:** Force calculation accuracy in AFT xStream is dependent not only on accurately defining the force sets, but can also be highly dependent on the [Sectioning](#) and [Fluid panel](#) settings. This is discussed further in the [High Pressure Steam - Forces example](#).

## Defining from Analysis Setup

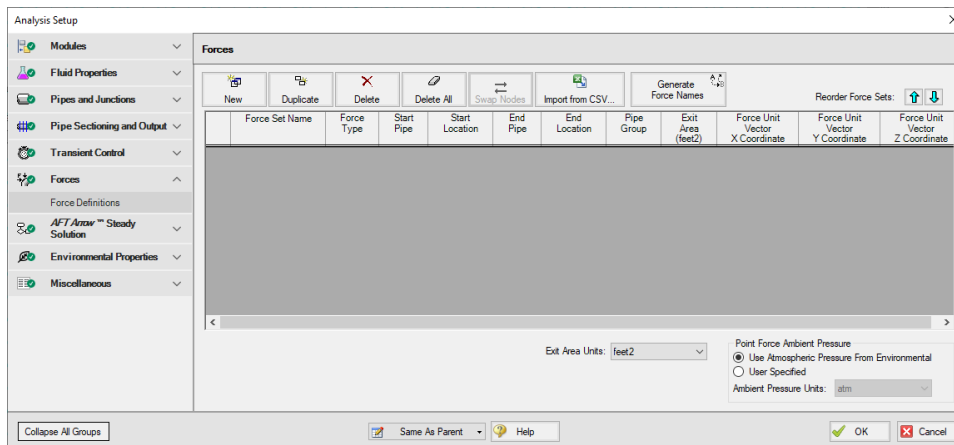


Figure 1: Force set data is defined on the Force Definitions panel in the Analysis Setup window

- **New** - Add a new force set to the table
- **Duplicate** - Copy an existing force set
- **Delete/Delete All** - Remove force sets

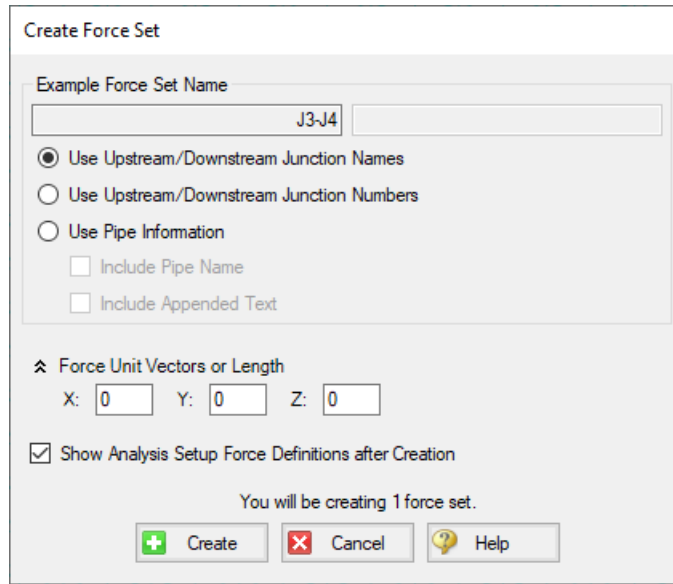
- **Swap Nodes** - The calculation will be the same, but the sign of the force will be reversed. This may be important for proper interpretation of results or for exporting data to other applications.
- **Import from CSV** - If force sets are defined in another application or by hand they can easily be imported with a comma separated value file. Click the Import from CSV button for an explanation on the required format.
- **Generate Force Names** - Automatically changes the names for the selected force sets to "*upstream junction name - downstream junction name*" using the names from the connected junctions on the workspace.
- **Point Force Ambient Pressure** - If using a Point force set, the ambient pressure is required to calculate the force balance. This defaults to the [Environmental Properties](#) value, but can be changed here.

Each individual force set has a number of required parameters:

- **Apply** - Whether to include the force set calculation. This allows disabling force sets without deleting them.
- **Force Type** - There are several force set types available. They are described in detail in [Force Set Types](#).
- **Force Set Name** - A unique name for the force set.
- **Start Pipe** - Where the first node is located.
- **Start Location** - Force Sets in xStream must have their endpoints at either the inlet or outlet of a pipe.
- **End Pipe** - The second (for difference types) node. Note that the selection must be valid for the specified force type. See [Force Set Types](#).
- **End Location** - Force Sets in xStream must have their endpoints at either the inlet or outlet of a pipe.
- **Pipe Group** - For force sets across 3+ pipes, a group is required and is selected here.
- **Exit Area** - For Exit type forces, an exit area is required.
- **Force Unit Vector Coordinates** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. xStream is still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the [Export Force Data](#) tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

### Defining from the Workspace

Force Sets for individual pipes can be defined directly from the workspace by right-clicking on a selected pipe. Choose the Create Force Set With Selected Pipe... option to launch the Create Force Set window, shown below in Figure 2. If multiple pipes are selected, an individual Difference Force Set will be created for each pipe applying the specified settings for each.



**Figure 2: Create Force Set window in the Workspace**

Each force has several required and optional parameters:

- **Pipe Name** - If the pipe has a name entered other than 'Pipe', the user can choose whether to include that pipe name or not in the Force Set name.
- **Force Set Name** - Users can add a name for the Force Set to better organize and manage Force Sets in the model.
- **Force Unit Vectors or Length** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. xStream is still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the [Export Force Data](#) tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.
- **Show Analysis Setup Force Panel after Creation** - If checked the user will be brought to the Force panel of the Analysis Setup window to view all Force Set information defined for that scenario.

## Related Topics

Transient Forces.....	30
-----------------------	----

## Related Examples

High Pressure Steam - Forces Example .....	606
--	-----

---

## AFT Arrow Steady Solution Group

The user has control over a number of parameters that influence global convergence for the AFT Arrow™ Steady Solution.

---

**Note:** These values should only be changed by experienced users or under advisement by AFT Support. **Poorly defined Solution Control settings can cause the model to converge on an incorrect solution without warning messages.**

---

- [Solution Method panel](#) - The calculation method can be defined on this panel.
- [Tolerance panel](#) - The iterative tolerances that define convergence can be defined on this panel.
- [Iterations panel](#) - Additional iteration controls that can affect whether or not a model converges, or how quickly it will converge, can be defined here. Most settings on this panel will not affect the final result.
- [Pressure and Temperature Limits panel](#) - Options to limit the pressure and/or temperature results in the solver.

## Related Topics

---

References..... 461

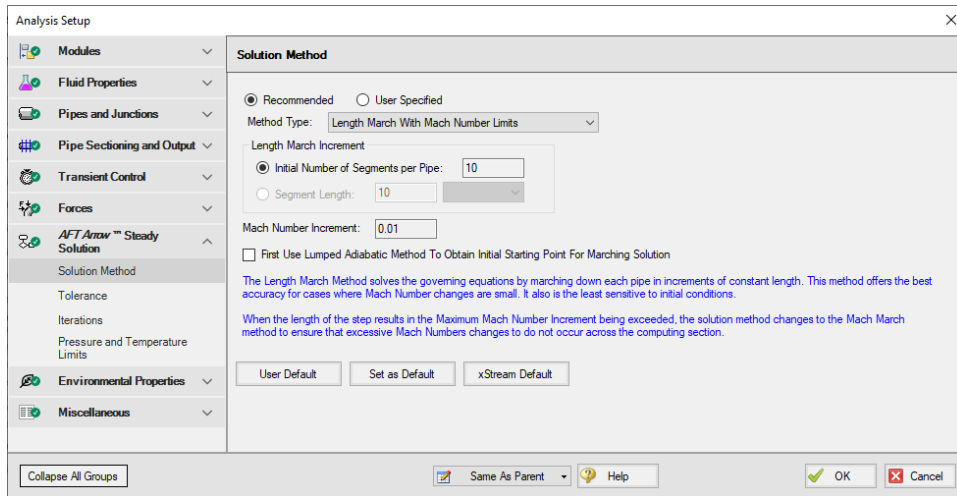
## Solution Method Panel

The Solution Method panel defines the calculation method and relevant parameters for the AFT Arrow™ Steady Solution (Figure 1). This panel is defined by default and typically should not be changed unless problems are encountered when running the model.

The **Method Type** can be set as Recommended or User Specified. The Recommended option (Length March With March Number Limits) is sufficient for most models. If desired the User Specified option can be selected to choose a different Method Type as are described below in the Solution Method Overview section below. Choosing User Specified also allows the march method increments to be changed.

A few other settings are defined by default for the user including the **Length March Increment** and **Mach Number Increment** settings. See the [Marching Methods](#) discussion below for more information on these parameters.

Lastly the "**First Use Lumped Adiabatic...**" option is available for all method types except the Lumped methods. This option solves the entire system with the Lumped Adiabatic method, and applies the results of that solution as a starting point to the marching method selected. This can help with convergence for some models. The results will be the same as using only the marching method. See the [Lumped Methods](#) section below for more information on Lumped Adiabatic.



**Figure 1: Default settings on the Solution Method panel**

## Solution Method Overview

xStream has six solution methods it can employ to find the AFT Arrow Steady Solution. Each method has certain advantages, and applicability to certain types of systems. While each option should theoretically come to a similar solution, there will be differences in the results between the methods. The magnitude of this difference depends heavily on the particular system being analyzed. All of the methods apply the same fundamental equations in differing ways.

- **Length March** - All of the fundamental equations are applied at fixed length increments - axial points - down a pipe. This method provides good accuracy if changes in Mach Number are small. It is the least sensitive to initial conditions.
- **Length March with Mach Number Limits** - This hybrid method starts with the Length March method, and dynamically switches to the Mach Number March method if the change in Mach Number between steps is large. This ensures excessive changes in Mach Number across computing sections are avoided.
- **Mach Number March** - The fundamental equations are still applied at axial points down the pipe, but in this case the distance between the points is fixed by a Mach Number increment. This method is more accurate when there are large changes in Mach Number, often occurring when the Mach Number is high and when the flow is sonically choked.
- **Mach Number March with Length Limits** - This hybrid method begins with the Mach Number March method, switching to Length March if the distance between axial points is too large. This helps ensure accuracy when the changes in Mach Number are small.
- **Lumped Adiabatic** - This method ignores all heat transfer and elevation changes. These effects are minimal in xStream as the pipes will be treated as adiabatic regardless, and simplifying the calculations this way greatly reduces the required solution time and likelihood of convergence.
- **Lumped Isothermal** - This method further simplifies the fundamental equations by assuming pipes are a fixed temperature throughout and again ignoring elevation.

### Marching Methods

---

The marching methods provide the most accurate solutions. AFT Arrow "marches" down a pipe in the model, solving the fundamental equations along the way.

Dividing the pipe into smaller segments is important for compressible flow because the fluid properties change throughout the pipe. Each segment considers the flow properties constant and applies the fundamental equations. This represents a discretization of the continuously changing flow parameters - necessary for an iterative Solver such as AFT Arrow. This discretization introduces a certain amount of error into the solution, which is closely related to how many segments are present, as well as how much the flow properties change over a single segment.

When dividing the pipes into segments, **axial points** are created along the pipe, at the ends of every segment. The results at each axial point can be viewed in the output.

- **Length March** - This method divides pipes into segments based on length. The segment length can be defined in two ways - either by dividing every pipe into a certain number of segments (**Initial Number of Segments per Pipe** setting), or dividing all pipes into equal-length segments (**Segment Length** setting). This method has the advantages that it is conceptually easier to understand and ensures some minimum number of segments in every pipe. Axial points are also often located at convenient locations. However, at high Mach Numbers, the large change in flow properties over a segment makes this method less accurate. For a detailed mathematical description of this method see [Length March Method - Detailed Discussion](#).
- **Mach Number March** - Unlike the Length March, Mach Number March determines the locations of the axial points dynamically based on the flow solution. There is no easy way to tell where the axial points will be without completing the solution. This method is superior when analyzing high Mach Number flows, as the flow properties tend to change dramatically over short distances. Effectively, this method adds axial points as necessary where the flow properties are changing rapidly. The disadvantage of this method is that the minimum number of sections cannot be enforced - if the Mach Number in a single pipe changes less than the increment specified, the entire pipe will be represented with a single segment. This is not necessarily inaccurate, but it is generally preferable to have more than one segment per pipe. For a detailed mathematical description of this method see [Mach Number March Method - Detailed Discussion](#).

### Hybrid Marching Methods

---

AFT Arrow offers *hybrid* marching methods to get the best of both marching methods in one solution. These methods dynamically switch from Length to Mach Number March or vice-versa during the solution.

This dynamic switch allows the user to ensure that a certain minimum number of segments is present in each pipe, and that the flow properties do not change significantly over the segment.

- **Length March With Mach Number Limits (AFT Arrow default)** - The default solution method begins the march down the pipe with equal length segments. If the Mach Number over any certain segment is detected to increase more than the specified limit, the march continues over the rest of the pipe at equal Mach Number increments. It is often the case that flow velocity in a pipe is

relatively low and constant in the first few segments of the pipe, but increases dramatically towards the exit. This method captures this behavior.

- **Mach Number March With Length Limits** - Very similar to the default method, this hybrid method begins with a Mach Number march, but if it is detected that the length to the next axial point is too far due to relatively constant velocity, additional axial points are forced into the pipe via the length limit. This helps ensure all pipes have some minimum number of sections.

The figure below shows an example of a pipe solved with Length March With Mach Number Limits. During the solution, the solver determined that taking an additional 10 foot step from point 6 (60 feet) to point 7 (70 feet) would violate the Mach Number limit of 0.01. Therefore, the march was adjusted (starting at point 6) to increment in Mach Number steps of 0.01.

Pipe -- Point	x (feet)	Mach #	Mass Flow (lbm/sec)	P Static (psia)	Rho Static (lbm/ft3)	Velocity (feet/sec)	H Stag. (Btu/lbm)	T Static (deg. F)
1 -- 0	0.00	0.2478	4.648	38.33	0.1811	290.3	202.5	93.22
1 -- 1	10.00	0.2537	4.648	37.42	0.1769	297.1	202.5	92.87
1 -- 2	20.00	0.2601	4.648	36.49	0.1727	304.5	202.5	92.49
1 -- 3	30.00	0.2670	4.648	35.53	0.1682	312.5	202.5	92.08
1 -- 4	40.00	0.2745	4.648	34.54	0.1637	321.2	202.5	91.61
1 -- 5	50.00	0.2828	4.648	33.52	0.1590	330.7	202.5	91.09
1 -- 6	60.00	0.2919	4.648	32.45	0.1541	341.2	202.5	90.50
1 -- 7	69.87	0.3019	4.648	31.36	0.1491	352.7	202.5	89.82
1 -- 8	78.73	0.3119	4.648	30.33	0.1444	364.1	202.5	89.15
1 -- 9	86.72	0.3219	4.648	29.37	0.1400	375.6	202.5	88.44
1 -- 10	93.93	0.3319	4.648	28.47	0.1359	387.0	202.5	87.72
1 -- 11	100.00	0.3412	4.648	27.68	0.1322	397.5	202.5	87.04

Figure 1: Length March dynamically switching to Mach Number March

## Lumped Methods

For faster - but less accurate - computation, AFT Arrow offers two of the traditional *lumped* methods. These methods divide all pipes into 10 segments, but the method solves the entire pipe in a single computation. Certain assumptions simplify the fundamental equations allowing a much faster solution at the cost of accuracy.

- **Lumped Adiabatic** - This method treats every pipe as adiabatic, disallowing heat transfer into or out of the pipe. If heat transfer is defined in the model, the definition is retained for convenience but is not applied during the solution. In addition, this model cannot accurately model body forces so therefore does not account for elevation changes or rotating systems. In many gas systems these assumptions are reasonable and the results from Lumped Adiabatic acceptable. For a detailed mathematical description of this method see [Lumped Adiabatic Method - Detailed Discussion](#).
- **Lumped Isothermal** - Similarly, this method treats all pipes as a constant temperature. Again, elevations are ignored. Note that heat transfer is applied to pipes (heat in or out) to meet the isothermal conditions. For a detailed mathematical description of this method see [Lumped Isothermal Method - Detailed Discussion](#).

- **Use Pipe Isothermal Temperatures When Available Instead of This Value** - If Isothermal models have been defined for individual pipes, that temperature will be applied to that pipe instead of the global temperature defined in the Solution Method panel.

### Comparing Marching Methods to Traditional Methods

---

There exist in industry practice a number of traditional methods and equations for calculating compressible flow in pipes.

A common equation is the Crane isothermal equation (see [Crane, 1988](#), pp. 1-8). AFT Arrow and equations such as the Crane isothermal equation differ in several respects. First is that the Crane equation assumes isothermal flow, while AFT Arrow can model isothermal flow or other, more general, boundary conditions. Second is that the Crane equation solves the entire pipe as one segment. This can be called a lumped method. It is thus roughly analogous to using AFT Arrow with a single computing section per pipe. Third, the Crane equation assumes a constant diameter run, whereas AFT Arrow allows you to change diameters. Fourth, it is not apparent how to apply the Crane equation to multi-pipe systems, including pipe networks.

The Crane equation solves the governing mass and momentum equations by making some assumptions. AFT Arrow is solving the same equations, but is not making any assumptions. It marches down each pipe, taking into account physical property changes and non-linear acceleration effects as it progresses. Thus, AFT Arrow is in general much broader and more accurate than the Crane equation. When the pipe flow conditions are within the assumptions of the Crane equation, then the Crane equation and AFT Arrow should agree closely.

Other equations on page 1-8 of Crane, 1988, compare similarly to AFT Arrow. They are all lumped methods with certain assumptions about thermal conditions in the pipe and properties of the gas.

These traditional equations can be used successfully in many engineering applications. They all represent a subset of AFT Arrow's marching methods, which solve the fundamental equations without limiting assumptions.

### Length March Method - Detailed Discussion

The Length March solution method for the [AFT Arrow Steady Solution Method](#) is described in detail below.

The stagnation pressure is given by the following:

$$P_o = P \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma/(\gamma-1)} \quad (22)$$

By differentiating Equation 22 the following is obtained:



$$\frac{dP}{P} = \frac{dP_o}{P_o} - \frac{\gamma M^2}{2 \left(1 + \frac{\gamma-1}{2} M^2\right)} \frac{dM^2}{M^2} \quad (23)$$

Now take Equation 23 and use it to replace the pressure term in Equation 10 to obtain:

$$\begin{aligned} \frac{dP_o}{P_o} = & -\frac{\gamma M^2}{2} \left( \frac{dM^2}{M^2} + \frac{dT}{T} \right) + \frac{\gamma M^2}{2 \left(1 + \frac{\gamma-1}{2} M^2\right)} \frac{dM^2}{M^2} - \frac{\gamma M^2}{2} \frac{f dx}{D} \\ & - \frac{\gamma M^2}{2} \frac{dZ}{Z} - \frac{\gamma M^2}{2} \frac{d\gamma}{\gamma} - \frac{g \sin \theta dx}{Z R T} \end{aligned} \quad (24)$$

The first two terms on the right hand side of Equation 24 relate to the [stagnation temperature \(Equation 15\)](#) as follows:

$$\frac{\gamma M^2}{2} \frac{dT_o}{T_o} = \frac{\gamma M^2}{2} \left( \frac{dT}{T} + \frac{dM^2}{M^2} \right) - \frac{\gamma M^2}{2} \frac{1}{1 + \frac{\gamma-1}{2} M^2} \frac{dM^2}{M^2}$$

Making this substitution, and also using [Equation 14](#)

$$\begin{aligned} \frac{dP_o}{P_o} = & -\frac{\gamma M^2}{2} \frac{dT_o}{T_o} - \frac{\gamma M^2}{2} \frac{f dx}{2} - \frac{\gamma M^2}{2} \frac{dZ}{Z} - \frac{\gamma M^2}{2} \frac{d\gamma}{\gamma} \\ & - \frac{g \sin \theta dx}{Z R T} \left( 1 + \frac{\gamma-1}{2} M^2 \right) \end{aligned} \quad (25)$$

Integrating Equation 25 yields:

$$\begin{aligned} \ln \frac{P_{o2}}{P_{o1}} = & -\frac{\gamma \bar{M}^2}{2} \ln \frac{T_{o2}}{T_{o1}} - \frac{\gamma \bar{M}^2}{2} \frac{f(x_2 - x_1)}{D} - \frac{\gamma \bar{M}^2}{2} \ln \frac{Z_2}{Z_1} - \frac{\gamma \bar{M}^2}{2} \ln \frac{\gamma_2}{\gamma_1} \\ & - \frac{g \sin \theta (x_2 - x_1)}{R \bar{Z} \bar{T}} \left( 1 + \frac{\gamma-1}{2} M^2 \right) \end{aligned} \quad (26)$$

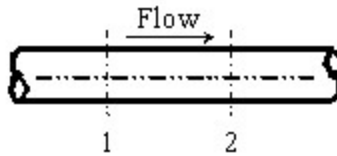
And with algebra the Length March equation (Equation 27) is obtained:

$$P_{o_2} = P_{o_1} e^{-C}$$

$$C = \frac{\gamma \bar{M}^2}{2} \left[ \ln \frac{T_{o_2}}{T_{o_1}} + \ln \frac{Z_2}{Z_1} + \ln \frac{\gamma_2}{\gamma_1} + \frac{f(x_2 - x_1)}{D} \right] + \frac{g \sin \theta (x_2 - x_1)}{R \bar{Z} \bar{T}} \quad (27)$$

## Example

Some examples are given here to clarify how the Mach Number March and Length March methods solve the governing equations.



**Figure 1: Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations.**

Both of these methods are referred to as marching methods. What does this mean? Figure 1 shows a representation of a computing section in a pipe. In this example, all parameters at Location 1 are known, and it is desired to calculate conditions at location 2 which is always downstream of location 1.

Whether we are applying the [Length March](#) or the Mach Number March (Equation [21](#), [37](#) or [41](#)), at location 1 we know these primary parameters:

$$\dot{m}_1, x_1, M_1, P_{o_1}, T_{o_1}, Z_1, \gamma_1$$

Because we know the parameters above, we also know:

$$\rho_1, V_1, P_1, T_1, h_{o_1}, h_1$$

Let's consider the Length March method first because it is easier to grasp. Using the [Length March equation](#), we know three things about location 2. First, we know the length step, as this is specified by the user. We also know that the mass flow rate is constant. Lastly, we know the heat transfer relationship specified by the user and can use this to determine a thermodynamic parameter.

For instance, if the pipe is modeled as adiabatic ( $q = 0$ ), we can determine the stagnation enthalpy at location 2 using the following assumptions.

If there is no elevation change, then:

$$z_2 = z_1$$

and according to the [Energy equation](#):

$$h_{o2} = h_{o1}$$

Even if there is elevation change, we know what it is and can just add that into the energy equation and calculate  $h_{o,2}$  explicitly.

If the pipe is isothermal, then:

$$T_2 = T_1$$

If there is convective heat transfer or known heat flux, we can solve Equation [36](#) or [40](#) for  $T_{o,2}$  which allows us to calculate  $h_{o,2}$  and we are at the same point as with adiabatic.

Thus, if any boundary condition other than isothermal is specified, the following is known at location 2:

$$\dot{m}_2, x_2, h_{o2} \quad (\textit{nonisothermal})$$

or, if isothermal, the following is known at location 2:

$$\dot{m}_2, x_2, T_2 \quad (\textit{isothermal})$$

With one thermodynamic parameter known at location 2, we guess at the stagnation pressure solution to the [Length March equation](#). This in effect gives us two known thermodynamic parameters, from which all the other parameters can be determined through state relationships. This requires significant iteration for each computing step.

For each pipe in the system, the marching is initiated at the pipe inlet where conditions are known. The known inlet conditions reflect the current solution state at the junction upstream of the pipe. This is location 1. Based on the user specified length step, the distance to location 2 is specified. The method just described is used to converge on the conditions at location 2. Once this convergence occurs, the previous location 2 becomes location 1 and a new location 2 is determined based on the step size. The marching continues to the end of the pipe.

If the user specifies an absolute length size (say 10 foot sections), a step may go past the end of the pipe. This will occur for any pipe that is shorter than 10 feet. For a longer pipe of say 12 feet, the first computing length will be 10 feet, while the second will be 2 feet.

It should be understood that in the Length March method, computing sections are equidistant along a pipe, aside from possible roundoff at the end.

**Note:** The process of converging on location 2 while marching down a pipe is referred to as a local iteration. It is local in that it does not relate to what is happening in any other part of the model. AFT xStream also has global iterations, which relate to iterations made on the entire system in trying to obtain complete system convergence. You can specify iteration limits for each type of convergence in the [Iterations panel](#).

### Example Application

Consider the model in Figure 2. This model is of an air system modeled as a real gas using Redlich-Kwong and Generalized enthalpy. The Solution Method panel specifies Length March method with 10 segments per pipe. Results using the Length March method are shown in Figure 3.

There are several items worth noting in Figure 3. First, note that the second column shows the x location for each computation step. Point 1 will be identical to the pipe's inlet values. For this model, there were 10 steps along a 100 foot pipe, thus each step is equally 10 feet long.

Second, notice how the velocity and Mach number increase along the pipe length.

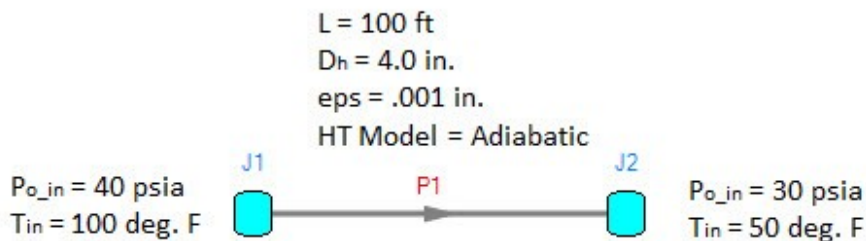
Third, notice how the static temperature drops. This is due to cooling of the gas as it expands with pressure drop.

Fourth, notice that the stagnation enthalpy is constant in accordance with adiabatic flow.

Fifth, notice how the static enthalpy drops along the pipe length. This shows that the flow is not isenthalpic.

Finally, notice how the stagnation temperature drops slightly. This is a result of real gas effects. A perfect gas will have constant stagnation temperature and enthalpy in adiabatic flow with constant elevation.

These results can be viewed graphically by changing to the Graph Results window and selecting to graph a profile along a flow path.



**Figure 2: Model of an air system dropping from 40 to 30 psia.**

## Analysis Setup Overview

Pipes		Axial Points									
Pipe -- Point (AFT Arrow Steady)	x (feet)	Mass Flow (lbm/sec)	Velocity (feet/sec)	Mach #	P Stag (psia)	P Static (psia)	T Stag (deg. F)	T Static (deg. F)	H Stag (Btu/lbm)	H Static (Btu/lbm)	Rho Static (lbm/ft <sup>3</sup> )
1 -- 0	0.00	4.884	299.7	0.2604	40.00	38.16	100.00	92.52	202.2	200.4	0.1867
1 -- 1	10.00	4.884	306.8	0.2666	39.14	37.25	99.98	92.14	202.2	200.3	0.1824
1 -- 2	20.00	4.884	314.4	0.2734	38.25	36.32	99.96	91.72	202.2	200.2	0.1780
1 -- 3	30.00	4.884	322.7	0.2807	37.35	35.36	99.93	91.26	202.2	200.1	0.1734
1 -- 4	40.00	4.884	331.8	0.2887	36.41	34.36	99.91	90.74	202.2	200.0	0.1687
1 -- 5	50.00	4.884	341.7	0.2974	35.44	33.33	99.89	90.16	202.2	199.9	0.1638
1 -- 6	60.00	4.884	352.6	0.3071	34.44	32.26	99.86	89.51	202.2	199.7	0.1587
1 -- 7	70.00	4.884	364.8	0.3179	33.40	31.14	99.84	88.75	202.2	199.5	0.1534
1 -- 8	80.00	4.884	378.4	0.3301	32.32	29.97	99.81	87.88	202.2	199.3	0.1479
1 -- 9	90.00	4.884	393.9	0.3439	31.19	28.74	99.78	86.85	202.2	199.1	0.1421
1 -- 10	100.00	4.884	411.8	0.3599	30.00	27.43	99.75	85.62	202.2	198.8	0.1359

**Figure 3: Axial Points (available in the Output window) for model in Figure 2 using Length March method**

### Length March Method Verification

The axial points shown Figure 3 can be used to verify that the fundamental equations are being solved accurately. Figure 4 shows the relevant verification parameters for a pipe carrying air from a tank at 40 psia and 100 deg. F to another tank at 30 psia. The pipe is vertical, 100 feet long, and 4 inches in diameter with a roughness of 0.001 inches, and is adiabatic. The high pressure tank is the top tank.

Pipes		Axial Points					
Pipe -- Point (AFT Arrow Steady)	x (feet)	P Static (psia)	Rho Static (lbm/ft <sup>3</sup> )	Velocity (feet/sec)	H Stag. (Btu/lbm)	f	
1 -- 0	0.000000	38.138676	0.18667805	301.37143	202.17825	0.014931718	
1 -- 1	10.000000	37.235443	0.18236247	308.50336	202.19110	0.014931604	
1 -- 2	20.000000	36.304802	0.17792010	316.20615	202.20395	0.014931362	
1 -- 3	30.000000	35.344063	0.17333873	324.56354	202.21681	0.014931089	
1 -- 4	40.000000	34.350040	0.16860390	333.67810	202.22966	0.014930783	
1 -- 5	50.000000	33.318951	0.16369845	343.67725	202.24251	0.014930434	
1 -- 6	60.000000	32.246201	0.15860157	354.72177	202.25536	0.014930034	
1 -- 7	70.000000	31.126141	0.15328781	367.01828	202.26820	0.014929573	
1 -- 8	80.000000	29.951607	0.14772496	380.83902	202.28105	0.014929035	
1 -- 9	90.000000	28.713531	0.14187238	396.54959	202.29391	0.014928400	
1 -- 10	100.000000	27.399845	0.13567621	414.65955	202.30676	0.014927636	

**Figure 4: Axial Points for AFT xStream model Verify\_Governing\_Equations.xtr using the Length March method.**

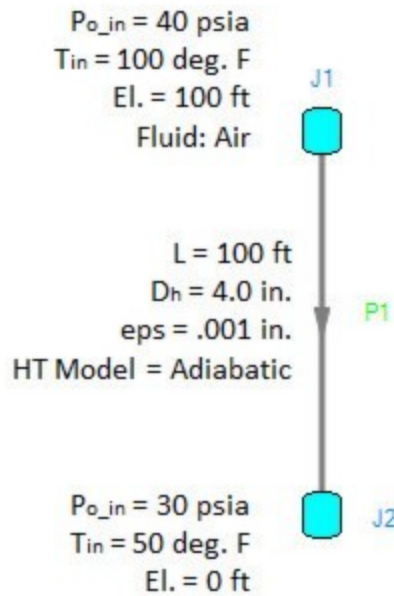
The model, Verify\_Governing\_Equations.xtr, is shown in Figure 5 can be obtained from AFT upon request (support@aft.com). Also included is an Output Control data configuration file called Verify\_Governing\_Equations\_1.dat that can be used to configure any model to that shown in Figure 1.

The Figure 4 results were obtained using the Length March method with 10 pipe segments. The results can be copied to the clipboard and pasted into a spreadsheet as shown in Table 1. Once in the spreadsheet, additional calculations can be performed. Table 1 shows these calculations and a balance verification using all terms in the momentum equation (Equation 2), continuity equation (Equation 1), and energy equation (Equation 30). Along with the previous files, this spreadsheet is included upon request and is named Verify\_Governing\_Equations\_Length.xls.

There are several terms in Table 1 that require clarification. Using coordinates as shown previously in Figures 1 of [Fundamental Equations](#) and Figure 1, density and velocity can be represented as follows:

$$\rho_{avg} = \frac{\rho_1 + \rho_2}{2}$$

$$V_{avg} = \frac{V_1 + V_2}{2}$$



**Figure 5: AFT xStream model solved in Figure 1 and verified in Table 1. Fluid is air. File is Verify\_Governing\_Equations.xtr.**

And using these terms, the four momentum equation terms from Table 1 are defined as:

dP Total: 
$$dP = P_1 - P_2$$

$$\text{dP Fric. (Friction): } \frac{\rho V_2}{2} \frac{f dx}{D} = \frac{\rho_{avg} V_{avg}^2}{2} \frac{f (x_1 - x_2)}{D}$$

$$\text{dP Accel (Acceleration): } \rho V dV = \rho_{avg} V_{avg} (V_1 - V_2)$$

$$\text{dP Grav. (Gravity): } \rho g dz = \rho_{avg} g (z_1 - z_2) \quad (53)$$

**Table 1: Verification showing how AFT xStream Length March method predictions satisfy momentum, mass and energy balance equations. Taken from spreadsheet Verify\_Governing\_Equations\_Length.xls.**

AFT ARROW OUTPUT						AVERAGED VALUES (CALCULATED)						MOMENTUM BALANCE				MASS BALANCE	ENERGY BALANCE		
P Static	Rho Static	Velocity	H Stag.			dp	V	dV	dho			dP Total	dP Fric.	dp Accel	dp Grav.	Balance	Balance	Balance	
x (feet)	(psia)	(lbm/ft3)	(feet/sec)	(ft/lbm)	f	ρ (lbm/ft3)	(lbm/ft3)	(feet/sec)	(feet/sec)	(ft/lbm)	dx (feet)	dz (feet)	(psi)	(psi)	(psi)	(psi)	(psi)	(lbm/sec)	(lbm/sec)
0.00	38.139	0.1886	301.394	202.259	0.0149	-	-	-	-	-	-	-	-	-	-	-	-	-	-
10.00	37.235	0.1823	308.529	202.271	0.0149	0.1845	0.00432	304.962	-7.135	-0.0128	-10.00	10.00	0.90320	-0.82941	-0.08664	0.01281	-0.00004	7.72E-08	-0.012
20.00	36.305	0.1779	316.236	202.284	0.0149	0.1801	0.00444	312.382	-7.707	-0.0129	-10.00	10.00	0.93062	-0.84959	-0.09358	0.01251	-0.00005	-6.15E-08	-0.012
30.00	35.344	0.1733	324.598	202.297	0.0149	0.1756	0.00458	320.417	-8.362	-0.0128	-10.00	10.00	0.96073	-0.87444	-0.10154	0.01219	-0.00006	2.67E-08	-0.012
40.00	34.350	0.1686	333.718	202.310	0.0149	0.1709	0.00474	329.158	-9.120	-0.0129	-10.00	10.00	0.99403	-0.89521	-0.11075	0.01187	-0.00007	-4.69E-08	-0.012
50.00	33.319	0.1637	343.724	202.323	0.0149	0.1661	0.00491	338.721	-10.006	-0.0128	-10.00	10.00	1.03112	-0.92123	-0.12151	0.01154	-0.00008	1.04E-07	-0.012
60.00	32.246	0.1586	354.777	202.336	0.0149	0.1611	0.00510	349.251	-11.053	-0.0128	-10.00	10.00	1.07282	-0.94987	-0.13423	0.01119	-0.00010	-8.20E-08	-0.012
70.00	31.126	0.1532	367.085	202.349	0.0149	0.1559	0.00532	360.931	-12.307	-0.0129	-10.00	10.00	1.12017	-0.98165	-0.14947	0.01083	-0.00012	1.53E-07	-0.012
80.00	29.951	0.1477	380.919	202.361	0.0149	0.1505	0.00557	374.002	-13.835	-0.0128	-10.00	10.00	1.17470	-1.01721	-0.16803	0.01045	-0.00009	-1.27E-07	-0.012
90.00	28.713	0.1418	396.648	202.374	0.0149	0.1447	0.00586	388.784	-15.729	-0.0128	-10.00	10.00	1.23832	-1.05744	-0.19104	0.01005	-0.00011	-4.59E-08	-0.012
100.00	27.399	0.1356	414.783	202.387	0.0149	0.1387	0.00620	405.715	-18.135	-0.0129	-10.00	10.00	1.31405	-1.10354	-0.22029	0.00963	-0.00015	-1.49E-09	-0.012

There is another term that is not relevant in this model and is thus not shown in Table 1. This term accounts for additional minor losses in the form of K factors, and is a modified form of the dP Acceleration term:

$$\frac{\rho V^2}{2} \frac{K dx}{L} = \frac{\rho_{avg} V_{avg}^2}{2} \frac{K (x_1 - x_2)}{L}$$

The momentum equation terms in Table 1 are added together for each computing section. If the momentum equation is being satisfied, the four terms should sum to zero (or a very small number). They do, as shown in Table 1 in the column titled Balance (psi).

## Analysis Setup Overview

Note that AFT xStream does not solve [Equation 49](#). Rather, when using the Length March method, AFT xStream solves [Equation 27](#), which contains (more accurate) logarithmic terms. Equation 27 is a more complex form of the momentum equation. The arithmetically averaged terms in Equations 49 and Table 1 are approximations which show that the momentum equation is being satisfied.

Table 1 has another column titled Balance (lbm/sec) which shows that the continuity equation (Equation 1) is also satisfied. This is calculated in Table 1 as follows:

$$\frac{d\rho}{\rho} + \frac{dV}{V} = \frac{\rho_1 - \rho_2}{\rho_{avg}} + \frac{V_1 - V_2}{V_{avg}} = 0$$

While not shown here, it was shown earlier in Figure 3 how AFT xStream maintains constant stagnation enthalpy for adiabatic flow with zero elevation change. According to the [energy equation](#) (Equation 30), when the elevation changes, the stagnation enthalpy is not constant in adiabatic flow. The energy balance is calculated as follows:

$$\frac{d\rho}{\rho} + \frac{dV}{V} = \frac{\rho_1 - \rho_2}{\rho_{avg}} + \frac{V_1 - V_2}{V_{avg}} = 0$$

In the Energy Balance column, Table 1 shows how this equation is satisfied. If heat transfer is applied, it is a simple matter to extend the spreadsheet behind Table 1 to verify that energy is balanced.

The individual terms in Equations 49 are also available for output in AFT xStream. The contributions to pressure drop can be reviewed individually, and can also be used for further verification. Figure 6 shows the axial points again for `Verify_Governing_Equations.xtr`, but now these pressure drop contributions are shown. The Output Control data configuration file used to obtain the Figure 6 format is `Verify_Governing_Equations_2.dat`.

Pipe -- Point (AFT Arrow Steady)	x (feet)	P Static (psia)	Rho Static (lbm/ft <sup>3</sup> )	Velocity (feet/sec)	H Stag. (Btu/lbm)	f	dP Static Friction (psid)	dP Static Component (psid)	dP Static Accel. (psid)	dP Gravity (psid)
1 -- 0	0.000000	38.138676	0.18667805	301.37143	202.17825	0.014931718	0.00000000	0	0.00000000	0.00000000
1 -- 1	10.000000	37.235443	0.18236247	308.50336	202.19110	0.014931604	0.82945919	0	0.086603239	-0.0128139053
1 -- 2	20.000000	36.304802	0.17792010	316.20615	202.20395	0.014931362	0.84963429	0	0.093535230	-0.0125098098
1 -- 3	30.000000	35.344063	0.17333873	324.56354	202.21681	0.014931089	0.87147665	0	0.101484083	-0.0121964859
1 -- 4	40.000000	34.350040	0.16860390	333.67810	202.22966	0.014930783	0.89524043	0	0.110678449	-0.0118730078
1 -- 5	50.000000	33.318951	0.16369845	343.67725	202.24251	0.014930434	0.92123854	0	0.121419959	-0.0115382755
1 -- 6	60.000000	32.246201	0.15860157	354.72177	202.25536	0.014930034	0.94986433	0	0.134114042	-0.0111909723
1 -- 7	70.000000	31.126141	0.15328781	367.01828	202.26820	0.014929573	0.98161858	0	0.149316922	-0.0108294915
1 -- 8	80.000000	29.951607	0.14772496	380.83902	202.28105	0.014929035	1.01715529	0	0.167825714	-0.0104518319
1 -- 9	90.000000	28.713531	0.14187238	396.54959	202.29391	0.014928400	1.05734646	0	0.190773994	-0.0100554619
1 -- 10	100.000000	27.399845	0.13567621	414.65955	202.30676	0.014927636	1.10338950	0	0.219909802	-0.0096371025

**Figure 6: Axial Points for `Verify_Governing_Equations.xtr` showing the individual pressure drop contributions.**

The Pipe Results table shows the total pressure drop contributions for each pipe when appropriate parameters are selected for display. See Figure 7.



Pipes		Axial Points							
Pipe (AFT Arrow Steady)	P Static In (psia)	Rho Static In (lbm/ft3)	Vel. In (feet/sec)	dH Stag. In (Btu/lbm)	H Stag. In (Btu/lbm)	dP Static Friction Wall (psid)	dP Static Component (psid)	dP Static Accel. (psid)	dP Gravity (psid)
1	38.14	0.1867	301.4	-0.1285	202.2	9.476	0	1.376	-0.1131

All Junctions		Tank		
Jct (AFT Arrow Steady)	P Stag. In (psia)	P Stag. Out (psia)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)
1	40.00	40.00	100.00	100.00
2	30.00	30.00	50.00	50.00

Figure 7: Pipe Results table shows total pressure drop contributions as defined in Equation 49.

## Related Topics

Nomenclature .....410

### Mach Number March Method - Detailed Discussion

The Mach Number March Solution Method for the [AFT Arrow Steady Solution Method](#) is described in detail below.

Modify [Equation 2](#) by substituting [Equations 3 and 5](#) and eliminating density,

$$\frac{dP}{P} + \frac{V^2}{2ZRT} \frac{f dx}{D} + \frac{V dV}{ZRT} + \frac{g \sin(\theta) dx}{ZRT} = 0 \quad (6)$$

By definition of Mach number for a real gas

$$V = M \sqrt{\gamma ZRT} \quad (7)$$

and from basic calculus the following relationship holds:

$$\frac{dM}{M} = \frac{1}{2} \frac{dM^2}{M^2} \quad (8)$$

Differentiating Equation 7 (and using Equation 8) results in,

$$\frac{dV}{V} = \frac{dM^2}{2M^2} + \frac{dT}{2T} + \frac{dZ}{2Z} + \frac{d\gamma}{2\gamma} \quad (9)$$

Now, substitute Equation 7 and 9 into Equation 6 and eliminate the velocity

$$\begin{aligned} \frac{dP}{P} + \frac{\gamma M^2}{2} \frac{f dx}{D} + \frac{\gamma M^2}{2} \frac{dM^2}{M^2} + \frac{\gamma M^2}{2} \frac{dT}{T} + \frac{\gamma M^2}{2} \frac{dZ}{Z} \\ + \frac{\gamma M^2}{2} \frac{d\gamma}{\gamma} + \frac{g \sin \theta dx}{ZRT} = 0 \end{aligned} \quad (10)$$

Differentiating the equation of state, [Equation 5](#) yields

$$\frac{dP}{P} = \frac{dT}{T} + \frac{dZ}{Z} - \frac{dV}{V} \quad (11)$$

Replace velocity by Mach number by substituting Equation 9 into Equation 11 to obtain

$$\frac{dP}{P} = \frac{1}{2} \left( \frac{dT}{T} + \frac{dZ}{Z} - \frac{d\gamma}{\gamma} - \frac{dM^2}{M^2} \right) \quad (12)$$

Eliminate pressure from Equation 10 using Equation 12 to obtain:

$$\begin{aligned} \frac{(\gamma M^2 - 1)}{2} \frac{dM^2}{M^2} + \frac{(\gamma M^2 + 1)}{2} \frac{dZ}{Z} + \frac{(\gamma M^2 - 1)}{2} \frac{d\gamma}{\gamma} + \frac{(\gamma M^2 + 1)}{2} \frac{dT}{T} \\ + \frac{\gamma M^2}{2} \frac{f dx}{D} + \frac{g \sin \theta dx}{ZRT} = 0 \end{aligned} \quad (13)$$

From the definition of stagnation temperature

$$T_o = T \left( 1 + \frac{\gamma - 1}{2} M^2 \right) \quad (14)$$

Differentiating the preceding yields

$$\frac{dT}{T} = \frac{dT_o}{T_o} - \frac{(\gamma - 1) M^2}{2 \left(1 + \frac{\gamma - 1}{2} M^2\right)} \frac{dM^2}{M^2} \quad (15)$$

Replace static temperature in Equation 13 with stagnation by using Equation 14 and Equation 15 to obtain:

$$\begin{aligned} & \frac{(\gamma M^2 - 1)}{2} \frac{dM^2}{M^2} + \frac{\gamma M^2}{2} \frac{f dx}{D} + \frac{(\gamma M^2 + 1)}{2} \frac{dZ}{Z} + \frac{(\gamma M^2 - 1)}{2} \frac{d\gamma}{\gamma} \\ & + \frac{(\gamma M^2 + 1)}{2} \left[ \frac{dT_o}{T_o} - \frac{(\gamma - 1) M^2}{2 \left(1 + \frac{\gamma - 1}{2} M^2\right)} \frac{dM^2}{M^2} \right] \\ & + \frac{g \sin \theta dx}{Z R T_o} \left(1 + \frac{\gamma - 1}{2} M^2\right) = 0 \end{aligned}$$

Collecting terms and simplifying yields:

$$\begin{aligned} & \frac{M^2 - 1}{2 \left(1 + \frac{\gamma - 1}{2} M^2\right)} \frac{dM^2}{M^2} + \frac{(\gamma M^2 + 1)}{2} \frac{dT_o}{T_o} + \frac{(\gamma M^2 + 1)}{2} \frac{dZ}{Z} + \frac{(\gamma M^2 - 1)}{2} \frac{d\gamma}{\gamma} \\ & + \frac{\gamma M^2}{2} \frac{f dx}{D} + \frac{g \sin \theta dx}{Z R T_o} \left(1 + \frac{\gamma - 1}{2} M^2\right) = 0 \end{aligned} \quad (16)$$

Rearrange Equation 16 to obtain

$$\begin{aligned} & \frac{dM^2}{M^2} - \frac{(1 + \gamma M^2) \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2} \left(\frac{dT_o}{T_o} + \frac{dZ}{Z}\right) \\ & - \frac{(1 - \gamma M^2) \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2} \frac{d\gamma}{\gamma} - \frac{\gamma M^2 \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2} \frac{f dx}{D} \\ & - \left(1 + \frac{\gamma - 1}{2} M^2\right) \frac{g \sin \theta dx}{Z R T_o} = 0 \end{aligned} \quad (17)$$

Four useful parameters that simplify notation are:

$$F_{T_o} = \frac{(1 + \gamma M^2) \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2}$$

$$F_\gamma = \frac{(1 - \gamma M^2) \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2}$$

$$F_f = \frac{\gamma M^2 \left(1 + \frac{\gamma - 1}{2} M^2\right)}{1 - M^2}$$

$$F_g = 1 + \frac{\gamma - 1}{2} M^2 \tag{18}$$

Using the four new parameters in Equation 18 simplifies to

$$\frac{dM^2}{M^2} - F_{T_o} \left( \frac{dT_o}{T_o} + \frac{dZ}{Z} \right) - F_\gamma \frac{d\gamma}{\gamma} - F_f \frac{f dx}{D} - F_g \frac{g \sin \theta dx}{Z R T_o} = 0 \tag{19}$$

Integrating Equation 19 yields

$$\ln \frac{M_2^2}{M_1^2} - (x_2 - x_1) \left( \bar{F}_f \frac{f}{D} + \bar{F}_g \frac{g \sin \theta}{R \bar{Z} \bar{T}_o} \right) - \bar{F}_{T_o} \left( \ln \frac{T_{o_2}}{T_{o_1}} + \ln \frac{Z_2}{Z_1} \right) - \bar{F}_\gamma \left( \ln \frac{\gamma_2}{\gamma_1} \right) = 0 \tag{20}$$

Equation 20 can be solved for x2:

$$x_2 = x_1 + \frac{\ln \frac{M_2^2}{M_1^2} - \bar{F}_{T_o} \left( \ln \frac{T_{o2}}{T_{o1}} + \ln \frac{Z_2}{Z_1} \right) - \bar{F}_\gamma \ln \frac{\gamma_2}{\gamma_1}}{\bar{F}_f \frac{f}{D} + \bar{F}_g \frac{g \sin \theta}{R \bar{Z} \bar{T}_o}} \quad (21)$$

where the bars over certain parameters indicate that the average value over the computing section is used.

Equation 21 is the Mach Number March equation. The Mach Number March is a variable step size method. As shown in Equation 21, the distance is the parameter being calculated.

### Modifications for Heat Transfer

In order to model heat transfer effects it is convenient in some cases to modify the previous equations.

The energy equation has the form:

$$\dot{m} d \left( h + \frac{1}{2} V^2 + g z \right) = q \quad (28)$$

The definition of stagnation enthalpy is

$$h_0 = h + \frac{1}{2} V^2 \quad (29)$$

which, when substituted into Equation 28, yields:

$$\dot{m} d (h_o + g z) = q \quad (30)$$

In most applications, the elevation term, gz, is very small and is neglected. This term will only become significant over pipes that have large elevation changes such as pipelines that go over mountains or down mineshafts, or in rapidly rotating systems.

With elevation neglected, it is not uncommon for engineers to conclude that adiabatic flow is isenthalpic (constant static enthalpy). It's not. Rather, the stagnation enthalpy is constant in adiabatic flow.

## Convective Heat Transfer

---

In pipe convective heat transfer, it is convenient to use the change in stagnation temperature rather than stagnation enthalpy by introducing the following approximation

$$c_p dT_o = dh_o \quad (31)$$

When heat transfer occurs, the effect of elevation change on the energy balance is small and is neglected. Substituting Equation 31 into Equation 30 yields

$$\dot{m} c_p dT_o = q \quad (32)$$

The heat transfer due to convection is given by the following:

$$q = U P_h (T_\infty - T_o) dx \quad (33)$$

where:

U = Overall heat transfer coefficient

Ph = Heated perimeter

The heated perimeter will frequently equal the wetted perimeter, which is the internal pipe circumference. It can change if part of the pipe is insulated (it will be less than the wetted perimeter) or there are fins on the pipe (it will be greater than the wetted perimeter).

Equating Equation 32 and 33 yields

$$\dot{m} c_p \frac{dT_o}{dx} = U P_w (T_\infty - T_o) \quad (34)$$

Rearranging Equation 34 yields:

$$\frac{dT_o}{T_o} = \frac{U P_h}{\dot{m} c_p} \left( \frac{T_\infty}{T_o} - 1 \right) \quad (35)$$

Integrating Equation 35 yields

$$\ln \frac{T_{o2}}{T_{o1}} = \frac{U P_h}{\dot{m} c_p} \left( \frac{T_\infty}{\bar{T}_o} - 1 \right) (x_2 - x_1) \quad (36)$$

This relationship for stagnation temperature can be substituted into Equation 20 to obtain:

$$x_2 = x_1 + \frac{\ln \frac{M_2^2}{M_1^2} - \bar{F}_{T_o} \ln \frac{Z_2}{Z_1} - \bar{F}_\gamma \ln \frac{\gamma_2}{\gamma_1}}{\bar{F}_{T_o} \frac{U P_h}{\dot{m} c_p} \left( \frac{T_\infty}{\bar{T}_o} - 1 \right) + \bar{F}_f \frac{f}{D} + \bar{F}_g \frac{g \sin \theta}{R \bar{Z} \bar{T}_o}} \quad (37)$$

In convective applications which use the Mach Number March method, the advantage of Equation 37 over [Equation 21](#) is that the stagnation temperature change is proportional to the distance step  $dx$  (as shown in Equation 35), for which we are solving. Equation 37 couples the effects of friction and heat transfer, and provides more reliable convergence, especially in convection heating.

## Constant Heat Flux

---

The constant heat flux boundary condition on a pipe results in a modified form of Equation 32

$$\dot{m} c_p dT_o = P_h q'' dx \quad (38)$$

The stagnation temperature change is thus:

$$\frac{dT_o}{T_o} = \frac{q'' P_h}{T_o \dot{m} c_p} dx \quad (39)$$

Integrating Equation 39 yields:

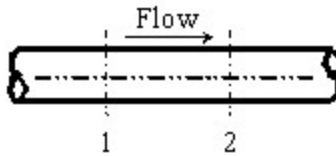
$$\ln \frac{T_{o2}}{T_{o1}} = \frac{q'' P_h}{\bar{T}_o \dot{m} c_p} (x_2 - x_1) \quad (40)$$

Substituting this into Equation 20 yields:

$$x_2 = x_1 + \frac{\ln \frac{M_2^2}{M_1^2} - \bar{F}_{T_o} \ln \frac{Z_2}{Z_1} - \bar{F}_\gamma \ln \frac{\gamma_2}{\gamma_1}}{\bar{F}_{T_o} \frac{q'' P_h}{T_o \dot{m} c_p} \left( \frac{T_\infty}{\bar{T}_o} - 1 \right) + \bar{F}_f \frac{f}{D} + \bar{F}_g \frac{g \sin \theta}{R \bar{Z} \bar{T}_o}} \quad (41)$$

## Example

Some examples are given here to clarify how the Mach Number March and [Length March](#) methods solve the governing equations.



**Figure 1: Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations**

Both of these methods are referred to as marching methods. What does this mean? Figure 1 shows a representation of a computing section in a pipe. In this example, all parameters at Location 1 are known, and it is desired to calculate conditions at location 2 which is always downstream of location 1.

Whether we are applying the [Length March](#) or the Mach Number March (Equation [21](#), [37](#) or [41](#)), at location 1 we know these primary parameters:

$$\dot{m}_1, x_1, M_1, P_{o1}, T_{o1}, Z_1, \gamma_1$$

Because we know the parameters above, we also know:

$$\rho_1, V_1, P_1, T_1, h_{o1}, h_1$$

Let's turn our attention to the Mach Number March method. Depending on the heat transfer model, AFT xStream will solve either Equation [21](#), [37](#) or [41](#). The most general of these is Equation 21, so in the fol-



lowing discussion it will be referred to as the Mach Number March equation, with the understanding that Equation 21 encompasses Equations 37 and 41.

Similar to the [Length March](#) method, it is desired to solve for conditions at location 2 based on known conditions at location 1 (see Figure 2). However, here the length step itself is not known. In fact, as shown in Equation 21 it is the computing length (i.e.,  $\Delta x$ ) for which we are solving. What is the reason for trying to solve for  $\Delta x$  rather than just specifying it? This question will be answered shortly. Before that, let's understand better what is happening when Equation 21 is applied.

The knowns at location 1 are the same as before, namely:

$$\dot{m}_1, x_1, M_1, P_{o1}, T_{o1}, Z_1, \gamma_1$$

with the other derived knowns easily obtainable:

$$\rho_1, V_1, P_1, T_1, h_{o1}, h_1$$

In the Mach Number March method, our step size is dictated by increments of Mach number. We thus take the Mach number at location 1 ( $M_1$ ) and add to it an increment. Since the maximum Mach number in the pipe systems modeled in AFT xStream is 1, the step size should be far smaller. The default step size is 0.01. Using the default, we know the Mach number at location 2:

$$M_2 = M_1 + \Delta M$$

$$M_2 = M_1 + 0.01$$

where  $\Delta M$  is the Mach number increment. As before, we know the mass flow rate at location 2 (it's the same as location 1), and we also have similar heat transfer relationships as discussed in the Length March method example previously.

Specifically, if the pipe is adiabatic, we know the following at location 2:

$$\dot{m}_2, M_2, h_{o2} \quad (\textit{adiabatic})$$

or, if isothermal, the following is known at location 2:

$$\dot{m}_2, M_2, T_2 \quad (\textit{isothermal})$$

and if the pipe has a convective or heat flux boundary condition,

$\dot{m}_2, M_2, h_{o2}$       (*convective & heat flux*)

However, in the convective and heat flux cases, we cannot definitely say what  $h_{o2}$  is because the amount of stagnation enthalpy change depends on the length step, for which we are solving. This requires additional iteration.

Similar to the Length March method, the Mach Number March method requires significant iteration for each computing step. Again, known inlet conditions become the first location 1, and once location 2 is solved, it becomes location 1 and the marching continues to the end of the pipe.

Since we are solving for  $\Delta x$ , it is entirely possible that the distance step will extend beyond the pipe length. This will occur for any pipe that has an end-to-end Mach number change of less than  $\Delta M$ . It will always occur for the final computing section of each pipe that does not reach choking. In such a case, the method reverts to the [Length March method](#) to compute conditions at the pipe exit.

It should be clearly understood that in the Mach Number March method, computing sections are not equidistant along a pipe. In fact, as the gas accelerates along the pipe, the Mach number increases more and more rapidly. This results in shorter and shorter computing section lengths as shown in the following application.

### Example Application

---

Consider once again the model in Figure 2. All conditions are the same as in the previous Length March method application, except for the solution method. If we solve this model using the Mach Number March method, with a Mach number increment of 0.01, the results in Figure 3 are obtained.

There are several items to note about Figure 3. First, note how the Mach number increases down the length of the pipe. At the inlet, the Mach number is 0.2604 (this is not specified, but determined as part of the solution for flow rate). Each computing point increases the Mach number by  $\Delta M$ , which is 0.01. This is what the Mach Number March method is, a method which marches in Mach number increments. With the step size dependent on the Mach number change, the number of calculation points in a pipe cannot be controlled. The 10 points shown in Figure 3 were chosen by AFT xStream, not the user.

Second, note the x locations of computing points. These numbers are not whole numbers, and as the end of the pipe is approached the distance between each computing section and the previous one becomes smaller and smaller. This is a result of the gas acceleration. At higher Mach numbers this becomes even more pronounced.

Figure 4 shows how the Mach number increases along the pipe length. The curve's upward curvature shows the acceleration effect. Figure 5 shows the relative computing section distances.

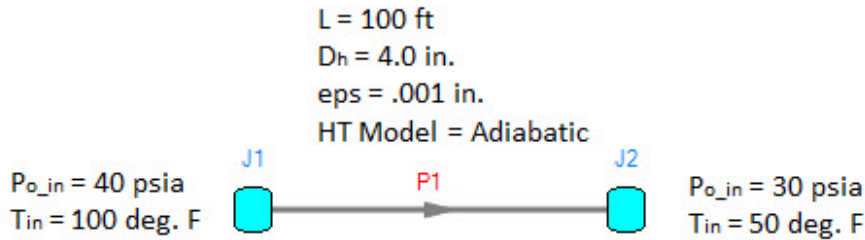
Third, notice this adiabatic flow case has constant stagnation enthalpy, but not constant static enthalpy. This is similar to the Length March method.

Last, notice how the mass flow rate is (in this case) the same as the Length March method results shown in Figure 3 of [Length March Method Example](#) (4.884 lbm/sec). Since both methods solve the same governing equations, one would expect the results to be comparable. However, the two methods will not

## Analysis Setup Overview

always agree. Each method has strengths and weaknesses, and results can differ somewhat when the flow conditions favor one of the methods.

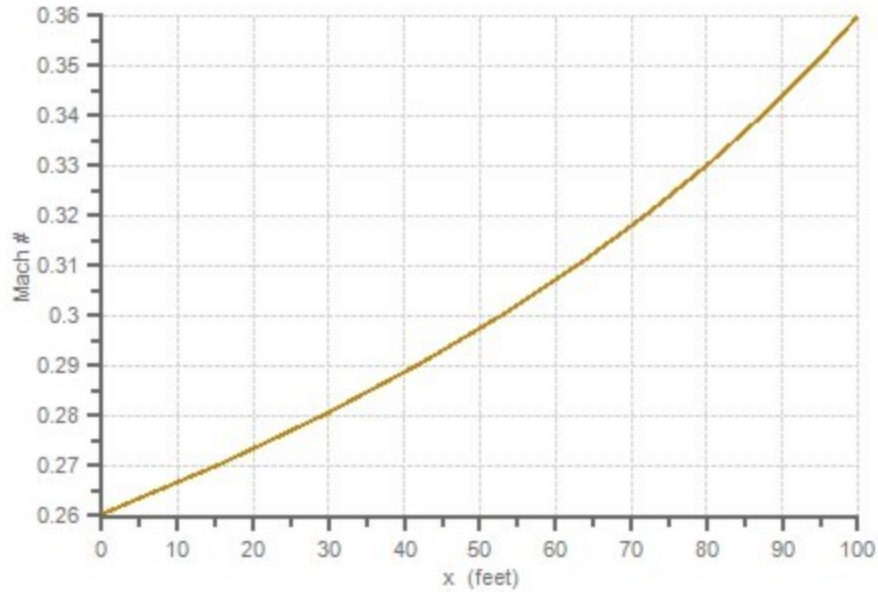
Is there any way to check that the results offer valid solutions of mass balance ([Equation 1](#)), momentum balance ([Equation 2](#)) and energy balance ([Equation 30](#))? There is. An example is given in the Mach Number March Verification section below.



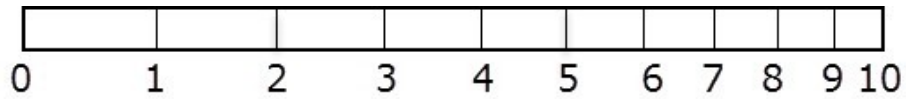
**Figure 2: Model of an air system dropping from 40 to 30 psia.**

Pipes		Axial Points										
Pipe -- Point (AFT Arrow Steady)	x (feet)	Mass Flow (lbm/sec)	Velocity (feet/sec)	Mach #	P Stag (psia)	P Static (psia)	T Stag (deg. F)	T Static (deg. F)	H Stag (Btu/lbm)	H Static (Btu/lbm)	Rho Static (lbm/ft <sup>3</sup> )	
1 -- 0	0.00	4.884	299.7	0.2604	40.00	38.16	100.00	92.52	202.2	200.4	0.1867	
1 -- 1	15.67	4.884	311.1	0.2704	38.64	36.73	99.97	91.91	202.2	200.3	0.1799	
1 -- 2	29.59	4.884	322.4	0.2804	37.38	35.40	99.93	91.28	202.2	200.1	0.1736	
1 -- 3	42.01	4.884	333.7	0.2904	36.22	34.16	99.91	90.63	202.2	200.0	0.1677	
1 -- 4	53.13	4.884	345.0	0.3004	35.13	33.00	99.88	89.97	202.2	199.8	0.1622	
1 -- 5	63.11	4.884	356.3	0.3104	34.12	31.92	99.85	89.28	202.2	199.6	0.1571	
1 -- 6	72.10	4.884	367.5	0.3204	33.18	30.90	99.83	88.58	202.2	199.5	0.1523	
1 -- 7	80.21	4.884	378.8	0.3304	32.29	29.94	99.81	87.86	202.2	199.3	0.1478	
1 -- 8	87.56	4.884	390.0	0.3404	31.47	29.04	99.79	87.12	202.2	199.1	0.1435	
1 -- 9	94.23	4.884	401.2	0.3504	30.69	28.20	99.77	86.36	202.2	199.0	0.1395	
1 -- 10	100.00	4.884	411.8	0.3599	30.00	27.43	99.75	85.62	202.2	198.8	0.1359	

**Figure 3: Axial Points (available in the Output window) for model in Figure 2 using Mach Number March method.**



**Figure 4:** The Mach number increases with length in this plot of the Figure 3 results



**Figure 5:** Relative pipe section locations for Figure 3 results. Notice how computing sections get closer as a result of gas acceleration as the end of the pipe is approached

### Mach Number March Method Verification

---

By changing the solution method for Verify\_Governing\_Equations.xtr in [Figure 5 of the Length March Verification](#) to the Mach Number March method, the verification process described in the previous section for the Length March method can be repeated. This is shown in Figure 6 and Table 1.

## Analysis Setup Overview

Pipes		Axial Points					
Pipe -- Point (AFT Arrow Steady)	x (feet)	P Static (psia)	Rho Static (lbm/ft3)	Velocity (feet/sec)	H Stag. (Btu/lbm)	f	
1 -- 0	0.000000	38.138699	0.18667814	301.36948	202.17825	0.014931721	
1 -- 1	15.597554	36.717525	0.17988965	312.74225	202.19830	0.014931537	
1 -- 2	29.455023	35.396305	0.17358769	324.09607	202.21611	0.014931159	
1 -- 3	41.812508	34.164791	0.16772211	335.43042	202.23198	0.014930761	
1 -- 4	52.870373	33.014107	0.16224931	346.74472	202.24620	0.014930346	
1 -- 5	62.796864	31.936489	0.15713139	358.03854	202.25896	0.014929913	
1 -- 6	71.734070	30.925156	0.15233518	369.31125	202.27043	0.014929463	
1 -- 7	79.802658	29.974121	0.14783147	380.56238	202.28081	0.014928997	
1 -- 8	87.105637	29.078119	0.14359453	391.79132	202.29019	0.014928514	
1 -- 9	93.731346	28.232458	0.13960156	402.99759	202.29871	0.014928017	
1 -- 10	99.755882	27.432987	0.13583227	414.18063	202.30644	0.014927504	
1 -- 11	100.000000	27.399927	0.13567655	414.65601	202.30676	0.014927232	

**Figure 6: Axial Points for AFT xStream model Verify\_Governing\_Equations.xtr using the Mach Number March method.**

**Table 1: Verification showing how AFT xStream Mach Number March method predictions satisfy momentum, mass and energy balance equations. Taken from spreadsheet Verify\_Governing\_Equations\_Mach.xls**

AFT ARROW OUTPUT							AVERAGED VALUES (CALCULATED)						MOMENTUM BALANCE					MASS BALANCE	ENERGY BALANCE		
P Static	Rho Static	Velocity	H Stag.				dp	V	d/dx	dho											
x (feet)	(psia)	(lbm/ft3)	(feet/sec)	(Btu/lbm)	f		(lbm/ft3)	(lbm/ft3)	(feet/sec)	(feet/sec)	(Btu/lbm)	dx (feet)	dz (feet)	dP Total (psi)	dP Fric (psi)	dp Accel (psi)	dp Grav (psi)	Balance (psi)	Balance (lbm/sec)	Balance (lbm/sec)	
0.00	38.139	0.1867	301.369	202.178	0.0149	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--
15.60	36.718	0.1799	312.742	202.198	0.0149	0.1833	0.00679	307.066	-11.373	-0.0200	-15.60	15.60	14.2117	-1.30300	-0.13815	0.01985	-0.00012	1.54E-09	-0.019	-0.019	
29.46	35.396	0.1736	324.096	202.216	0.0149	0.1767	0.00630	318.419	-11.354	-0.0178	-13.86	13.86	1.32122	-1.20041	-0.13791	0.01701	-0.00010	1.02E-07	-0.017	-0.017	
41.81	34.165	0.1677	335.430	202.232	0.0149	0.1707	0.00587	329.763	-11.334	-0.0159	-12.36	12.36	1.23151	-1.10856	-0.13767	0.01464	-0.00007	-1.72E-07	-0.015	-0.015	
52.87	33.014	0.1622	346.745	202.246	0.0149	0.1650	0.00542	341.088	-11.314	-0.0142	-11.06	11.06	1.15068	-1.02539	-0.13743	0.01267	-0.00006	1.11E-07	-0.014	-0.014	
62.80	31.936	0.1571	358.039	202.259	0.0149	0.1597	0.00512	352.392	-11.294	-0.0128	-9.93	9.93	1.07762	-0.95150	-0.13718	0.01101	-0.00004	-3.37E-08	-0.012	-0.012	
71.73	30.925	0.1523	369.311	202.270	0.0149	0.1547	0.00480	363.675	-11.273	-0.0115	-8.94	8.94	1.01133	-0.88406	-0.13692	0.00960	-0.00004	-4.05E-08	-0.011	-0.011	
79.80	29.974	0.1478	380.562	202.281	0.0149	0.1501	0.00450	374.937	-11.251	-0.0104	-8.07	8.07	0.95104	-0.82281	-0.13665	0.00841	-0.00002	-5.64E-09	-0.010	-0.010	
87.11	29.078	0.1436	391.791	202.290	0.0149	0.1457	0.00424	386.177	-11.229	-0.0094	-7.30	7.30	0.89600	-0.76703	-0.13638	0.00739	-0.00002	9.97E-08	-0.009	-0.009	
93.73	28.232	0.1396	402.998	202.299	0.0149	0.1416	0.00399	397.394	-11.206	-0.0085	-6.63	6.63	0.84566	-0.71608	-0.13610	0.00652	0.00000	-3.18E-08	-0.008	-0.008	
99.76	27.433	0.1358	414.181	202.306	0.0149	0.1377	0.00377	408.589	-11.183	-0.0077	-6.02	6.02	0.79947	-0.66941	-0.13582	0.00576	0.00000	-5.19E-08	-0.007	-0.007	
100.00	27.400	0.1357	414.656	202.307	0.0149	0.1358	0.00316	414.418	-0.475	-0.0003	-0.24	0.24	0.03306	-0.02751	-0.00577	0.00023	0.00001	-3.04E-08	0.000	0.000	

### Related Topics

Nomenclature.....410

## Lumped Adiabatic Method - Detailed Discussion

When this [AFT Arrow Steady Solution Method](#) is used elevation data is ignored by the Solver. The output will still show the elevation as entered by the user. Since elevation effects are usually not very important in gas systems, neglecting elevation changes is frequently an acceptable approximation.

An analytical solution to [Equations 1, 2 and 5](#) for adiabatic flow neglecting elevation changes is given by the following (e.g., see [Saad, 1993](#), page 209):

$$\int_0^L \frac{f}{D} dx = \int_{M_1}^{M_2} \frac{1 - M^2}{\gamma M^4 \left(1 + \frac{\gamma - 1}{2} M^2\right)} dM^2 \quad (42)$$

This equation is valid for real gases as long as the following approximation is accepted:

$$\Delta h_o \approx c_p \Delta T_o$$

using a suitably averaged value of  $c_p$ . Integrating from 0 to L obtains:

$$\frac{fL}{D} = \frac{1}{\gamma} \left( \frac{1}{M_1^2} - \frac{1}{M_2^2} \right) + \frac{\gamma + 1}{2\gamma} \ln \frac{M_1^2}{M_2^2} \left( \frac{1 + \frac{\gamma - 1}{2} M_2^2}{1 + \frac{\gamma - 1}{2} M_1^2} \right) \quad (43)$$

Equation 43 is then solved for each pipe, using an average value of friction factor and  $g$  for each pipe. The solution of Equation 43 typically is iterative where the unknown is  $M_2$ .

## Related Topics

References..... 461

### Lumped Isothermal Method - Detailed Discussion

When this [AFT Arrow Steady Solution Method](#) is used elevation data is ignored by the AFT Arrow Steady solver. The output will still show the elevation as entered by the user. Since elevation effects are usually not very important in gas systems, neglecting elevation changes is frequently an acceptable approximation.

Another special case of flow in pipes is that of isothermal flow. In isothermal flow, the gas static temperature remains constant. The tendency is for gas to cool as it flows along a pipe. For the temperature to remain constant, an inflow of heat is required.

When the temperature remains constant, it removes one of the unknowns from [Equations 1, 2 and 5](#). From Equation 5, the density becomes directly proportional to pressure, and an analytical solution can be obtained if elevation changes are neglected (e.g., see [Saad](#), 1993, pp. 264-269):

$$\int_0^{L_T} \frac{f}{D} dx = \int_{M_1}^{M_2} \frac{1 - \gamma M^2}{\gamma M^4} dM^2 \quad (44)$$

where the T subscript on L emphasizes isothermal. Integrating from point 0 to L,

$$\frac{f L_T}{D} = \frac{1 - \left( \frac{M_1^2}{M_2^2} \right)}{\gamma M_1^2} - \ln \frac{M_2^2}{M_1^2} \quad (45)$$

Isothermal flow behavior results in a different relationship for sonic choking as compared to adiabatic flow. In fact, to truly maintain isothermal flow up to the sonic point requires an infinite amount of heat transfer. This results in the strange but mathematically correct conclusion that sonic choking occurs at a Mach number less than 1 for isothermal flow. Sonic choking will occur at  $1/\sqrt{\gamma}$  for isothermal flow, where  $\gamma$  is the isentropic expansion coefficient (Saad, 1993, pp. 267). Practically speaking, isothermal flow will not remain isothermal at high velocities.

Similar to the [Lumped Adiabatic](#) method, Equation 45 is solved for each pipe, using an average value of friction factor and  $g$  for each pipe. The solution of Equation 45 typically is iterative where the unknown is  $M_2$ .

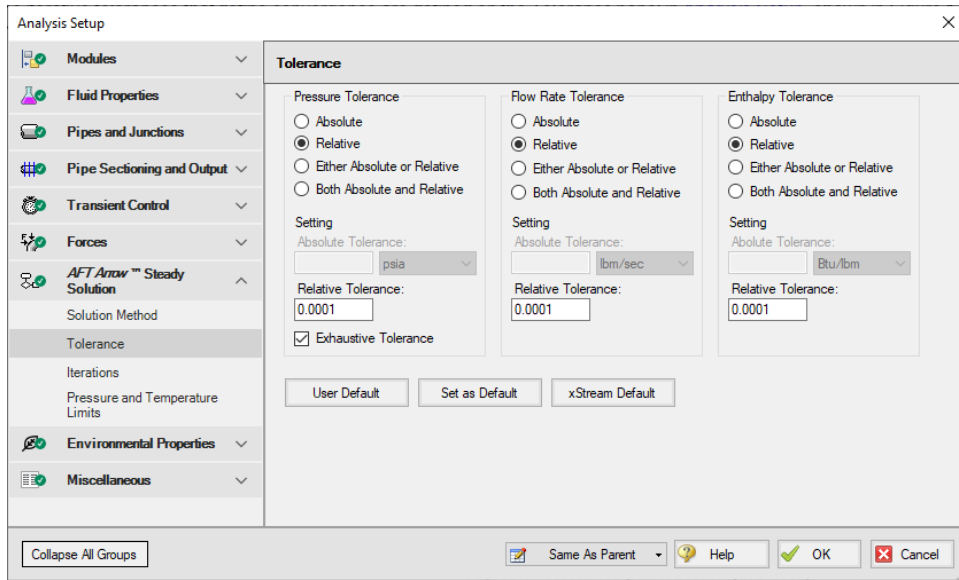
When the user chooses the Lumped Isothermal method, an isothermal temperature is required in the [AFT Arrow Steady Solution panel](#). This temperature is used for all pipes.

## Related Topics

References.....	461
-----------------	-----

## Tolerance Panel

The iterative tolerances that define convergence are defined on the Tolerance panel in [Analysis Setup](#), shown in Figure 1. These values can be modified for Pressure Tolerance, Flow Rate Tolerance, and Enthalpy Tolerance.



**Figure 1: Tolerance panel in Analysis Setup**

An iterative approach is used to obtain pipe flow solutions. When approaching any analysis iteratively, a stopping point must be defined. The simplest approach is to stop after a certain number of iterations. However, this either results in uncertain results because the result is still changing, or wasted processing time because there are too many iterations.

A solution is to use iterative tolerance. This tolerance is related to the change between iterations. When this change is smaller than the iterative tolerance, the solution is said to be converged. These settings are specified in the [Solution Method panel](#).

## Tolerance and Solution Accuracy

Iterative tolerance does not have any relationship to an engineering tolerance.

An engineering tolerance is a familiar term to most engineers - it is a permissible limit in variation of a parameter. A simple example is the length of a steel bar - an engineer might specify that this length must be  $3 \pm 0.01$  meters. A steel bar manufactured by this specification is known to be within the engineering tolerance of the "true" or desired length. In other words, engineering tolerance describes an accuracy.

Iterative tolerances **do not** directly describe an accuracy. Convergence **only** means that the change between iterations was within the specified limit.

With sound iterative methods, and an appropriately specified iterative tolerance, the iterative solution will be close to the true solution. However, even with good settings, the iterative tolerance does not define how close the results are to the true solution.

If there is concern that the results may not be close enough to the true answer, the tolerances could be lowered and the results compared. If the results do not change appreciably, an appropriate tolerance was likely selected.



## Avoiding False Convergence

---

It is important to keep the tolerance at least one order of magnitude smaller (two orders or more is recommended) than the [relaxation](#). A false convergence can occur when the change from the old value to the new value is small enough to be within the tolerance because it is restricted by the relaxation.

## Absolute Tolerance

---

Absolute tolerance looks at the *absolute* change in quantity of a given parameter. If the change is smaller than the directly specified value, the parameter is considered converged.

This approach works equally well for both small and very large values, making iterative solvers more likely to find convergence. However, specifying an appropriate absolute tolerance requires some knowledge of the true solution. The tolerance should be set 3 or 4 orders of magnitude below the solution, if there are near zero flows in the system this will require a very low tolerance.

## Relative Tolerance

---

Relative tolerance checks the *relative* change of a given parameter. Each change is divided by the number itself, giving a change proportional to the magnitude.

This approach does not require any knowledge about the true solution, and for that reason is the default method.

Table 1 shows the difference between relative and absolute change while iterating. Notice that the absolute change has units, while relative change does not.

Convergence criteria can also be based on either (or both) absolute or relative change. Table 1 shows how this is applied.

**Table 1: Example of four tolerance methods for a mass flow rate example. Absolute tolerance = 0.0005 lbm/s, Relative tolerance = 0.0001**

Iteration #	Mass Flow Rate (lbm/sec)	Relative Change	Absolute Change (lbm/sec)	Meets Abs. Tolerance?	Meets Rel. Tolerance?	Meets Abs. OR Rel. Tolerance?	Meets Abs. AND Rel. Tolerance?
1	20.0000	-	-	-	-	-	-
2	12.0000	0.666667	8.000000	NO	NO	NO	NO
3	14.0000	0.142857	2.000000	NO	NO	NO	NO
4	13.5000	0.037037	0.500000	NO	NO	NO	NO
5	13.6000	0.007353	0.100000	NO	NO	NO	NO
6	13.5900	0.000736	0.010000	NO	NO	NO	NO
7	13.5913	0.000096	0.000130	NO	YES	YES	NO
8	13.5907	0.000044	0.000600	NO	YES	YES	NO
9	13.5909	0.000015	0.000200	YES	YES	YES	YES

## Exhaustive Tolerance

AFT xStream finds a solution for all junction pressures when Exhaustive Tolerance is enabled.

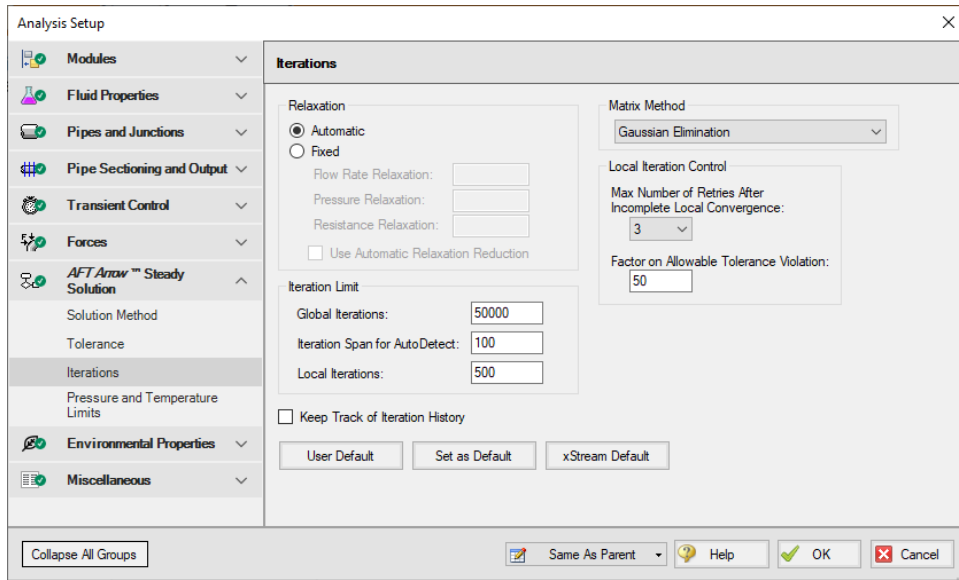
Disabling this option will only check junction pressures that are in the [solution matrix](#). Only junctions that contain a split in flow are included in the matrix. Both options should come to a reasonably similar converged result, but the more rigorous approach of checking all junctions is selected by default.

## Related Blogs

[Having model convergence problems? Start here!](#)

### Iterations Panel

Relaxation, iteration limits, and matrix method settings are defined on the Iterations panel in [Analysis Setup](#). Most settings on this panel will not affect the final result. The default state of the Iterations panel is shown in Figure 1.



**Figure 1: Iterations panel in Analysis Setup**

## Relaxation

Relaxation slows iterative progress toward a solution reducing the destabilizing effect of non-linear systems.

- **Automatic** - Implements an adaptive relaxation scheme that reduces the relaxation dynamically in response to highly non-linear features of the problem.
- **Fixed** - Allows for user specified Flow Rate and Pressure Relaxation

When applied to the flow solution, relaxation  $r$  is used as follows:

$$\dot{m}_{new} = r (\dot{m}_{new} - \dot{m}_{old}) + \dot{m}_{old}$$

A relaxation of 1 is the same as no relaxation. Each iteration's prediction is used directly in the next iteration.

A relaxation of 0 is not allowed, as this would prevent the parameter from changing.

- **Flow Rate Relaxation** - For highly non-linear systems, a flow rate relaxation of 0.1 is advised. Experience shows that using flow rate relaxation factors between 0.5 and 1 has limited usefulness. If a flow rate relaxation of 1 does not work, you should generally try flow rate relaxation parameters between 0.1 and 0.5. In more extreme cases, a flow rate relaxation factor smaller than 0.1 may be required.
- **Pressure Relaxation** - Because of the nature of the matrix solver, experience indicates that if pressure relaxation is used, it should **always** be set to 0.5 or 1 to maintain stability.

- **Resistance Relaxation** - Typically should be left at 1. In some cases with very long pipelines, reducing the value as low as 0.1 can improve convergence.

### Iteration Limit

---

- **Maximum Iterations** - 50000 by default, sufficient for the vast majority of models. Most models converge in a relatively small number of iterations. This limit does not directly affect the solution.
- **Iteration Span for AutoDetect** - If the AFT Arrow Steady Solver detects that no meaningful progress has been made to convergence within this many iterations, relaxation is automatically lowered. This also automatically adjusts [tolerance](#).
- **Local Iterations** - The maximum number of local iterations allowed before allowing the Solver to continue without local convergence. See **Local Iteration Control** below for details.

### Iteration History

---

Iteration History shows a history of which pipe or junction is most [out of tolerance](#), which is helpful for troubleshooting. After enabling Iteration History on the Iterations panel in [Analysis Setup](#), the Iteration History can be viewed from the [Solution Progress window](#) by pausing the calculation and selecting the Other Actions button.

### Matrix Method

---

Network analysis requires solving complex matrices potentially thousands of times. Different methods may solve faster or slower, but have no bearing on results. The default method of Gaussian Elimination is very robust and usually provides the fastest convergence. The available Matrix Methods are:

- Gaussian Elimination
- Gaussian Elimination With Pivoting
- LU Decomposition

### Local Iteration Control

---

In the AFT Arrow Steady Solver, properties such as density, specific heat, and viscosity will vary over an interval. Calculations done on such an interval should use average values for the interval. However, the average values cannot be directly calculated without knowing the values at both the beginning and ending of the interval. Because the parameters at the end of the interval are not known initially, the local properties must be solved for iteratively.

In certain models, these local iterations may not converge. The **Local Iterations** limit provides a means to allow the Solver to continue if this occurs.

If the Local Iteration limit is reached, the Solver will continue with the global solution using the non-converged local solution. This does not affect the final results unless it occurs on the final global pass through the Solver. If a local iteration problem does occur on the final pass, the solution is retried. A **Max**

**Number of Retires After Incomplete Local Convergence** will be carried out to attempt to solve the problem. If the retries are not successful, the results will be presented as converged along with a warning including information on the amount out of tolerance.

A default **Factor on Allowable Tolerance Violation** of 50 is specified here. Local iterations within this factor of the built-in tolerance values are deemed adequately acceptable. Setting this value to 1 will show a warning for any tolerance violation.

### Related Blogs

---

[Having model convergence problems? Start here!](#)

### Pressure and Temperature Limits Panel

If the maximum and minimum temperatures or pressures in a system are known, these can be entered here. If limits are present, the AFT Arrow Steady Solver will not allow solutions outside of these bounds.

This is most useful when operating near the edge of available physical property data. During iteration, the Solver may attempt to extrapolate past the available data, returning an error. Limiting the allowable range can keep the search within the proper bounds.

---

**Note:** This setting can affect results. It is critical that the limits are appropriately selected to not artificially constrict the solution.

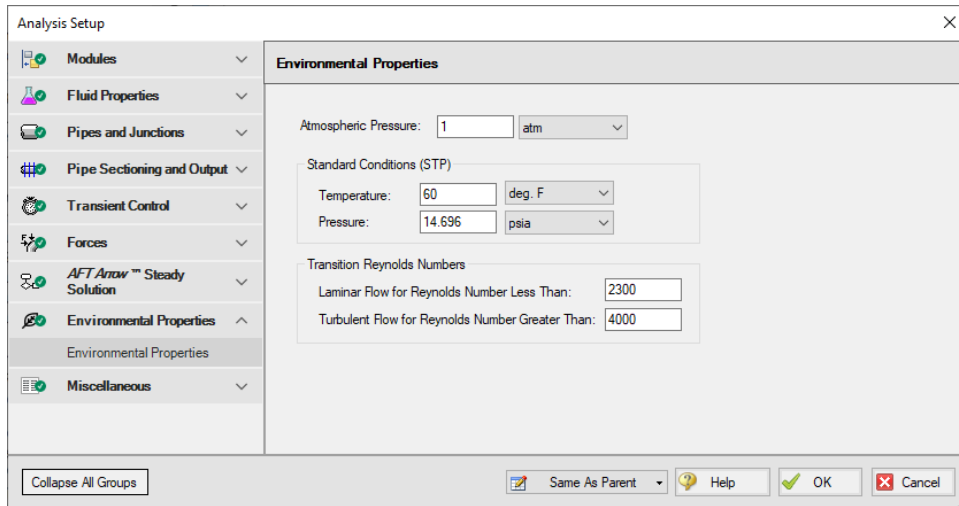
---

### Environmental Properties Group

The Environmental Properties Group in [Analysis Setup](#) has one item: [Environmental Properties panel](#). By default, the Environmental Properties Group is fully defined and will have a green checkmark next to its name.

### Environmental Properties Panel

The Environmental Properties panel, shown in Figure 1, controls the atmospheric pressure, acceleration properties, standard conditions, and transition Reynolds number used to model the system.



**Figure 1: Environmental Properties panel in the Analysis Setup window**

### Atmospheric Pressure

AFT xStream uses a default atmospheric pressure of 1 standard earth atmosphere, or 14.696 psia. This pressure is also used to convert absolute pressures to gauge pressures. Keep in mind that atmospheric pressure varies with altitude. A project at Sea Level should not have the same atmospheric pressure as a project at 8500 feet.

### Standard Conditions (STP)

The standard temperature and pressure (STP) are used for defining common usage flow rate units including standard cubic feet per minute (scfm).

AFT xStream uses a default standard pressure of 14.696 psia. This value is used consistently across industry, and the user likely will not need to change it. AFT xStream uses a default standard temperature of 60° F. This value changes across industries, and the user may wish to change it.

Another common usage flow rate is normal flow rate units. The definition of normal uses the same standard pressure as above, but 32° F (0° C) for temperature. The normal pressure and temperature cannot be changed by the user.

### Transition Reynolds Numbers

AFT xStream allows you to modify the transition Reynolds numbers. The default values are Reynolds number less than 2300 is laminar and greater than 4000 is turbulent. In the transition zone a linear interpolation is assumed. This applies for friction factors as well.

### Miscellaneous Group

The Miscellaneous Group in [Analysis Setup](#) has two items: the [Advanced Parameters panel](#) and [Advanced Transient Parameters panel](#). Both panels contain advanced parameters intended for troubleshooting only and should be used with caution.

### Advanced Parameters

The Advanced Parameters panel offers various advanced options to adjust the AFT Arrow Steady Solver.

These parameters should not be changed under normal circumstances and will seriously impact results. They can be useful for troubleshooting certain models. [Contact AFT](#) for more information.

### Advanced Transient Parameters Panel

The Advanced Transient Parameters panel is only visible when Show Advanced Options is checked.

The options on this panel are intended for troubleshooting use and should only be changed with caution. In almost every model, the default values for these parameters is appropriate. There are rare circumstances where the user may wish to change them.

A description of some of the most commonly used advanced parameter options is provided here. For information and guidance on other options please [contact AFT](#).

- **Create Transient Debug File** - Disabled by default. When enabled, xStream will write relevant diagnostic information to a text file, including information on tolerance and convergence settings, records of when/if saturation conditions were reached, etc. The transient debug file will be created in the same location as the model file, and follows the naming structure "*Model Name*\_#\_GasTranDebug.txt", where Model Name is the user specified model name, and # represents a number assigned to differentiate between the transient debug files generated for different scenarios. It is recommended to send this file to AFT when requesting support on a model file.
- **Limit Properties to Saturation Conditions** - Disabled by default. If enabled, xStream will attempt to limit the temperature and pressure to saturation conditions to allow the transient solution to continue. This will not produce accurate results as xStream cannot account for two-phase flow. However, this may be useful for troubleshooting purposes to be able to analyze the transient results up to the point that saturation conditions were reached.
- **Halt Solver When Saturation Conditions Encountered** - Disabled by default. If enabled, xStream will halt the solver if the temperature and/or pressure conditions exceed the saturation line. This is useful to provide the user the chance to immediately stop the transient and troubleshoot the model, as any transient results calculated after the time that saturation conditions were reached will be invalid.

## Working with Pipe and Junction Objects

This section focuses on interactions with the Pipe and Junction objects in the AFT xStream workspace. Topics include how to create pipes and junction objects, how to alter junctions already on the workspace, and how to ensure the inputs for objects are both correct and complete.

### Contents

---

<a href="#">Selecting and Finding Objects</a> .....	140
<a href="#">Pipe Drawing Modes</a> .....	144

## Selecting and Finding Objects

AFT xStream has a number of methods to assist in selecting multiple objects at once as well as finding objects for recall later. The contents of this section can be broken down into the two categories shown below. Follow the links to learn more.

- Methods for [selecting objects](#)
  - Select Flow Path
  - Select Special
  - [Groups](#)
- Methods for [finding objects](#)
  - Workspace Find
  - Workspace Search
  - Find from Model Data or Output
  - Jump Feature
  - Bookmarks

### Related Topics

---

Selecting Objects .....	140
Moving Objects .....	159
Editing Objects .....	160
Defining Objects .....	161

## Selecting Objects

A single left click on any object will select that object.



Selecting multiple objects can be accomplished by:

- Holding the shift or control key and left clicking on objects
- Dragging the cursor over the objects, creating a selection area
  - Dragging left-to-right creates a selection area that will only select objects entirely bounded by the area
  - Dragging right-to-left creates a selection area that selects any objects completely or partially inside the area
- Using Select All from the Edit Menu, right-clicking on Workspace to open context menu, or keyboard shortcut (Ctrl+A)
- Using Select Last Selection from the Edit Menu or with F12 to reselect the previously selected objects
- Using **Select Flow Path**
- Using **Select Special**
- By selecting a [Group](#)

### Select Flow Path

---

A flow path is a sequence of pipes and junctions where there is no branching or flow sources or sinks.

The pipes and/or junctions in a flow path may all be easily selected by selecting a pipe on the Workspace then clicking the Select Flow Path item on the [Edit menu](#), [Toolbar](#), or the right-click context menu.

This feature is very useful in duplicating or deleting sequences of pipes and junctions. It can also be used to quickly define groups for graphing or for selected Output or Model Data listings.

### Select Special

---

Select Special is a tool for selecting or deselecting [objects](#) based on certain criteria. It can be opened from all windows/menus that give you the option to choose what pipes and/or junctions are selected. These windows/menus include the Output Control window, the Model Data window, the Visual Report Control window, the Select Graph Data window, the [Edit menu](#), right clicking on the [Workspace](#), or pressing F6 on the keyboard.

Once Select Special is open, all pipe names and numbers and junction names and numbers are shown in numerical order under Pipe Selections and Junction Selections, respectively. At the top of the Select Special window is the Selection Type option. This option allows you to choose what criteria you would like to use to select certain pipes and junctions. You can choose from the following criteria to select certain pipes and/or junctions.

- Selected on Workspace
- Closed in Model
- Special Conditions Set
- Number Range
- Junction Type

- Pipe Input Properties
- Notes
- Undefined Objects
- User-Colored Pipes
- Transient Data Exists
- Groups
- Pipe Output Properties (Only available when output is available)
- Junction Output Properties (Only available when output is available)

Choosing certain Selection Types will prompt a second or sometimes third option below the Selection Type to allow you to further specify the type of pipe or junction you want. For example, selecting Junction Type in the Selection Type box will prompt the Junction Type box to appear. The Junction Type box will allow you to specify what type of junctions you would like to select.

By default, the Selection Type is Selected on Workspace and any pipes or junctions selected on the Workspace are selected in the Select Special window with a black check mark next to the box corresponding to the pipe or junction.

Once you have fully specified the Selection Type, you must click Select Pipes and/or Select Junctions to automatically select the corresponding objects.

The selected pipes and junctions under Pipe Selections and Junction Selections do not automatically become unselected when you switch from one Selection Type to another. Therefore, you must first choose Deselect Pipes and/or Deselect Junctions located above the Pipe Selections and Junction Selections boxes, respectively. For example, if you select all of the pipes on the Workspace prior to opening the Select Special window, all of the pipes will be selected in the Select Special window when you open it. If you then choose Junction Type as the Selection Type and then specify All Junctions as the Junction Type, all junctions will be selected under Junction Selections, but all pipes previously selected will also remain selected. In this example, if you would like only junctions to be selected with none of the selected pipes on the Workspace to be selected in the Select Special window, first click the Deselect Pipes button, which is located above the Pipe Selections list.

Options to select all pipes/junctions, to deselect all pipes/junctions, and to invert the selected pipes/junctions are located below both Pipe Selections and Junction Selections boxes. Inverting the pipes/junctions deselects all currently selected pipes/junctions and selects all currently deselected pipes/junctions. This feature is helpful if you want to select all pipes/junctions that **do not** fit a certain criteria. For example, if you want to select all open objects in the model, you could specify Closed in Model in the Selection Type box, then click Select Pipes and Select Junctions. All pipes and junctions that are closed in the model will be selected. Because you want to select all open pipes and junctions in the model but only the closed pipes and junctions are selected, you would simply click Invert below both the Pipe Selections and Junction Selections lists to deselect all closed pipes and junctions and select all open pipes and junctions.

### Related Topics

---

Selecting and Finding Objects .....	140
Moving Objects .....	159
Editing Objects .....	160

Defining Objects..... 161

## Finding Workspace Objects

### Workspace Find

---

The Find window utility helps you find a particular pipe or junction by its ID number and name. The Find utility becomes more useful for models that expand over multiple Workspace pages.

You can access the Find utility by selecting Find on the [View menu](#), by clicking the button on the [Toolbar](#), or by using the keyboard shortcut "Ctrl+F". After you double click on the pipe or junction you are searching for in the Find utility or when you click the Find button, the Workspace is moved so that the pipe or junction is located near the center of the screen. Additionally, the pipe or junction you are searching for is selected in the Selected Items color specified in the [User Options](#) window in the [Tools menu](#). By default, this color is red.

### Workspace Search

---

The Search window utility helps you search for a particular pipe or junction by its ID number, name, or notes you have inputted in the properties window. The Search utility becomes more useful for models that expand over multiple Workspace pages.

You can access the Search on the Common [Toolbar](#). The search feature will list all objects related to your search term beneath itself in a list of search results. You can hover over each search results to see a preview of input and output (if the model has been run). Clicking on or pressing enter while the object is highlighted in the search results will select the object in the workspace and center the focus of the workspace on the object.

### Find From Model Data or Output

---

Objects can be quickly located on the Workspace by right-clicking on a cell in Model Data or row header in Output and selecting "Find on Workspace..." the view will change to show the Workspace, with the object of interest selected.

### Jump Feature

---

By clicking the Jump button in a [Properties window](#), you can move directly to another Properties Window of another a pipe or junction that you specify.

The Jump button is located in the upper right corner of the Properties Window.

## Bookmarks

---

Use the Bookmark tools on the View menu to set Bookmarks and to display different bookmarked pipes or junctions.

Bookmarks can be added on the [Workspace](#) in relation to individual pipes or junctions. You can quickly navigate around the Workspace by moving from Bookmark to Bookmark.

## Groups

Groups of [pipes and junctions](#) can be named for later recall. This is done using the Group tools on the Edit menu. The easiest way to create a named Group is as follows:

1. First select the pipes and junctions you want in the group
2. Choose Groups -> Create from the Edit menu or Toolbar.
3. You will be prompted for a name for the group. Enter a meaningful name and click OK.
4. The Group Manager then opens. The selection of pipes and junctions in the Group will match those currently selected on the Workspace. If you want to modify the selections once more, you can do it here.

You can include Subgroups inside of your Groups. Subgroups are existing Groups. If a Group contains one or more Subgroups and the Group is selected on the Workspace, then all pipes and junctions in the Group and Subgroup(s) will be selected. Additionally, if you include a Subgroup in your Group and then later change the contents of the Subgroup, the Group that contains the Subgroup will also be changed.

After you have created a Group, you can then quickly reselect that Group of pipes or junctions on the Workspace by choosing Group -> Select from the Edit menu or Toolbar and choosing the name of the Group.

The Group Manager can be opened by choosing Group -> Manager from the Edit menu or Toolbar. The Group Manager allows you to modify, create, delete or rename Groups.

Another place that Groups can be selected from is in the [Select Special](#) window.

## Related Topics

---

Graph Results Window.....	311
Global Pipe Editing.....	152
Global Junction Editing.....	153
Force Set Types.....	33

## Pipe Drawing Modes

AFT xStream has 3 different pipe drawing modes. These pipe drawing modes are

- [2D Freeform](#)
- [2D Orthogonal](#)

- [Isometric](#)

This section will discuss the features of each drawing mode as well as how to adjust the pipe presentation in each including [Snap to Grid](#).

### Related Blogs

[Iso love the new Pipe Drawing Mode!](#)

## Snap to Grid

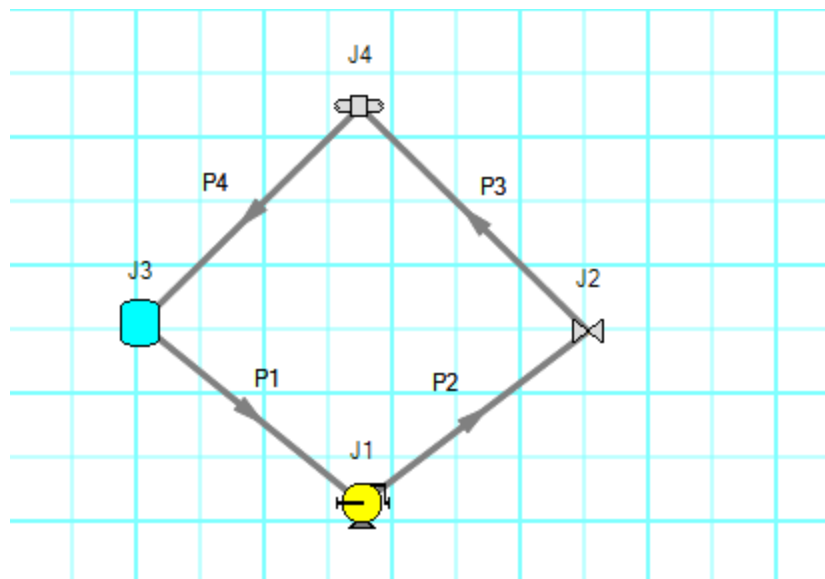
Toggled from Arrange > Snap to Grid.

All junctions, pipe endpoints, and pipe vertices will be snapped to the grid intersection points when Snap to Grid is on. Turning Snap to Grid on in an existing model will optionally attempt to move all existing junctions to the grid points.

## 2D Freeform Grid

The standard grid, enabled by default in the Arrange -> Pipe Drawing Mode menu.

This Pipe Drawing Mode allows placing pipes or junctions anywhere on the Workspace, with pipes at any angle.



*Figure 1: 2D Freeform with Snap to Grid off*

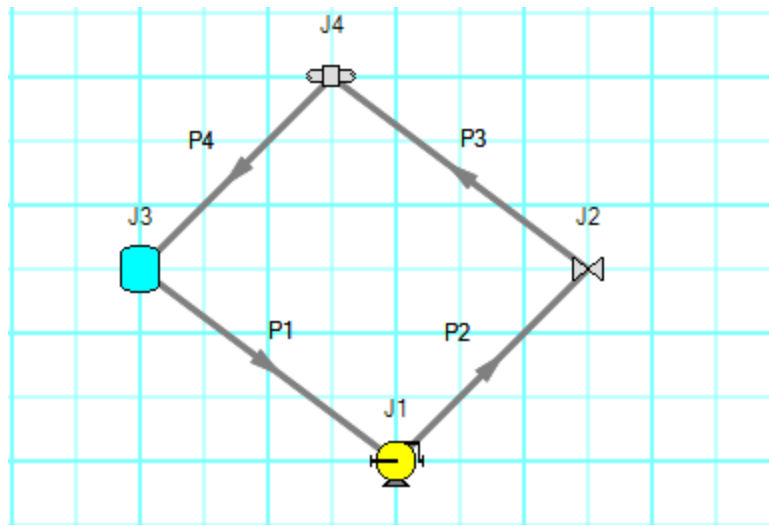


Figure 2: 2D Freeform with Snap to Grid on

## 2D Orthogonal

The 2D Orthogonal Pipe Drawing Mode forces all pipes drawn in the model to be either horizontal or vertical. This is enabled in the Arrange -> Pipe Drawing Mode menu.

In 2D Orthogonal mode, segments are automatically inserted into pipes to enforce drawing only along the horizontal and vertical.

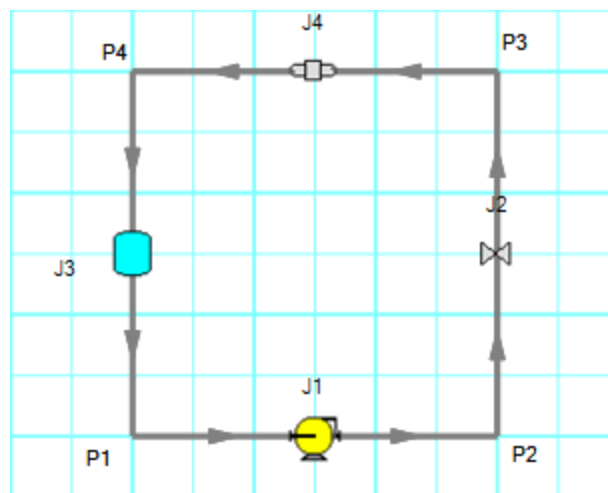



Figure 1: Model in 2D Orthogonal mode with Snap to Grid on

### Tips for Using the Orthogonal Drawing Feature

---

Orthogonal drawing can create some confusion for the user because segments from different pipes may overlap each other and hence become indistinguishable.

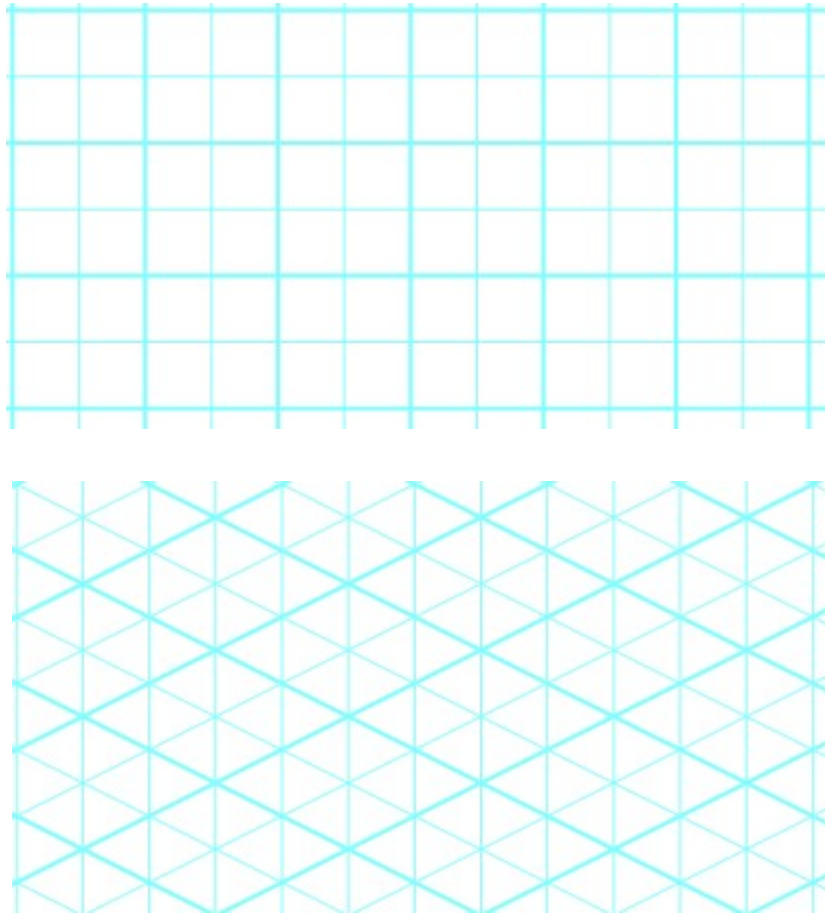
- Use the Add/Remove Segments button  to add new vertices
- Use a [fine grid](#) and [Snap to Grid](#) features
- If you have a junction with several connecting pipes that interfere, [enlarge the junction](#) to 150% or 200% of normal size

### Isometric

AFT xStream allows the user to place pipe or junction objects anywhere in the Workspace.

In order to make building clean looking models easier, objects are snapped to the 2D Freeform grid by default. However, it may be convenient or desired to demonstrate the three dimensional nature of a system. A typical 2D grid has two sets of gridlines representing the x and y axes. An isometric grid has three sets of gridlines that are offset by 60°, representing the x, y, and z axes.

Enable the isometric grid by selecting Pipe Drawing Mode -> Isometric from the Arrange menu. There will be a prompt about converting non-isometric pipes - this is a required step, but may result in some unintended object placement. It is best practice to set the grid type before building a model and retain that type of grid if possible. It is also very helpful to show the Grid (Arrange -> Show Grid) when building an isometric model.



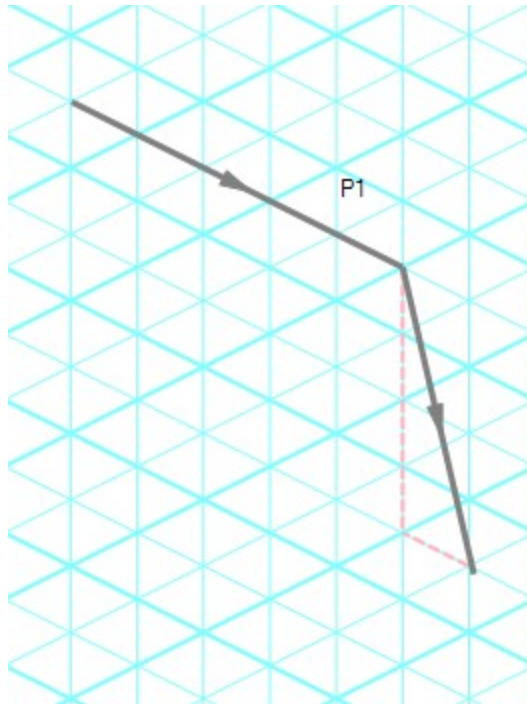
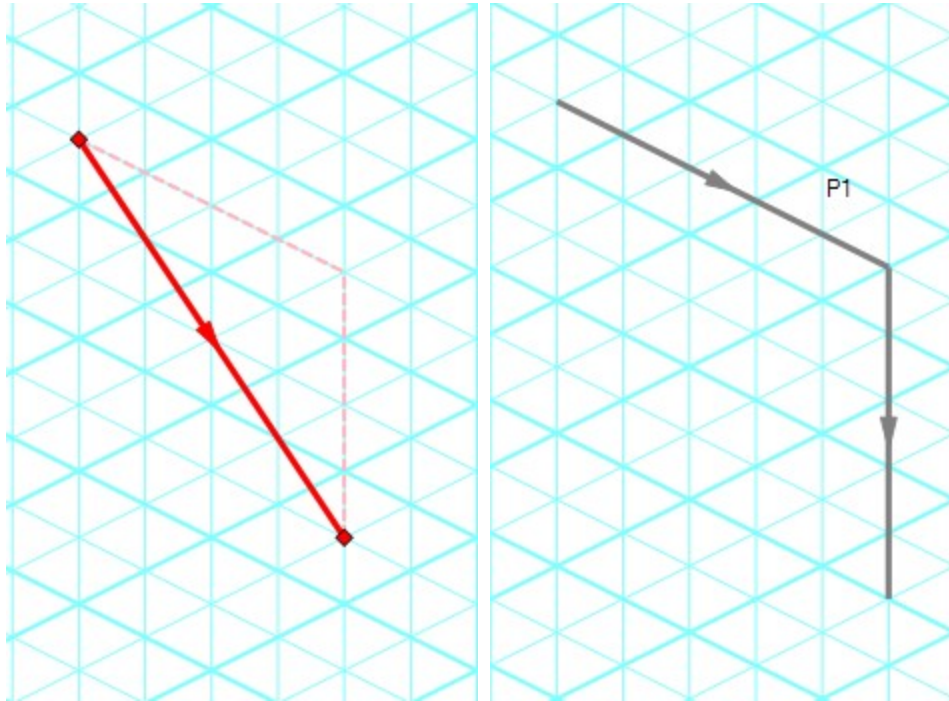
**Figure 1: 2D (left) and isometric (right) grids**

When drawing on the isometric grid, pipes are automatically segmented to lie on the gridlines as shown below. Dragging one of the handles on the pipe shows a preview line of where the pipe will be placed if adjustment needs to be made.

The placement of the preview line can be cycled through all possibilities by pressing the arrow keys or by scrolling the mouse scroll wheel.

Holding the alt key while dragging a pipe endpoint will retain the existing vertex and automatically add a new segment to the pipe - the placement of the line can be controlled with the arrow keys or scroll wheel as before.





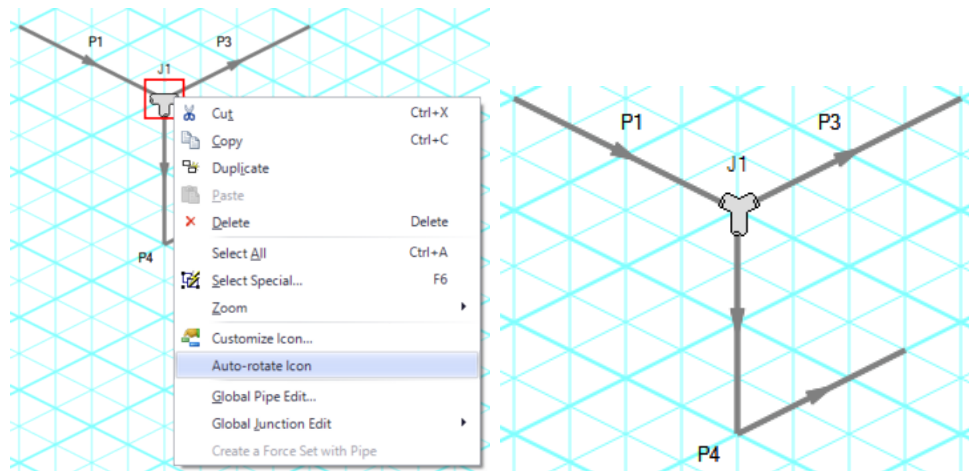
**Figure 2: Automatic layout of isometric pipes**

The default action is for pipes in the workspace to snap to the isometric gridlines. However, if desired, a given pipe can be set to not snap to the grid by clicking Edit -> No Snapping for Pipe or by right-clicking a selected pipe and clicking the No Snapping for Pipe option.

## Placing Junctions

Placing junctions onto the Workspace follows the usual rules - however, the visual appearance of the icon is more complex than in an orthogonal grid. When icons are added after pipes are placed, AFT xStream will rotate the icon automatically following the workspace arrangement. When pipes are connected to icons already placed in the workspace, right-click and select Auto-rotate Icon to adjust the icon. For manual control of icon appearance, right-click and select Customize Icon to choose the desired appearance.

**Note:** It may be easiest to make use of the Rotate button in the Customize Icon window to find the appropriate icon. With many possibilities to choose from, finding the proper rotation can be difficult.



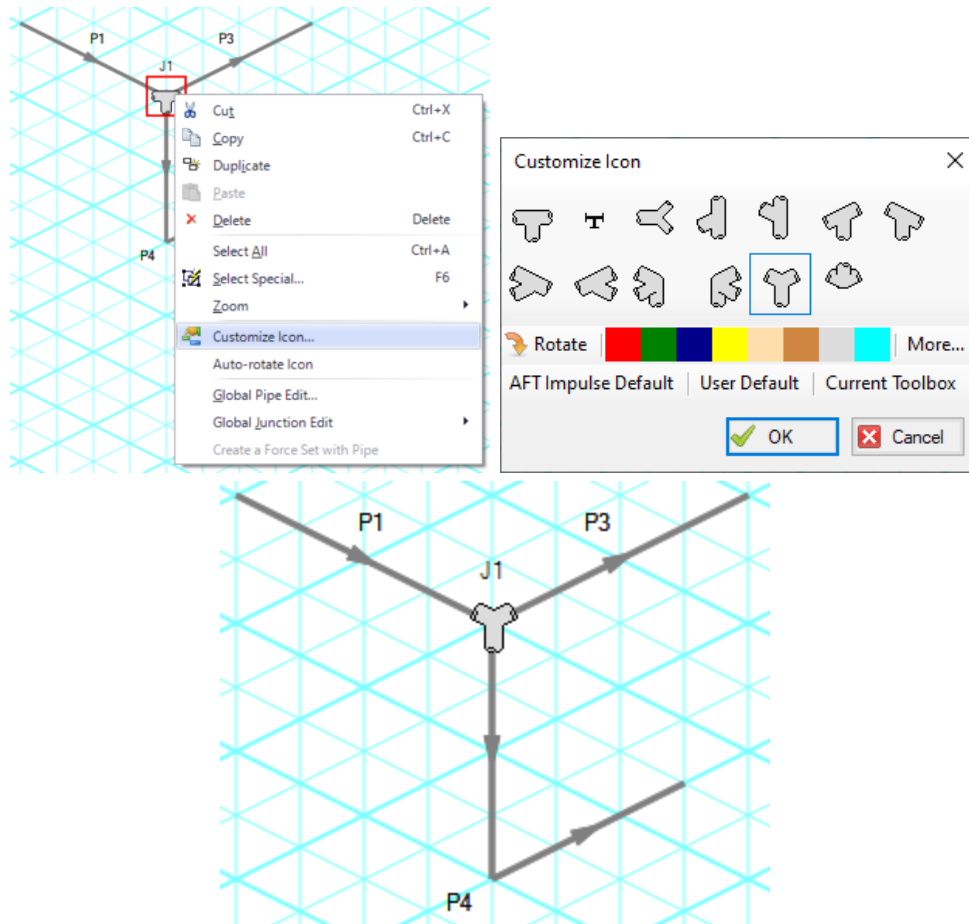


Figure 3: Icon rotations in isometric space

## Related Examples

High Pressure Steam - Fluid Property Models Example .....590

## Related Blogs

[Iso love the new Pipe Drawing Mode!](#)

## Global Edit

AFTxStream has the powerful capability to apply input parameters across a wide range of objects. This can be done for pipes by using the [Global Pipe Edit](#) or across junctions of the same type by using the [Global Junction Edit](#).

### Related Blogs

---

[Mass Modify your Model with Global Edit](#)

## Global Pipe Editing

The Global Pipe Edit window allows you to edit data for multiple pipes at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process.

Here is how you use it:

1. Open the Global Pipe Edit window from the Edit menu, or from the right-click menu on the Workspace.
2. Select the pipes you want to edit from the Pipe list on the left (if pipes were selected in the Workspace, they will be automatically selected here)
3. Click the Select Pipe Data button. This will open a template [Pipe Properties Window](#) in which you can enter the data you want to change on each of the selected pipes. Alternatively, if you want to copy the data from an existing pipe, you can choose "Existing Pipe", then select the desired pipe number and skip to step 5.
4. Enter the pipe data you want to change and click the OK button in the Pipe Properties window.
5. The Global Pipe Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
6. Click the Apply Selections button.

---

**Note:** You cannot change some data, like the pipe ID number, using this global method.

---

**Note:** There is no undo for global pipe editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

---

### Related Examples

---

Gas Turbine Fuel System Example .....575

### Related Blogs

---

[Mass Modify your Model with Global Edit](#)

### Global Junction Editing

The Global Junction Edit window allows you to edit data for multiple junctions at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process, or the instructions are provided below in this topic.

Junction data falls into the following two broad categories:

- **Common Data**- data that applies to all junctions regardless of junction type; examples include elevation and initial pressure data.
- **Junction Specific Data**- data that applies only to certain junctions; examples include compressor curves and control valve setpoints.

To open the Global Junction Edit window, browse to the Global Junction Edit option on the Edit menu. From there, choose the category of junction you want to edit. The available categories include those junctions selected on the Workspace, any junction type used in the model, and common junction data. Global Junction Edit can also be launched by right-clicking on the Workspace.

---

**Note:** There is no undo for global junction editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

---

### Editing Common Junction Data

---

You can open the Global Junction Edit window for common data editing in two ways. First, you can choose Common Data from the Global Junction Edit menu selection on the Edit menu, or you can choose one of the other specific junction types on the menu and then choose the Common option for the Junction Template in the upper left.

Here is how to globally edit junction common data:

1. Open the Global Junction Edit window.
2. Select the Common option for Junction Template in the upper left.
3. Select the junctions you want to edit from the Junction list on the left. Note that all junctions in the model are shown, no matter what the junction type.
4. Click the Select Common Junction Data button. This will open the Common Junction Data Edit window in which you can enter the data you want to change on each of the selected junctions.
5. Enter the junction data you want to change and click the OK button in the Junction Properties Window.
6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
7. Click the Apply Selections button.

---

**Note:** You cannot change some data, like the junction ID number, using this global method.

---

### Editing Specific Junction Data

---

You can open the Global Junction Edit window for specific data editing in two ways. First, you can choose the junction type from the Global Junction Edit menu selection on the Edit menu, or you can choose Common Data on the menu and then choose the desired junction template, or Existing Junction data for the desired junction type.

Here is how to globally edit specific types of junctions:

1. Open the Global Junction Edit window.
2. Select the Junction Template option in the upper left. Alternatively, if you want to copy the data from an existing junction, you can choose "Existing Junction", then select the desired junction number and skip to step 6.
3. Select the type of junction you want to edit in the dropdown list in the upper left. For example, you could choose an Assigned Pressure junction.
4. Click the Select Assigned Pressure Data button (the button name will change based on the type of junction selected). This will open the appropriate junction Properties Window, in this case the Assigned Pressure Properties Window.
5. Enter the junction data you want to change and click the OK button in the Assigned Pressure Properties Window (or whatever window type is displayed).
6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
7. Select the junctions you want to edit from the Junction list on the left. Note that only junctions of type selected in the upper left are shown (Assigned Pressure junctions in this case).
8. Click the Apply Selections button.

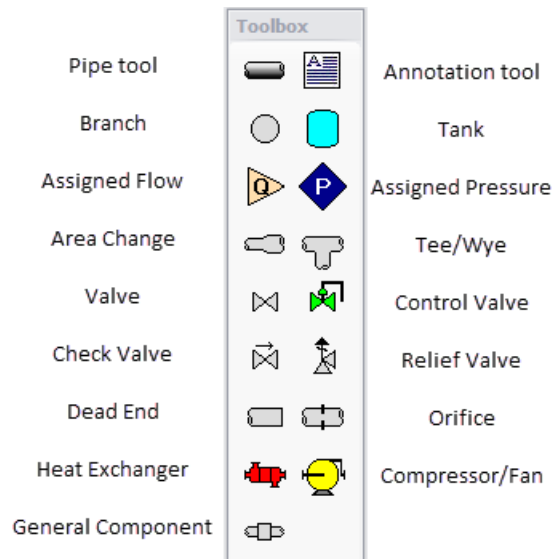
### Related Blogs

---

[Mass Modify your Model with Global Edit](#)

#### Toolbox

The bundle of tools at the left of the [Workspace](#) window that allows you to create pipes and junctions. The toolbox can be docked to the left, right, or floated within the workspace by clicking on the arrows on the top of the toolbox.



**Figure 1: Toolbox items**

## Creating Pipes - the Pipe Drawing Tool

The tool in the upper left corner of the [Toolbox](#) is the Pipe Drawing tool. With this tool you can draw new pipes on the Workspace, starting and ending at any desired location.

To draw a pipe, click the Pipe Drawing tool. The mouse pointer will change to a crosshair when you move it over the Workspace. Position the cursor at the pipe's starting point, hold down the left mouse button while dragging the cursor to the pipe end point, then release the mouse button.

To draw multiple pipes, double click the Pipe Drawing tool. This will keep the tool engaged until you click the Pipe Drawing tool again or press the escape key.

The length of the line on the Workspace has no relationship to the physical length of the pipe. The Workspace pipe is merely an abstract representation of a pipe, however, each pipe drawn must have a visible length on the Workspace. If the pipe drawn does not have a visible length, AFT xStream eliminates the line immediately after you release the mouse.

When a single pipe is selected, two black squares (or more if the pipe is segmented), called handles, appear. When the cursor is placed over a pipe handle it changes to a crosshair and the pipe handle can be moved or stretched by dragging with the left mouse button.

Another way to create new pipes is to duplicate existing pipes with the Duplicate or the Copy and Paste features found on the [Edit menu](#). When a new pipe is derived from a previous pipe, the new pipe retains the input associated with the original pipe.

Each pipe must be connected to two junctions. There are no exceptions to this rule.

## Pipe ID Numbers

---

When a new pipe is created, it is assigned a default ID number. When the pipe is displayed on the Workspace, the ID number is preceded by the letter "P" to indicate that it is for a pipe. From the [Pipe Properties Window](#), you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the pipe and cannot be replicated in any other pipes on the Workspace. The pipe number remains with the pipe in descendant scenarios in the [Scenario Manager](#). An arrow near the ID number indicates the [reference flow direction](#) for the pipe. The reference positive flow direction can be reversed by choosing [Reverse Direction](#) from the Edit menu or by selecting the reverse direction button on the [Toolbar](#).


**Tip:** The pipe or junction ID number text can be dragged to a new location to improve visibility. The new location is saved with the model. If you want the text of one or more pipes and/or junctions returned to the default location, select those pipes/junctions on the Workspace and choose Reset Label Locations from the Arrange menu. From this box, click the Reset Labels For Currently Selected Pipes and Junctions. If you want all pipe and junction text returned to the default locations, choose Reset Label Locations from the Reset Label Locations box on the [Arrange menu](#).

---

## Pipe Segments

---

A pipe can be segmented or "bent" into numerous sections. This only affects the visual representation and does not affect the calculated results. It does not add an "elbow" to the model or the resulting pressure drop effects of an elbow. This feature is useful in graphically bending the pipe around another part of the model or graphically showing that there are embedded elbows as [fittings & losses](#).

You can add a segment to a pipe in several ways. The easiest way to add a pipe segment is to first select the pipe, then click on the Add/Remove Segment icon  from the Toolbar. A segment will automatically be added. One or more segments can be added by clicking on the down-facing arrow on the Add/Remove Segment icon from the Toolbar, and then choosing the number of segments to be added. Alternatively, you can first select the pipe, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to choose Add a Segment or Add Multiple Segments.

The easiest way to remove a segment is to first select the pipe, then click on the down-facing arrow on the Add/Remove Segment icon from the Toolbar. A drop down list will appear that will allow you to choose Remove Segment. If you wish to remove all segments, click Remove All Segments. Alternatively, you can first select the pipe from which you want to delete one or more segments, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to remove one or all segments by clicking on either Remove Segment if you want to remove one segment, or by clicking on Remove All Segments if you want to delete all segments.

## Keyboard Shortcuts

---

See the [Keyboard Shortcuts for Workspace](#) topic for more information on additional Pipe Drawing Tool shortcuts.



---

## Related Topics

---

Keyboard Shortcuts for Workspace ..... 293

### Joining or Splitting Pipes

When pipes are present in the [Workspace](#), users can both combine and split up pipes. This can be done either to clean up the model visually or to further manipulate the model.

---

### Joining Pipes

---

When multiple pipes are placed in the workspace separated by junctions without having other [flowpaths](#) splitting off, those pipes and intermediate junctions can be combined into a single pipe. The new pipe will have a total length equal to the sum of the combined pipe lengths, and any losses associated with the intermediate junctions will be summed and entered as additional [Fittings & Losses](#).

The pipes being combined must have the same material, size, type, and friction model to be combined. In addition, the pipes must be pointing in the same direction.

Joining pipes may cause some junction data to be lost such as special conditions, loss factors for reverse flow, and ignored transients.

To join pipes and junctions in the model, select the "Join Pipes/Junctions" option from the Edit menu. The Join Pipes window will be shown with more information on pipes that can be combined in the model. The behavior of the Join Pipes/Junctions window will vary depending on if junctions are selected on the Workspace when the Join Pipes/Junctions option is selected.

---

**Note:** For maintenance releases prior to August 2022 the Join Pipes/Junctions window did not exist. Instead pipes and junctions selected in the Workspace would be joined automatically when the "Join Selected Pipes/Junctions" option was selected in the Edit menu.

---

- If junctions are selected on the Workspace, the Join Pipes/Junctions window will open with only the selected junctions shown in the list.
- If no junctions are selected on the Workspace, the Join Pipes/Junctions window will open with all junctions that can be combined shown in the list. Any junctions that can be combined and won't cause a loss in data at that junction will be selected automatically.

Once the Join Pipe/Junctions window is opened the selections can be modified before joining the pipes. The options below the table can be enabled/disabled in order to filter the shown junctions based on if they are selected on the workspace, if they will loose data when combined, or if they can't be combined.

If any junctions are selected that would loose data when combined, an additional button will appear near the bottom of the window that allows junctions that would have data loss to be deselected. It is recommended to select this button before joining the pipes/junctions unless you are certain the lost data is no longer required.

To join the pipes at the junctions selected in the Join Pipes/Junctions window, click the Join Selected Junction(s) button at the bottom of the window.

---

## Splitting Pipes

---

An existing pipe in the workspace can be split into multiple pipes by [holding the Shift](#) key while placing a junction on the pipe. This will split the pipe in two and insert the selected junction at the specified length. Any fittings and losses defined in the pipe can be placed in either of the split pipe segments.

## Related Topics

---

Keyboard Shortcuts for Workspace..... 293

### Creating Junctions

Below the pipe drawing and the annotation tools on the [Toolbox](#) are junctions, which allow you to model a large variety of pipe system components. To add a junction to a model, drag the desired icon from the Toolbox and drop it anywhere on the [Workspace](#).

As you move the mouse pointer over the Toolbox, a tooltip identifies the type of junction under the mouse pointer.

When you duplicate a junction using the Duplicate, Copy, or Paste features on the [Edit menu](#), the new junction retains the input associated with the original junction.

## Junction ID Numbers

---

Junctions are shown on the Workspace with default ID numbers, which you can change in the [Junction Properties Window](#). The letter "J" precedes the ID number to signify that it is for a junction. From the junction's Properties window, you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the junction and cannot be replicated in any other junctions on the Workspace. The junction number remains with the junction in descendant scenarios as long as the junction type is not changed.

Identical ID numbers can be assigned to both a pipe and a junction because the "P" or "J" will distinguish the two.

Unlike pipes, junctions do not have reference positive flow directions. Junctions for which the flow direction is important (such as control valves) derive the flow direction from the flow direction of the connecting pipes.

## Related Topics

---

Keyboard Shortcuts for Workspace..... 293

### Connecting Objects

During construction of a new model, objects can be placed anywhere on the [Workspace](#). In order to assemble a model that is ready for analysis, you must connect the objects properly.

Remember that connectivity only exists between [junctions and pipes](#). There are no junctions that connect to junctions, and no pipes that connect to pipes.

AFT xStream will attempt to automatically connect pipes and junctions when they are moved. This eliminates unnecessary opening of the pipe and junction Properties windows. If xStream cannot make the connection automatically, then the following three steps may be required to establish a connection between a junction and a pipe:

1. Graphically connect the objects on the Workspace. The pipe endpoint must terminate within the boundaries of a junction icon.
2. After the pipe is graphically connected to two junctions, double-click the pipe to open its [Properties window](#), then click OK to accept the connected junctions as determined by AFT xStream. It is most efficient to do this when you are entering the pipe's data in its Properties window.
3. Open the Properties window of the corresponding junction and accept the pipe connectivity determination. In most junctions, this pipe connectivity determination will be located in the upper right hand corner of the Properties Window. In a few junctions, such as the [Assigned Pressure Junction](#) and [Tank Junction](#), the pipe connectivity will be shown under the tab containing Loss Coefficients. You can do this as you enter the junction's data in its Properties Window.

If the model seems to not be connected properly, you can also use the [Extended Model Check](#) feature on the View menu.

When you accept AFT xStream's connectivity interpretation, the connection advances from a graphical connection to a model connection.

The model connectivity you establish on the Workspace is retained only as long as you maintain the graphical objects in their current visual relationship to each other. If you move a pipe or a junction and break the graphical connection, AFT xStream disconnects the two objects. However, if you move a group of connected objects, their model connectivity is maintained.

## Moving Objects


Objects on the [Workspace](#) can be moved individually or as groups. Clicking a Workspace object selects it. Pipe objects change color when selected. Junction objects are outlined in red when selected. These colors are configurable in the Workspace section of [User Options](#) (Tools menu).

To move an object, select it, drag it within the Workspace, and drop it in the desired location. When an object is dragged beyond the existing Workspace area, the Workspace is expanded accordingly.

A pipe object can be stretched by grabbing the handles (small red squares) at the pipe endpoints and moving an endpoint to a new location. To make these handles visible, select the pipe you want to move.

There are numerous ways to select a group of objects for movement or other operations.

To prevent accidental movement on the Workspace, lock the objects. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction. The Lock feature is

accessed from the Arrange Menu or the lock button  on the [Toolbar](#). To lock an object on the Workspace, select it and then activate the Lock feature either from the Arrange menu or from the lock button on the Toolbar. If you would like to lock multiple objects at the same time, ensure that all of the objects are selected prior to activating the Lock feature.

---

## Keyboard Modifiers

---

If you hold down the CTRL key when dragging a junction, the endpoints of any connected pipes are not moved with the junction, thus breaking the graphical connection.


## Related Topics

---

Keyboard Shortcuts for Workspace.....	293
Selecting and Finding Objects.....	140
Selecting Objects.....	140
Editing Objects.....	160
Defining Objects.....	161

## Editing Objects

The [pipes and junctions](#) you place on the [Workspace](#) can be edited with the editing commands from the Edit menu or the [Toolbar](#). Objects can be cut, copied, pasted, duplicated, and deleted. These operations can be performed on individual objects or on groups. The Edit menu provides the option to undo each editing operation in the Workspace, but will not undo any changes made in the pipe/junction properties

windows. Alternately you can use the Undo button  on the Toolbar.

The Copy Graphics feature on the [Edit menu](#) will copy the specified Workspace objects to the Windows clipboard. This image can then be pasted into other Windows applications.

## Related Topics

---

Keyboard Shortcuts for Workspace.....	293
Selecting and Finding Objects.....	140
Selecting Objects.....	140
Moving Objects.....	159
Defining Objects.....	161

## Morphing Junctions

Occasionally you may want to change a junction's type in your model. You can do this by morphing the old junction into a new one.

If you hold down CTRL key and select a junction from the Toolbox and drop it onto an existing Workspace junction, the old junction will morph into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.

## Related Topics

---

Keyboard Shortcuts for Workspace.....	293
---------------------------------------	-----

## Defining Objects

To specify the required property data for an object, open its Properties window. This can be done in six ways:

1. Double-click the object on the Workspace
2. Select the object (by clicking on it once, or using the [Selection Box](#)) and press ENTER
3. Select the object and click the Open Pipe/Jct Window icon on the [Toolbar](#)
4. Double-click the left column of the appropriate table in the [Model Data window](#) (where the object ID number is shown)
5. [Jump](#) from another Properties window
6. Double-click the connected pipe or junction number in a Properties window

Parameters that functionally describe the pipe or junction are entered in the Properties window. Some parameters in the Properties windows are required and others are optional. The optional parameters offer you added control over your model and the ability to obtain specialized information.

The [highlight feature](#) shows the required input in pipes and junctions.

## Undefined Objects Window

---

You can open the [Undefined Objects window](#) to view all undefined objects and the undefined properties. This is available on the [View](#) menu.

## Use Status Feature

---

Each pipe and junction Properties window has a Status tab. The Status tab shows what required information for the pipe or junction is missing.

## Satisfying Connectivity Requirements

---

In addition to entering the required information for each object in the model, you must satisfy these connectivity requirements:

- You must accept the connectivity information shown in each object's Properties window
- Each junction must have the appropriate number of pipes connected to it. For example, an area change junction must have exactly two connecting pipes, whereas a tank may have one to twenty-five connecting pipes.
- Pipes must always be connected to two junctions, one at each pipe endpoint.


Should you ever delete any of the required information in the Properties window or move a connected object from its connection, the object's status reverts to undefined.

For example, say you place a [check valve](#) on the Workspace. A check valve requires inlet and outlet piping. AFT xStream reflects this by requiring each check valve to be assigned an upstream pipe and a

downstream pipe. Entering only the Valve model details and setting an elevation is necessary but not sufficient information to define the check valve according to AFT xStream's requirements. Once you have entered the check valve data and properly connected it, the check valve is defined.

## Show Object Status

---

The Show Object Status feature helps you identify which objects on the Workspace are defined and which are not. This feature can be toggled on and off by selecting it from the [View menu](#) or by clicking the floodlight button  on the [Toolbar](#).

Turning Show Object Status on displays the undefined junction and pipe ID numbers in the Undefined Color, which is red by default. The color can be changed through the [User Options window](#) on the [Tools menu](#). Show Object Status helps you rapidly assess where your model requires additional information.

Leaving Show Object Status always toggled on is not recommended for large models, as it will slow the Workspace graphics response. When this feature is on, every Workspace graphical operation you perform must be accompanied by a check of all pipe and junction connections. This check takes time, and could degrade your graphics performance.

## Highlight Required Input

The highlight feature functions to identify required input data for the Properties Window for each pipe and junction. You can toggle the highlight feature on and off in the following ways:

- From the Properties Window
  - Double-click anywhere in the Properties Window
  - Press the F2 function key
- From the View Menu
  - Toggle the Highlight in Pipe and Jct Windows to Off

The highlighting feature may be especially useful when you are first learning to use AFT xStream or when you are having difficulty obtaining a defined object status.

## Inspecting Objects

The Inspection feature is a time saving feature for reviewing the data associated with a Workspace object. Inspection displays the data for an object in read-only format (see Figure 1).

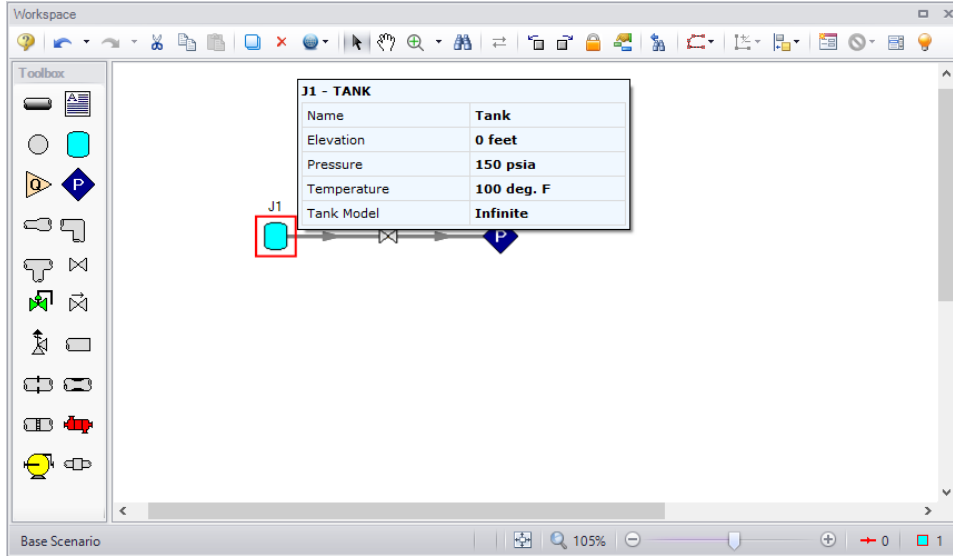
To inspect an object, position the mouse pointer on the object and hold down the right mouse button. Inspecting is quicker than opening the Properties Window if you want to look at an object's input data but not edit it.

The Inspection feature is also available in each Properties Window. This is useful if you want to check the input data of objects connected to the one you are working on.

For example, when working in a [Valve Properties](#) window, you may wish to quickly check the upstream and downstream pipe diameters previously defined. Holding down the right mouse button on each connected pipe ID number will show you that pipe's primary input parameters.

The Inspection feature works similarly in [Pipe Properties](#) windows when you inspect connected junctions. Judicious inspecting can save you time when assembling or debugging a model.

If output data exists it will also be displayed.



**Figure 1: Inspecting from the Workspace with right mouse button**

## Related Blogs

[AFT Responds to 2020 Customer Survey Feedback](#)

## Notes

Each object can have notes associated with it. This is useful for listing assumptions, reference documents, drawings, measurements, etc.

## Related Topics

Pipe Properties Window.....	166
Junction Window Overview.....	192

## Status

The status tab lists all undefined parameters that are required to define the pipe or junction.


## Related Topics

---

Pipe Properties Window.....	166
Junction Window Overview.....	192

## Workspace Pan

The user can use Workspace Pan mode to move around, or pan, the model. If the Workspace has been enlarged beyond the default one screen size (Workspace scrollbars will be visible), then the [Workspace](#) can be panned by activating Workspace Pan mode. There are several ways to activate Workspace Pan Mode:

- Click the Workspace Pan button  on the toolbar
- Select the Workspace Pan option in the View menu
- If your mouse has a center button (a scroll wheel), hold the scroll wheel button down
- Hold CTRL and use the arrow keys

Once in pan mode, the cursor will change to an open hand that can be used to grab and move a point in the Workspace. The Workspace can be panned by holding down the left mouse button (if the Workspace Pan mode has been activated using the Workspace Pan button on the toolbar or the Workspace Pan option has been selected in the View menu) and moving the mouse in the desired direction, or by holding down the mouse scroll wheel (if using the scroll wheel to keep Workspace Pan mode activated) and moving the mouse in the desired direction.

To switch off the pan mode, click the mouse scroll wheel or the Workspace Pan button on the toolbar a second time. If Workspace Pan mode is maintained using the mouse scroll wheel, it will automatically turn off once you release the mouse scroll bar.

## Related Topics

---

Keyboard Shortcuts for Workspace.....	293
---------------------------------------	-----

## Related Blogs

---

[AFT Responds to 2020 Customer Survey Feedback](#)

## Moving Pipe and Junction Labels

Each pipe and junction on the [Workspace](#) also displays text (or label) referring to the number and/or name of the object. This label can be dragged to improve model aesthetics. AFT xStream keeps track of the relative location of the label to the object so that if the object is moved, the label moves with it.

## Returning Label to Default Location

---

To return the label to its default location, you can:




- Right-click the label for an individual object and choose Reset Label Location
- Use Arrange -> Reset Label Locations and respond to the dialog

## Special Conditions

Some junctions have Special Conditions which alter the normal state of the junction. For example, a [valve](#) can be closed, a compressor turned off, or a relief valve opened.

The special conditions are set in one of three ways:

- By selecting the junction(s) on the [Workspace](#) and choosing Special Conditions from the [Edit Menu](#)
- By selecting the junction(s) on the Workspace and clicking the Special Condition icon  on the [Toolbar](#)
- By opening the junction's Properties window and clicking the appropriate condition on the Optional tab

The junctions which have special conditions set are shown using a special symbol before the ID number (an X by default). This symbol can be customized in the Tools/[User Options](#) window.

When the special condition results in a section of the model being closed or "turned off", the pipes which will have zero flow are display as dotted lines. This indicates visually that they are cut off from flow; the junctions are outlined with a dotted line. The default dotted line can be changed in the Tools/User Options window.

The user can specify Special Conditions for [Control Valves](#) to be either closed or fully open. Similarly, compressors/fans can be turned off and either allow flow through or stop the flow.

### Special Condition: Ignore

---

The [Relief Valve](#) junction has a unique kind of Special Condition called Ignore. This makes the steady-state and transient solvers ignore the presence of the junction. Thus not only is the data ignored, the junction itself is ignored. This is convenient when locating a relief valve by trial and error. Rather than having to delete the junction, it can be ignored.

As an example, assume that one is trying to find the best location for a relief valve. If one does this with a single relief valve junction, it will be required to run a case, view the results, then relocate the junction by changing the pipe lengths and rerunning. If one uses the Special Condition Ignore, one can place multiple relief valve junctions in the model and then set all of them except one to Ignore so that only one of the junctions is used at a time.

## Pipes

This section focuses on specifying and modifying the Pipe object in the Workspace. Topics include an overview of the Pipe Properties window, how to input pipe specification data, a description of the methodology used for heat exchange within pipes, and how to insert fittings and losses within the pipes.

### Contents

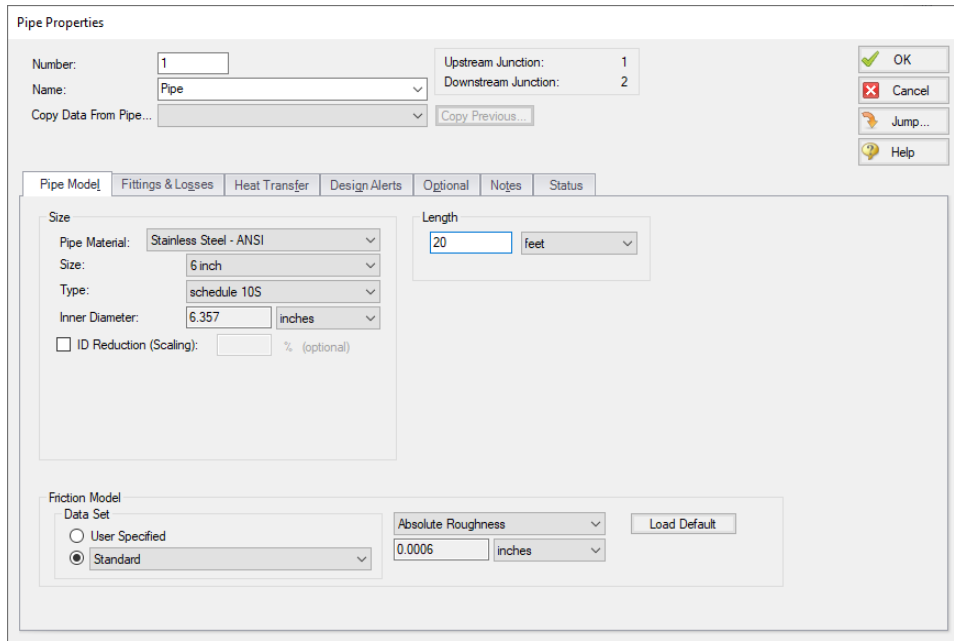
---

<a href="#">Pipe Properties Window</a> .....	166
<a href="#">Common Pipe Input Parameters</a> .....	167
<a href="#">Pipe Model</a> .....	168
<a href="#">Fittings &amp; Losses</a> .....	169
<a href="#">Heat Transfer in Pipes Overview</a> .....	171
<a href="#">Insulation for Pipes</a> .....	188
<a href="#">Optional Input for Pipes</a> .....	188
<a href="#">Add/Remove Pipe Segment</a> .....	189
<a href="#">Reverse Direction</a> .....	190
<a href="#">Pressure Drop in Pipes - Detailed Discussion</a> .....	190

## Pipe Properties Window

Each pipe on the [Workspace](#) has engineering information associated with it. Each pipe must have an ID number, a length, a diameter, a roughness specification, and two connecting junctions. If the pipe is not purely cylindrical, additional geometry information is also required.

The Pipe Properties Window allows the user to define all data for a pipe on the Workspace. An example on a fully defined Pipe Properties Window is shown below in Figure 1. The various Pipe inputs are discussed in the respective pipe pages listed in the Related Topics section below.



**Figure 1: Fully Defined Pipe Properties Window**

## Related Topics

Notes.....	163
Status.....	163

## Common Pipe Input Parameters

The common input parameters are located above the tabs in the [Pipe Properties Window](#).

## Number

Every pipe must have a unique ID number. When new pipes are created, a pipe number is automatically allocated.

The pipe number can be changed by the user to any unique value greater than zero and less than 100,000. Pipe numbers are only identifiers and have no affect on the model behavior.

## Name

A name can be assigned to each pipe for reference purposes. The default name is simply “Pipe.” Names do not need to be unique. The name can be shown on the Workspace, in the Visual Report and in the Output window.

### Copy Data From Pipe...

---

The Copy Data From Pipe drop-down list allows you to copy all or some of an existing pipe's properties to the current pipe. Selecting an existing pipe will prompt the user on what parameters should be changed. The Copy Previous button functions similarly, copying parameters from the previously edited pipe.

In a child scenario, Copy Data From Pipe lists "Parent Pipe Data," allowing the user to copy all or some parameters from the parent pipe.

### Connected Junctions

---

All pipe objects have a [reference positive direction](#), as indicated on the Workspace by an arrow in the center of the pipe. The upstream and downstream junctions are identified with this direction. The connected junctions area displays these junctions. Every pipe must be [connected to two junctions](#) in order to be completely defined.

## Pipe Model

### Size

---

The size section of the Pipe Model tab specifies the pipe geometry.

- **Pipe Material**
  - **User Specified** - If selected, the geometry of the pipe must be directly specified.
  - **Library Material** - Many common pipe materials are included in the standard [Pipe Material library](#). These and any other accessible library materials can be selected. When a library material is selected, the parameters defining the geometry cannot be modified.
- **Size** - Select from a list of defined nominal sizes for the selected Pipe Material. Only available for library materials.
- **Type** - Select from a list of defined types (classes/schedules) for the selected Pipe Material. Only available for library materials.
- **ID Reduction (Scaling)** - Reduction in hydraulic diameter due to scaling can be accounted for by entering a percent reduction in this field. Zero represents no reduction.

### Length

---

The Length is defined as the distance between the two connecting junctions. This length has no relationship to the visual length of the pipe drawn on the [Workspace](#).

**Note:** In contrast with other AFT products, xStream does not contain the Zero-Length Connector (ZLC) option at this time.

---

### Friction Model

---

Multiple [friction data sets](#) can be compiled for each library pipe material. These different data sets are then made available in the Data Set drop down list. For AFT standard materials, the friction data set is generally "Standard."

The Friction Model can be modified for any individual pipe, and the default friction model can be set in [Parameter Options](#). The Friction Model for a library material can be modified by selecting the User Specified radio button in Data Set, and must be directly specified for User Specified materials.

- **Absolute roughness (default)** - Absolute average roughness height. Values of pipe roughness can be found in many pipe handbooks or from manufacturer's data.
- **Relative roughness** - Relative roughness is the absolute roughness divided by the pipe diameter.
- **Hydraulically smooth** - Hydraulically smooth implies that roughness is negligible. This is not the same as frictionless.
- **Explicit Friction Factor** - Friction factor for the pipe, you can enter the value explicitly.
- **Frictionless** - For troubleshooting purposes, it is occasionally useful to model a pipe as having no friction. There are limitations to where such a pipe can be located in your model.

For detailed information on how these Friction Models are defined, see [Pressure Drop in Pipes - Detailed Discussion](#).

## Fittings & Losses

Fittings & Losses allows the user to define additional losses directly in a pipe. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings.

To specify library losses, click Specify Fittings & Losses to open the Pipe Fittings & Losses window. An additional K Factor loss can also be user specified.

### Pipe Fittings & Losses

---

The Fittings & Losses library contains many standard loss values. Because many of these values are dependent on pipe diameter, a pipe diameter must be specified to open the Pipe Fittings & Losses window.

Losses are divided into several categories represented by tabs. To add fittings to the pipe, select the type, and adjust the quantity column next to the desired fitting.

After accepting the changes by clicking OK, a summary of the pipe's Fittings & Losses will be displayed in the pipe's properties window.

**Note:** Area Change fittings do not change the size of the pipe, they only represent the loss factor based on the pipe diameter and the area ratio of the loss. When including area changes, notice the illustration of how the loss is defined, where  $A_{pipe}$  is the area of the pipe and  $A_s$  is the area of the loss. These area changes are useful to model fittings between a pipe and another junction like a compressor.

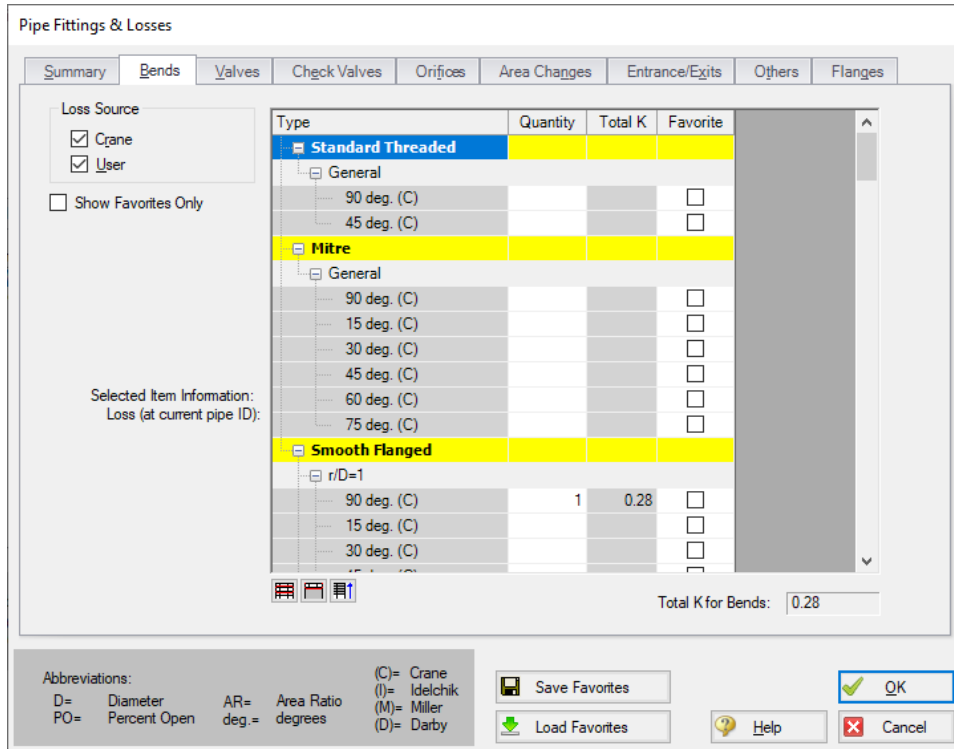


Figure 1: Pipe Fittings & Losses

## Fittings & Losses Workspace Symbol

When a pipe has any loss value assigned through the Fittings & Losses tab, a symbol ([default of &](#)) is added before the pipe number.

## Favorite Fittings & Losses

In the Pipe Fittings & Losses window, there is a column labeled Favorite for each of the loss types. Check the box for favorite to add it to the list of favorite fittings and losses. On the left side of the window there are several options to filter the list. For example, one may only want to see [Crane](#) fittings. In which case, Crane should be checked, and all others unchecked on the left hand side of the window. If a list of only favorite fittings is desired, check the box to "Show Favorites Only".

---

## Related Topics

---

Fittings & Losses Library.....	398
Specifying Losses.....	198
References.....	461

## Heat Transfer in Pipes Overview

Each pipe responds thermally to the environment outside of the pipe. AFT xStream offers several models that allow you to model the heat transfer characteristics of a pipe:

- **Adiabatic at Internal Surface** - An adiabatic pipe is one that has perfect insulation and thus no heat transfer takes place. The temperature into the pipe is taken from the upstream junction. [Insulation data](#) can be entered but is not used for thermal purposes. This model assumes the fluid in the pipe does not thermally interact with the pipe wall.
- **Convective Heat Transfer** - This model allows you to specify heat transfer data for the internal fluid, pipe wall, three layers of insulation and external convection.
- **Convective Heat Transfer and Heat Flux** - This model allows you to model convective heat transfer while also specifying a constant heat flux.
- **Convective Heat Transfer with Heat Tracing** - This represents adding some type of heating layer onto the pipe's outer surface. The amount of heating may be entered as a constant heat flux or by providing information on the power and number of wire turns. Insulation layers can then be added on top of the heat tracing (heat flux source).

Note that similar to flow rate definition, heat flow is positive when flowing into the pipe, and negative when flowing out.

## Viewing Calculated Thermal Results in Output

---

A table is displayed in the Output window Pipe area showing the temperature distribution in the pipe and insulation for the steady-state system.

# Pipes

Pipe (AFT Arrow Steady)	Results Diagram	Name	T Fluid Inlet (deg. F)	T Fluid Outlet (deg. F)	T Inlet Wall Inside (deg. F)	T Outlet Wall Inside (deg. F)	T Inlet Outer Wall (deg. F)	T Outlet Wall Outside (deg. F)	T Ambient (deg. F)
1	Show		70.00	69.95	70.00	69.95	70.00	69.95	70.00
2	Show		139.28	139.25	139.19	139.15	139.16	139.12	70.00
3	Show		139.24	139.24	139.15	139.14	139.12	139.11	70.00
4	Show		139.56	139.55	139.36	139.35	139.34	139.32	70.00
5	Show		138.93	138.91	138.73	138.71	138.70	138.68	70.00
6	Show		139.39	139.38	139.27	139.26	139.24	139.24	70.00
7	Show		139.23	139.22	139.13	139.12	139.11	139.09	70.00
8	Show		138.57	138.56	138.47	138.46	138.44	138.43	70.00
9	Show		139.59	139.58	139.36	139.35	139.33	139.32	70.00
10	Show		139.22	139.21	139.12	139.11	139.09	139.08	70.00
11	Show		138.56	138.54	138.46	138.44	138.43	138.42	70.00

Figure 1: Thermal Data table in Output window Pipe area shows the steady-state temperature distribution in pipe

Clicking the button next to Show in the Results Diagram column brings up a Results Diagram for that particular pipe:

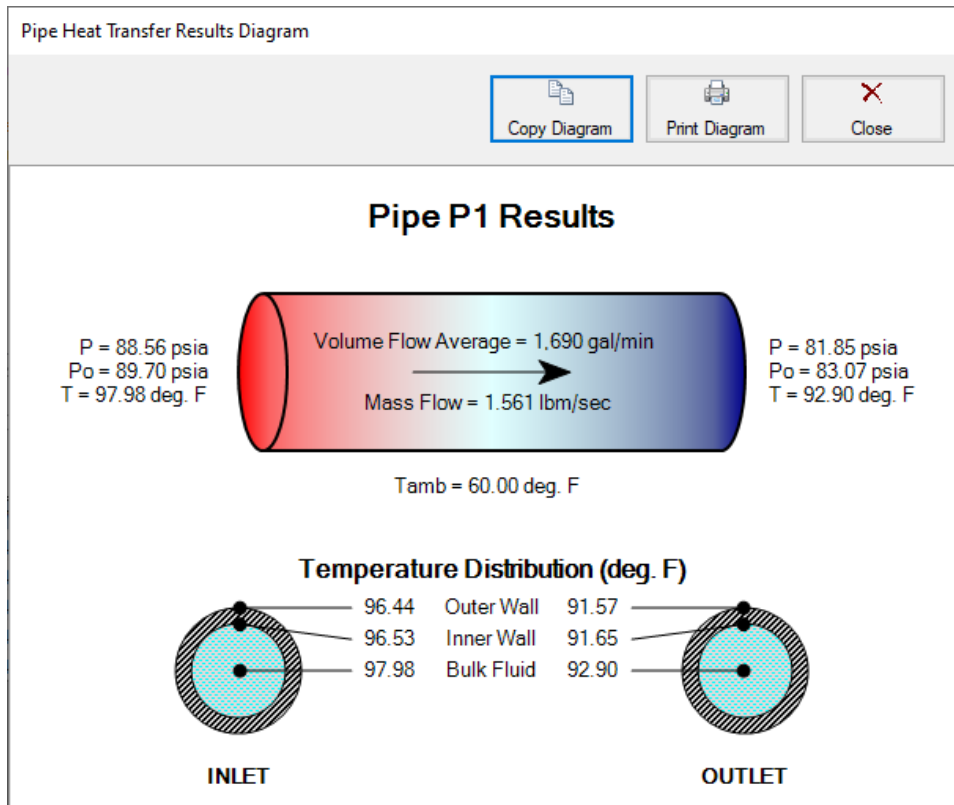


Figure 2: Pipe Heat Transfer Results Diagram



---

## Related Topics

---

References..... 461

## Convective Heat Transfer

There are 3 convective based models for pipe heat transfer:

- **Convective Heat Transfer** - Represents a pipe thermally affected solely by convection.
- **Convective Heat Transfer and Heat Flux** - Behaves the same as the Convective Heat Transfer model, but also includes the effect of an external heat flux such as radiation.
- **Convective Heat Transfer with Heat Tracing** - Similar to the Convective Heat Transfer and Heat Flux model, but instead of applying the flux on the outermost surface of the pipe/insulation, the flux can be applied directly to the pipe. As the name suggests, this represents a device such as a heated wire. If the flux is not known directly, it can be determined [based on Wire Turns and Length](#).

## Defining the Convective Model

---

When one of the convective heat transfer models is selected, there are a number of items that must be defined:

- **Resistance Geometry** - Radial or Linear
- **Number of Insulation Layers** - There can be up to three layers of insulation defined, in any combination of internal or external.
- **External Heat Flux** - Only available if Convective Heat Transfer and Heat Flux is selected. A constant flux applied to the outermost layer of resistance.
- **Heat Tracing on Pipe (Flux)** - Only available if Convective Heat Transfer with Heat Tracing is selected. A constant flux is applied at the outer wall of the pipe.
- **Ambient Conditions** - The temperature and velocity of the fluid around the pipe. The fluid can be either air or water. These values are used for calculating the convection coefficient if correlation is selected as the convection Data Source. This correlation can be used for pipes over which boundary layers develop freely, free of constraints imposed by other surfaces. All of the external fluid properties are evaluated at the film temperature using data from [NIST REFPROP](#).
- **Resistance Table** - The table at the bottom of the window defines all of the resistance layers used in the thermal model.
  - **Rows**
    - **Units** - This special row defines the units of any value in the column below a cell.
    - **Fluid Internal** - The fluid itself has a convection coefficient that affects the transfer of energy to the pipe wall. By default, the Gnielinski Correlation is used to determine this value. Alternatively, the Dittus-Boelter Correlation can be used, or a User Specified value can be entered.

- **Pipe Wall** - The wall of the pipe has a thermal conductivity that affects the heat transfer. The built-in pipe materials have thermal conductivity values defined, and this value is automatically selected. If a User Specified pipe is used, the conductivity will need to be entered.
- **Insulation** - There can be several insulation rows, depending on the selection of Number of Insulation Layers. The Insulation can be selected from the [Insulation library](#), in which case the conductivity is read from the library. User Specified insulation requires this value to be defined.
- **External** - Convection occurring external to the pipe and any insulation layers. By default the Convection Coefficient is calculated with the Churchill-Bernstein Correlation. The Correlation can also be set to Free-Horizontal or Free-Vertical which requires only an ambient temperature, and no fluid velocity. The Free Horizontal and Free Vertical orientations are in reference to the orientation of the pipes. If the convection coefficient is known, a value can be entered by the user.
- **Columns**
  - **Apply** - This can allow enabling/disabling of including certain layers in the calculation. This is useful for comparing states or scenarios without redefining the thermal model.
  - **Conductivity/Convection Data Source** - Specifies from where relevant information is attained. This can be a Library, Correlation, or User Specified.
  - **Conductivity** - Applies to the Pipe Wall and Insulation. If Data Source is a Library, the particular insulation must be selected. If User Specified, the conductivity must be entered. Note that obtaining the thermal conductivity from an insulation or pipe material library permits temperature dependent thermal conductivity to be employed. With this option, AFT xStream dynamically selects the insulation thermal conductivity during model solving based on the local temperatures that exist. Conversely, user specified values are always constant.
  - **Thickness** - Applies to Pipe Wall and Insulation. If a Library is used for the Pipe Material, this value is automatically populated. All insulation layers require a thickness.
  - **HT Area Ratio** - This represents any "extra" heat transfer area that may exist. For example, if the pipe has cooling fins this ratio would be larger than one.
  - **Convection Coefficient** - Applies to Fluid Internal and External. Both values can be either Correlation or User Specified.

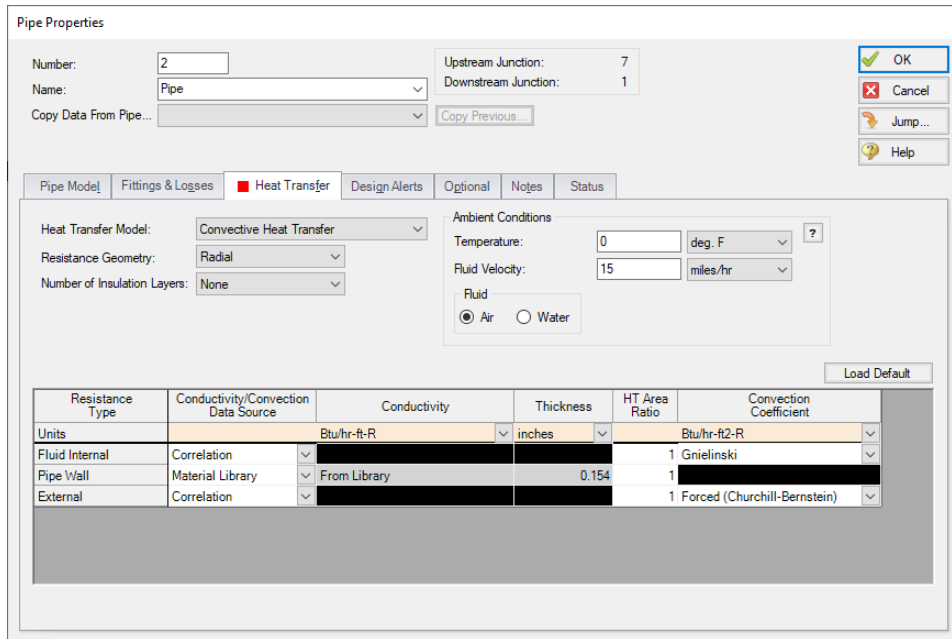


Figure 1: Convective Heat Transfer Layout

## Modeling a Constant Wall Temperature

Assign an ambient temperature and disable the External Resistance by unchecking the box in the Apply column. This will force the outside wall of the pipe to be equal to the ambient temperature. Note that this differs from isothermal in that the fluid temperature can still change - only the outside wall of the pipe has a fixed temperature.

## Related Blogs

[Let the Heat Flow: Modeling Heat Transfer in Pipes in AFT Fathom and AFT Arrow](#)

### Heat Tracing Flux Based On Wire Turns And Length.html

The Convective Heat Transfer with Heat Tracing model requires additional information to represent the thermal effect of a heat tracing.

xStream allows users the ability to model Heat Tracing by either entering the heat flux directly, or by providing information on the number of turns and power per unit length of wire, which xStream will use to determine the heat flux as described below.

# Pipes

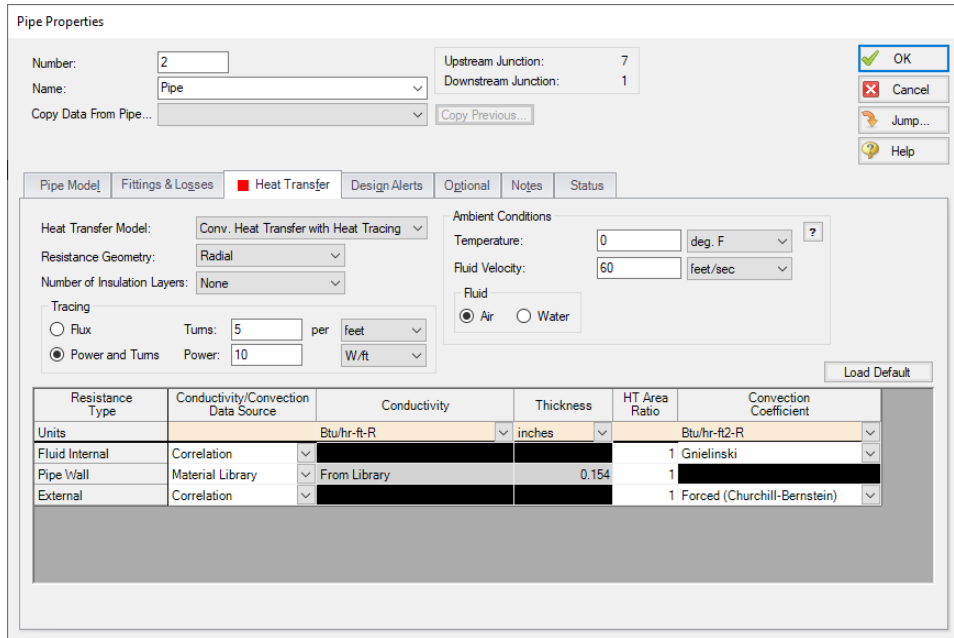


Figure 1: Pipe Properties window with the Conv. Heat Transfer with Heat Tracing model enabled

## Power and Turns - Detailed Discussion

When you select the option to model Heat Tracing based off a heated wire, xStream models that heat flux based on the theory described below.

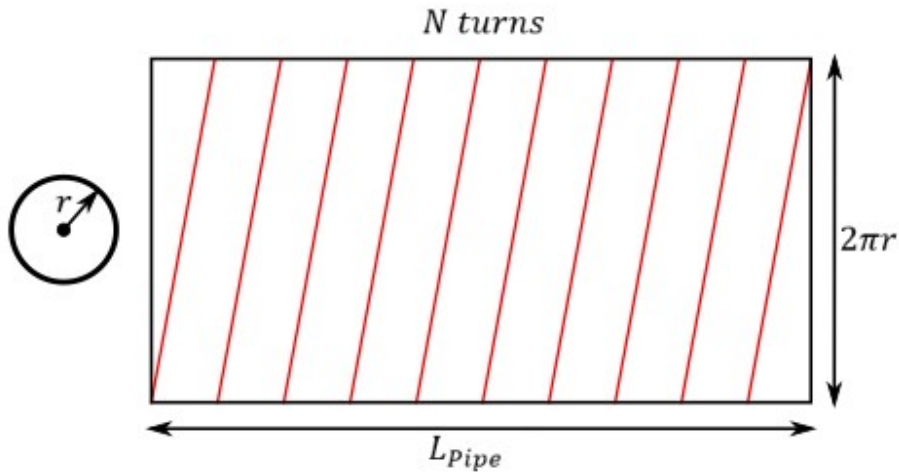


Figure 2: Wire tracing geometry

---

## Length of Wire

---

First, it may be necessary to determine the length of the wire itself, based on a number of turns.

- $N$  - total number of turns on the pipe
- $n$  - number of turns per unit length
- $L_{Pipe}$  - total length of the pipe
- $r$  - external radius of the pipe
- $L_{Wire}$  - length of the wire
- $L_{Turn}$  - length of one wire turn

The above figure shows a pipe with heat tracing that has been "split open" and laid into a flat rectangle to make these equations more intuitive.

The total turns and turns per length are related by the pipe length:

$$N = n L_{Pipe}$$

Each turn of the wire creates a right triangle on the "flattened" pipe, so we can find the length of each segment by the Pythagorean theorem.

$$L_{Turn}^2 = \left( \frac{L_{Pipe}}{N} \right)^2 + (2 \pi r)^2$$

The total length of the wire is related to the number of turns and the segment length.

$$L_{Wire} = N L_{Turn}$$

Incorporating this relationship and rearranging:

$$L_{Wire}^2 = L_{Pipe}^2 + (2 \pi r N)^2$$

Using the number of turns per unit length:

$$L_{Wire}^2 = L_{Pipe}^2 + (2 \pi r n L_{Pipe})^2 = L_{Pipe}^2 (1 + (2 \pi r n)^2)$$

or:

$$L_{Wire} = L_{Pipe} \sqrt{1 + (2 \pi r n)^2}$$

### Flux of Wire

With the length of the wire found, the flux based on power per length can be found.

- $P_{Total}$  - total power usage of the wire, along the entire pipe length
- $p$  - power per unit length
- $Q''$  - heat flux
- $A$  - area heat flux is applied to

The heat flux is defined as the heat delivered per area. Here, we assume that the "total power" used by the heat trace is equivalent to the total heat delivered. Therefore:

$$Q'' = \frac{Q}{A} = \frac{P_{Total}}{A}$$

The total power is equal to the power per length times the length, and area is related to pipe geometry:

$$Q'' = \frac{p L_{Wire}}{2 \pi r L_{Pipe}}$$

Substituting in for  $L_{Wire}$ :

$$Q'' = \frac{p L_{pipe} \sqrt{1 + (2 \pi r n)^2}}{2 \pi r L_{Pipe}}$$

Simplifying:

$$Q'' = p \sqrt{\frac{1}{(2 \pi r)^2} + n^2}$$

## Steady State Heat Transfer In Pipes

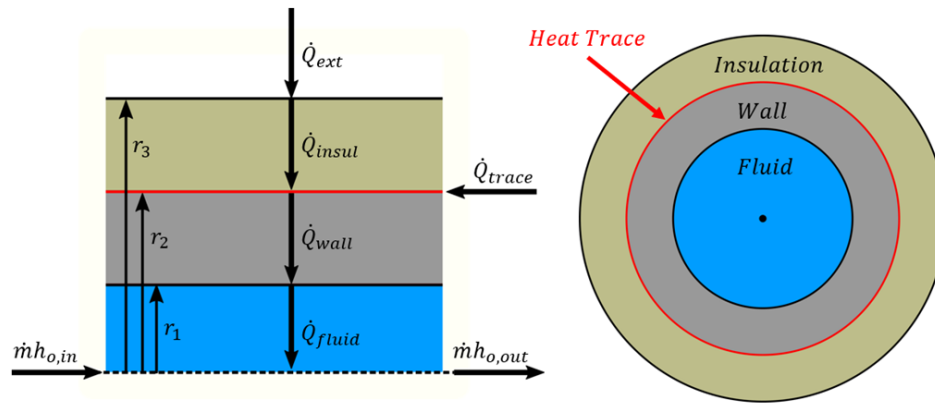


Figure 1: Diagram of a pipe with Heat Tracing and Insulation

There are many possible combinations of heat transfer that can be defined in AFT xStream. In any of these models there are various heat flows that can be considered.

Figure 1 shows the heat rates for one of the more complex cases – convective heating with a heat trace on the outer face of the pipe wall.

This topic will discuss heat transfer in terms of steady state heat transfer calculations. Balancing these heat rates and temperatures via iterative methods allows AFT xStream to come to a unique solution. The [Transient Heat Transfer](#) topic expands on how the convective heat transfer calculations are carried out during the transient.

### Heat Rates

As a hydraulic solver, the primary heat rate of interest is  $\dot{Q}_{fluid}$ , the heat added to the fluid itself. Adding heat to the fluid has the effect of raising its temperature.

We can derive an expression for  $\dot{Q}_{fluid}$  with the first law of thermodynamics. For convenience, we use specific quantities (per unit mass)

$$e_{out} - e_{in} = q - w$$

The energy  $e$  is composed of internal energy  $u$  and kinetic energy  $V^2/2$ . Note that the gravity term  $gz$  is neglected from this equation to simplify the derivation.

$$e = u + \frac{V^2}{2}$$

$$u_{out} + \frac{V_{out}^2}{2} - \left( u_{in} + \frac{V_{in}^2}{2} \right) = q - w$$

The work done by the fluid is comprised of pressure work  $pv$  and shaft work  $w_{shaft}$

$$w = (pv)_{out} - (pv)_{in} + w_{shaft}$$

Combining and rearranging

$$\left( u_{out} + (pv)_{out} + \frac{V_{out}^2}{2} \right) - \left( u_{in} + (pv)_{in} + \frac{V_{in}^2}{2} \right) = q - w_{shaft}$$

Using the definition of enthalpy

$$h = u + pv$$

$$h_{out} + \frac{V_{out}^2}{2} - h_{in} - \frac{V_{in}^2}{2} = q - w_{shaft}$$

In the steady state analysis, we consider that the velocity through the control volume remains constant. In reality, changing temperature will cause the density of the fluid to change, and thus the velocity. For this reason, pipes in AFT xStream are thermally sectioned. This means that certain properties are considered constant throughout the section as an approximation – having several sections in a pipe helps increase the accuracy of this approximation.

There is also no shaft work being done on the fluid.

$$\Delta h = q$$

We further assume that this can be treated thermodynamically as a constant pressure process. That is,  $c_p$  is constant in the thermal section, like density. With this assumption we can use the relationship

$$\Delta h = c_p \Delta T_{o,fluid}$$

To find that



$$q = c_p \Delta T_{o,fluid}$$

Rather than using specific quantities, it is useful to relate heat rate and mass flow. Multiply both sides by  $\dot{m}$

$$\dot{Q}_{fluid} = \dot{m} c_p \Delta T_{o,fluid}$$

The other heat rates in the system can be related to  $\dot{Q}_{fluid}$  and other user defined values like  $\dot{Q}_{flux}$  or  $\dot{Q}_{trace}$  by examining various control volumes and surfaces:

$$\begin{aligned} \dot{Q}_{fluid} &= \dot{Q}_{trace} + \dot{Q}_{ext} && \text{CV around wall \& insulation} \\ \dot{Q}_{wall} &= \dot{Q}_{fluid} && \text{CS at inside of wall} \\ \dot{Q}_{ext} &= \dot{Q}_{insul} && \text{CS at outside of insulation} \end{aligned}$$

We now have a system of equations that allow us to determine all the heat rates in the system given a mass flow (determined from a mass balance), a specific heat (defined versus temperature) and a change in temperature. However, we do not know what this change in temperature is. To find it, we need to relate the surface temperatures to the heat rates.

## Surface Temperatures and Resistance

Thermal resistances are useful because they directly relate the heat rate between two points with the temperature between those points.

$$R_{layer} = \frac{\Delta T_{layer}}{\dot{Q}}$$

The value of resistance depends on the mechanism of heat transfer

$$R_{convection} = \frac{1}{h A}$$

$$R_{conduction} = \frac{\Delta r}{k A}$$

For conduction, the area and thus resistance changes with increasing  $r$ , so an integration must be done

$$\int_0^R dR_{conduction} = \int_{r_1}^{r_2} \frac{dr}{k A_{eff}} = \int_{r_1}^{r_2} \frac{dr}{k A_{ratio} 2 \pi r L}$$

$$R_{conduction} = \ln \left( \frac{r_2}{r_1} \right) \frac{1}{k A_{ratio} 2 \pi L}$$

These resistances can all be calculated with user-defined data. AFT xStream helps simplify this by offering several correlations that can be used to determine the heat transfer coefficients in the above equations from more readily available parameters.

With resistances and  $\dot{Q}_{fluid}$ , all the surface temperatures can be determined. By bounding the system with a given  $T_{ambient}$  we can work outside in to find the required  $T_{fluid}$  to obtain those heat rates.

$$\frac{T_{ambient} - T_3}{R_{convection}} = \dot{Q}_{ext}$$

$$\frac{T_3 - T_2}{R_{insulation}} = \dot{Q}_{insulation}$$

$$\frac{T_2 - T_1}{R_{wall}} = \dot{Q}_{wall}$$

$$\frac{T_1 - T_{fluid}}{R_{fluid}} = \dot{Q}_{fluid}$$

If the result for  $T_{fluid}$  matches the values used for  $\Delta T_{fluid}$  used for determining  $\dot{Q}_{fluid}$ , the correct solution has been found. If they are different,  $\Delta T_{fluid}$  needs to be adjusted and the process repeated until reaching a stable solution.

## Overall Heat Transfer

Wherever there are sequential thermal losses, it is convenient to combine these losses. This is straightforward:

$$R_{total} = \sum R_{layer}$$

Additionally, an overall heat transfer coefficient  $U$  can be defined which behaves similarly to  $h$  or  $k$

$$R_{total} = \frac{1}{UA}$$

Using the heat rate and resistance relationship, as well as the fact the heat transfer is into the fluid

$$\dot{Q}_{total} = \frac{\Delta T_{radial}}{R_{total}} = \dot{Q}_{fluid}$$

$$UA(T_{external} - T_{fluid}) = \dot{m} c_p \Delta T_{fluid}$$

Transforming the equation into a differential form on radius and length

$$UP(T_{external} - T_{fluid}) dx = \dot{m} c_p dT_{fluid}$$

Rearranging and integrating

$$\int_0^L \frac{UP}{\dot{m} c_p} dx = \int_{T_{in}}^{T_{out}} \frac{1}{T_{external} - T_{fluid}} dT_{fluid}$$

$$\frac{UPL}{\dot{m} c_p} = -\ln\left(\frac{T_{external} - T_{out}}{T_{external} - T_{in}}\right)$$

We arrive at an expression relating the temperature change along the pipe and temperature change through the pipe:

$$e^{\frac{-UPL}{\dot{m} c_p}} = \frac{T_{external} - T_{out}}{T_{external} - T_{in}}$$

## Correlations

Thermal resistances depend on the conduction and convection coefficients  $k$  and  $h$  which can be directly specified or calculated via a correlation.

The convection coefficient for internal fluid convection can be calculated with the Gnielinski Correlation or the Dittus-Boelter Correlation which can be seen in [Incropera](#). These correlations determine the Nusselt number of the fluid – a ratio of convective to conductive heat transfer. Conductive properties are required input – with a Nusselt number and conductivity defined, the convection coefficient can be calculated.

$$Nu = \frac{\text{Convection}}{\text{Conduction}} = \frac{h}{k/L}$$

These correlations also make use of the Prandtl number, a ratio of viscous diffusion to thermal diffusion.

$$Pr = \frac{\text{Viscous diffusion}}{\text{Thermal diffusion}} = \frac{\mu/\rho}{k/(c_v \rho)} = \frac{c_p \mu}{k}$$

Gnielinski

$$Nu = \frac{(f/8)(Re - 1000)Pr}{1 + 12.7(f/8)^{0.5}(Pr^{2/3} - 1)} \quad \text{valid for } \left( \begin{array}{l} 0.5 \leq Pr \leq 2000 \\ 3000 \leq Re_D < 5 * 10^6 \end{array} \right)$$

Dittus-Boelter

$$Nu = 0.023 Re^{4/5} Pr^n \quad \text{where} \quad \begin{pmatrix} n = 0.4 \text{ for heating} \\ n = 0.3 \text{ for cooling} \end{pmatrix} \quad \text{valid for} \quad \begin{pmatrix} 0.6 \leq Pr \leq 160 \\ Re \geq 10^4 \end{pmatrix}$$

External convection behavior differs from the above. AFT xStream provides the Churchill-Bernstein correlation for determining a convection coefficient from ambient temperature and fluid velocity. This correlation can be used for pipes over which boundary layers develop freely, free of constraints imposed by other surfaces. All of the external fluid properties are evaluated at the film temperature.

$$\overline{Nu} = 0.3 + \frac{0.62 Re^{1/2} Pr^{1/3}}{\left[1 + \left(\frac{0.4}{Pr}\right)^{2/3}\right]^{1/4}} \left[1 + \left(\frac{Re}{282,000}\right)^{5/8}\right]^{4/5} \quad \text{valid for } (Pr * Re \geq 0.2)$$

## Related Topics

References..... 461

## Related Blogs

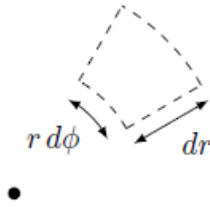
[Let the Heat Flow: Modeling Heat Transfer in Pipes in AFT Fathom and AFT xStream](#)

## Transient Heat Transfer in Pipes

AFT xStream uses a thermal resistance network to account for convective and conductive heat transfer and calculate the steady state heat rates. xStream then uses a discrete solution method to account for radial and axial heat transfer through the pipes and fluid during the transient. This topic discusses the solution methods used by xStream to determine the heat rate and temperatures during the transient. See the [Steady State Heat Transfer in Pipes](#) page for information on how the heat rate is derived, and how the convection coefficients are calculated.

## Transient Heat Rate

Consider a cylindrical control volume as shown below in Figure 1. Note that dz is perpendicular to the page and thus is not shown.



**Figure 1: Cylindrical control volume for heat rate calculation through the pipe wall/insulation, with  $dz$  extending out of the page**

The heat energy in the control volume due to a change in temperature can be written as:

$$\dot{Q} = m c_p \frac{\partial T}{\partial t} = \rho c_p \frac{\partial T}{\partial t} r dr d\phi dz \quad (1)$$

The heat energy can also change in response to heat flux and generation:

$$\dot{Q} = \dot{E}_{gen} + \dot{Q}_r - \dot{Q}_{r+dr} \quad (2)$$

By simplifying the heat flux balance and equating it to the heat energy due to temperature change the following is found:

$$\dot{E}_{gen} - \frac{\partial \dot{Q}}{\partial r} dr = \rho c_p \frac{\partial T}{\partial t} r dr d\phi dz \quad (3)$$

From Fourier's law for conduction the heat rate is defined as

$$\dot{Q} = -k r d\phi dz \frac{dT}{dr} \quad (4)$$

By substituting Fourier's law into equation 3 above and dividing the whole equation through by  $dr d\phi dz$ , the following heat diffusion equation is derived for the radial dimension.

$$\frac{\dot{E}_{gen}}{r} + \frac{1}{r} \frac{\partial}{\partial r} \left( k r \frac{dT}{dr} \right) = \rho c_p \frac{\partial T}{\partial t} \quad (5)$$

---

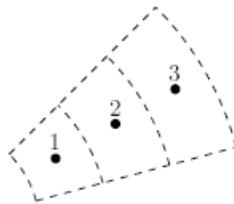
## Transient Pipe Temperatures

---

xStream will account for heat transfer in the radial and axial directions. However radial heat transfer is dominant, so the below discussion will focus on calculating radial heat transfer.

To calculate the heat rate and temperature profile across the pipe walls and insulation the heat diffusion equation (Equation 5) must be solved. There is not a simple analytical solution for the heat diffusion equation derived above, so a discrete solution method will be used.

The pipes and insulation layers will each be sectioned radially similar to the example in Figure 2 below, with points 1, 2, and 3 representing calculation nodes at which the temperature and heat rate will be explicitly calculated. Each node represents the temperature, density, and thermal conductivity of the control volume they are within.



**Figure 2: Partial pipe cross-sectional area representing the radially sectioning for heat transfer calculations**

There are a few assumptions that are made to calculate the heat rate and temperature.

First, for the purpose of calculating a heat rate xStream assumes a linear temperature profile between each set of consecutive temperature nodes. In reality the temperature would vary logarithmically through a single material. However, using a logarithmic temperature function would be computationally difficult, and provide a negligible increase in accuracy.

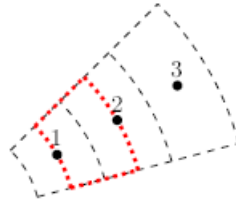
Second, the temperature is assumed to be constant at the boundary of different materials, such as where the insulation touches the pipe wall. In reality there would be some contact resistance creating a difference in temperatures between the pipe and insulation, but this is typically a negligible effect.

Lastly, for calculation nodes that exist at a boundary between pipe materials, the thermal conductivity and density are assumed to be an average (by volume) of the staggered control volume (Figure 3). This assumption is made to simplify the solution by creating a continuous temperature profile. From testing this assumption provides reasonable results.

---

**Note:** The discrete solution method used for calculating radial conduction can potentially become unstable and give physically unrealistic results if the number of radial calculation nodes is insufficient. The default number of nodes should be sufficient for most cases. The number of thermal nodes can be changed on the [Sectioning Panel](#).

---



**Figure 3: Partial pipe cross-sectional area with the staggered control volume outlined in red**

## Insulation for Pipes

Insulation data can be entered on the Heat Transfer tab of a pipe. The insulation data is only included in the heat transfer calculations if a convective heat transfer model is entered. It may be entered for other heat transfer models, but will only be used for cost calculations. Internal insulation will reduce the flow area which AFT xStream will account for in such cases. Insulation data is from the [Nayyar Piping Handbook 2000](#) pages B.325-B.344.

See [Heat Transfer in Pipes](#) for more details.

## Related Blogs

---

[Let the Heat Flow: Modeling Heat Transfer in Pipes in AFT Fathom and AFT Arrow](#)

## Optional Input for Pipes

### Initial Flow Guess

---

The Initial Flow Guess for a pipe can be specified. An appropriate initial guess can decrease the number of iterations required to arrive at a converged solution. This initial guess will only affect the steady-state solution process.

The Solver generates initial guesses on its own, which is usually sufficient to get the Solver going in the right direction so a converged solution results. If this is not the case, user defined guesses can help obtain a solution.

### Pipe Line Size and Color

---

Each pipe may have its line size and color changed from the defaults configured in [Workspace Options](#).



---

## Display on Workspace

---

By default, only the pipe number is displayed on the [Workspace](#). In addition to this, the pipe name, nominal size, and type/schedule can be displayed.

---

## Design Factors

---

A design factor can be used to add safety margins into the analysis. For pipes, two design factors exist:

- Pipe Friction: friction factor is multiplied by the design factor assigned to the pipe.
- Pipe Fittings & Losses: total loss factor (K factor or Equivalent Lengths) is multiplied by the design factor assigned to the pipe.

---

## Parallel Pipes

---

A single [pipe](#) object can be specified as a number of parallel pipes. When multiple pipes are defined, the parameters in the pipe properties window represent only one of the parallel pipes.

---

## Use Temperature Override for Zero Flow

---

Enabling the Temperature Override option will force the temperature in the pipe to the specified value for the initial steady state solution. The override will only occur for pipes with zero flow in the steady state.

If the temperature override option is not used the temperature in the pipe will be set to the ambient temperature if the heat transfer model is set as Convective Heat Transfer, or to the temperature of the connected boundary junction if any other heat transfer model is used. See [Heat Transfer in Pipes](#) for more information.

---


## Related Topics

---

Optional Input.....	197
Make Initial Flow Rate Guesses for Pipes.....	526

## Add/Remove Pipe Segment

Visual segments can be added to pipes.

To add or remove segments, use the Arrange -> Add/Remove Segments submenu. These options are also available by clicking the Add/Remove Segments button  on the Workspace Toolbar.

---

**Note:** These segments are useful for creating a clear layout of the system, but have no impact on the solution - a visual "bend" is not the same as adding a [Branch junction](#).

---


---

## Related Topics

---

Keyboard Shortcuts for Workspace ..... 293

### Reverse Direction

Every pipe has a [reference positive flow direction](#) as indicated by its arrow. This direction can be easily reversed by selecting Reverse Direction from the [Arrange menu](#) or clicking the Reverse Direction button  on the Workspace [Toolbar](#).

### Pressure Drop in Pipes - Detailed Discussion

Depending on the pipe friction model selected, the calculation of pressure drop in a pipe differs.

#### Darcy-Weisbach Loss Model

---

**Note:** The pressure drop for compressible flow depends on several equations, as discussed in [Review of Compressible Flow Theory](#). The effect of pipe friction is taken into account as part of the full compressible method. The Darcy-Weisbach pressure loss equation for incompressible flow is shown below for simplicity.

---

AFT xStream utilizes the Darcy-Weisbach loss model to relate the Darcy Friction Factor, the pipe geometry, fluid density, and fluid velocity to pressure drop in the pipe. The Darcy Friction Factor differs from the Fanning Friction Factor by a factor of 4.

$$\Delta P = f \frac{L}{D} \frac{1}{2} \rho V^2$$

This model requires the calculation of a friction factor, as described below.

- **Roughness-Based Methods** - This method calculates the friction factor based on the roughness of the pipe wall. Different equations are used based on flow regime - laminar flow uses the standard laminar relationship, while turbulent flow uses the implicit Colebrook-White equation. In the transition range between laminar and turbulent flow, a linearly interpolated value is used. The default transition Reynold's Numbers can be modified in the [Environmental Properties panel](#) of Analysis Setup.

$$f = \frac{64}{Re} \quad \textit{Laminar, } Re < 2300 \textit{ (default)}$$

$$f = \left[ 1.14 - 2 \log \left( \frac{\epsilon}{D} + \frac{9.35}{Re \sqrt{f}} \right) \right]^{-2} \quad \textit{Turbulent, } Re > 4000 \textit{ (default)}$$

- **Absolute Roughness (default)** - The absolute average roughness height  $\epsilon$  is specified directly.
- **Relative Roughness** - The roughness is specified as a ratio  $\epsilon/D$ .
- **Hydraulically Smooth** - The ratio  $\epsilon/D$  is set equal to zero.
- **Explicit Friction Factor** - The friction factor to be used in the Darcy-Weisbach equation is specified directly

## Frictionless

---

The pipe will not have any pressure drop across it. This is inherently **unrealistic** behavior, but can be useful for troubleshooting purposes.

## Design Factors

---

In each pipe you can specify a [Design Factor](#) for the pipe friction. This is a multiplier that is applied to the friction factor calculated with the preceding methods.

## Related Topics

---

Features for Modeling Irrecoverable Losses.....	59
Conventions for Specifying Base Area.....	198
Nomenclature.....	410

## Junctions

This section focuses on the Junctions in the workspace as well as the theories that govern the junctions. Topics included discuss the types of junctions that xStream provides, the required inputs for each junction, and theory topics that indicate how the inputs to the junction are used.

### Contents

---

<a href="#">Junction Window Overview</a> .....	192
<a href="#">Junction Loss Models</a> .....	201

## Junction Window Overview

The different types of junctions on the Toolbox offer a great degree of freedom in assembling a pipe network model. While there is some redundancy in capabilities between the junctions, each of the types of junctions offers specialized features to allow you to prepare a good conceptual model of the physical system of interest. Table 1 compares the features of the junction types.

There is a junction properties window for each junction type. Each junction Properties window falls into one of [two basic window formats](#), depending on the number of connecting pipes allowed for that junction type.

---

**Note:** AFT xStream does not prevent negative flow through junctions. Should this occur, AFT xStream will use the same loss factor or pressure drop data referenced to the defined upstream pipe, just as in forward flow. Because it is possible that the loss factor or pressure drop data would not be the same for both flow directions, the flow solution could be misleading. For this reason it is important that you either properly define the pipe flow directions for junctions or pay close attention to the flow directions that result. AFT xStream will not always give a warning for this, though it will for certain junctions.

---

## Junctions

**Table 1: Junction Property Windows and Features**

Junction Type	Connecting Pipes	Special Conditions	CdA	Custom Library	Window Format
Area Change	2		Yes	Yes	1
Assigned Flow	1	Yes		Yes	1
Assigned Pressure*	1/1-25		Yes	Yes	1
Bend/Elbow	2		Yes	Yes	1
Branch	2-25			Yes	2
Check Valve	2	Yes	Yes	Yes	1
Compressor/Fan	2	Yes		Yes	1
Control Valve	2	Yes		Yes	1
Dead End	1				1
General Component	2	Yes	Yes	Yes	1
Heat Exchanger	2	Yes		Yes	1
Orifice	1-2		Yes	Yes	1
Relief Valve	1-2	Yes	Yes	Yes	1
Screen	2		Yes	Yes	1
Tank	1-25			Yes	2
Tee/Wye**	3				2**
Valve	1-2	Yes	Yes	Yes	1
Venturi	2		Yes	Yes	1

\*Assigned Pressure Junction connects to only one pipe for static pressure, and up to 25 for stagnation

\*\*Tee/Wye junction has a unique format closer to Format #2

### Related Topics

Notes.....	163
Status.....	163
Common Junction Input Parameters.....	196

## Two Window Formats

### Format #1: Junctions With One or Two Connecting Pipes

The first basic window format is for junction types that allow only one or two connecting pipes (see Figure 1). While junctions do not have an explicitly defined flow direction like pipes do, typically those with two pipes or less adopt a direction from the connecting pipes. For these junction types, upstream and downstream pipes are recognized by AFT xStream based on the [reference positive flow direction](#) of the connecting pipes. The upstream and downstream pipes are displayed separately near the top of the window.

In this first basic window type, it is generally important to have the pipes' reference positive flow directions specified in the physically correct directions. A good example of this is a [Compressor junction](#). A compressor will add pressure to the system, but it does so only in the direction of positive flow through the connecting pipes. The compressor junction interprets where to add the pressure based on the directions of the connecting pipes.

To [inspect](#) in this window format, position the mouse pointer over the connected pipe ID and hold down the right mouse button.

Valve Properties

Number: 4  
 Name: Valve  
 Library Jct:   
 Copy Data From Jct...   
 Upstream Pipe: 7  
 Downstream Pipe: 8  
 Elevation: 0 feet

OK  
 Cancel  
 Jump...  
 Help

Loss Model | **Transient** | Optional | Design Alerts | Notes | Status

Subsonic Loss Model  
 Cv (ANSI/ISA) ?  
 Kv

Loss Source  
 Fixed Cv  
 From Open Percent Table (on Optional Tab)  
 % Open:   
 Cv: 3000  
 Xt: 0.7  
 Guide for Xt...

Open Percentage Data

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

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

Flow Area Units: feet2  
 Open Pct Increment: 10  
 Autofill

Exit Valve (optional)  
 Exit Pressure:   
 Exit Temperature:

Figure 1: Properties window format #1 (fixed connecting pipe display).

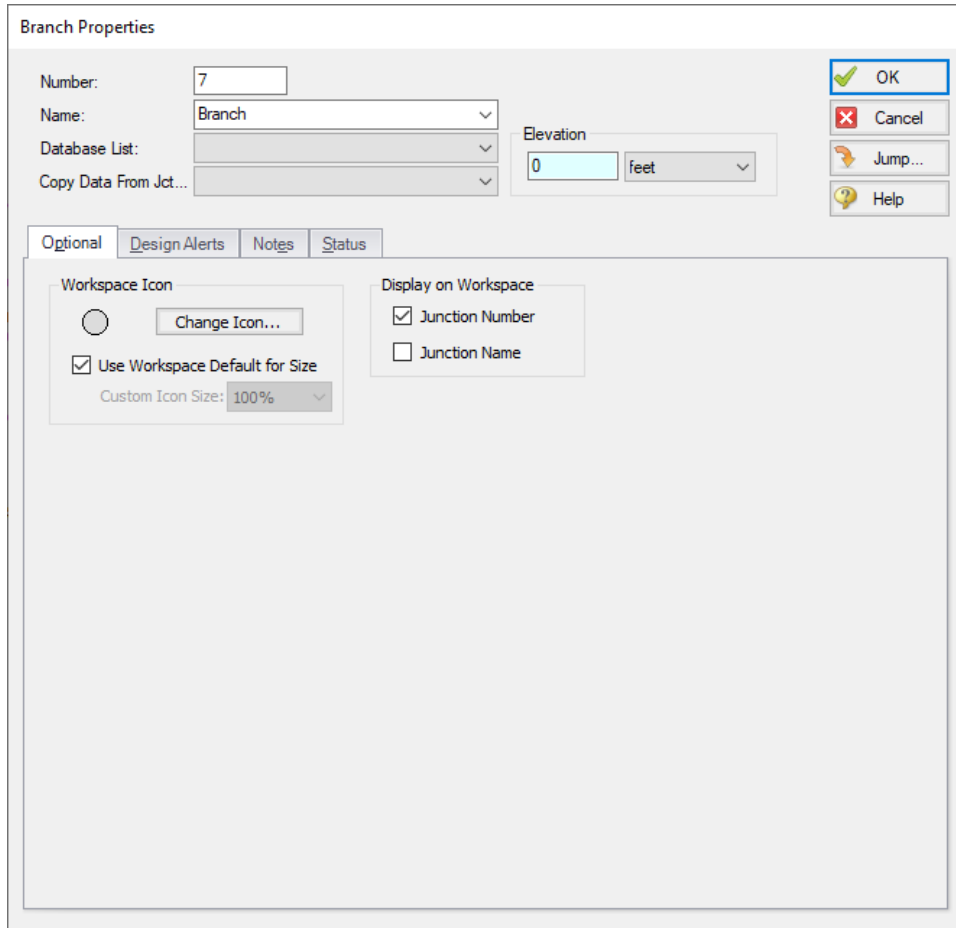
---

## Format #2: Junctions With More Than Two Connecting Pipes

---

The second basic window format for junction Properties windows is for junctions that allow more than two connecting pipes (see Figure 2). These junctions typically allow up to 25 connecting pipes; the exception is the [Tee/Wye junction](#), which allows only three.

An example of this second window format is the [Branch Properties](#) window, shown in Figure 2, which has three connecting pipes.



**Figure 2: Properties window format #2**

## Common Junction Input Parameters

### Number

---

Every junction must have a unique ID number. When new junctions are created, a junction number is automatically allocated.

The junction number can be changed by the user to any unique integer value greater than zero and less than 100,000. Junction numbers are only identifiers and have no effect on the model behavior.

### Name

---

A name can be assigned to each pipe for reference purposes. Names do not need to be unique. The name can be shown on the Workspace, in the Visual Report and in the Output window.

### Library List

---

A list of all available [custom components](#). It displays components from any connected library.

Selecting a component from this list automatically retrieves the junction settings and populates the properties window. Changing the input manually will deselect the component in the list.

### Copy Data From Jct

---

The Copy Data From Junction dropdown list allows you to copy all or some of an existing junction's properties to the current junction. Selecting an existing junction will prompt the user on what parameters should be changed.

In child [scenarios](#), Copy Data From Junction lists "Parent Junction Data," allowing the user to copy all or some parameters from the parent junction.

### Pipe Connectivity

---

The connected pipes are displayed. This is helpful to determine potential [connection](#) issues and easily identify which pipes are upstream and downstream of the junction.

### Elevation

---

Every junction must have an elevation reference. This value is the centerline elevation of the connected pipe(s).



Junction elevations are used to account for pressure changes due to gravity, allowing calculation of absolute pressures in pipes.

A default elevation can be specified in Parameter Options, automatically setting new junctions to the specified elevation.

## Jumping to Another Junction

---

Quickly move to the next junction. See [Jump Feature](#).

## Related Topics

---

Notes.....	163
Status.....	163

## Optional Input

## Workspace Display

---

By default, only the junction number is displayed on the [Workspace](#). In addition to this, the junction name can be displayed.

## Special Conditions

---

The [Special Condition](#) can be changed for some junctions. For example, a valve can be set to closed, preventing all flow. A symbol (default of "X") will be placed before the junction label indicating that a special condition is applied.

## Workspace Icon

---

You may want to change the icon to better represent a specific junction type. This is accomplished by clicking the Change Icon button on the Optional tab. The Change Icon window appears, allowing you to choose a new icon from a list of available icons.

You may want to change the size of the junction to distinguish it from other junctions. This is accomplished by clearing the use Workspace Default for Size checkbox and selecting the desired size in the drop down list.

## Related Topics

---

Optional Input for Pipes.....	188
-------------------------------	-----

## Specifying Losses

Local losses can be specified in two ways. The most flexible way of including a loss factor in an AFT xStream model is to associate it with a junction specifically defined for that type of loss. This allows you to define a loss as a function of a flow parameter such as Reynolds number or volumetric flow rate, or as a function of the area change ratio between the upstream and downstream pipes. This method results in the greatest clarity.

Secondly, you can include loss factors with pipes. When you specify [Fittings & Losses](#) in the [Pipe Properties window](#), the loss is assumed to be distributed evenly along the pipe length, much like friction losses. For this reason a pipe-associated loss factor is referred to as a distributed loss. The losses at junctions, on the other hand, are point losses.

It should be noted that for compressible flow, the location of a fitting/loss along a pipe will change the pressure drop due to that fitting/loss. This is because the velocity along a pipe changes for compressible flow. For this reason, the recommended method for representing losses is to define them in a junction.

### Related Topics

Fittings & Losses Library.....	398
Fittings & Losses.....	169
Discharge Coefficient Loss Model.....	201
K Factor Loss Model.....	202
Cv/Kv and XT Loss Model.....	205
Resistance Curve Loss Model.....	208

## Conventions for Specifying Base Area

[Loss factors](#) are area dependent. When specifying a loss factor at a junction, the base area is of critical importance. By default AFT xStream adheres to the following convention: whenever it makes sense physically, the base area is always the upstream pipe flow area. However, this can be modified by the user. In most Junction Properties windows (on the Optional tab), the base area can be specified as the upstream pipe, downstream pipe, user specified area, or user specified diameter.

### Related Topics

Features for Modeling Irrecoverable Losses.....	59
Pressure Drop in Pipes - Detailed Discussion.....	190

## Reference Density

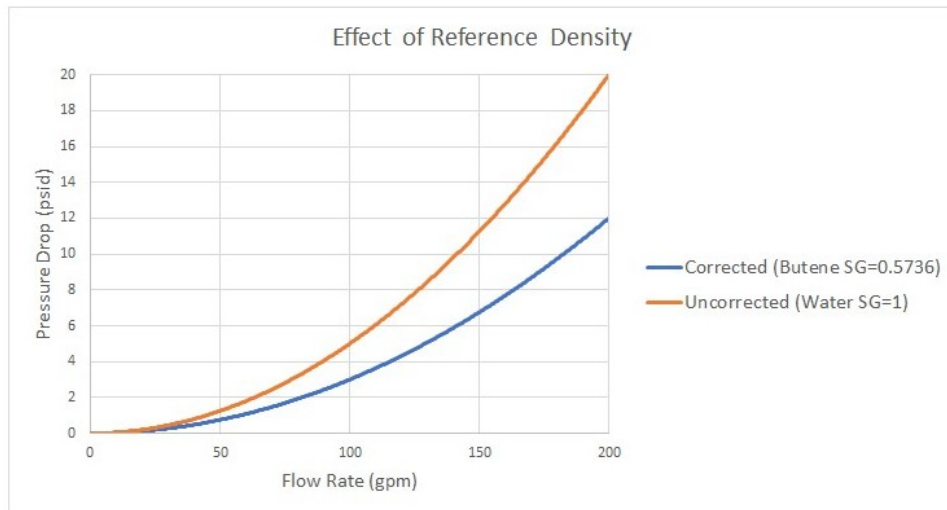
A relationship can be drawn for many components between *flow energy* and *flow rate*. This can be a decrease (e.g., a loss across a valve) or increase in energy (e.g. energy added by a compressor). The specific parameter used to represent *flow energy* can vary depending on application. It can be given in terms of *pressure change* or *head change*, and there is a critical difference between the two.

Consider a centrifugal turbomachine (a pump or radial compressor): Kinetic energy is imparted to the fluid by accelerating it along the radius of the impeller. If the rate of rotation is held constant, a denser fluid will

have more energy imparted to it than a lighter weight fluid. This would give rise to higher pressures. In other words, the *pressure change* depends on the density of the fluid.

Sometimes curves are provided in terms of head. This is convenient because head is a unit of specific energy - or energy per weight. Therefore, it does not vary with fluid density like pressure does. The head added to a fluid by a pump or compressor at a fixed speed, for example, will be the same for fluids of different densities.

However, head relationships are not always available or meaningful. If a *pressure vs flow* relationship is used, a **reference density** must be specified. This is the fluid density used during testing and creation of the curve.



**Figure 1: Visual comparison of Pressure Loss Curves for differing densities. Head curves would be identical for this component**

---

**Note:** Incompressible liquids are used in the example in Figure 1, but the principles apply similarly to compressible flow.

---

### Affected Parameters

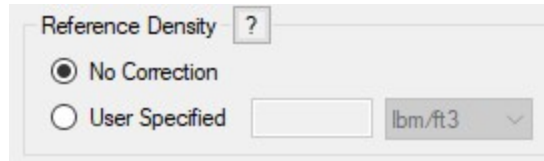
---

There are several parameters that are affected by the density of the fluid:

- **Pressure** - As discussed above
- **Mass Flow** - The mass term contains information about the density of the fluid
- **Power** - More power is demanded by a compressor to meet the head curve for a denser fluid. While head is independent of density and therefore often reported by manufacturers, power is not independent of density. Therefore, a proper specification sheet that includes power must also include the reference fluid density.

## Accounting for Reference Density

[Compressors](#) and any [Resistance Curve](#) allow for the specification of a reference density if one of the above parameters is selected. The selection appears below the curve fit area as shown in Figure 2.



**Figure 2: Reference Density Selection**

There are two options:

- **No Correction (default)** - This selection assumes that the component was tested with the fluid defined in the [Fluid Panel](#) and is operating at the reference conditions.
- **User Specified** - This directly specifies the reference density. If the fluid or operating conditions differ from the testing conditions, the entered curve will be corrected.

## Related Topics

Curve Fitting.....66

### Equal Percentage Valves

Equal Percentage valves have a valve plug shaped so that each increment in valve percent open (lift) increases the flowrate by a certain percentage of the previous flow.

The relationship between valve lift and orifice size is not linear but logarithmic.

It is expressed mathematically as follows:

$$C_v = \frac{e^x}{\tau} C_{v,max}$$

$$x = \ln(\tau)H$$

Where:

- $\tau$  = valve rangeability (ratio of the maximum to minimum controllable flowrate)
- $H$  = valve percent open or lift (0 = closed; 1= fully open)
- $C_{v,max}$  = full open valve loss value

## Related Topics

Control Valve.....	247
Valve.....	263

## Junction Loss Models

There are a number of loss models available in xStream to account for frictional losses at the junctions in the model. See the below topics for more information on how losses are calculated for each of the available models.

## Discharge Coefficient Loss Model

Orifice subsonic losses for the Orifice, Relief Valve and Venturi junctions can be defined with a discharge coefficient,  $C_d$ .

### Subsonic Loss

When solving the system, AFT xStream creates a subsonic  $C_dA$  for the orifice using the  $C_d$  specified for the subsonic loss model, and the specified orifice area. xStream then solves the pressure drop by iterating the following equation for pressure drop for a given mass flow rate. The equation below for mass flow rate can be derived from isentropic relationships, replacing the static pressure at the restriction with the downstream stagnation pressure,  $P_{o,out}$

$$\dot{m} = C_d A \sqrt{2 P_{o,out} \rho_{o,in} \frac{\gamma}{\gamma - 1} \left( \frac{P_{o,in}}{P_{o,out}} \right)^{-1/\gamma} \left[ \left( \frac{P_{o,in}}{P_{o,out}} \right)^{(\gamma-1)/\gamma} - 1 \right]}$$

The assumption that the static pressure at the restriction can be replaced with the downstream stagnation pressure,  $P_{o,out}$ , is necessary to find a solution, as the velocity must be known to find the static pressure, and the mass flow must be found before the velocity can be a known quantity. It is important to note that this assumption neglects static pressure recovery, which could be a significant factor.

### Sonic Loss

The above equation for the mass flowrate can be simplified when the orifice is sonically choked. When the Mach number at the restriction is 1, the equation can be simplified as shown below. The value of  $C_dA$  used here will be the  $C_dA$  for sonic choking, which is entered as a separate value from the discharge coefficient,  $C_d$ , as is discussed in the [Subsonic vs. Sonic Pressure Losses](#) topic.

$$\dot{m} = C_d A \sqrt{\gamma P_o \rho_o \left( \frac{2}{\gamma+1} \right)^{(\gamma+1)/(\gamma-1)}}$$

**Note:** The  $C_d A$  for sonic choking may be different from the subsonic  $C_d A$  loss model option in xStream. The discharge coefficient can vary at different pressure ratios due to the vena contracta moving closer to or farther from the orifice restriction. For the highest accuracy the  $C_d A$  used for subsonic and sonic losses should be tested and entered separately. See the "[Modeling Choked Flow Through an Orifice](#)" white paper on AFT's website for more information.

## Transient Theory

The above equations for the mass flow rate through an orifice are applied in the transient simulation similar to the approach used for other loss models. Inlet conditions are guessed at, a mass and energy balance is applied to find the outlet conditions, and then those conditions are used to determine the mass flow through the orifice for that guess. Iteration proceeds until the guesses for inlet conditions are consistent with the solved mass flow equation.

## Related Topics

Orifice.....	253
Relief Valve.....	256
Subsonic vs. Sonic Pressure Losses.....	454

## K Factor Loss Model

AFT xStream models component losses according to the following equation:

$$\Delta P_{loss} = K \frac{1}{2} \rho V^2 \tag{1}$$

where  $K$  is commonly referred to as the loss factor. Note that for calculation purposes this pressure loss is a *stagnation* pressure loss.

Table 1 lists the sources for the  $k$  factor loss models used in AFT xStream. The losses implemented directly in the code were chosen on the basis of ease of use. Many loss factor types are functions of the flow, and thus too general to be easily incorporated.

**Table 1: Loss model references**

Junction Type	References
Area Change	Crane 1988, A-26; Idelchick 2007, 246 & 254
Orifice	Idelchick 2007, Chapter 4

[Crane \(1988\)](#) offers good general purpose correlations for modeling irrecoverable losses in pipe systems. A well prepared pipe flow analyst would be advised to have copies of either or both of these references.

Another lesser known source of loss factor information is Idelchik's Handbook of Hydraulic Resistance ([Idelchik 2007](#)). This reference is indispensable for the engineer who must make detailed assessments of pipe systems in which so-called minor losses play a significant roll. The reference is voluminous in scope and full of tables, charts and equations for calculating loss factors for almost any pipe arrangement.

[Miller's Internal Flow Systems \(1990\)](#) is another reference offering good general purpose hydraulic data.

---

## K Factor Losses in Compressible Flow

Most handbook K factor data is based on incompressible flow tests or theory. Applying standard K factors to compressible flow components carries some uncertainty. There are not any universally accepted methods available to determine K factors for truly compressible flow except in very specific situations.

xStream offers common handbook values for K factors, and you should use caution in how you apply the data. Also recognize that when you use the "[Fittings & Losses](#)" in pipes, computationally these K factors are spread out along the whole pipe as distributed losses. In compressible flow, the location of a fitting along a pipe will affect the actual pressure loss since velocity is not constant along a pipe. The Fittings & Losses do not account for this.

---

## Transient Theory

When a junction is defined using a K factor, the junction inlet will be treated as a static pressure boundary. A guess for the inlet static pressure is made, which can be used to solve for the inflow conditions using the compatibility equations as shown for the [Assigned Pressure junction](#) at the downstream end of a pipe. With the known inlet conditions, the inlet stagnation pressure and stagnation enthalpy can be calculated and used to solve for outlet stagnation pressure using the K factor loss equation above.

With the outlet stagnation pressure and enthalpy the outlet conditions can be solved using the method for an [assigned pressure boundary](#).

With both the inlet and outlet conditions known the inlet and outlet mass flow rates can be calculated and compared. If the mass flow rate difference is too large, then iterations on the inlet static pressure will be completed until the mass flow rate tolerance is met.

## Exit Junction Transient Theory

If a K factor loss is defined for a junction that is specified as an exit type junction, such as a relief valve, then a different solution method must be applied as there is no downstream pipe.

The user specified exit conditions are assumed to be stagnation properties. The K factor loss model can be rearranged and the compatibility equations substituted as follows.

$$P_{o,in} - P_{o,out} - \frac{1}{2} K \rho_{in} V_{in}^2 = 0$$

$$(C_{wave} + B_{wave} V_{in}) \left( 1 + \frac{\gamma - 1}{2\gamma} \frac{(C_{path} + B_{path} (C_{wave} + B_{wave} V_{in})) V_{in}^2}{(C_{wave} + B_{wave} V_{in})} \right)^{\gamma/(\gamma-1)} - P_{o,out} - \frac{1}{2} K (C_{path} + B_{path} (C_{wave} + B_{wave} V_{in})) V_{in}^2 = 0$$

The combined equations can then be rearranged to solve for velocity and density at the junction inlet.

If flow reverses at the exit junction then the solution will be similar to the solution with two pipes connected, except that the effective pipe area at the exit side of the junction will be assumed to be equivalent to the area of the pipe connected to the entrance of the junction.

## Sonic Choking Transient Theory

Either [restriction choking](#) or [expansion choking](#) can occur at junctions modeled with K factor loss, depending on the geometry of the junction.

When the flow is choked iterations on the inlet pressure will be performed similar to the non-choked transient solution. However, the [sonic area](#) will be used as the dependent variable for the iterations. Once the inlet conditions are determined, the mass flow rate and stagnation enthalpy can be used to solve for the outlet conditions.

## Related Topics

Area Change Loss Model.....	220
General Component.....	251
Heat Exchanger.....	252
Control Valve.....	247
Relief Valve.....	256
Valve.....	263
Orifice Loss Model.....	254
Transient Sonic Choking.....	443



Sonic Choking Detailed Description .....	450
References .....	461

## Cv/Kv and XT Loss Model

AFT xStream allows the user to enter valve loss coefficient data in terms of  $K_V$  or  $C_V$ . Conversions are handled seamlessly in the background.  $K_V$  is the metric equivalent to  $C_V$ .

The valve  $C_V$  loss model type is unique in that this loss model can be used as part of both the [subsonic and sonic loss](#) calculations across the valve.

### Subsonic Loss

The relationship between valve  $C_V$ ,  $X_T$ , and flow is provided by [ANSI/ISA-75.01.01-2012](#) standard as follows:

$$\dot{m} = C N_6 F_P Y \sqrt{X_{sizing} P_1 \rho_1}$$

Where  $C$  is the flow coefficient ( $C_V$  or  $K_V$ ) and  $N$  is a numerical constant found in Table 1 of the ANSI standard referenced above. The units for the flow, pressure and density will be based off of the numerical constant chosen.

$F_P$  is a correction factor for piping geometry. This correction will be treated as 1 unless the Valve ID has been entered by the user.

$X_{sizing}$  is the ratio of the pressure drop across the valve to the inlet pressure of the valve for subsonic flow:

$$X = \frac{\Delta P}{P_1}$$

$Y$  is the expansion factor, which accounts for the change in density as the gas passes through the vena contracta, and is calculated as follows:

$$Y = 1 - \frac{X_{sizing}}{3 X_T}$$

Where  $X_T$  is the pressure ratio,  $X$ , when the valve has just begun to choke.

$X_T$  can be a complicated topic. If more information on it is needed, a helpful blog post on the topic can be found [here](#).

## Sonic Loss

---

If choked flow occurs across the valve, then the mass flow equation above is still used. However, the  $X_{sizing}$  is now equivalent to  $X_T$ , which is the pressure ratio when the valve first begins to choke.

## Estimated Cv/Kv in Output

---

In the Output window the estimated  $C_v/K_v$  can be displayed if the  $C_v/K_v$  was not defined by the user. The estimated  $C_v/K_v$  value is calculated using the mass flow rate equation shown above, with  $X_{sizing}$  estimated as follows

$$X_{sizing} = \frac{\gamma_1}{1.4} \left( \frac{\Delta P}{P_1} \right)$$

## Transient Theory

---

A valve junction has a pressure loss across the valve. When the valve closes this loss becomes infinite. With two connecting pipes, the upstream and downstream pipe parameters must be determined through iteration. The valve will be treated as a mass flow boundary. A guess for the velocity is made, which can be used to solve for the inflow conditions using the compatibility equations as shown for the [Assigned Flow junction](#) at the pipe inlet. This will provide values for the conditions at the valve outlet, including the mass flow at the valve.

The outlet static pressure can now be used to calculate the pressure ratio  $X$  and the expansion coefficient  $Y$  as are shown above. The calculated  $X$  and  $Y$  can then be used in the to determine the ANSI/ISA predicted mass flow as shown above.

The mass flow calculated from the assigned flow boundary solution and the mass flow calculated from the ANSI/ISA equation can now be compared. The difference between these two values will be driven to zero using iterations on velocity.

## Exit Valves Transient Theory

---

If the valve is an exit valve, then the downstream pressure is known. The user specified exit properties are assumed to be static. The [compatibility equations](#) can be substituted into the ANSI/ISA mass flow equation shown above, which is then rearranged to solve for the inlet pressure at the valve.

$$\begin{aligned} & (C_{path} + B_{path} P_{in}) A_{in} \left( \frac{P_{in} - C_{wave}}{B_{wave}} \right) \\ & = C_V N_6 F_P \left( 1 - \frac{1}{3 X_{choked}} + \frac{1}{3 X_{choked}} \frac{P_{out}}{P_{in}} \right) \sqrt{(C_{path} + B_{path} P_{in}) (P_{in} - P_{out})} \end{aligned}$$

If the flow reverses then a similar substitution can be done with the appropriate compatibility equation

$$\rho_{out} = \frac{C_V N_6 F_P B_{wave}}{A_{out} (P_{out} - C_{wave})} \left( 1 - \frac{1}{3 X_{choked}} + \frac{1}{3 X_{choked}} \frac{P_{out}}{P_{in}} \right) \sqrt{\rho_{in} P_{in} - \rho_{in} P_{out}}$$

Iterations are then performed by varying  $P_{out}$  (the pressure at the inlet of the connected pipe).

## Sonic Choking Transient Theory

[Restriction choking](#) can occur at any time at the valve junction for either a regular or exit type valve. If the valve becomes choked, the valve Cv/Kv equation for choked flow must be used.

$$\dot{m} = \frac{2 C_V N_6 F_P}{3} \sqrt{X_{choked} P_{in} \rho_{in}}$$

The mass flow term can be eliminated to find the equation in terms of pressure, density, and velocity only

$$\rho_{in} A_{in} V_{in} = \frac{2 C_V N_6 F_P}{3} \sqrt{X_{choked} P_{in} \rho_{in}}$$

The [wave and particle path compatibility equations](#) can then be substituted to eliminate density and velocity.

$$(C_{path} + B_{path} P_{in}) A_{in} \left( \frac{P_{in} - C_{wave}}{B_{wave}} \right) = \frac{2 C_V N_6 F_P}{3} \sqrt{X_{choked} P_{in} (C_{path} + B_{path} P_{in})}$$

Though not shown here, this equation can be rearranged into a polynomial and solved for static pressure. Once the static pressure is known the choked valve can be solved using the typical [transient sonic choking](#) calculation.

At the end of the calculation  $X$  will be recalculated. If  $X$  is greater than  $X_{choked}$  then the solution is valid. Otherwise, the valve is not choked, and the non-choked valve calculation are repeated using the choked valve solution to bracket the velocity solution.

## Related Topics

Nomenclature.....	410
Valve.....	263
Control Valve.....	247
Check Valve.....	230
Method of Characteristics.....	435
Sonic Choking.....	449
Sonic Choking Detailed Description.....	450
References.....	461

**Related Blogs**

[Making Sense of xT – Valve Loss in Compressible Flow](#)

**Resistance Curve Loss Model**

The resistance curve loss model is available in the General Component, Screen, and Venturi junctions. It is useful when measured flow vs loss data is available for a component.

It is not uncommon to obtain from an equipment manufacturer a single data point. For example, the data might be 3 psi pressure drop at 1 lbm/sec. Along with this data point, another always comes along for free. The second data point is zero pressure drop at zero flow.

The pressure drop characteristics of most equipment follows the square law. As the flow rate increases, the pressure increases with the square of the flow rate.

Therefore, in our example, an increase in the flow rate to 2 lbm/sec would result in a pressure drop of 12 psi. Although this may not be exactly correct, it is much better than assuming a flat 3 psi drop for all flow rates, which we know for certain is not correct.

The **Fill As Quadratic** button found in the Curve Fit window can be used to create a quadratic curve out of a single data point as was shown in the below example. To do this, just enter the single data point in the table, click the Fill As Quadratic, then click Generate curve Fit to finish creating the curve.

*Table 1: Example data points for pressure drop modeling using manufacturer data*

Data Point	Flow Rate (lbm/sec)	Pressure Drop (psid)	Comment
1	0	0	Always valid for passive equipment
2	1	3	Given by manufacturer
3	2	12	Extrapolated assuming square law

**Related Topics**

---

Curve Fitting .....	66
Troubleshooting Resistance Curves .....	524
General Component .....	251

## Defining Junction Transients

This section contains information related to defining transient behavior for junctions. Topics include discussions on standard junction transient information and transient initiation options. For information on overall transient simulation controls see the [Pipe Sectioning and Output group](#) and [Transient Control group](#).

### Contents

---

## Junction Transient Data

Many junction types have an area where active transient data can be specified. **Active transient** in this context means that the behavior of a component over time is being directly defined by the user. It is also possible for components to react **passively** during the simulation as well - for example, a relief valve opening under excess pressure or a finite tank changing pressure as mass flows into or out of it.

**Table 1: Junctions that support transient data**

Junction Type	Type of Transient
Assigned Flow	Flow rate/Temperature
Assigned Pressure	Pressure/Temperature
Infinite Tank	Pressure/Temperature
Finite Tank	Heat Rate/Volume
Valve	Loss/Open %
Heat Exchanger	Heat Rate
Compressor	Speed

The active transient data is entered on the junction's Transient tab.

## Defining Junction Transients

Valve Properties

Number: 7 Upstream Pipe: 12  
Name: Valve Downstream Pipe: None  
Library Jct: Elevation: 0 feet  
Copy Data From Jct...: OK  
Cancel  
Jump...  
Help

Loss Model **Transient** Optional Design Alerts Nojes Status

Transient Special Condition  
 None  
 Ignore Transient Data

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

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

Data Point	Time (seconds)	% Open
1	0	100
2	5	0
3		
4		
5		
6		

Xt Source  
 User  
 Open % Table  
Fill Xt from table

Edit Table Show Graph...  
 Repeat Transient  
Transient

**Figure 1: Transient tab for a Valve**

In most cases, transient data can be entered as absolute values or as a percentage of steady-state, as defined by the first options in Transient Data.

The first data point always needs to match the steady-state value which is usually entered on the Loss Model tab. xStream fills in that initial data point for the user, but it should always be checked.

Lastly, take note that several junctions require multiple inputs for transient data to be used. This is different from many liquid based transient solvers, and should be kept in mind when establishing transients in xStream.

---

**Note:** If a junction does not have any active transient behavior, there is no need to enter any data on the Transient Data tab.

---

### Repeat Transient

If the transient data is periodic, you can enter the data for one cycle of the period and then tick the check box for Repeat Transient. This will cause the one cycle of transient data to be repeated once it has reached the end. The repetition will continue until the end of the simulation.

### Transient Special Condition

There are instances where you will only wish to use the defined transient data in certain cases or scenarios. Instead of deleting the transient data, specify the Transient Special Condition as Ignore Transient Data. The data will be left in the window for future use, but will not activate during the run.

---

## Initiation of Transient

---

There are multiple ways that the data entered in Transient Data can be applied during the simulation. The default selection is Time for most junctions, meaning that the behavior vs simulation time is directly defined by the user.

It is also possible to initiate a transient based on a system event. This will execute the Transient Data when a user defined parameter reaches a certain setting. For example, a valve can be set to close only when a specific flow rate becomes too low.

See [Initiation of Transients](#) for detailed information.

---

## Related Topics

---

Assigned Flow.....	221
Assigned Pressure.....	223
Valve.....	263
Heat Exchanger.....	252
Compressor/Fan.....	232
Control Valve.....	247
Check Valve.....	230

---

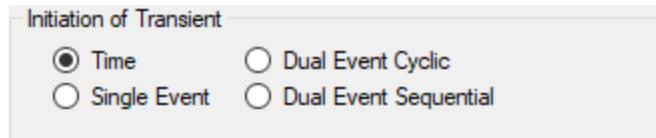
## Related Blogs

---

[And, Not, Or - Starting Transient Events with Complex Logic](#)

### Initiation of Transients

In general, transients can be initiated according to a certain time or according to certain events that happen in the system. These two categories are referred to as time based and event based transients. This is defined in the Initiation of Transient area of a junction's transient tab.



**Figure 1: Type of transient initiation**

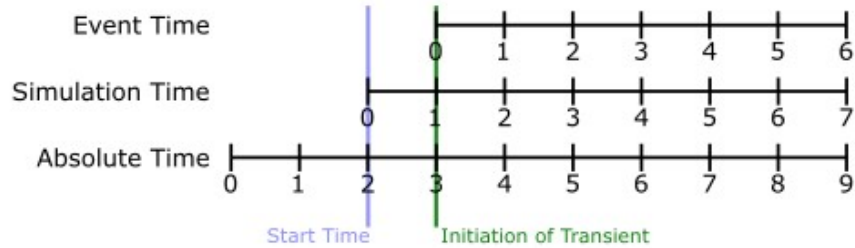
It is helpful to consider the different time scales that are used when referring to the initiation of a transient, as shown in Figure 2.

Absolute and Simulation Time are often identical. The simulation Start Time is set in the [Simulation Duration panel](#) to an Absolute Time of zero seconds by default, but it could be non-zero. In Figure 2, the Simulation Start Time is set to a value of 2 seconds – the entire Simulation Time timeline is shifted forward 2 seconds.



## Defining Junction Transients

Event Time refers to the time data defined in Transient Data this could be equal to Simulation Time (and Absolute Time) depending on how the event is defined. For example, all three time bases will match for a valve closure event starting at the beginning of a simulation with Start Time equal to zero. However, the event may not start until a certain system condition is met and zero Event Time could be well into the simulation.

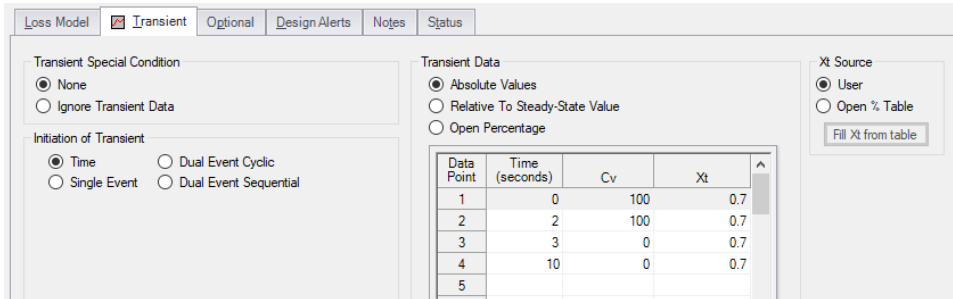


**Figure 2: Time basis**

## Time

Time based transients modify a parameter directly according to Simulation Time.

For example, consider a valve closure transient. The valve starts to close at two seconds into the simulation, and the time it takes for the valve to close is one second. After that, the valve stays closed. To define this event with the Time option, the Start Time must be set to zero, and the Transient Data must look something like the data shown in Figure 3.



**Figure 3: Defining a valve closure at a specific time**

## Single Event

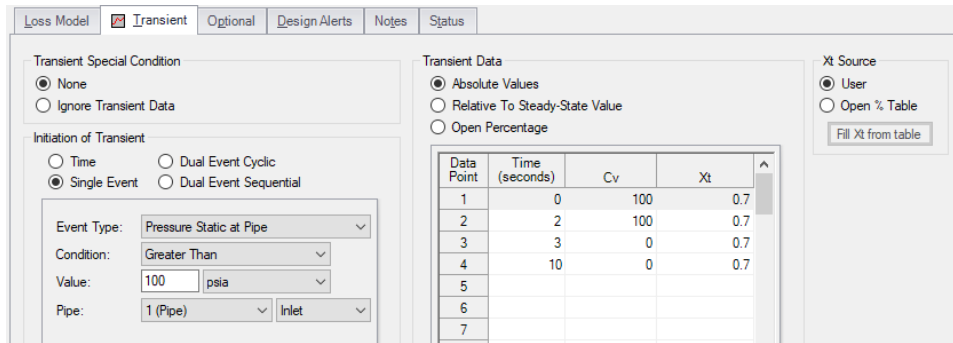
Single Event initiated transients do not begin until a certain condition is met.

To completely define an Event, four parameters are required:

## Defining Junction Transients

- **Event Type** - What parameter is being considered to initiate the transient. See Table 1 below for a list of the possible Event Types.
- **Condition** - The logical relationship between Event Type and Value (e.g. greater than, less than, etc)
- **Value** - The specific magnitude of the Event Type parameter at Location
- **Location** - Where in the model the Value should be observed

Given the same valve closure profile described in Figure 3, we can set the valve to remain open until (and if) the flow at the inlet of P1 exceeds 20 lbm/sec. Note that the Transient Data is identical - but the initiation of the event may happen at a very different time.



**Figure 4: Event definition**

**Table 1: Event types**

<b>** Multi-Condition **</b>	Pressure Static Diff. at Junction
Check Valve State	Pressure Static Diff. Between Pipes
Compressor/Fan Speed	Relief Valve State
Cv/Kv	Tank Pressure
Control Valve State	Time Absolute
Mass Flow Rate in Pipe	Time Difference
Pressure Stag. Diff. at Junction	Valve Open Percent
Pressure Stag. Diff. Between Pipes	Velocity in Pipe
Pressure Stagnation at Pipe	Volumetric Flow Rate in Pipe
Pressure Static at Pipe	

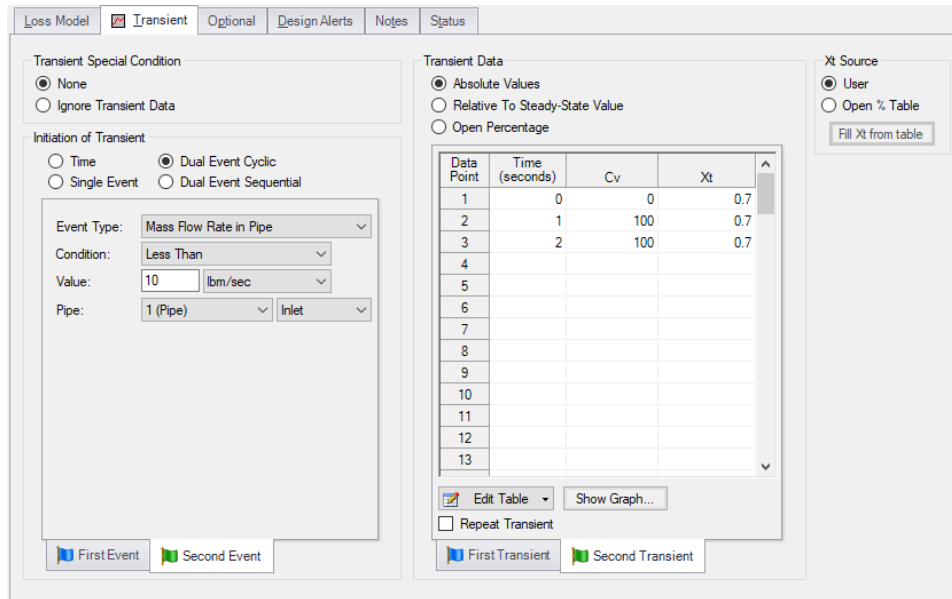
## Dual Event Cyclic

The Dual Event options, as the names imply, require the definition of two events. The definition for each event follows the same structure as a Single Event.

Dual Event Cyclic will continue monitoring both triggering events throughout the simulation and will apply the corresponding transient data when the conditions are met.

To continue with the valve example, we can define our valve not only to close when a flowrate is exceeded, we can tell it to open when the flow drops below a certain setting.

## Defining Junction Transients



**Figure 5: Dual Event set up. Note that a Second Event requires additional data for the Second Transient**

### Dual Event Sequential

Dual Event Sequential is nearly identical to Dual Event Cyclic. In fact, all of the required user input is the same.

The difference is that the two defined events are only applied once. That is – after the second event is complete, no more transients will be applied even if the conditions are met.

### Junctions With Inherent Event Logic

There are two junction types in AFT xStream which have built-in, or inherent, event logic. These are the check valve and relief valve. The user does not need to specify the nature of the events, and in fact is not allowed to.

The inherent event logic is very similar to the Dual Event Cyclic logic described previously.

- **Check valve** - The [check valve](#) has two built-in events. The first is that it closes when backflow starts to occur. The second is that it reopens when sufficient pressure differential occurs. These transients are assumed to be instantaneous.
- **Relief valve** - The [relief valve](#) has one built-in event. It opens when the set pressure is reached. This transient is assumed to be instantaneous.

### Related Blogs

---

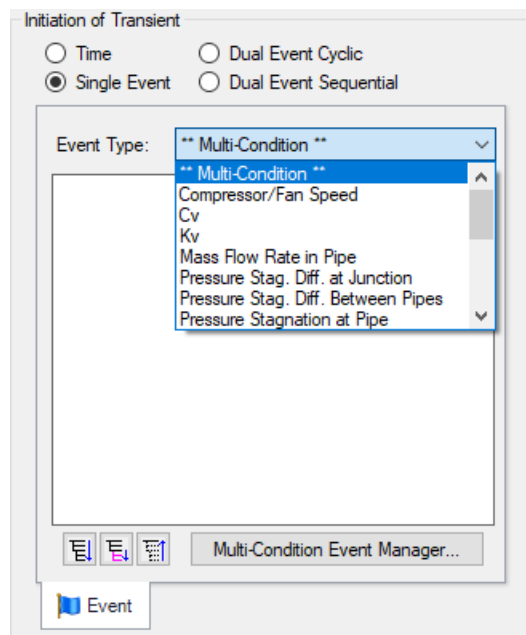
[And, Not, Or - Starting Transient Events with Complex Logic](#)

## Multi-Condition Events

One of the [Event Types](#) available is Multi-Condition. This feature allows starting a transient based on more than one parameter in the model.

For example, it may be desirable to open a valve when either of two pressures is exceeded.

To define a Multi-Condition Event, first select Multi-Condition as the Event Type from the junction's Transient tab.



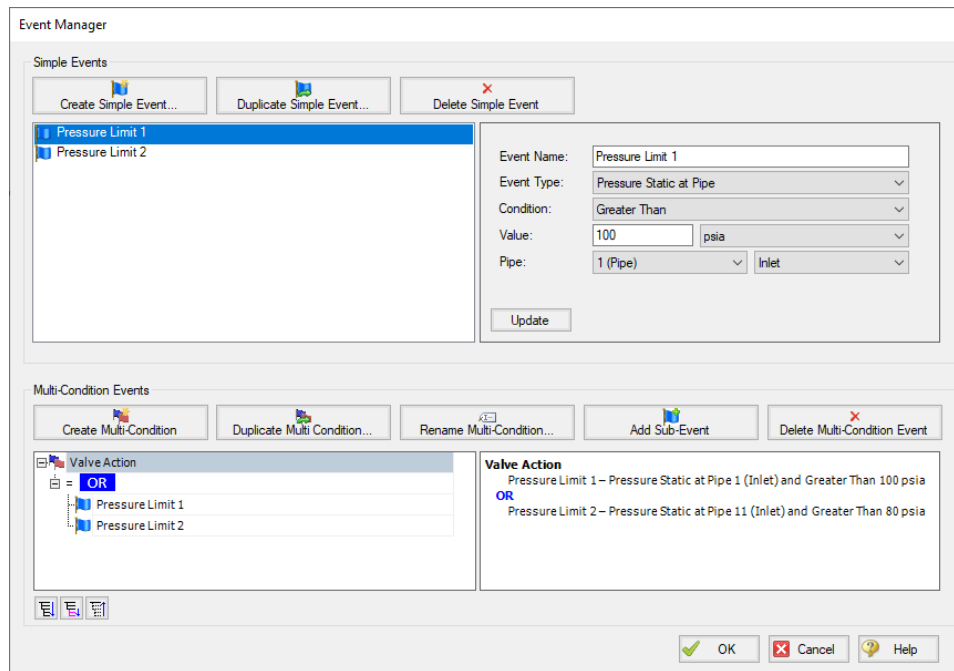
**Figure 1: Selecting Multi-Condition Event Type**

### Event Manager

---

Clicking Multi-Condition Event opens the Event Manager, where Simple Events and Multi-Condition Events can be defined.

The Event Manager defines events model-wide. That is, if the active scenario is changed, the same events will be accessible.



**Figure 2: The Event Manager**

### Simple Events

The Event Manager allows for the definition of Simple Events and Multi-Condition Events.

Simple Events are nearly identical to normal (non Multi-Condition) events defined in a Transient tab. They require a Parameter, Condition, Value, and Object as well as a unique Name.

Clicking "Create Simple Event..." adds a new Simple Event to the list with default values. These values should be updated with the desired values and applied with the Update button. Simple Events can be duplicated to help reduce repeated data entry.

### Multi-Condition Events

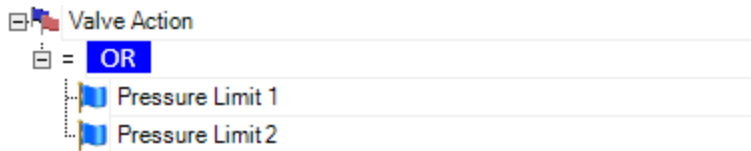
Multi-Condition Events are a collection of other Events with a logical condition applied to them. When creating a new Multi-Condition Event, a prompt for a Name appears. After entering a name, an empty Multi-Condition is created with a default logical condition of AND. This logical condition can be:

- **AND** - All of the Simple Event conditions must be met for the Multi-Condition Event to trigger.
- **NOT** - The Multi-Condition Event triggers when none of the conditions evaluate to true.
- **OR** - One or more of the Simple Event conditions must be met for the Multi-Condition Event to trigger.



**Figure 3: Conditional logic for Multi-Condition Events**

Any number of Simple Events can be added to a Multi-Condition Event as Sub-Events:

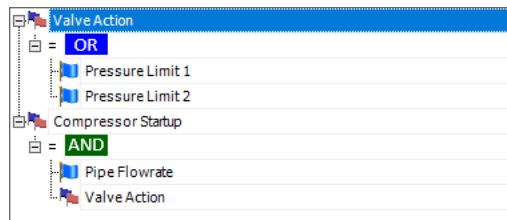


**Figure 4: Multiple Events**

## Complex Multi-Condition Events

---

More complex events can be defined by adding one or several Multi-Condition Events as Sub-Events to another Multi-Condition Event.

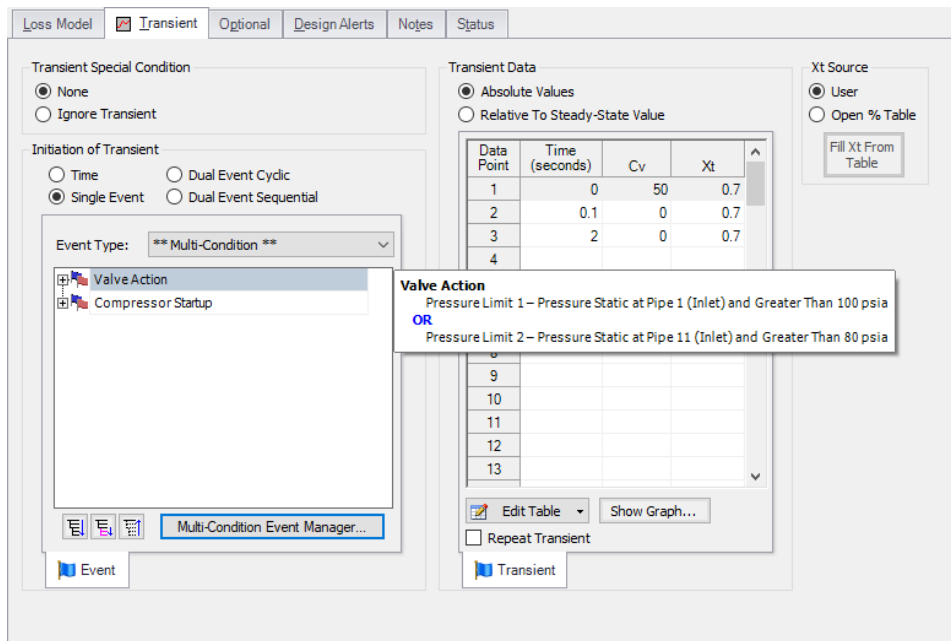


**Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))**

## Using Multi-Condition Events

---

All Multi-Condition Events are listed in the junction's Initiation of Transient section. Only one Multi-Condition Event can be selected as the Initiating Event for any particular Transient Data.



**Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event**

### Related Blogs

[And, Not, Or - Starting Transient Events with Complex Logic](#)

### Area Change

The Area Change junction type must always have two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs at the area transition between two pipes, whether expansion or contraction.

The Area Change Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the two connecting pipes.

The Area Change Properties window offers two standard types of area change losses: the Conical Transition and the Abrupt Transition.

The critical parameter that influences the magnitude of the area change loss is the area ratio between upstream and downstream pipes. Until you have input the information for the upstream and downstream pipes that allow for flow area determination, the standard Area Change junction loss factor is not calculated. For conical transition geometries, you also need to specify the conical angle. This angle has a range of 0-180 degrees, with a 0 degree angle representing a transition taking place over an infinitely long length, and a 180 degree angle representing an abrupt transition.

If the standard loss factors provided are inadequate for your application, you may specify your own custom loss factor referenced to the base area you specify.

**Note:** It is possible to have a vena contracta through a physical area change. If high velocities are possible, the Mach number may reach sonic conditions and the flow will choke. The optional CdA input parameter is intended to represent this vena contracta. The minimum possible sonic flow area is entered as the CdA and if sonic conditions are reached, AFT Arrow will solve the system on a [choked flow](#) basis.

## Related Topics

Common Junction Input Parameters..... 196

### Area Change Loss Model

Two standard area change loss geometries are available: the conical transition and the abrupt transition.

The conical expansion correlation ([Crane 1988](#), A26) is

$$K_{up} = 2.6 \left( \sin \frac{\mu}{2} \right) \left( 1 - \frac{A_{up}}{A_{down}} \right)^2 \quad \mu < 45^\circ$$

$$K_{up} = \left( 1 - \frac{A_{up}}{A_{down}} \right)^2 \quad \mu > 45^\circ$$

The conical contraction correlation ([Crane 1988](#), A26) is

$$K_{up} = \frac{0.8 \left( \sin \frac{\mu}{2} \right) \left( 1 - \frac{A_{down}}{A_{up}} \right)}{\left( \frac{A_{down}}{A_{up}} \right)^2} \quad \mu < 45^\circ$$

$$K_{up} = \frac{0.5 \left( 1 - \frac{A_{down}}{A_{up}} \right) \sqrt{\sin \frac{\mu}{2}}}{\left( \frac{A_{down}}{A_{up}} \right)^2} \quad \mu > 45^\circ$$

The abrupt expansion correlation ([Idelchik 2007](#), 246) is



$$K_{up} = \left(1 - \frac{A_{up}}{A_{down}}\right)^2$$

The abrupt contraction correlation ([Idelchik 2007](#), 254) is

$$K_{up} = \frac{0.5 \left(1 - \frac{A_{down}}{A_{up}}\right)^{0.75}}{\left(\frac{A_{down}}{A_{up}}\right)^2}$$

---

**Note:** While a 180 degree conical transition is the same as an abrupt transition, Crane's conical contraction equation gives a different result at 180 degrees compared to Idelchik's abrupt contraction equation. Therefore, it is up to the user to determine which one to use.

---

All K factor equations are provided here in terms of the upstream pipe. The K factor in terms of the downstream pipe can be found as follows:

$$K_{down} = K_{up} \left(\frac{A_{down}}{A_{up}}\right)^2$$

## Related Topics

---

Nomenclature.....	410
K Factor Loss Model.....	202
References.....	461

## Assigned Flow

The Assigned Flow junction type allows you to connect one pipe and to input a known steady-state and/or transient flowrate entering or leaving at a particular location.

The Assigned Flow Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipe information in a fixed format

The Assigned Flow junction type allows you to specify positive mass flow rates. Negative flow rates are not accepted for steady flow, although they are for transient flow. If, for example, you have an outflow

type junction and your system is physically flowing in, you cannot assign a negative flow rate to the junction. You must reverse the connecting pipe reference flow direction to be consistent with the actual direction and change from an outflow type to an inflow type.

### Transient Data

---

On the transient data tab a flowrate transient can be entered. A special feature is the ability to model a sinusoidal flowrate transient. Input for the amplitude and frequency is required. If data is present in the transient data table, the sinusoidal transient is summed with the data in the transient data table.

A chopped sine wave takes the absolute value of the sine function. The end effect looks like doubling the frequency in the positive domain only, and having nothing in the negative domain.

For more information on transient data, including event transients, see [Junction Transient Data](#).

### Special Conditions

---

You can set a [Special Condition](#) for an Assigned Flow junction, which will turn the flow off and make it act like a Dead End during steady-state.

### Related Topics

---

Common Junction Input Parameters.....	196
Junction Transient Data.....	210

### SSL Module

#### Settling Slurry Module

An optional add-on module to AFT Impulse which allows users to simulate settling slurries.

### Assigned Flow Transient Theory

With an assigned flow, the flow rate is known at the pipe inlet or outlet. The unknowns are then the velocity, the pressure, and the density.

If the flow rate is known at the upstream end of the pipe, the [negative wave compatibility equation is used](#) to determine the pressure based on an initial guess for velocity.

$$P_{i,new} = C_{\bar{wave}} + B_{\bar{wave}} V_{known}$$

The density can then be determined directly from the equation of state as a function of pressure and temperature. Once the pressure and density are calculated, the mass flowrate can be recalculated using the [continuity equation](#). If the difference between the defined and calculated mass flow rate is too large the velocity is adjusted and the process is repeated until the specified flow is met within a reasonable tolerance.

Conversely, if the flow is known at the downstream end, the [positive wave compatibility equation is used](#) to calculate the velocity based on an initial guess for pressure.

$$V_{i,new} = \frac{P_{i,new} - C_{wave}^+}{B_{wave}^+}$$

The density can then be found using the [particle path compatibility equation](#).

$$\rho_{i,new} = C_{path}^+ + B_{path}^+ P_{known}$$

Similar to the process above with a known inflow the mass flow can be recalculated using the [continuity equation](#), and the pressure will be iterated on until the calculated mass flow is within a reasonable tolerance of the specified mass flow.

---

## Sonic Choking Transient Theory

---

An outlet assigned flow junction can experience [endpoint choking](#) during the steady state, or can become choked during the transient due to transient events in the system. An inlet assigned flow junction cannot be choked in the steady state, but may experience choking during the transient as a result of rapid transients in the system. If sonic choking occurs at the assigned flow junction a different solution method must be enacted. See [Transient Sonic Choking](#) for more information.

---

## Related Topics

---

Nomenclature.....	410
Method of Characteristics.....	435
Sonic Choking Detailed Description.....	450

## Assigned Pressure

The Assigned Pressure junction allows for the specification of a fixed boundary pressure. The Assigned Pressure window follows the second of the two basic [Properties window formats](#). A table on the Loss Coefficients tab displays the connecting pipe information, and allows loss factors to be entered to account for losses due to fittings or components at the entry/exit to each of the connected pipes.

This junction is very similar to the Infinite Tank in that it fixes a pressure at a boundary. However, the Assigned Pressure allows the definition of either a static or stagnation pressure boundary, whereas tanks are always stagnation pressure. Typically, any pressure gauge on a pipe will be reporting static pressure - it is important to select static pressure if this is the actual boundary in the real system. A Tank would be inappropriate to represent such a piping connection with non-zero flow.

If stagnation is selected, the Assigned Pressure accepts the connection of up to twenty-five pipes. Static selection limits the connection to one pipe.

## Transient Data

On the transient data tab a flowrate transient can be entered. A special feature is the ability to model a sinusoidal flowrate transient. Input for the amplitude and frequency is required. If data is present in the transient data table, the sinusoidal transient is summed with the data in the transient data table.

A chopped sine wave takes the absolute value of the sine function. The end effect looks like doubling the frequency in the positive domain only, and having nothing in the negative domain.

For more information on transient data, including event transients, see [Junction Transient Data](#).

## Related Topics

Role of Pressure Junctions - Detailed Discussion (Long).....	480
Role of Pressure Junctions and How They Work.....	61
Tank.....	259
Common Junction Input Parameters.....	196
Junction Transient Data.....	210

## Related Online Content

- [Static vs. Stagnation Pressure Video](#)

## Assigned Pressure Transient Theory

With an assigned pressure, the pressure is known at the pipe endpoint. The unknowns are therefore the velocity, which can be obtained by solving the appropriate compatibility equation, and the density.

If the static pressure is known at the upstream end of the pipe, the [negative compatibility equation is used](#).

$$V_{i,new} = \frac{P_{known} - C_{wave}^-}{B_{wave}^-}$$

The density can then be determined directly from the equation of state as a function of pressure and temperature.

Conversely, if the static pressure is known at the downstream end, the [positive compatibility equation is used](#).

$$V_{i,new} = \frac{P_{known} - C_{wave}^+}{B_{wave}^+}$$

The density can then be found using the [particle path compatibility equation](#).

$$\rho_{i,new} = C_{path}^+ + B_{path}^+ P_{known}$$

If the known pressure changes with time, then a current pressure is obtained for the current time step and used in the equations.

If the known pressure is a stagnation pressure instead of the static pressure, additional complexities are introduced as the velocity must be known to convert the given stagnation pressure to a static pressure. Therefore, iterations must be performed using the corresponding compatibility equations as shown above combined with the [stagnation property relationships](#).

## Sonic Choking Transient Theory

---

An outlet assigned pressure junction can experience [endpoint choking](#) during the steady state, or can become choked during the transient due to transient events in the system. An inlet assigned pressure junction cannot be choked in the steady state, but may experience choking during the transient as a result of rapid transients in the system. If sonic choking occurs at the assigned pressure junction a different solution method must be enacted. See [Transient Sonic Choking](#) for more information.

## Related Topics

---

Nomenclature .....	410
Method of Characteristics .....	435
Sonic Choking Detailed Description .....	450

## Bend

The Bend junction type must always have two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs when the flow direction changes through a bend or elbow.

The Bend Properties window follows the first of the two basic [Properties Window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the two connecting pipes.

The Bend Properties Window offers three standard types of bend losses: the Smooth Bend, the Standard Elbow (the standard short radius threaded elbow), and the Mitre Bend. The window shows a schematic of the selected geometry. Until you have input the information for the upstream pipe that allows flow area determination, the standard Bend junction loss factor is not calculated. For smooth radius bends, you also need to specify  $r/D$ , which is the radius of curvature over the pipe diameter. At  $r/D$  less than one the Reynolds number must be known to calculate the loss factor, and therefore cannot be displayed until output is available.

For bends that are not at a 90-degree angle, you can enter the angle and the standard loss factor will be calculated.

If the standard loss factors provided are inadequate for your application, you may specify a custom loss factor referenced to the upstream pipe flow area.

**Note:** In principle, the upstream and downstream pipe diameters should be the same. However, the Bend junction only uses the reference area specified under Basis Area for Loss Model, even if the physical flow is in the opposite direction. If you are working with a user-specified loss, your geometry may have different diameter pipes connecting to the elbow. In this case, you would purposely define different upstream and downstream pipe areas. However, you would still want to be careful that the custom loss factor you assign is referenced to the correct area that is specified in Basis Area for Loss Model.

## CdA for Sonic Choking

As the fluid goes around the bend, it can separate from the wall. This results in a reduced physical flow area. When assessing sonic choking, it is this reduced area that is important. The optional CdA input parameter is intended to represent this reduced flow area. The minimum possible flow area is entered as the CdA and if sonic conditions are reached AFT xStream will solve the system on a [choked flow](#) basis.

## Bend Loss Model

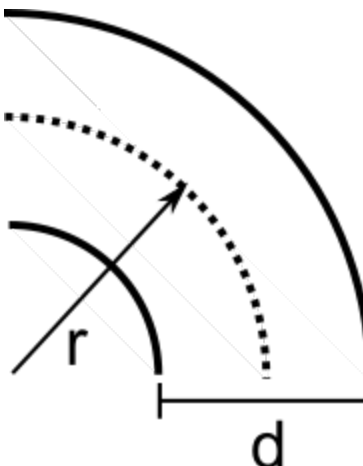
### 90 degree bends

The three [Bend](#) loss correlations are all for turbulent Reynolds numbers ([Crane 1988](#), A-29).

The K-factors for a **smooth flanged bend** with  $r/D \geq 1$  are provided in Table 1.

*Table 1: K-factors for 90 degree smooth flanged bends*

$r/d$	$K$
1	$20 f_T$
1.5	$14 f_T$
2	$12 f_T$
3	$12 f_T$
4	$14 f_T$
6	$17 f_T$
8	$24 f_T$
10	$30 f_T$
12	$34 f_T$
14	$38 f_T$
16	$42 f_T$
20	$50 f_T$



The diagram illustrates a 90-degree smooth flanged bend. It shows a quarter-circle arc with a solid outer boundary and a dashed inner boundary. The radius of the bend is labeled 'r' and the diameter of the pipe is labeled 'd'. An arrow points from the center of the bend towards the dashed inner boundary, indicating the flow path.

The **standard threaded elbow** is given by:

$$K = 30 f_T$$

where  $f_T$  is the turbulent friction factor given in Table 2.

**Table 2: Pipe friction factors used for Crane formulas**

Nominal Size	Friction Factor $f_T$
1/2"	0.027
3/4"	0.025
1"	0.023
1 1/4"	0.022
1 1/2"	0.021
2"	0.019
2 1/2", 3"	0.018
4"	0.017
5"	0.016
6"	0.015
8"-10"	0.014
12"-16"	0.013
18"-24"	0.012

The **Mitre bend** is given by:

$$K = 60 f_T$$

### Non-90 degree bends

---

The two non-90 degree bend loss correlations are for turbulent Reynolds numbers (Crane 1988, A-29).

A **smooth flanged bend** with  $r/D \geq 1$  is given by:

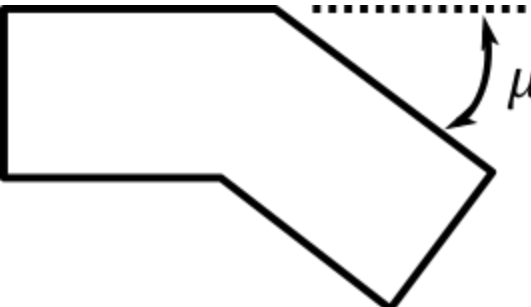
$$K_B = (n - 1) \left( 0.25 \pi f_T \frac{r}{D} + 0.5 K \right) + K$$

where  $n$  is the number of 90 degree bends,  $K$  is the loss factor for one 90 degree bend (Table 1), and  $f_T$  is given by Table 2.

The **Mitre bend** is given by Table 3.

Table 3: K values for Mitre bends

$\mu$ (deg.)	$K$
0	$2 f_T$
15	$4 f_T$
30	$8 f_T$
45	$15 f_T$
60	$25 f_T$
75	$40 f_T$
90	$60 f_T$



### Smooth Flanged Bends, $r/D < 1$

The smooth, flanged bend with  $r/D < 1$  loss factor uses a correlation applicable for turbulent Reynolds numbers (Miller 1990, 209)

The loss coefficient is calculated based on Miller's class 2 bend model, which is sufficient for general design purposes, and is given by

$$K_b = K_b^* C_{Re} C_0 C_f$$

$C_0$  and  $C_f$  are assumed to be 1 for class 2 bends.  $K_b^*$  is given by the nomograph on page 207 of Miller. For  $r/D > 0.7$  or  $K_b^* < 0.4$ ,  $C_{Re}$  is given by Miller Figure 9.3. Otherwise,  $C_{Re}$  is given by the equation below.

$$C_{Re} = \frac{K_b^*}{K_b^* - 0.2 (C_{Re} \text{ for } r/D = 1) + 0.2}$$

where the  $C_{Re}$  for  $r/D = 1$  is also found from Miller Figure 9.3.

### Related Topics

References..... 461

### Branch

The Branch junction allows up to twenty-five connecting pipes. The Branch junction is the most flexible junction type for building network models that have flow splits. The Branch junction must have at least two connecting pipes.



The Branch Properties window follows the second of the two basic [Properties window formats](#).

This junction will always be treated as lossless.

## Related Topics

Common Junction Input Parameters..... 196

### Branch Transient Theory

At a branch, there can be multiple pipes. Two additional relationships are needed, the conservation of mass and conservation of energy. The total mass flow in and out of the junction must sum to zero. The branch junction is treated as adiabatic, so the energy must sum to zero as well.

A constant stagnation pressure  $P_o$  is asserted at the branch since the branch is treated as lossless, meaning there is no stagnation pressure loss. By asserting a constant stagnation pressure the branch can essentially be treated as an assigned pressure boundary, though the "known" stagnation pressure must be iterated on.

Using the stagnation pressure solution the flow rate and density can be solved for with the appropriate compatibility equations as is done for the [Assigned Pressure Junction](#).

The total mass flow in can then be calculated

$$\dot{m}_{in} = \sum_{m=1}^M \rho_m V_m A_m$$

and the total energy flow is found using

$$\dot{e}_{in} = \sum_{m=1}^M \rho_m V_m A_m \left( h_m + \frac{V_m^2}{2} \right)$$

Using the above equations and enforcing energy conservation, the stagnation enthalpy can be found as follows

$$h_{o,out} = h_{o,in} = h_m + \frac{V_m^2}{2}$$

$$h_{o,out} = \frac{\dot{e}_{in}}{\dot{m}_{in}}$$

While the conservation of energy is inherently enforced in the solution using the above equation, conservation of mass is not inherently included. Mass conservation is instead achieved using iteration.

The total mass outflow can be written similar to the mass inflow summation above. The total mass outflow can then be calculated similar to the mass inflow using the compatibility equations, known stagnation pressure solution and known stagnation enthalpy.

$$\dot{m}_{out} = \sum_{k=1}^K \rho_k V_k A_k$$

An iterative method for stagnation pressure is used to drive the mass inflow and outflow difference to zero.

$$P_{o,new} = P_{o,old} - \frac{\Delta \dot{m}_{old}}{\frac{\partial \Delta \dot{m}}{\partial P_o}}$$

## Sonic Choking Transient Theory

---

[Expansion choking](#) can occur at a branch if the total cross-sectional area of the inflow pipes is less than the total cross-sectional area of the outflow pipes. If sonic choking occurs at the branch junction a different solution method must be enacted. See [Transient Sonic Choking](#) for more information.

## Related Topics

---

Nomenclature.....	410
Method of Characteristics.....	435
Sonic Choking Detailed Description.....	450

## Check Valve

The Check Valve junction type requires two connecting pipes. This junction type allows you to model the effect of a check valve which closes to prevent back flow.

The Check Valve Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Check Valve junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes for calculating loss. The user can define the base area for the loss model, though the default is set to use the upstream flow area.

A check valve is a device that allows flow in only one direction. AFT xStream assumes that the Check Valve is initially open (unless modeled as Closed on the Optional tab). If the flow solution indicates that the velocity at the check valve is insufficient to keep the check valve open based upon the check valve input, AFT xStream closes the valve. The valve can close during the steady-state or transient portion of the simulation.

AFT xStream allows you to model the check valve by entering a User Specified closing velocity.

The Loss Model for the open valve can be specified as a Cv, Kv or K factor by choosing an option on the Valve Model tab and entering the Full Open value.

An optional input in the Check Valve Properties window is the CdA. This parameter describes the effective area restriction in the valve for the purpose of calculating [sonic choking](#). In most cases, information on the CdA must be obtained from the manufacturer or from test data. Note that the CdA input field is disabled if Cv or Kv loss is used, as the Xt value is used for sonic choking calculations in that case.

### User Specified

---

The User Specified option opens and closes based on user-defined values for velocity to close and a delta pressure to reopen. For steady-state analysis the valve is either fully open, or fully closed. In a transient analysis, the valve opens and closes instantaneously.

The inputs are as follow:

- **Forward Velocity to Close Valve** - By default this value is zero. Some valves are designed to start closing at some reverse velocity - you can specify that here as a negative value. Entering a positive value in this field would imply that the check valve closes while the flow is still continuing forward.
- **Delta Pressure to Reopen** - Frequently after a check valve closes, there is pressure across the valve at which it will open again. You can specify that delta pressure here.

### Special Conditions

---

A check valve can have a [Special Condition](#) of None, Closed (stays closed or allowed to reopen in transient), or Open (stays open).

### Related Topics

---

- [Parameters Common to All Junction Properties Windows](#)
- [Junction Transient Data](#)
- [K Factor Loss Model](#)
- [Kv, Cv, and XT Loss Model](#)
- [Inherent Event Logic](#)

### Compressor/Fan

The Compressor/Fan junction allows you to model the pressure added to a system by a compressor, and model many transient events involving compressors/fans. Because all compressors/fans are different, AFT xStream does not provide any standard types. Pressure and flow data that describes the device must be obtained from the manufacturer or from test data. To reduce repeated data entry, frequently-used compressors/fans can be added to the [Junction Library](#).

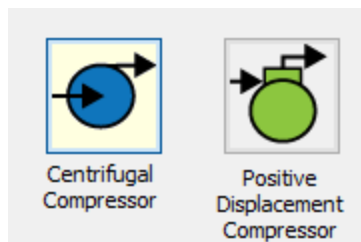
### Common Input Information

Like all junctions, Compressor/Fan junctions require a junction number, junction name, and inlet and outlet elevations. Similar to other junctions, compressors can have [Design Alerts](#) and [Notes](#) applied to them. They require two connecting pipes. Note that the direction of the connected pipes define which side of the compressor/fan is suction and which is discharge.

### Compressor Model

This tab requires the selection and definition of what type of compressor/fan the junction represents. There are two models currently available:

- [Centrifugal \(Rotodynamic\)](#) - Represents a typical centrifugal compressor where the head added to the fluid is related to the flow through the compressor via a compressor curve.
- [Positive Displacement](#) - Represents a steady state approximation of a positive displacement compressor. The flow is fixed to a constant value, and whatever head is necessary to obtain that flow will be supplied.



**Figure 1: Compressor/Fan Models**

### Variable Speed

This tab allows the speed of the compressor to vary. These options are only available for a centrifugal compressor.

- **Fixed Speed (%)** - Directly modify the compressor curve according to the [Affinity Laws](#). The Compressor/Fan Curve entered in [Compressor/Fan Configuration](#) is always assumed to be the 100%

speed curve.

### Transient

---

The Compressor/Fan junction allows for Speed vs Time transients. This allows the user to directly specify how the speed of the compressor/fan changes over time. This can be used to model startups, trips, or control transients.

For more information on transient data, including event transients, see [Junction Transient Data](#).

All of the transient options need to know when to start with [Initiation of Transient](#) and allow the definition of a [Transient Special Condition](#).

### Optional

---

There are several optional parameters that can be defined for a compressor/fan. Some of these are available to [most junctions](#) such as Display on Workspace, and Workspace Icon.

Also present are several compressor junction specific options:

- **Special Condition**
  - **None** - The compressor/fan operates normally.
  - **Compressor Off No Flow** - The compressor/fan is shut off and blocked. This is identical to a closed valve.
  - **Compressor Off With Flow Through** - The compressor/fan is shut off but allows flow to continue through. This is identical to a lossless connection.

Additionally, there is the option to insert the efficiency data of the compressor/fan motor.

### Related Topics

---

Common Junction Input Parameters.....	196
Junction Transient Data.....	210
Initiation of Transients.....	212

### Related Blogs

---

[Modeling Fans, Blowers and Compressors - Oh My!](#)

## Centrifugal Compressor

### Centrifugal Compressor Curve

A Compressor Curve is required to model a centrifugal compressor. This is the typical Head or Pressure vs. Flow information that will be available from any compressor manufacturer.

To enter these curves, click [Enter Curve Data](#). Once this data has been defined, previews of the applied curves can be seen here.

**Note:** AFT xStream does not have a means to account for stall or surge lines for the compressor. Therefore, it is possible that xStream will allow the compressor to operate at unrealistic conditions. It is advised that the user check the results to ensure that the compressor did not reach the surge or stall lines during the run, [design alerts](#) can be useful to assist with this check.

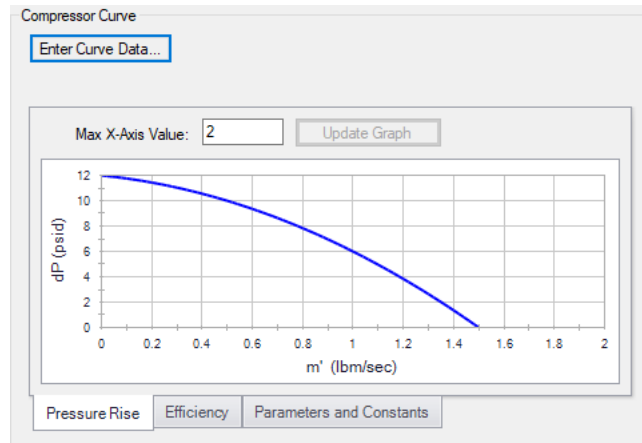


Figure 1: Standard Compressor Curve

### Added Pressure

If the compressor curve is entered using pressure rise information as opposed to a head rise, the pressure rise will need to be specified as either a stagnation or static pressure. This specification is performed on the Compressor Model tab under the Centrifugal Compressor specification. Note that mechanical energy rises are typically entered in xStream as pressure rises instead of head rises because the usefulness of the head term in variable-density systems is diminished.

### Compression Process Thermodynamics

In xStream the compressor will always be modeled as adiabatic and 100% efficient. The compressor efficiency is treated as an isentropic efficiency, so 100% efficiency in this case means the compressor

operation is isentropic. If the compression process is adiabatic, the temperature rise is given by the following isentropic relationship:

$$\frac{T_2}{T_1} = \left( \frac{P_2}{P_1} \right)^{(\gamma-1)/\gamma}$$

Note that these pressures and temperatures are static and not stagnation. Gamma is the isentropic expansion coefficient or specific heat ratio, depending on the [Fluid Accuracy Options](#).

**Note:** The [Compressor Configuration window](#) provides the option to enter efficiency option for the sole purpose of maintaining input when models are imported from other AFT products. The compressor efficiency data will not be used for calculation purposes.

### Related Blogs

[The Truth About Air Compressors](#)

[Modeling Fans, Blowers, and Compressors - Oh My!](#)

## Compressor/Fan Configuration

The compressor configuration window is used to define the specific operation of a centrifugal compressor. It is opened with the Enter Curve Data button from the [Compressor/Fan Properties](#) Compressor Model tab when [Centrifugal Compressor](#) is selected.

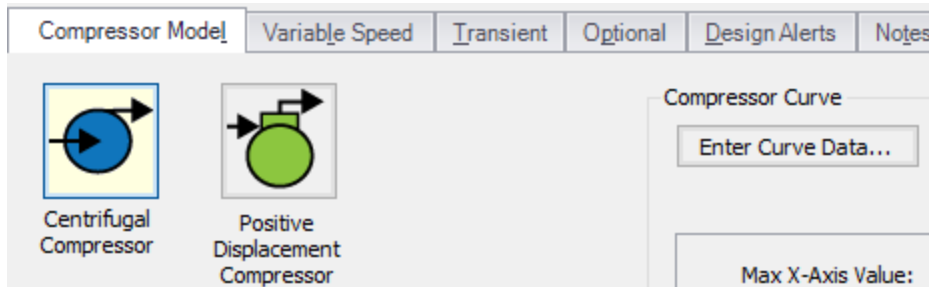


Figure 1: Enter Curve Data

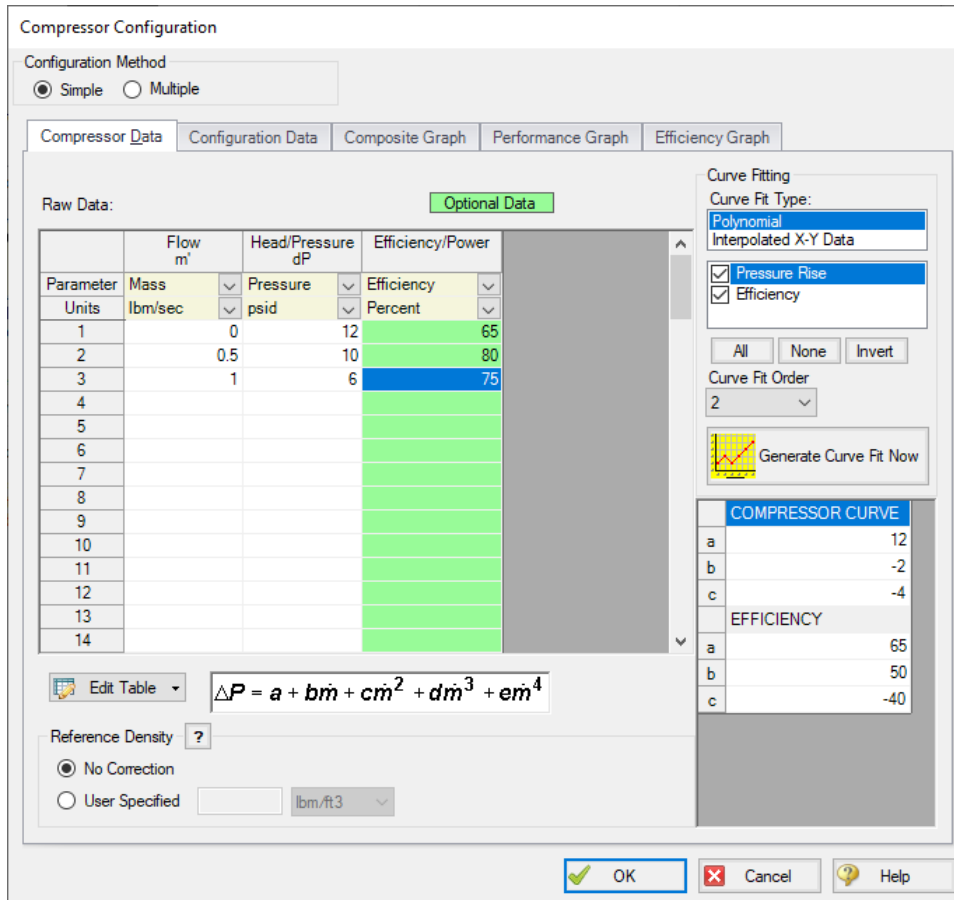
### Configuration Method

By default, a compressor/fan curve is defined using the Simple configuration. This means that the user inputs only one compressor/fan curve for the junction.

If the Configuration Method is changed to Multiple, additional input becomes available on the right of the configuration window. This allows defining different curves to represent the behavior of the compressor/fan at different speeds. See a detailed description in [Compressor/Fan Multiple Configuration](#). This feature is especially useful when multiple curves for a given compressor/fan junction must be evaluated as it prevents the user from having to delete and enter curve data.

### Compressor Data

The first tab of the compressor configuration controls the compressor's most basic behavior - the compressor curves.



**Figure 2: Compressor Data**

A compressor/fan can have two curves defined:

- **Flow vs. Head/Pressure/Compression Ratio**- Required. This is what is commonly referred to as the "compressor/fan curve" and defines how much pressure the turbomachine can add to a fluid at any given flowrate.



- **Flow vs. Efficiency/Power** - (Optional) Data entered in this column allows for quick assessment of power used at the operating point and the proximity to Best Efficiency Point (BEP).

After entering the raw values into the table, the data needs to be [curve fit](#).

If Mass Flow, Pressure, or Power are used in the curve definitions, [Reference Density](#) becomes an additional required input.

### Configuration Data

This tab controls some additional behavior or input for a compressor junction. Entering or changing values on this tab is optional.

The screenshot shows the 'Compressor Configuration' dialog box with the 'Configuration Data' tab selected. The 'Configuration Method' is set to 'Simple'. The 'Best Efficiency Point (BEP)' section has 'Calculate from Efficiency/Power Data' selected. The 'Preferred Operating Region (POR)' section has 'None' selected. The 'Allowable Operating Region (AOR)' section has 'None' selected. There are input fields for 'End Of Curve Flow Rate (optional)' in lbm/sec, and 'Low End' and 'High End' values for both POR and AOR, expressed as percentages of BEPQ or in lbm/sec. The dialog includes 'OK', 'Cancel', and 'Help' buttons at the bottom.

**Figure 3: Configuration Data**

- **Best Efficiency Point (BEP)** - The Best Efficiency Point (BEP) by default is calculated from the entered efficiency or power curve. This is generally the preferred option as it best relates to the true meaning of BEP. A User Specified value can be entered if no efficiency or power curve is available, or if it is desired to force the BEP to a certain flow.

## Defining Junction Transients

- **End Of Curve Flow Rate (optional)** - This value is used to create a warning in the output if it is exceeded. It does not affect the solution - the model will behave identically however this value is defined.
- **Preferred Operating Region (POR)** - % of BEP values falling outside of the defined range will generate a warning in the model output. This range can be defined by the user.
- **Allowable Operating Region (AOR)** - Similarly, this flow range creates a warning in the output when the user specified flow range is not adhered to.

## Graphs

The following tabs display the curves that have been fit to the given data. If multiple curves are defined, the Composite Graph tab becomes available. The other graph tabs show the individual curves, along with points representing the raw data. It is a good idea to check that these graphs to confirm the curve fit is reasonable for any potential operating region.

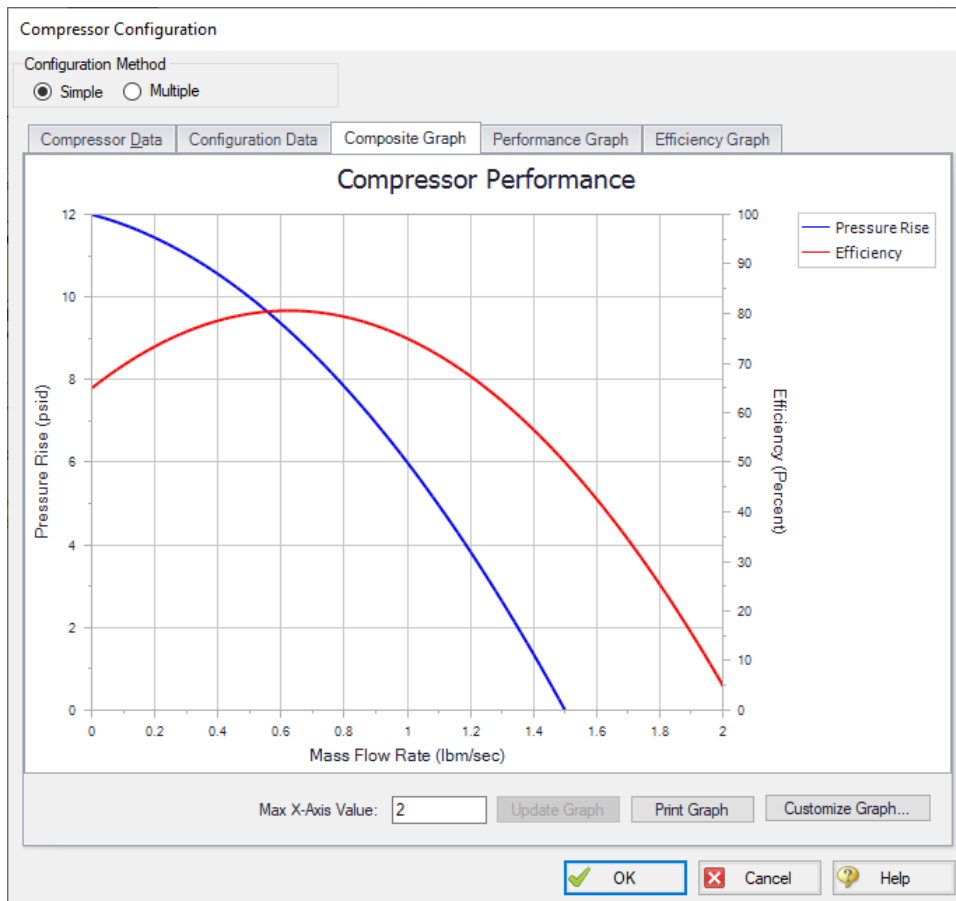


Figure 4: Composite Graph

## Related Topics

Curve Fitting.....	66
Reference Density.....	198

### Related Blogs

---

[Modeling Fans, Blowers and Compressors - Oh My!](#)

### Positive Displacement Compressor

Positive Displacement compressors/blowers can be considered to provide a fixed flow for steady state analysis. The positive displacement compressor/blower allows for a fixed flow to be defined in a straight-forward manner.

To model a positive displacement compressor/blower, select the Positive Displacement option on the Compressor Model tab of the [Compressor/Fan Properties window](#).

### Parameter

---

- **Volumetric Flow Rate** - Fix the volumetric flow rate through the compressor/blower.
- **Mass Flow Rate** - Fix the mass flow rate through the compressor/blower.

### Fixed Condition

---

- **Fixed Flowrate** - The value and unit of the fixed flow amount.
- **Nominal Efficiency** - A fixed efficiency, useful to calculate [power requirements](#). The positive displacement compressor thermodynamics will be treated as adiabatic regardless of the defined efficiency value.

## Defining Junction Transients

The screenshot shows the 'Compressor/Fan Properties' dialog box. At the top, there are fields for 'Number' (3), 'Name' (Compressor/Fan), 'Library Jct.', and 'Copy Data From Jct.'. To the right, there are fields for 'Upstream Pipe' (1), 'Downstream Pipe' (2), and 'Elevation' (0 feet). On the far right are buttons for 'OK', 'Cancel', 'Jump...', and 'Help'. Below these fields is a tabbed interface with 'Compressor Model' selected. Under 'Compressor Model', there are two icons: 'Centrifugal Compressor' and 'Positive Displacement Compressor'. The 'Positive Displacement Compressor' is selected. Below the icons are options for 'Added Pressure' (Stagnation or Static) and 'Compression Process Thermodynamics: Adiabatic'. A red caution message reads: 'Caution: User efficiency data is used only for Steady State reporting - the Compressor/Fan junction is considered adiabatic during the simulation.' To the right of the icons is a 'Positive Displacement Compressor' configuration panel. It has a 'Parameter' section with radio buttons for 'Volumetric Flow Rate' and 'Mass Flow Rate' (selected). Below that is a 'Fixed Condition' section with 'Fixed Flow Rate' (0.7 lbm/sec) and 'Nominal Efficiency' (90 Percent (optional)).

**Figure 1: Positive Displacement Compressor/Blower Model**

## Related Blogs

[Modeling Fans, Blowers and Compressors - Oh My!](#)

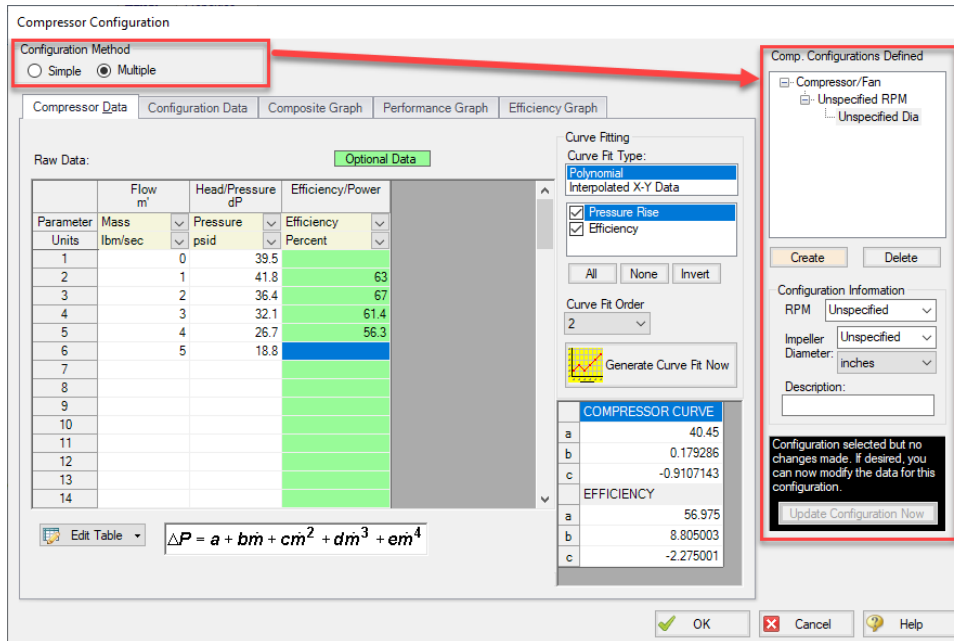
## Multiple Configurations

A single [compressor junction](#) can contain information for multiple compressor configurations - impeller trims and operating speeds. This is useful to compare different variations of a compressor without needing to redefine a new compressor junction for every case. The configuration of the compressor can easily be changed without losing any information.

## Defining Multiple Configurations

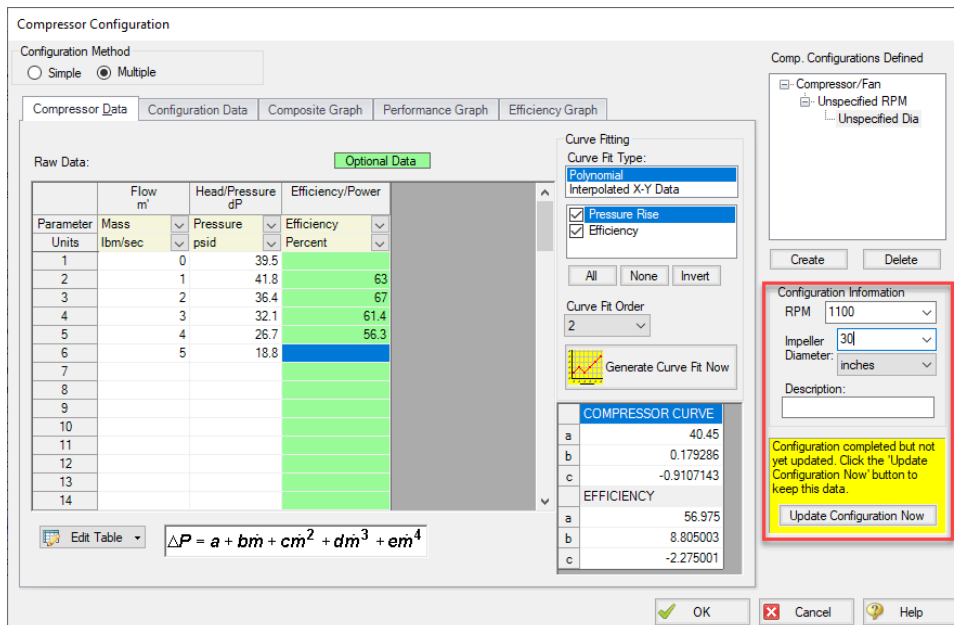
When opening the [compressor configuration](#), change the Configuration Method from Simple to Multiple. When doing so, an additional user input panel appears on the right hand side of the window.

## Defining Junction Transients



**Figure 1: Multiple Configuration Method Selected**

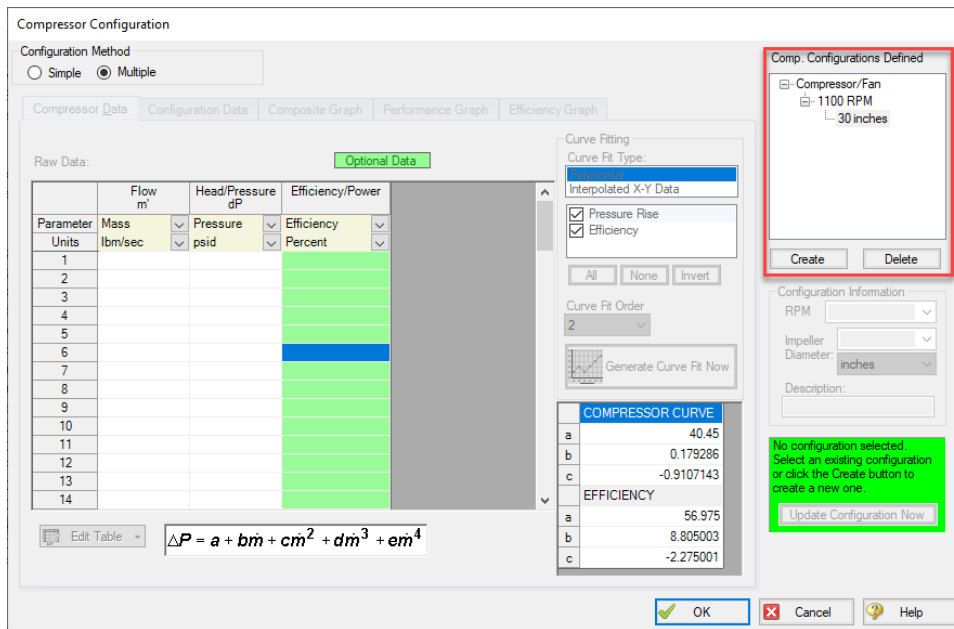
In order to make use of the multiple configuration feature, an RPM and Impeller Diameter under Configuration Information should be filled in. A Description can optionally be added. After modifying any information that affects the definition of this configuration the Update Configuration box turns yellow:



**Figure 2: Update Configuration**

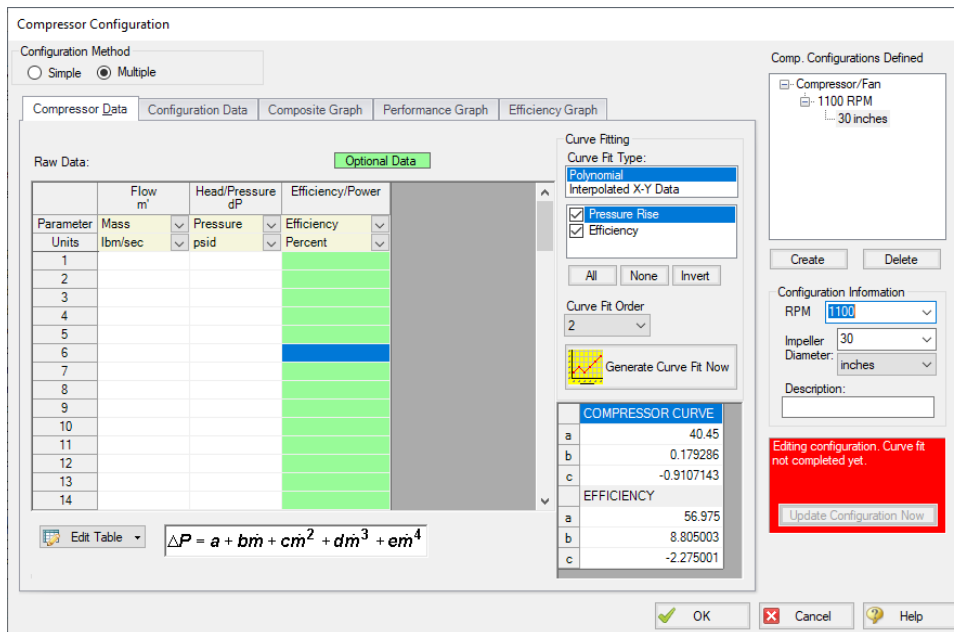
## Defining Junction Transients

Clicking Update Configuration Now adds the Configuration to the Compressor Configurations listed in the top-right.



**Figure 3: Current Compressor Configurations**

To create an additional configuration, click Create.



**Figure 4: Creating a new Configuration**

## Defining Junction Transients

The compressor curves must be created and curve fit, as with a single configuration compressor. After completing this and clicking Update Configuration Now, the new configuration will be created and shown in the Compressor Configurations area. Configurations can be edited by selecting them in this area

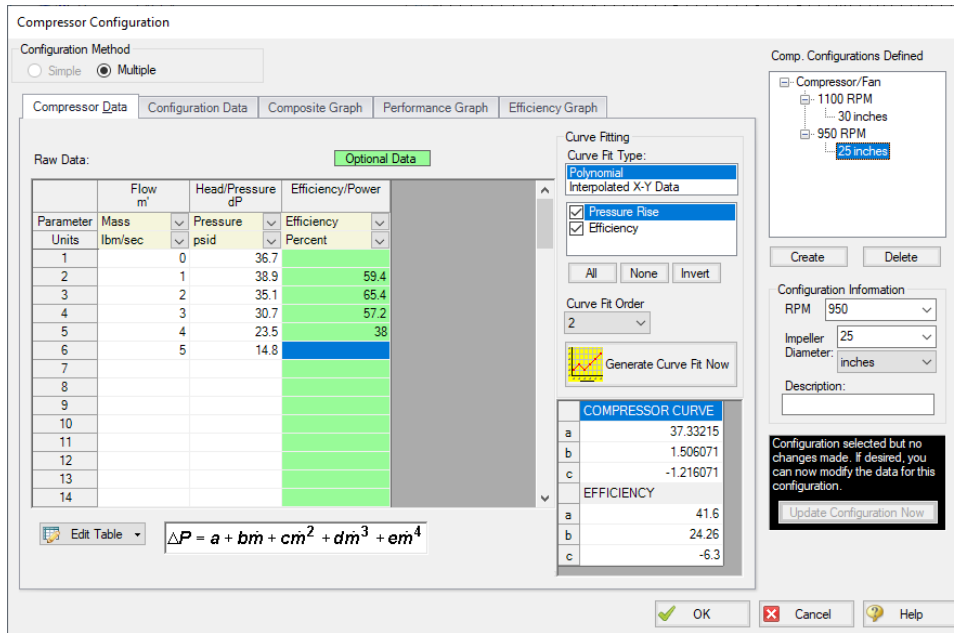


Figure 5: Editing a Configuration

Now that multiple configurations are available, they can be selected on the Compressor Model tab:



Figure 6: Selection of a Configuration

## Affinity Laws

Compared to compressors, it is more common for fans or blowers to operate with variable speeds (adjustable in the Variable Speed tab in the Compressor/Fan Properties window). Affinity laws (also called homologous laws) allow an estimation of the compressor/fan performance at speeds other than the design speed.

Before discussing the affinity laws, it should be noted that the affinity laws are an approximation. In many cases, compressor/fan manufacturers will have performance data for a number of speeds. These data should be used if available. When not available, the affinity laws can be used with an understanding that an approximation is being made. The affinity laws are known to have much lower accuracy for compressors and blowers than for fans.

Compressor/fan curves can generally be curve fit to a polynomial:

$$\Delta H = a + bQ + cQ^2 + dQ^3 + eQ^4$$

The speed ratio is defined in terms of the design impeller speed,  $N_1$  and the desired operating speed  $N_2$  as:

$$s = \frac{N_1}{N_2}$$

### Affinity Laws

---

For the following ratios, subscript 1 represents the parameter from the defined curve (assumed to be at 100% speed). Subscript 2 represents the given parameter at the desired operating speed  $N_2$ .

- **Flow rate -**

$$\frac{Q_1}{Q_2} = s$$

- **Compressor/fan head -**

$$\frac{\Delta H_1}{\Delta H_2} = s^2$$

- **Power consumption -**

$$\frac{P_1}{P_2} = s^3$$

- **Efficiency -**



$$\frac{\eta_1}{\eta_2} = 1$$

### Application to Transients

---

During a transient event, the speed of the compressor/fan may change. In such cases, the affinity laws are used to adjust the head/pressure curves appropriately for the new speed. This is a process that often requires additional iteration.

### Related Blogs

---

[To Affinity and Beyond!](#)

### Transient - Speed vs. Time

Certain configurations of a [compressor](#) allow the direct specification of Speed vs. Time.

### Transient Data

---

For Speed vs. Time, the variation of speed must be directly defined in Transient Data. The speed of the compressor will be forced to meet this data at each time step of the transient simulation, with the compressor curves being adjusted every time step according to the [affinity laws](#).

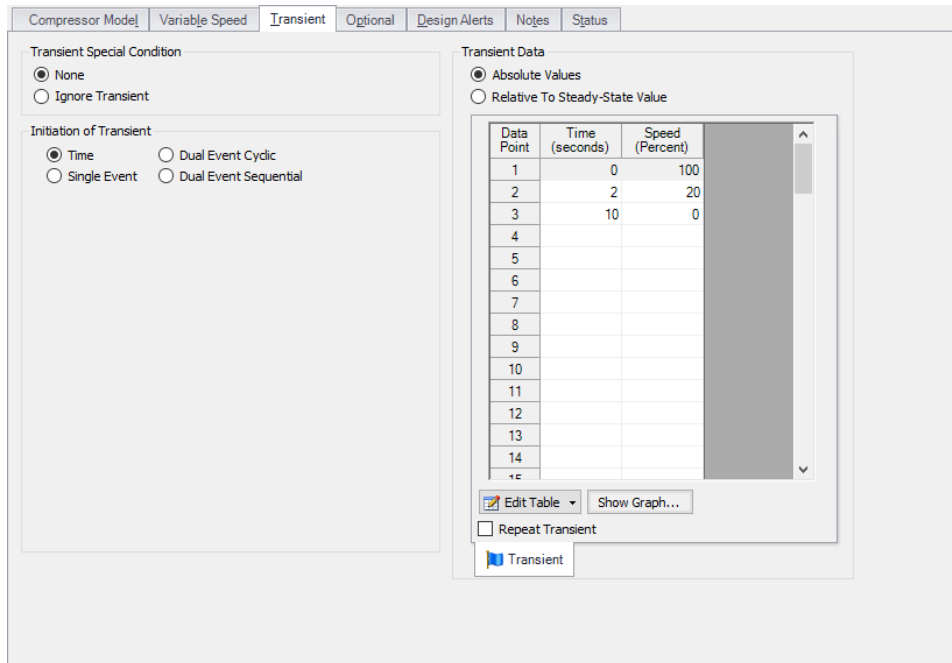


Figure 1: A Speed vs. Time transient

## Related Topics

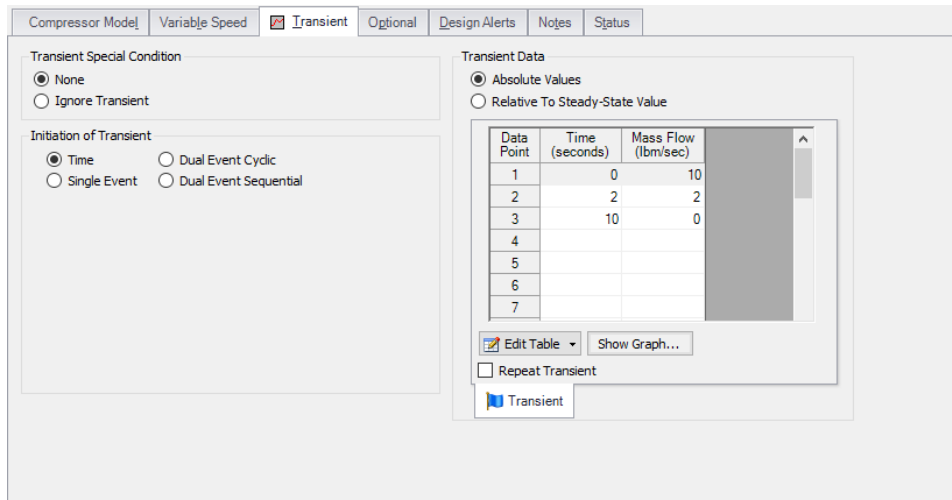
Initiation of Transients.....212

### Transient - Setpoint vs. Time

When modeling a [Positive Displacement Compressor](#) with a steady setpoint, it is possible to vary the setpoint over time. This replaces the [Speed vs. Time](#) transient as defining both the speed and the setpoint would over constrain the system.

## Transient Data

For Setpoint vs. Time, the variation of the setpoint must be directly defined in Transient Data. The selected parameter will be forced to meet this data at each time step of the transient simulation. The resulting speed variation over time will be visible in the output.



**Figure 1: A Setpoint vs. Time transient**

## Related Topics

Initiation of Transients.....212

## Control Valve

Control Valve junctions are always internal to the system, with two connecting pipes. This junction type allows modeling of valves that offer special pressure or flow control characteristics at a location in the pipe system.

The Control Valve Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Control Valve junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

## Control valve types

xStream offers four types of control valves: Pressure Reducing Valves (PRVs), Pressure Sustaining Valves (PSVs), Flow Control Valves (FCVs), and Pressure Drop Control Valves (PDCVs). Loss information for a control valve is not required, because control valves are dynamic devices that change their geometry in response to the pipe system behavior. The loss that results is that required to maintain the control parameter. A Loss When Fully Open can be specified, and is the loss that will occur should the valve fail to a full open state.

- A **PRV** is a device that controls the pressure in a pipe system. The PRV maintains a constant control pressure downstream of the junction as long as the upstream pressure exceeds the control pressure. If the upstream pressure is lower than the control pressure, the ability to control pressure is lost.

- A **PSV** is similar to a PRV in that it controls pressure in a pipe system. While the PRV maintains a constant downstream pressure, the PSV maintains a constant upstream pressure. If the downstream pressure rises higher than the control pressure, the ability to control pressure is lost.
- An **FCV** is a device that maintains a constant flow rate in a pipe system. By setting the junction to an FCV type and entering a flow rate, the junction will limit the flow through the connecting pipes to be equal to the control flow rate. The FCV can lose its ability to control flow when the pressure drop across it becomes zero or backward flow begins.
- A **PDCV** is a device that maintains a constant (stagnation) pressure drop. For this option, the valve is forced to always meet its setpoint. An indicator that an unrealistic pressure drop has been demanded is a failure to obtain a converged solution.

### PRV/PSV Static vs Stagnation Pressure

---

The control pressure for a PRV or PSV can be either [static or stagnation](#) . The default selection is static, as this is the most frequent application in industry.

### Loss When Fully Open

---

By default, the valve will be set to model the valve with no loss if it fails open. This can be changed by choosing a loss model on the Control Valve Model tab to use Cv, Kv, or a K factor. When Cv, or Kv, is selected, the option becomes available to pull the value from an open percentage table on the Optional tab.

### Open Percentage Table

---

The Optional tab allows data to be entered for special control valve characteristics. Specifically, the valve Cv and Flow Area can be Specified vs. the Open Percentage of the valve. This data does not affect AFT xStream's flow solution. However, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary (in the Output window) always displays the valve Cv, and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point.

AFT xStream can assist in creating this Cv vs Open percent curve. This is accessible in the optional tab by selecting "Edit Table" and then "Create Cv vs. Open Percent...". This then opens a window in which a user defined, linear, [equal percentage](#), or Pre-defined curve can be defined and transferred to the Cv vs. Open Percentage table.

### Special Conditions

---

Control valves have two [special conditions](#): Fully Open - No Control, or Closed.

### Action if Setpoint Not Achievable

---

It is possible to build a system where the control valve cannot control the system to the specified setpoint, either in the steady-state or during the transient analysis.

When the control valve cannot meet the specified setpoint in the steady-state system, its behavior will depend on why it could not control the system. If the valve was fully open and would need to open further to meet the setpoint, the valve will open fully and lose control. An example of this situation is a FCV with a setpoint of 100 scfm. If the valve sees 90 scfm at 99.9% open, it cannot control to 100 scfm, will lose control, and open fully. In the opposite direction, if the valve was fully closed and would need to close further to meet the setpoint, the valve will close fully and lose control. An example of this situation is a PSV with a setpoint of 3 atm. If the valve sees 2.5 atm at 0.1% open, it cannot control to 3 atm, will lose control, and close fully.

In a transient simulation, a control valve can regain control if system conditions change such that the control valve can again control to its setpoint.

### Related Topics

---

Common Junction Input Parameters.....	196
K Factor Loss Model.....	202
Cv/Kv and XT Loss Model.....	205
Junction Transient Data.....	210

### Related Blogs

---

[Get in Control with Control Valves](#)

#### Control Valve Transient Theory

We can use the methods for known flow or pressure to calculate the conditions at a control valve. AFT xStream flow and pressure control valves allow the control setting to be changed with time. Whether constant or changing, the flow or pressure is known and can be used to solve the equations.

#### Flow Control Valves

---

With the known flowrate, one can substitute into the [Compatibility Equations](#) and solve for the upstream and downstream pressures, respectively, at the valve. The valve loss solution can then be compared against the user-specified loss range to determine if the valve can control to the setpoint for that time step.

Once the valve loses control, it acts like a regular [valve](#). However, it is possible for the control valve to later regain control if the upstream pressure raises sufficiently.

## Pressure Control Valves

---

Pressure control can be specified on the upstream or downstream side of the valve, or as a differential pressure across the valve. All three control types use identical approaches to find a solution at each time step which satisfies the control setpoint.

An iterative approach is required to find a solution that meets the controlled parameter. A flowrate through the valve is guessed at, the positive characteristic is used to determine the upstream pressure, and the negative characteristic used to determine the downstream pressure. The upstream or downstream pressure is compared against the valve setpoint, and further iterations are used to find a flow solution which meets the setpoint.

If the valve is fully open or fully closed yet cannot meet the control setpoint, the valve will lose control and behave as a regular [valve](#). It is possible for the control valve to later regain control if the upstream pressure raises sufficiently. Note that the Pressure Drop Control Valve will never lose control and open fully, as there must be a control pressure loss by definition.

## Related Topics

---

Nomenclature .....	410
Method of Characteristics .....	435

## Dead End

The Dead End junction must always have one connecting pipe. This junction type is one way to define flow in a pipe as truly zero.

In steady flow, dead ends have no flow and thus play no role in the solution to the system. However, in transient flow pipes that terminate in dead ends can have flow. Thus dead ended lines can and do influence the transient response of the system. Unless you know otherwise for a specific system, dead ends should always be included in the model.

The Dead End Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The only required information is the connecting pipe and the elevation.

## Related Topics

---

Common Junction Input Parameters .....	196
--	-----

## Dead End Transient Theory

A dead end is similar to an [assigned flow](#), except the flow rate is zero ([Wylie, et al., 1993](#), pp. 44). The unknown is then the pressure, which can be solved by calling the appropriate compatibility equation. If the dead end is at the upstream end of the pipe, the [negative wave compatibility equation is used](#) with zero flow.

$$P_{i,new} = C_{wave}^-$$

Conversely, if the dead end is at the downstream end, the [positive wave compatibility equation is used](#) with zero flow.

$$P_{i,new} = C_{wave}^+$$

The path characteristic becomes completely vertical, but is still represented by the [particle path compatibility equation](#), using the pressure from the appropriate wave compatibility equation.

$$\rho_{i,new} = C_{path}^+ + B_{path}^+ P_{i,known}$$

## Related Topics

---

Nomenclature.....	410
Method of Characteristics.....	435
References.....	461

## General Component

The General Component junction type is internal to the system and requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through equipment that may not be explicitly represented on the [Toolbox](#) and to specify loss factors as a function of a flow parameter.

The General Component Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

When you select [Resistance Curve](#), the General Component Properties Window makes additional features available. You can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT xStream modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

No transient data can be entered for a General Component. From a transient point of view, the junction is a static element.

An optional input in the General Component Properties window is the CdA. This parameter describes an effective area restriction for the purpose of calculating [sonic choking](#). In most cases, information on the CdA must be obtained from the manufacturer or from test data.

## Related Topics

---

Curve Fitting.....	66
Common Junction Input Parameters.....	196
K Factor Loss Model.....	202
Resistance Curve Loss Model.....	208
Troubleshooting Resistance Curves.....	524
Reference Density.....	198

### Heat Exchanger

The Heat Exchanger junction type is internal to the system and requires two connecting pipes. This junction models the irrecoverable loss that occurs through heat exchanger equipment along with a known heat rate added to or removed from the fluid.

The Heat Exchanger Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

### Heat Transfer

---

On the Thermal Data tab you can define a fixed heat rate into the system.

Note that similar to flow rate definition, heat flow is positive when flowing into the system, and negative when flowing out.

### Transient Data

---

On the Transient Data tab, a heat rate transient can be entered. Note that if there is a heat rate and flow through the junction reaches zero (either because flow is halted or because flow reverses), the solution will fail as heat can no longer flow out of the junction, causing heat to accumulate infinitely. For more information on transient data, including event transients, see [Junction Transient Data](#).

### Special Conditions

---

The [Special Condition](#) for a Heat Exchanger junction functions differently than other junctions. Here you can turn off the heat transfer for the heat exchanger.

### Related Examples

---

Gas Turbine Fuel System Example.....	575
--------------------------------------	-----

### Related Topics

---

Curve Fitting.....	66
--------------------	----



Common Junction Input Parameters.....	196
K Factor Loss Model.....	202
Junction Transient Data.....	210
Reference Density.....	198

## Orifice

The Orifice junction type requires two connecting pipes, unless it is defined as an exit orifice, in which case only one connecting pipe is allowed. This junction type models the irrecoverable loss that occurs through a plate-type flow restriction. It also allows the specification of loss factors as a function of a flow parameter and an orifice loss for pipes with an area change.

The Orifice Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Orifice junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

Orifices that are located at exit points are specified as exit orifices. These orifices require an associated back pressure definition. Exit orifices can only be connected to a single upstream pipe. To specify an exit orifice, select the Exit Orifice checkbox on the Loss Model tab. Then specify the exit pressure and temperature.

## Orifice Types

AFT xStream provides two standard orifice types: Sharp-Edged and Round-Edged. These loss factors are calculated based on correlations. In addition to the standard orifices that AFT xStream provides, users can enter custom orifice loss factors.

The constant loss characteristics of an orifice can be specified as a [discharge coefficient \(Cd\)](#) or as a [loss factor \(K\)](#). The standard orifice geometries include a functional dependence on the upstream and downstream pipe areas, which may be different.

An optional input in the Orifice Properties window is the CdA for Sonic Choking. This parameter describes the effective area restriction in the orifice for the purpose of calculating [sonic choking](#). In most cases, information on the CdA must be obtained from handbook or test data.

---

**Note:** The  $C_dA$  for sonic choking may be different from the subsonic  $C_dA$  loss model option in xStream. The discharge coefficient can vary at different pressure ratios due to the vena contracta moving closer to or farther from the orifice restriction. For the highest accuracy the  $C_dA$  used for subsonic and sonic losses should be tested and entered separately. See the "[Modeling Choked Flow Through an Orifice](#)" white paper on AFT's website for more information.

---

## Related Topics

Common Junction Input Parameters.....	196
Discharge Coefficient Loss Model.....	201
Subsonic vs. Sonic Pressure Losses.....	454

## Orifice Loss Model

### Sharp-Edged Orifice

The orifice loss factors are all for turbulent Reynolds numbers. The sharp-edged orifice shown in Figure 1, is given by the following equation (Idelchik 2007, 259):

$$K_{up} = \left( \frac{A_{up}}{A_{orifice}} \right)^2 \left[ 0.707 \left( 1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375} + \left( 1 - \frac{A_{orifice}}{A_{down}} \right) \right]^2$$

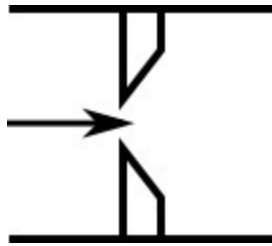


Figure 1: Sharp-edged orifice (in this case  $A_{up} = A_{down}$ )

### Round-Edged Orifice with Different Upstream and Downstream Diameters

When the upstream and downstream pipe diameters are different for a round-edged orifice as shown in Figure 2, the round-edged orifice equation for different pipe areas is used and is given by the following (Idelchik 2007, 258)

$$K_{up} = \left( \frac{A_{up}}{A_{orifice}} \right)^2 \left[ K' \left( 1 - \frac{A_{orifice}}{A_{up}} \right)^{0.75} + \left( 1 - \frac{A_{orifice}}{A_{down}} \right) \right]^2 + 2 \left( 1 - \frac{A_{orifice}}{A_{down}} \right) \sqrt{K' \left( 1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375}}$$

where

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge} / D_{orifice}}$$

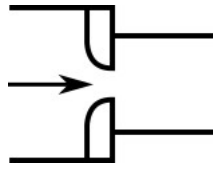


Figure 2: Round-edged orifice with different upstream and downstream pipe areas

### Round-Edged Orifice with Identical Upstream and Downstream Diameters

When the upstream and downstream pipe diameters are the same for a round-edged orifice as shown in Figure 3, the **round-edged orifice equation for the same pipe areas** is used and is given by the following ([Idelchik 2007,262](#))

$$K_{up} = \left( \frac{A_{up}}{A_{orifice}} \right)^2 \left[ 1 - \frac{A_{orifice}}{A_{up}} + \sqrt{K'} \left( 1 - \frac{A_{orifice}}{A_{up}} \right)^{0.75} \right]^2$$

where

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge} / D_{orifice}}$$

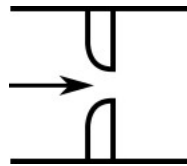


Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

For other orifice configurations, see Chapter 4 of Idelchik's Handbook of Hydraulic Resistance (2007).

### Other Loss Models

For information on the discharge coefficient loss model see the [Discharge Coefficient Loss Model](#) topic. For information on the orifice loss when the flow is choked see the [Sonic Choking](#) description.

### Related Topics

K Factor Loss Model.....	202
References.....	461

### Relief Valve

The Relief Valve junction allows users to model a device that opens at a specified pressure in the system.

The Relief Valve Properties window follows the first of the [two basic Properties Window formats](#), displaying the connected pipes in a fixed format. The Relief Valve junction adopts a flow direction from the connecting pipes.

A Relief Valve junction in AFT xStream will always behave as an Exit Rupture Disk, with the relief valve located at the end of a pipe leg. Below the Set Pressure, the valve is closed and behaves as a Dead End. When the Set Pressure is reached, the valve will instantly open and remain open for the rest of the MOC Transient simulation. If the valve is open during the AFT Arrow Steady Solution, it will remain open for the entire MOC Transient simulation.

### Valve Setpoints

---

The valve setpoints determine when the relief valve will open. These setpoints are defined using pressure.

There are two setpoints the user needs to define:

- **Exit Pressure** - the ambient pressure downstream of the relief valve when open.
- **Set Pressure** - the upstream pressure at which the valve opens.

---

**Note:** The Relief Valve junction operates exclusively using stagnation pressure. The Set Pressure and Exit Pressure are defined as [stagnation pressures](#) and all calculations are based on stagnation pressure.

---

### Loss Model

---

The Relief Valve loss information is entered on the Loss Model tab. The Basis Area for each Loss Model is by default the upstream pipe.

The following loss model options are available:

- **CdA (Constant)** - the relief valve instantly changes from closed to a fixed CdA. This CdA is defined as the product of an Orifice Effective Area (A) and a Discharge Coefficient (Cd).
  - The Orifice Effective Area can come from an [API 526](#) letter designation or it can be User Specified.
  - The Discharge Coefficient must always be specified by the user. The value is typically specified by the manufacturer.
- **K Factor** - the relief valve instantly changes from closed to a fixed K Factor.

**Note:** The  $C_dA$  for sonic choking may be different from the subsonic  $C_dA$  loss model option in xStream. The discharge coefficient can vary at different pressure ratios due to the vena contracta moving closer to or farther from the orifice restriction. For the highest accuracy the  $C_dA$  used for subsonic and sonic losses should be tested and entered separately. See the "[Modeling Choked Flow Through an Orifice](#)" white paper on AFT's website for more information.

---

### Special Conditions

---

The relief valve has two Special Conditions:

- **Failed Open** - the valve is forced to remain open regardless of pressure conditions.
- **Ignore Relief Valve** - the valve is forced to remain closed regardless of pressure conditions.

For more information see [Special Conditions](#).

### Related Topics

---

Common Junction Input Parameters.....	196
Discharge Coefficient Loss Model.....	201
Subsonic vs. Sonic Pressure Losses.....	454
K Factor Loss Model.....	202
Junction Transient Data.....	210
Initiation of Transients.....	212
References.....	461

### Relief Valve Transient Theory

The Relief Valve junction in AFT xStream allows users to model a rupture disk in the system, and behaves similarly to a regular exit valve. When closed, the relief valve behaves identically to a closed exit valve or a dead end. When open, the relief valve relieves to the user-specified Exit Pressure.

When the Relief Valve junction is defined using the CdA Subsonic Loss Model, the losses across the junction are determined as described in the [Discharge Coefficient Loss Model](#) topic.

When the Relief Valve junction is defined using the K Factor Subsonic Loss Model, the losses across the junction are determined as described on the [K Factor Loss Model](#) topic.

### Related Topics

---

Method of Characteristics.....	435
--------------------------------	-----

### Screen

The Screen junction type requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through a screen, mesh, or filter flow restriction. It also allows you to specify loss factors as a function of a flow parameter.

The Screen Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Screen junction does not have an explicit flow direction; it adopts a flow direction from the connecting pipes.

AFT xStream provides four standard screen types for your use: sharp-edged perforated plate, round-edged perforated plate, circular wire screen, and silk thread screen. The loss factors for these screens are taken from handbook values. In addition to the standard screens that AFT xStream provides, you may enter your own custom screen loss factor that is consistent with the application.

When you select [Resistance Curve](#), the Screen Properties window makes additional features available. Using these new features you can input Screen loss factors or pressure drops that vary with flow. To enter these factors, you can [fit a curve](#) to available data, or use interpolated x-y data. You can choose any of the optional flow and pressure parameters provided; you can also specify the most convenient units.

When a variable loss is specified, AFT xStream modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

### CdA for Sonic Choking

---

An optional input in the Screen Properties window is the  $C_dA$ . This parameter describes the effective area restriction in the screen for the purpose of calculating [sonic choking](#). In most cases, information on the  $C_dA$  must be obtained from the vendor or test data.

### Related Topics

---

Common Junction Input Parameters.....	196
Subsonic vs. Sonic Pressure Losses.....	454

### Screen Loss Model

The screen loss factors have much in common with orifice loss factors. In fact, the **sharp-edged perforated plate** and **round-edged perforated plate** screen losses are the same as for orifices with similar area ratios and equal upstream and downstream areas. Refer to the [orifice loss factor equations](#) for these relationships. The relationships are good for screen area Reynolds numbers above 100,000.

The **circular metal wire screen**, shown in Figure 1, is given by the following equation for screen area Reynolds numbers above 1000 ([Idelchik 2007](#), 598):

$$K_{up} = 1.3 \left( 1 - \frac{A_{screen}}{A_{up}} \right) + \left( \frac{A_{up}}{A_{screen}} - 1 \right)^2$$

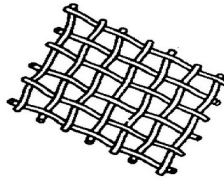


Figure 1: Circular metal wire screen

The **silk thread screen**, shown in Figure 2, is given by the following equation for screen area Reynolds numbers above 500 ([Idelchik 2007](#), 598):

$$K_{up} = 2.1 \left( 1 - \frac{A_{screen}}{A_{up}} \right) + \left( \frac{A_{up}}{A_{screen}} - 1 \right)^2$$

For other screen configurations, see Chapter 8 of Idelchik's [Handbook of Hydraulic Resistance \(2007\)](#).

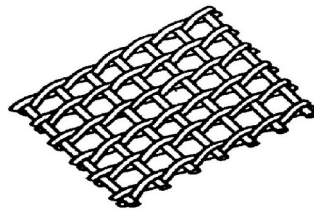


Figure 2: Silk thread screen

## Related Topics

---

References..... 461

## Tank

The Tank junction type allows you to connect up to twenty-five pipes. A minimum of one connecting pipe is required.

The Tank Properties window follows the second of the two basic [Properties window formats](#).

## Tank Model

---

There are two possible Tank Models available in the Tank junction, Infinite Tank and Finite Tank.

- **Infinite Tank** - Infinite tanks refer to a massive volume of gas whose pressure and temperature do not change appreciably as a result of gas inflow or outflow during the time frame of the simulation. An example is a large room or the atmosphere. The Infinite Tank model is convenient for specifying a fixed pressure in your system. This tank model applies a defined pressure at the junction location in the model. When solving a pipe flow system, an Infinite Tank causes the rest of the system to distribute the flow in a manner consistent with the defined pressure.
- **Finite Tank** - Finite tanks refer to a body of fluid which is small enough that its pressure and temperature change significantly during the time frame of the simulation as a result of gas inflow or outflow. An example is a tank which is pressurized by a compressor as the simulation progresses. Finite tanks require specification of tank geometry. Typically, the pressure and temperature are defined initially, and then calculated over time based on the tank geometry.

Both tank models will require the initial stagnation pressure and stagnation temperature/enthalpy to be defined.

If the Finite Tank option is chosen, then the volume and thermodynamic behavior of the tank become required inputs as discussed below.

### Known Parameters Initially

---

For the Finite Open Tank option, the options become active to specify whether the pressure and temperature are defined by the user for the steady state.

- **Initial Conditions Known** - This option should always be chosen for tanks at the inlet or outlet of the system which are acting as boundaries, Or for tanks with a known operating temperature and pressure in the steady state.
- **Initial Conditions Calculated From Steady State** - This option should be used for tanks that are inline or at intermediate points in the system at which the pressure and temperature are not known.

### Thermodynamic Process

---

The tank thermodynamics can be specified using one of three options:

- **Adiabatic** - The static enthalpy and volume in the tank will be fixed based on the user specified initial conditions in the tank. The variation in all other parameters will be calculated throughout the transient.
- **Isothermal** - The temperature and volume in the tank will be fixed based on the user specified initial conditions in the tank. The variation in all other parameters will be calculated throughout the transient.
- **Known Heat and Volume vs. Time** - The heat rate and volume are explicitly defined by the user throughout the transient. The variation in all other parameters will be calculated throughout the transient.

### Related Topics

---



Role of Pressure Junctions - Detailed Discussion (Long).....	480
Role of Pressure Junctions and How They Work.....	61
Assigned Pressure.....	223
Common Junction Input Parameters.....	196

## Tank Transient Theory

An infinite tank is equivalent to an [Assigned Pressure junction](#) defined with a stagnation pressure. The solution for a finite tank is more complex, and is discussed below.

### Adiabatic Finite Tank

For a finite tank a fixed volume is known, along with an initial stagnation pressure and temperature/enthalpy. The known values can be used to calculate initial values for the density, enthalpy, and energy.

$$\rho_o, h_o = f(P_o, T_o) \quad (1)$$

$$m = \rho_o \mathbb{V} \quad (2)$$

$$E = \mathbb{V} (h_o \rho_o - P_o) \quad (3)$$

The total mass and energy in the tank is calculated at the current time step, and the mass flow rate and energy flow rate out of the tank is assumed to be constant between the old and current time step. This assumption is reasonable when the time step is reasonably small, and the parameters in the tank are changing relatively slowly. With the known tank mass and the constant tank volume the density can be calculated for the current time step using Equation 2. The pressure and temperature are related to the total energy using Equations 1 and 3 as shown above. We can iterate on the pressure and temperature to solve these equations.. After each iteration the new pressure and temperature are used to recalculate the density, which is checked against the density calculated previously from Equation 2 to confirm when the iteration is complete.

Once the pressure and temperature have been calculated for the current time step, the new pressure and temperature can be used to calculate how the mass flow rate and energy flow rate will change for the future time step  $t + 1$ .

### Isothermal Finite Tank

The isothermal tank solution will proceed similar to the adiabatic finite tank solution outlined above. However, in this case the temperature is fixed as well as the volume.

The mass at the current time step can be calculated assuming the mass flow rate from the previous time step remained constant. The mass and volume can then be used to calculate the density at the current time step, which can be used to iterate on the enthalpy state equation which is represented by Equation 1 above.

Once the pressure has been calculated for the current time step, the new pressure and temperature can be used to calculate how the mass flow rate and energy flow rate will change for the future time step  $t + 1$ .

### Known Heat and Volume vs Time

---

The Known Heat and Volume vs. Time option allows the user to specify the tank volume over time plus an external rate of heat added to (or removed from) the tank. A similar process to the Adiabatic Finite Tank solution is used, where the external heat rate is used along with the mass flow into and out of the tank to determine the new pressure, temperature, and enthalpy inside the tank. The known tank volume is also coupled into the solution to iterate on tank mass and gas density as the tank volume changes.

### Sonic Choking Transient Theory

---

An outlet tank junction can experience [endpoint choking](#) during the steady state, or can become choked during the transient due to transient events in the system. If sonic choking occurs at the tank junction a different solution method must be enacted. See [Transient Sonic Choking](#) for more information.

### Related Topics

---

Nomenclature.....	410
Method of Characteristics.....	435
Sonic Choking Detailed Description.....	450

## Tee/Wye

The Tee/Wye junction type requires three connecting pipes.

### Loss Factor

---

The Tee/Wye is modeled as a Branch with no loss factors entered for the connecting pipes.

### Related Topics

---

Common Junction Input Parameters.....	196
---------------------------------------	-----

## Tee/Wye Transient Theory

The Tee/Wye junction is a static element, and its solution method is the same as the [Branch junction](#).

## Related Topics

---

Method of Characteristics ..... 435

## Valve

The Valve junction type requires two connecting pipes, unless you specify it as an exit valve, in which case only one connecting pipe is allowed. This junction type allows you to model the irrecoverable loss that occurs through a flow control component. You also have the ability to change the loss with time to open or close the valve in full or in part.

The Valve Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the connecting pipes.

## Loss Model

---

The valve loss model can be defined using one of two different valve parameters: Cv or Kv.

The upstream pipe is selected by default as the Base Area for Loss Model.

## Exit Valves

---

Valves that are used as exit flow control mechanisms are specified as exit valves. These valves require an associated back pressure definition. Exit valves can only be connected to a single upstream pipe. To specify an exit valve, check the Exit Valve check box, located at the bottom of the Loss Model tab. After this box is checked, you will be required to input the exit pressure and temperature.

## Transient Data

---

On the transient data tab a transient Cv, Kv, or open percent vs. time table can be entered to simulate the opening or closing of the valve. For more information on transient data, including event transients, see [Junction Transient Data](#).

AFT xStream can also create a Cv vs. Time profile based off of the open percentage table. To access this in the transient tab select "Edit Table" and then "Create Cv vs. Time from Open Percent...". This option enables the Cv vs. Open percent to be defined from several options:

- **Transferred** - If a table has already been defined on the Optional tab of the Valve Properties window, it can be copied over
- **User Defined** - Data can be entered directly into the table, or imported from existing data
- **Linear** - A linear curve is generated based on the user defined full open Cv and percent increment
- **Equal Percentage** - Generated as an [equal percentage](#) curve

- **Pre-Defined** - Pre-defined curves for several common valve Cv vs. Open profiles can be chosen and adjusted for a defined full open Cv value based on data from [Val-Matic 2009](#).

After the Cv vs. Open Percent is defined a transient can be defined based on duration, increment for the curves data points, and whether the valve is opening or closing. A linear Cv vs. Time curve will then be generated for the given data.

### Open Percentage Table

---

The Loss Model tab allows data to be entered for special valve characteristics. Specifically, the valve Cv, Xt, and Flow Area can be Specified vs. the Open Percentage of the valve. This data does not affect AFT xStream's flow solution. However, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary (in the Output window) always displays the valve Cv, and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point.

AFT xStream can assist in creating this Cv vs Open percent curve. This is accessible in the optional by selecting "Edit Table" and then "Create Cv vs. Open Percent...". This then opens a window in which a user defined, linear, [equal percentage](#) or pre-defined curve can be defined as described in the Transient Data section and transferred to the Cv vs. Open Percentage table. If a Cv vs. Open Percent table was already defined on the transient tab in the window to create a Cv vs. Time profile as described in the Transient Data section, then this curve will be automatically transferred.

### Special Conditions

---

The [Special Condition](#) for a valve always closes it.

### Related Topics

---

Common Junction Input Parameters.....	196
K Factor Loss Model.....	202
Cv/Kv and XT Loss Model.....	205
Junction Transient Data.....	210
References.....	461

### Related Examples

---

Gas Turbine Fuel System Example.....	575
--------------------------------------	-----

### Related Blogs

---

[Making Sense of xT – Valve Loss in Compressible Flow](#)

### [That Valve's Got Character – Applying Pre-defined Characteristic Curves](#)

## Venturi

The Venturi junction type requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through a venturi area change, which is frequently used in conjunction with pressure taps in flow measurement devices. It also allows you to specify loss factors as a function of a flow parameter.

The Venturi Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Venturi junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

## Loss Models

---

AFT xStream does not provide any standard venturi loss models. Venturi flow characteristics vary from vendor to vendor, so it is difficult to find generally applicable models. You must enter your own custom venturi loss factor that is consistent with the application. This information should be obtained from the manufacturer or from test data.

For convenience, you can specify constant loss characteristics of a venturi as a discharge coefficient ( $C_d$ ) or as a loss factor ( $K$ ).

If you would like to model the pressure loss through the Venturi as varying with the flow rate, select the radio button next to [Resistance Curve](#) under the Loss Model tab of the Venturi Properties window. Once Resistance Curve is selected, additional features appear that allow you to input pressure drops versus flow data. To enter this data, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT xStream modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, along with the most convenient units.

## CdA for Sonic Choking

---

An optional input in the Venturi Properties window is the  $C_dA$  for sonic choking. This parameter describes the effective area restriction in the venturi for the purpose of calculating [sonic choking](#). In most cases, information on the  $C_dA$  must be obtained from test or manufacturer data.

---

**Note:** The  $C_dA$  for sonic choking may be different from the subsonic  $C_dA$  loss model option in xStream. The discharge coefficient can vary at different pressure ratios due to the vena contracta moving closer to or farther from the orifice restriction. For the highest accuracy the  $C_dA$  used for subsonic and sonic losses should be tested and entered separately. See the "[Modeling Choked Flow Through an Orifice](#)" white paper on AFT's website for more information.

---

## Related Topics

---

Common Junction Input Parameters..... 196  
Subsonic vs. Sonic Pressure Losses..... 454

## Interface and User Options

This section focuses on the how the items within the various menus and tabs that comprise AFT xStream can be used to customize the user's experience within the software. Topics discussed include the options present in each of the drop-down menus, the tools available in the 5 Primary Window tabs, and descriptions of the customizable parameters in the User Options menu.

### Contents

---

<a href="#">Menus and Toolbars</a> .....	267
<a href="#">Print Preview</a> .....	278
<a href="#">Primary Window Overview</a> .....	279
<a href="#">Status Bar</a> .....	280
<a href="#">Quick Access Panel</a> .....	280
<a href="#">Model Status Light</a> .....	288
<a href="#">Solution Progress Window</a> .....	288
<a href="#">Workspace Window</a> .....	288
<a href="#">Model Data Window</a> .....	296
<a href="#">Output Window</a> .....	300
<a href="#">Graph Results Window</a> .....	311
<a href="#">Visual Report Window</a> .....	331
<a href="#">User Options Overview</a> .....	337

### Menus and Toolbars

Many of the actions required to set up an AFT Impulse model can be completed from the various menus located at the top of the window. Additionally, these menus can provide helpful features such as saving, importing and exporting files, and accessing various help documents such as the product help files and examples. These menus include

- [The File menu](#)
- [The Edit menu](#)
- [The View Menu](#)
- [The Analysis menu](#)
- [The Tools menu](#)
- [The Library menu](#)
- [The Arrange menu](#)
- [The Window menu](#)
- [The Help menu](#)

This topic also gives a high level overview of [Toolbars](#), including the Main Toolbar as well as each Primary Windows toolbar.

### File Menu

- **New** - Opens new model.
- **Open** - Calls Open file common dialog window so pre-existing model can be selected for opening.
- **Save** - Saves input model to disk using current model file name.
- **Save As** - Opens Save As common dialog so you can specify a file name to save model to disk.
- **Save Graph As** - (Graph Results Only) - Opens Save Graph As common dialog so user can specify a file name to save graph as a graphics image file to disk.
- **Save Graph Data As** - (Graph Results Only) - Opens Save Graph Data As common dialog so user can specify a file name to save graph x-y data to text file on disk.
- **Import Graph Data** - (Graph Results Only) - Allows for data as .txt and .dat to be imported for creating graphs
- **Export Model Data** - (Model Data Only) - Opens formatting window and Save As dialog where you can specify data from the Model Data window to be saved to a delimited file appropriate for spreadsheet use. Available in Model Data window.
- **Export Output** - (Output Only) - Opens formatting window and Save As dialog where you can specify data from the Output window to be saved to a delimited file appropriate for spreadsheet use.
- **Export Force File** - (Output Only) - Opens [Export Force File](#) window for the selected file type from those shown below.
  - **CAESAR II Force File**
  - **TRIFLEX Force File**
  - **ROHR2 Force File**
  - **AutoPIPE Force File**
- **Import Excel Change Data** - (Workspace and Model Data Only) - Imports data from Excel file which modifies certain input data. See [Importing Excel Data](#) for more info.
- **Excel Export Manager** - Opens the [Excel Export Manager](#).
- **Merge** - (Workspace Only) - Opens a Merge dialog where you can specify another model file to merge into the current model.



- **Print Preview** - Opens Print Preview which allows you to specify special formatting for printed graphics and text, as well as preview printed content before sending it to the printer.
- **Start Batch Run** - The Start Batch Run button on the File Menu opens a [Batch Run](#) dialog where you can select a different scenarios or models and initiates a process whereby xStream runs the multiple scenarios/models automatically. This tool can also be used to export output to Excel using the settings already configured in the Excel Export Manager.
- **Load Background Picture** - (Workspace Only) - Loads a background picture into the Workspace. Formats supported include bitmap, windows metafile, enhanced metafile, JPEG, GIF and RLE. Available in Workspace window.
- **Import Piping Layout From**
  - **Piping Component File (PCF)** - [Import a Piping Component File](#)
  - **Caesar Neutral File (CII)** - [Import a Caesar Neutral File](#)
  - **GIS Shapefile (SHP)** - Allows for [Importing GIS Shapefile](#) (geospatial maps) as .shp file.
  - **EPA Net File (INP)** - [Import an EPA Net File](#)
- **Export Piping Layout To**
  - **EPA Net File (INP)** - [Export an EPA Net File](#)
- **Exit** - Exits AFT xStream.

### Related Topics

---

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

### Edit Menu

- **Undo** - (Workspace Only) - Undoes the previous workspace manipulation action. xStream will keep track of a historical list of changes, and users can revert the workspace back to any of those previous states.
- **Redo** - (Workspace Only) - Reverts the workspace following Undo actions. xStream will keep track of undo actions, and users can redo any changes that have been undone.
- **Cut** - (Workspace Only) - Cuts the selected pipes and junctions on the Workspace and places them on the clipboard.
- **Copy** - (Workspace, Model Data, and Graph Results Only) - Copies the selected window contents to the clipboard.
- **Copy Graphics** - (Workspace and Visual Report Only) - Copies selected [Workspace](#) or [Visual Report](#) content to the Windows clipboard. Formats supported include bitmap, windows metafile, and enhanced metafile.
- **Copy X-Y Data** - (Graph Results Only) - Copies numerical data from graph to the Windows clipboard. Available in the [Graph Results window](#).

- **Paste** - (Workspace Only) - Pastes clipboard pipes and junction contents (from previous cut or copy operation) into the Workspace model. Paste preview mode allows the pasted pipes and junctions to be moved and specifically placed where desired and will indicate interference with other junction by changing color. A green box indicates that there is no interference and a red box indicates that the junctions will be overlapping and are not in a proper location. Available in the [Workspace window](#).
- **Delete** - (Workspace Only) - Deletes the selected pipes and junctions on the Workspace.
- **Duplicate** - (Workspace Only) - Duplicates the selected pipes and junctions on the Workspace.
- **Duplicate Special** - (Workspace Only) - Opens [Duplicate Special](#) dialog window where you can specify the increment by which all pipes and junction numbers will be increased when the model is duplicated, the creation of a new group from the duplicated objects, and whether to hide the duplicated objects in the output.
- **Select All** - (Workspace Only) - Selects all pipes and junctions on the Workspace.
- **Select Last Selection** - (Workspace Only) - The most recent pipes and/or junctions selected may be reselected by selecting Select Last Selection from the Edit menu.
- **Select Special** - (Workspace Only) - Opens [Select Special](#) window where group of pipes and junctions can be built up.
- **Select Flow Path** - (Workspace Only) - Selects the sequence of pipes and/or junctions in the [flow path](#) connected to the selected pipe(s) on the Workspace.
- **Select Shortest Path** - (Workspace Only) - The shortest flow path between two junctions can be found and selected by xStream. To enable this option, two junctions must be selected in the Workspace which xStream will find a path between.
- **Customize Icon** - (Workspace Only) - Allows user to change junction icon color, orientation, and icon image
- **Special Conditions** - (Workspace Only) - Sets or clears [Special Conditions](#) for selected pipes and junctions on the Workspace.
- **Groups** - (Workspace Only) - Allows you to create and manage [groups](#) of pipes and junctions.
- **Renumber** - (Workspace Only) - Allows renumbering pipes and junctions on the Workspace using one of the [Renumbering](#) methods.
- **Join Selected Pipes/Jcts-** (Workspace Only) - A continuous series of connected pipes and junctions can be combined into a single pipe object to simplify the Workspace. See [Combining and Splitting Pipes](#).
- **Global Pipe Edit** - (Workspace and Model Data Only) - Opens [Global Pipe Edit window](#) where editing of multiple pipes can be performed.
- **Global Junction Edit** - (Workspace and Model Data Only) - Opens [Global Junction Edit window](#) where editing of multiple junctions can be performed.
- **Delete Annotation** - (Visual Report Only) - Deletes the selected pipes and junctions on the [Visual Report](#).
- **Create Annotation** - (Visual Report Only) - Allows you to create an [annotation](#) on the Visual Report window.
- **Transfer Results to Initial Guesses** - (Output Only) - Copies solution [results into initial guesses](#) for pipes and junctions.

- **Make Selection Same As Parent** - (Workspace Only) - Replaces all input data for the selected pipes/junctions with data from the pipes/junctions in the current scenario's parent.
- **Clear Initial Guesses** - Clears all initial guesses in pipes and junctions.

### Related Topics

---

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

### Related Blogs

---

[AFT Responds to 2020 Customer Survey Feedback](#)

### View Menu

- **Last View** - (Workspace Only) - Switches to the most recent Workspace view. The view is changed by scrolling, zooming or using the [Workspace Find](#). The Last View in the View Menu will take you back to the previous view of the Workspace. You can toggle between two views by continuously clicking Last View.
- **Select Objects** - (Workspace Only) - Changes cursor to selection tool to select junctions, pipes, etc.
- **Overview Map** - (Workspace Only) - Opens overview map in the [Quick Access Panel](#).
- **Workspace Pan** - (Workspace Only) - Changes cursor to pan tool to move around workspace.
- **Show Magnifier** - (Workspace Only) - Turns on the [Magnifier Tool](#) in the Workspace
- **Show Object Status** - (Workspace Only) - Toggles [Show Object Status](#), which checks status of all pipes and junctions in model.
- **List Undefined Objects** - (Workspace Only) - Opens [Undefined Objects window](#) which displays undefined pipes and junctions and their properties.
- **Extended Model Check** - Performs and exhaustive check of model consistency and completeness.
- **Highlight in Pipe and Jct Windows** - (Workspace Only) - Toggles [Highlight feature](#) for pipe and junction Properties Windows.
- **Bookmarks** - (Workspace Only) - Allows creation and management of bookmarks.
- **Find** - (Workspace Only) - Opens window to [find](#) specific pipes and junctions.

### Related Topics

---

Workspace Window.....	288
-----------------------	-----

---

Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

## Analysis Menu

- **Analysis Setup** - Opens the [Analysis Setup window](#).
- **Time Simulation** - Allows selection of the Simulation Mode used to run the model.
- **Pulsation Frequency Analysis - (PFA Module Only)** - Allows pulsation analysis settings to be ignored or used in the model.
- **Run Model** - When the model is complete, the Run Model selection sends the model to the Solver for solution.

## Related Topics

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

## Tools Menu

- **Scenario Manager** - Opens the [Scenario Manager](#) window.
- **Last Scenario** - This allows you to toggle between two scenarios by loading the most recently loaded scenario. To do this, load one scenario using the [Scenario Manager](#) or the [Quick Access Panel](#), then load the other. Use the Last Scenario in the Tools Menu to then toggle between the two scenarios by continuously clicking Last Scenario.
- **Compare Scenarios** - This option quickly sets up a [comparison](#) between the current scenario and a related scenario.
- **Scenario Comparison Tool** - This tool opens the [Modify Comparison](#) window, allowing users to compare differences between any two or more scenarios in a model.
- **Generate Disconnected Scenario** - (Output Only) Generates a scenario which is no longer connected into the [Scenario Manager](#) hierarchy and thus does not inherit changes from parents.
- **Integration Query** - Opens Integration Query window, allowing users to have xStream perform a numerical integration of results over specified time frames.
- **Output Control** - Opens the [Output Control Window](#).
- **Output Print Content** - (Output Only) - Configures print content for Output Window.
- **Output Language** - Selects language for display of output results.
- **Model Data Control** - (Model Data Only) - Opens [Model Data Control Window](#).
- **Model Data Print Content** - (Model Data Only) - Configures print content for the [Model Data Window](#).

- **Select Graph Parameters** - (Graph Results Only) - Opens window to allow user to select which parameters to graph.
- **Quick Graph Formatting** - (Graph Results Only) - Opens window to allow user to change quick graph title and axis names, fonts, and scale values.
- **Visual Report Control** - (Visual Report Only) - Opens [Visual Report Control Window](#).
- **Design Alerts** - Opens the [Design Alerts](#) window.
- **Multi Condition Events** - Opens the [Multi Condition Events Window](#), which allows users to make transients based on more than one parameter in the model.
- **Annotation Manager** - (Workspace and Visual Report Only) - The Annotation Manager, available on the Tools menu, allows you to specify whether an [annotation](#) will be shown or hidden, and in which scenarios it should appear.
- **Fluid Property Viewers** - Gives options to open either the [SteamCalc Viewer](#) or [Chempak Viewer](#).
- **Activate Modules** - Opens [Modules panel](#) where optional modules can be activated or deactivated.
- **eLicense Information** - Visible if AFT xStream was opened using an eLicense - shows general information about the eLicense.
- **Keep Network eLicense Checked Out** - If a network eLicense is used, it can be kept Checked Out to this machine. This will prevent it from being lost when going offline, and prevent it from being accessible by other users. The license will be good for the Checkout Period - generally one week.
- **Model Diagnostics** - Opens the [Model Diagnostics Window](#), which allows users to check various settings that may have been changed during the model-building process and that may effect the model results
- **User Options** - Opens [User Options](#) window.

### Related Topics

---

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

### Related Blogs

---

[Get the Most Out of AFT's Fluid Property Viewers](#)

### Library Menu

- **Add Junction to Library** - (Workspace Only) - Prompts you to enter the name to be used in the Junction Library for the currently selected junction. After naming, [adds junction to the library](#).

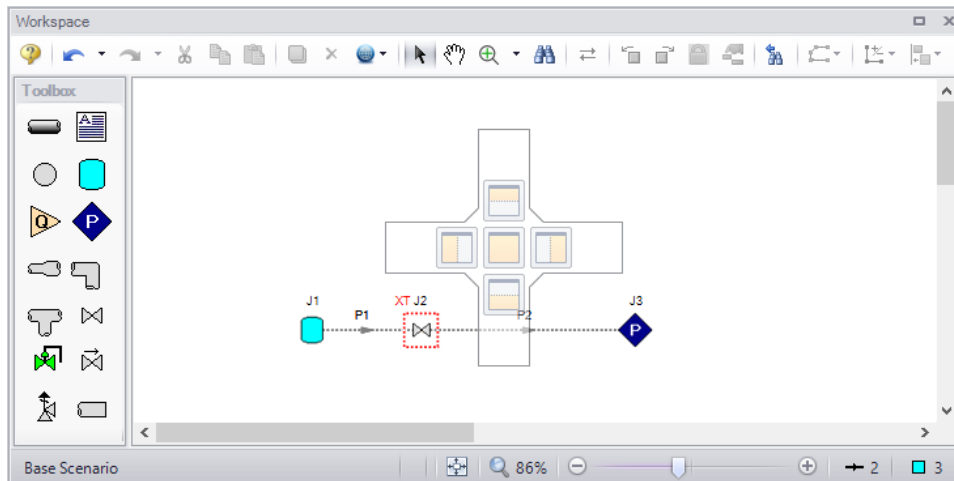
- **Edit Junctions** - Opens [Edit Junctions window](#) where you can delete or rename entries in the Junction Library.
- **Edit Fluids**- Opens [Edit Fluids window](#) where you can add, delete, modify or review custom fluids.
- **Edit Pipe Materials** - Opens [Edit Pipe Materials window](#) where you can add, delete, modify or review custom pipe materials.
- **Edit Insulations** - Opens [Edit Insulations window](#) where you can add, delete, modify or review custom insulation materials.
- **Edit Fittings & Losses** - Opens [Edit Fittings & Losses window](#) where you can add, delete, modify or review custom pipe fittings and losses.
- **Library Manager** - Opens the [Library Manager](#) window.

## Related Topics

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

## Arrange Menu

- **Primary Windows** - Allows you to configure the Primary windows in two panes in a horizontal or vertical orientation. The primary windows can be further rearranged by dragging the tabs into the center of the screen and adjusted by moving the tab into the desired position in the docking guide (see Figure 1).



**Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time**

- **Sort Table By** - (Output Only) - Allows user to order tables in order of any variable and in ascending or descending order. Available in the [Output window](#).
- **Rotate Icon Right or Left** - (Workspace Only) - Rotates icons for all selected junctions on the Workspace. Available in the [Workspace window](#).
- **Rotate Selection Right or Left** - (Workspace Only) - Rotates selected pipes and junctions as a group. Available in the [Workspace window](#).
- **Align** - (Workspace Only) - Aligns selected pipes and/or junctions on the Workspace. You can choose to align pipes and junctions, just pipes, or just junctions horizontally or vertically on the workspace. Available in [Workspace window](#).
- **Pipe Drawing Mode** - (Workspace Only) - Select the type of grid layout that the model will use.
  - **2D Freeform** - Junctions can be placed anywhere on the Workspace.
  - **2D Orthogonal** - Toggles [orthogonal drawing](#) capability on the Workspace. This toggle is a shortcut to enabling the feature in [User Options](#) window. See Orthogonal Drawing Overview for more information.
  - **Isometric** - Uses an [isometric grid](#).
- **Snap to Grid** - (Workspace Only) - Toggles Snap to Grid feature on the Workspace. When ON, all Workspace pipes, junctions and annotations snap to a grid specified in [User Options](#) window. Available in the [Workspace window](#).
- **Show Grid** - (Workspace Only) - Toggles Show Grid feature on the Workspace. Shows or hides grid that is specified in [User Options](#) window. Available in the [Workspace window](#).
- **Make Selected Pipes and Jcts Orthogonal** - (Workspace Only) - Makes all selected pipes and pipes connected to selected junctions on the Workspace orthogonal. Available in the [Workspace window](#).
- **Flip Vertex** - (Workspace Only) - Available when orthogonal drawing is enabled. This will flip the vertex of all selected pipes on the Workspace. Available in the [Workspace window](#).
- **Scale/Flip Workspace** - (Workspace Only) - [Scales](#) (shrinks or expands) or flips either selected or all pipes and junctions on the Workspace. Available in the [Workspace window](#).
- **Set Pipe and Junction Locations to Parent** - (Workspace Only) - Sets locations of pipe and junction labels to the same position as the parent [scenario](#). Available in the [Workspace window](#).
- **Reset Label Locations** - (Workspace Only) - Sets locations of pipe and junction labels to their default locations. Available in the [Workspace window](#).
- **Disable Pipes Snapping to Isometric Grid** - (Workspace Only) - Toggles Snap to Grid feature on the Workspace for the selected pipe. When Snap to Grid is disabled for that pipe, the pipe end point no longer snaps to grid points and pipe segments do not follow the Isometric lines. Available in the [Workspace window](#).
- **Pipe Segments** - (Workspace Only) - Allows user to Add Segment (split single into two pipes), Add Segment at Right Angle, Add Multiple Segments, Remove Segment, or Remove all Segments. Available in the [Workspace window](#).
- **Lock Object** - (Workspace Only) - Locks selected pipes and junctions on the Workspace so they cannot be moved. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction. Available in the [Workspace window](#).
- **Reverse Direction** - (Workspace Only) - Reverses the [reference positive flow direction](#) of all selected pipes on the Workspace. Available in the [Workspace window](#).

## Related Topics

---

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

## Related Blogs

---

[AFT Responds to 2020 Customer Survey Feedback](#)

### Window Menu

- **Workspace** - Makes the Workspace window active.
- **Model Data** - Makes the Model Data window active.
- **Output** - Makes the Output window active.
- **Graph Results** - Makes the Graph Results window active.
- **Visual Report** - Makes the Visual Report window active.

## Related Topics

---

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

### Help Menu

- **Open Help** - Opens help site home page
- **Use Offline Help** - Indicates that the PDF help file will be used instead of the Online AFT Help File when the help buttons in topics are clicked. Note that the PDF help file must be downloaded from the PDF Downloads page on the online help site prior to using this option.
- **Quick Start Guide** - Opens quick start guide pdf in English or metric units
- **Keyboard Shortcuts** - Opens keyboard shortcut help page in local help system
- **Show Examples** - Opens to example section of help file
- **Video Tutorials** - Opens video tutorials on aft.com in English or metric units and English or Spanish language
- **Technical Seminars** - Opens information on training courses on aft.com



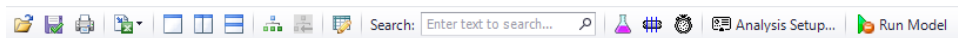
- **Flow Expert Package** - Opens information on aft.com for individualized help from AFT engineers for training, modeling, consulting, and more
- **Online Learning Center** - Opens learning center on AFT website with links to resources such as blogs, video tutorials, webinars, etc.
- **AFT Home Page** - Opens home page on aft.com
- **AFT Blog** - Opens blog home page on aft.com
- **Send Email to Support** - Links to AFT support in your default email program
- **Latest Release** - Opens page on aft.com to download the newest AFT xStream release
- **AFT Notifications** - Opens page on aft.com where you can sign up for email notifications for new software updates and releases
- **About AFT xStream** - Opens window in AFT xStream with information on current AFT xStream version, license number, etc.

### Related Topics

Workspace Window.....	288
Model Data Window.....	296
Output Window.....	300
Graph Results Window.....	311
Visual Report Window.....	331

### Toolbars

The Toolbars offer quick access to the features used most frequently in AFT xStream. The figures below show the Toolbars. There are actually six toolbars in all. One is the Common Toolbar, and it is always shown. Its features are common to all five Primary windows. In addition, there is one Toolbar for each [Primary window](#), offering features appropriate for work in that window. You only see two Toolbars at a time in AFT xStream. The function of each icon in a given Toolbar appears when you pass the mouse over the icon button without clicking on the button. Each of the functions on the Toolbar is also available through the menu system.



**Figure 1: Common Toolbar**



**Figure 2: Workspace Toolbar**



**Figure 3: Model Data Toolbar**



**Figure 4: Output Toolbar**



**Figure 5: Graph Results Toolbar**



**Figure 6: Visual Report Toolbar**

## Print Preview

The Print Preview window allows you to access special print features such as Print Preview and Fit to One Page. It is accessed from the [File menu](#) or using the shortcut CTRL + P.

The Print Preview window also offers you a preview of the print content and format for all [Primary Windows](#). By clicking the Print Preview button you can review your printed material at several zoom states. Individual pages can be printed.

The Fit to Page(s) feature causes the graphic image generated from the Workspace or Visual Report windows to be compressed onto the specified number of pages. This is a convenient feature for creating reports of your model. AFT xStream supports whatever printer page sizes are supported by your printer, so use of legal paper or larger may allow you to get more visible details. The graphics can also be centered vertically and/or horizontally. You also can fit your content onto multiple pages.

The Page Setup option in the Print Preview window allows you to set the page orientation, paper size/-type, and margins.

Additional special printing features affect the Workspace and Visual Report. The Selected on Workspace option works with Workspace printing by selectively printing the pipes and junctions currently selected on the Workspace.

By default, AFT xStream prints the junction icons for the Workspace and Visual Report with the same colors as displayed on the screen. It also prints the background color as the same color as your Workspace or Visual Report background. You can also specify that the junction icons be printed in black and white (still with the color background) or that the entire printout be black and white.

If you added a background image from the File menu, you can include this picture in the printout.

Finally, if you have a grid displayed on the Workspace or Visual Report, you can include the grid in the printout.

Page options allow the use of a company logo, user comments, titles, and different options for borders.

---

## Related Blogs

---

[Quick Guide to Custom Output](#)

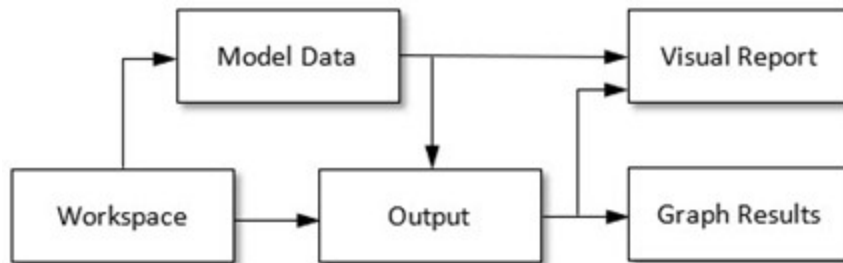
### Primary Window Overview

AFT xStream has five Primary Windows. The Primary Windows are tabbed and can be pulled out of the main window onto separate screens. The Primary Windows are permanent in that you work in one of these windows at all times. The Primary Windows work together to provide tools for entering model input, analyzing results for accuracy, and preparing results for documentation.

Of the five primary windows, two are input windows, two are output windows, and one displays both input and output information. Figure 1 shows the relationship between work-flow using the primary windows.

- The [Workspace window](#) allows you to build the model visually and see the model layout.
- The [Model Data window](#) is the text-based complement to the Workspace window. The Model Data window shows input data in text form. This window works hand-in-hand with the Workspace window to provide exceptional flexibility in manipulating data.
- The [Output window](#) displays the results of the analysis in text form and lets you produce attractive, effectively organized printed output.
- The [Graph Results window](#) provides full-featured Windows plotting capability. Here results can be viewed in a variety of ways for evaluating system transients or identifying important trends. The Graph Results window also allows graph customization.
- The [Visual Report window](#) merges the output data with the pipe system layout from the Workspace to present a unique perspective of the results. This window also allows customized layout of the information for documentation and presentation purposes. An alternate use for the Visual Report window is to show textual input data combined with the input schematic.

You can change between the five Primary Windows by selecting them from the [Window menu](#), the toolbar, or by clicking the appropriate tab within the AFT xStream window.



*Figure 1: Primary window work-flow in AFT xStream*

---

## Related Topics

---

Overview of AFT xStream..... 17

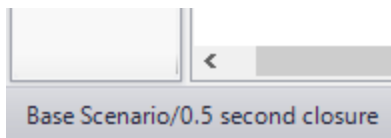
xStream Learning Resources.....	22
Overview of Fluid Transients.....	25
Philosophy of Computer Modeling.....	60

## Related Examples

Tank Blowdown Example (English Units).....	536
--	-----

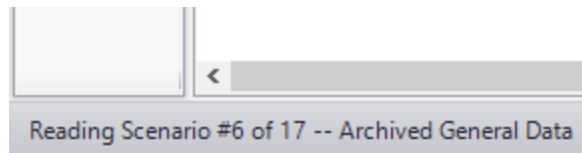
## Status Bar

AFT xStream's Status Bar displays contextual information about the current scenario or operation.



*Figure 1: Current scenario displayed in the Status Bar*

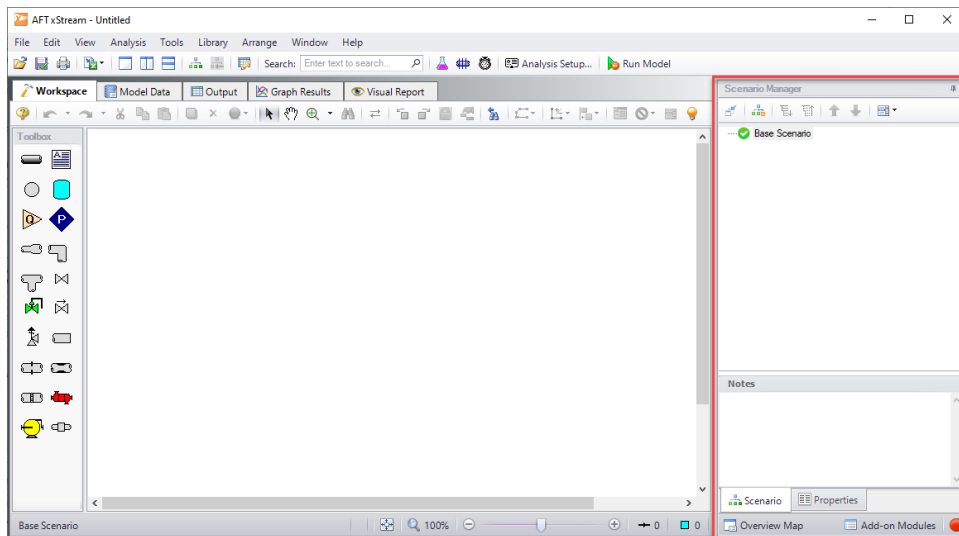
The Status Bar indicates the status of certain operations - for example, when opening a model the progress of loading the pipes, junctions, and scenarios is displayed.



*Figure 2: Status Bar displaying progress when opening a model*

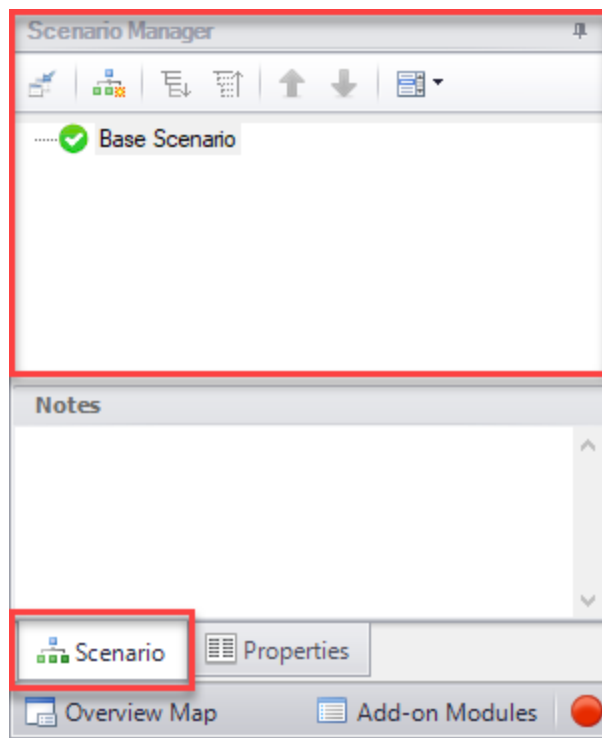
## Quick Access Panel

The Quick Access Panel as shown in Figure 1, contains several useful features. The Quick Access Panel allows you to manage your scenarios in a timely fashion, navigate significantly large models with the Workspace Overview Map, activate modules, view a pipe or junctions input properties and output results, control what graph parameters are displayed, and manage Graphs using the [Graph List Manager](#).



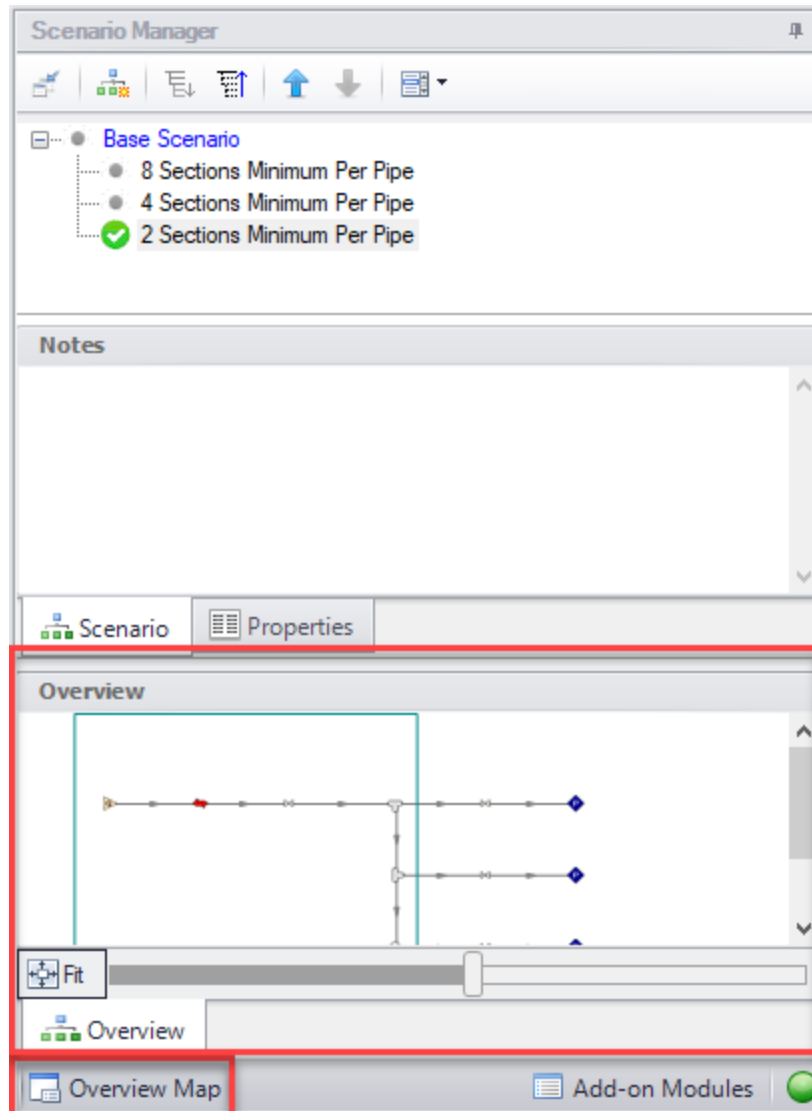
**Figure 1: The Quick Access Panel**

By clicking on the "Scenario" tab in Figure 2 below, you have the full scenario customization ability using the [Scenario Manager](#). You can create child scenarios, expand or collapse various families of scenarios, and utilize many other Scenario Manager features here as well.



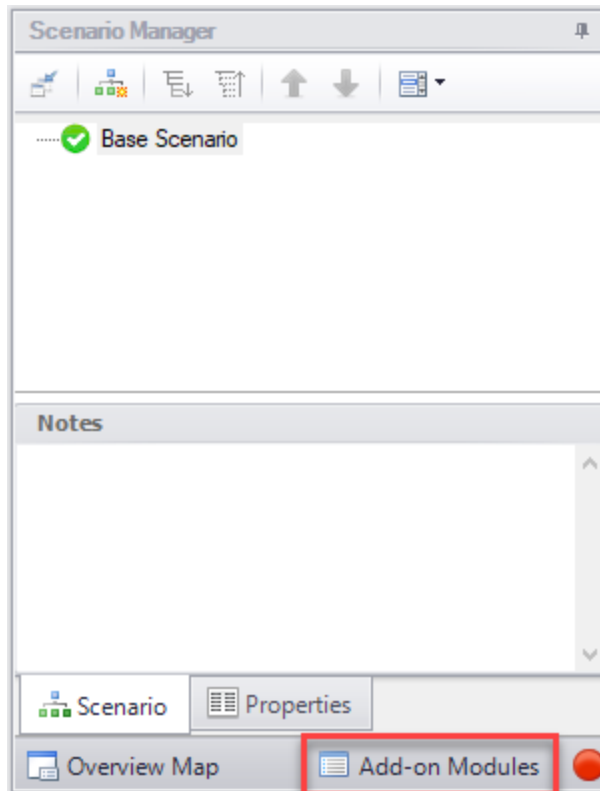
**Figure 2: The Scenario Manager**

When a model is significantly large in size, it can be troublesome to have to constantly drag the model around on the Workspace to get to a specific area of the model you want to view. By clicking on the "Workspace Overview Map" button in the red box in Figure 3, you will bring up a map that is a bird's eye view of the model. Drag the small square in the Workspace Overview Map over various parts of the model, then zoom in and out of the model. You can also click the "Fit" button to see the full model zoomed out on the Workspace, and use the mouse wheel to zoom in and out.



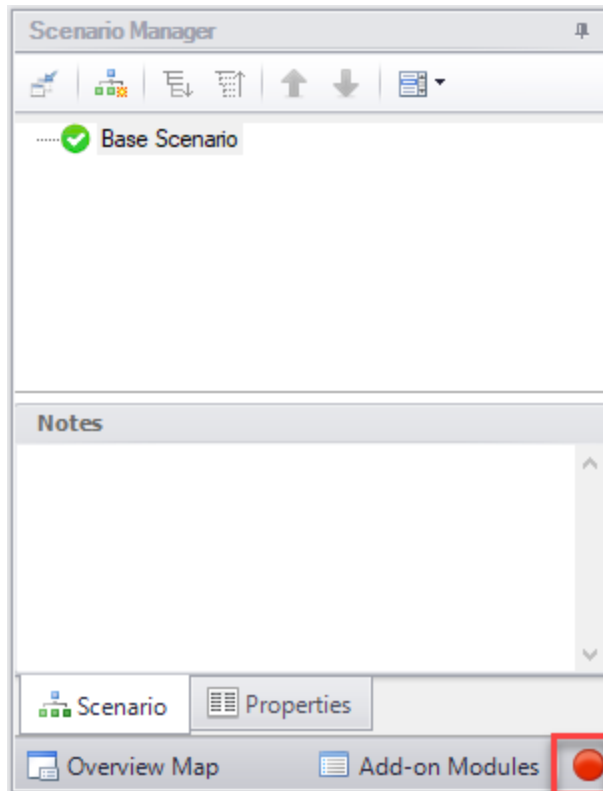
**Figure 3: The Workspace Overview Map**

Clicking the Modules button in the bottom right of the Quick Access Panel in Figure 4 allows you to activate or deactivate the [PFA](#) module in the [Analysis Setup window](#).



**Figure 4: Activating the Modules from the Quick Access Panel**

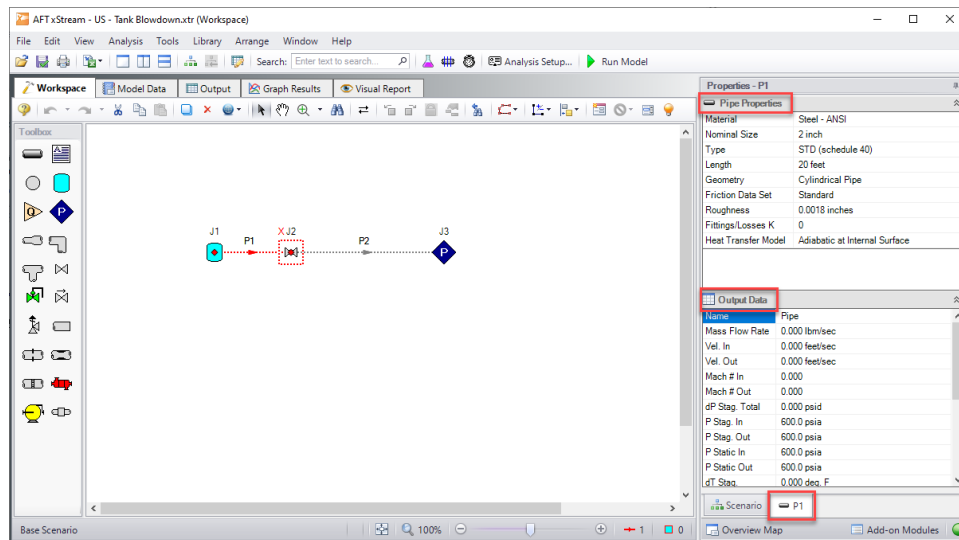
In the bottom far right of the Quick Access Panel in Figure 5 is the [Status Light](#). Clicking on the Status Light will display the [Analysis Setup window](#). When each item in the Analysis Setup window is fully defined, it will have a green check mark. When the model is fully defined in the Workspace and all necessary items in the Analysis Setup window are completed, the Status Light will turn from red to green. Depending on the Analysis type and module selection, additional items will appear in the Analysis Setup window.



**Figure 5: The Status Light and the Analysis Setup window**

As seen in Figure 1, there is a "Properties" tab in the Quick Access Panel. When pipes and junctions are drawn on the Workspace, you can quickly view the Pipe or Junction Properties by first selecting a pipe or junction on the Workspace, then clicking on "Properties" tab. This will display the Object Property data as well as any Output data if the model has been run. Figure 6 below shows an example of what the Properties tab would look like when clicking on pipe P1 on the Workspace after it has been defined and the model was run.

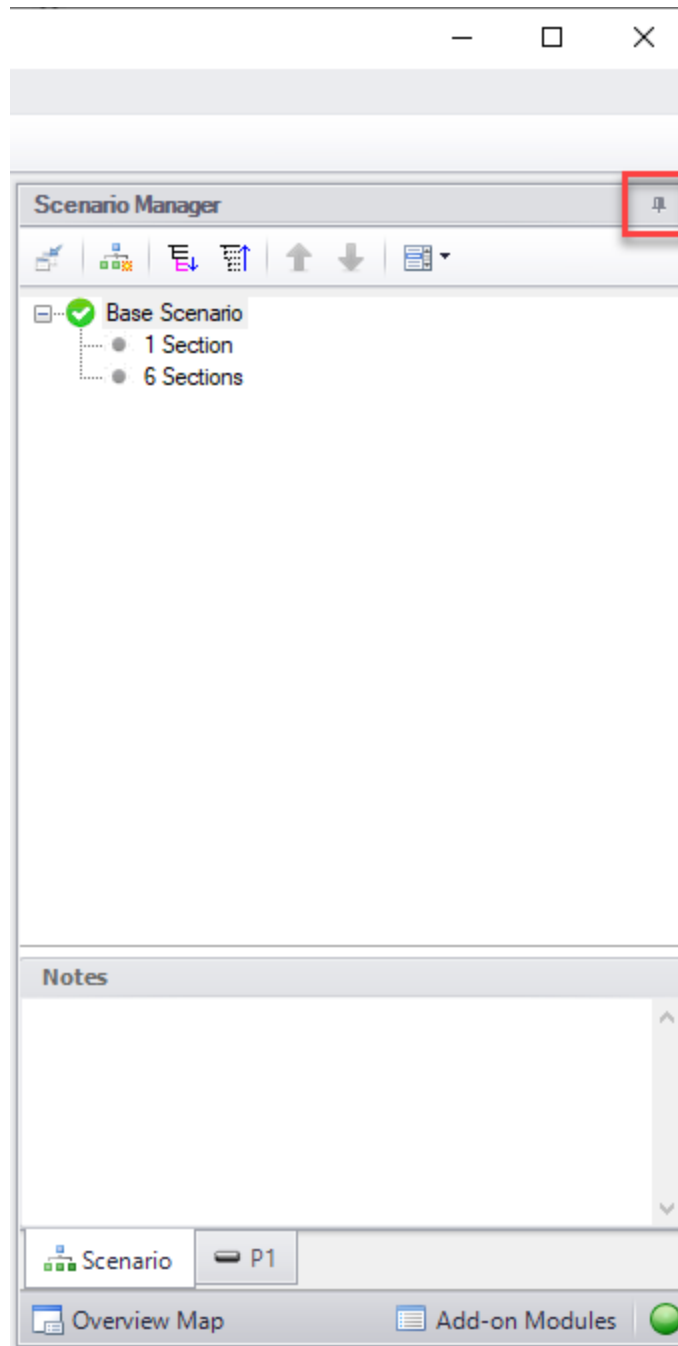




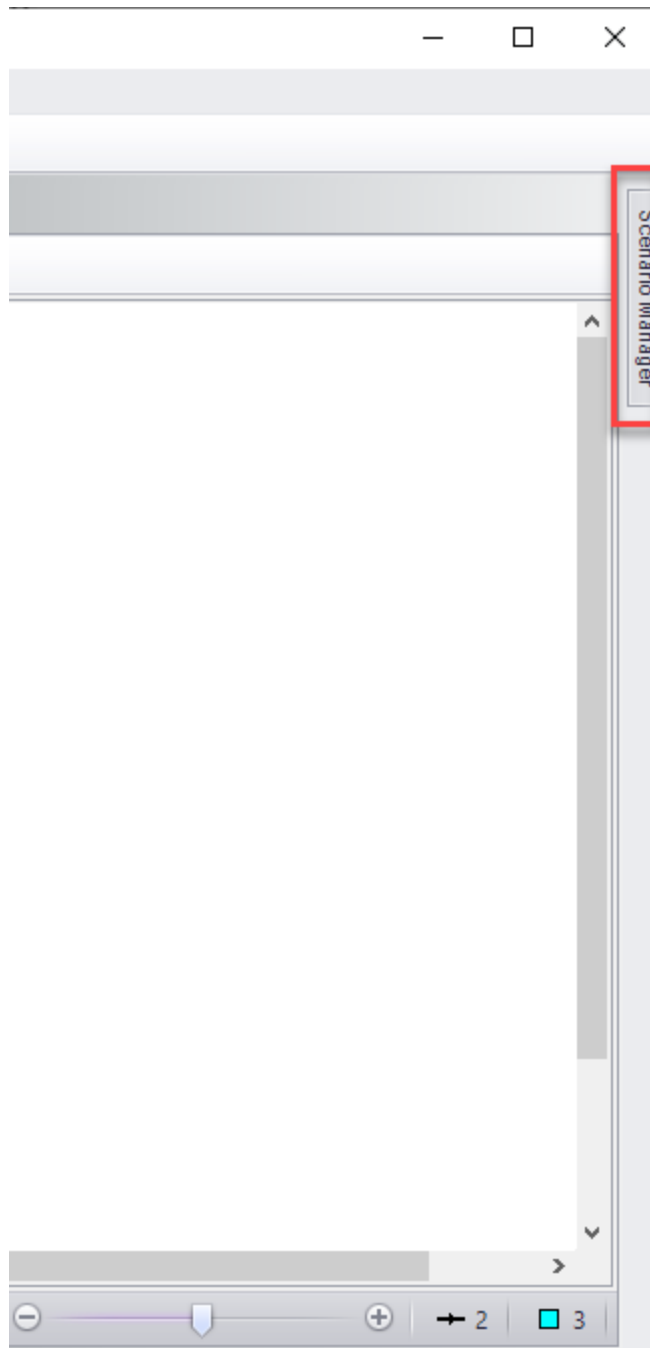
**Figure 6: Viewing the Pipe (or Junction) Properties and Output Data from the Quick Access Panel**

### Minimizing the Quick Access Panel

By Default, AFT xStream opens with the Quick Access Panel automatically expanded. The Quick Access Panel can be easily collapsed and pinned to the side of the Workspace to allow for a larger usage area of the Workspace. Figure 7 shows the Quick Access Panel initially expanded. By clicking on the thumbtack pin button highlighted with the red box, the Quick Access Panel will then become collapsed as shown in Figure 8. After the Quick Access Panel is collapsed, it can be temporarily expanded again simply by hovering over the "Scenario" sidebar with the mouse.



**Figure 7: The Workspace with the Quick Access Panel Expanded**



*Figure 8: The Workspace with the Quick Access Panel collapsed*

## **SSL Module**

### **Settling Slurry Module**

An optional add-on module to AFT Impulse which allows users to simulate settling slurries.

## PFA Module

### Pulsation Frequency Analysis Module

An optional add-on module to AFT Impulse which allows users to determine resonant frequencies in their system.

#### Model Status Light

The light at the right of the [Status Bar](#) located at the bottom of the AFT xStream window, which indicates whether or not the model can be sent to the Solver.

Clicking on the Model Status Light will show the [Analysis Setup window](#) where the completion status of different panels can be seen.

#### Solution Progress Window

The Solution Progress window indicates the current state of the solution iterations. During each of the three stages of the solution process, the Solution Progress window shows the following information.

- **Arrow Steady Solution** - The [tolerance](#) for the most out of tolerance junction or pipe is listed for the appropriate tolerances. General information about the solution method and solution control settings for the AFT Arrow Steady Solution are also shown. Note that both absolute and relative criteria are always displayed. These tolerances can be helpful for selecting the appropriate criteria on difficult to converge models.
- **MOC Steady Solution** - The absolute and relative velocity change across each block iteration is shown for the pipe with the largest velocity change. Information is also reported about if the [MOC Steady Solution](#) criteria are met and about any potential loss of mass or energy in the MOC Steady Solution.
- **MOC Transient Solution** - A progress bar of the transient simulation is shown.

The area in the bottom of the window will give important feedback on the model solution. It is important not to dismiss messages in this area, as they could have an impact on results. This information will also be printed to the General and Warnings sections of the Output window as is applicable.

When the model solution is complete, options to view [Output](#), [Graph Results](#), and [Visual Report](#) become available.

Pausing the iterations allows the modification of certain [AFT Arrow Steady Solution](#) parameters during the run, as well as viewing the iteration history. These options are available under Other Actions, and are only applicable to the AFT Arrow Steady solution.

#### Workspace Window

The Workspace window is the sole area where you specify the model layout and connectivity. The Workspace window has two separate functional areas: the [Toolbox](#) on the far left and the Workspace itself encompassing the rest of the screen (see Figure 1).

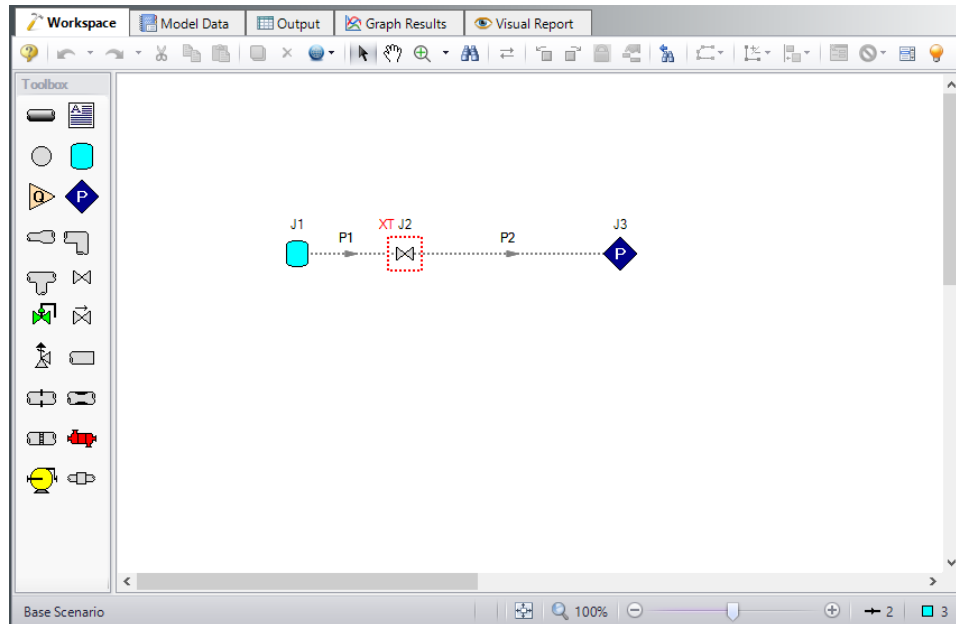


Figure 1: The Workspace window

## Display and Printing Features

The Workspace image can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The [Print Preview](#) window allows the above outputs to be applied to all or selected Workspace objects.

## Related Topics

Creating Pipes - the Pipe Drawing Tool.....	155
Annotations.....	292
Optional Input.....	197
Workspace.....	339

## Merging Models

Two models can be merged together using the command on the File menu. Here's how:

- Open the first model
- Choose Merge on the [File menu](#)
- From the file list, select the model to be merged

If there are duplicate numbers, the objects in the second model will be changed. The second model will be selected (highlighted) so that you can move it into position (be careful not to click off the selected portions or the second model may be deselected).

---

**Tip:** To move a selected group of objects, use a junction icon, instead of a pipe, to drag the group. This will avoid missing the pipe and clicking on an empty area of the Workspace which will deselect your group.

---

### Merging Models with Multiple Scenarios

---

If the model you are merging from has multiple [scenarios](#), only the base scenario can be merged. If you want to merge one of the children of that model, do the following:

1. Open the model which you want to merge.
2. Open the [Scenario Manager](#).
3. Select the child scenario you want to merge.
4. On the Other Actions button (or from the right-click menu), choose the Save Scenario to File Without Children.
5. Enter a name and click OK. This process creates a new xStream model with the desired scenario as the base scenario.
6. Open the original model and merge in the new model file created in Step 5.

### Renumbering Workspace Objects

Objects on the [Workspace](#) can be renumbered by choosing Renumber from the Edit menu. There are several ways that Renumbering can proceed.

#### Renumber Wizard

---

The Renumber Wizard allows you to renumber pipes and/or junctions by simply clicking on them on the Workspace. Normal Workspace functions are suspended when the Renumber Wizard is active. The Renumber Wizard is activated on the [Edit menu](#).

When the Renumber Wizard is active the settings can be modified in the [Quick Access Panel](#). Click Done Renumbering in the Quick Access Panel to finish renumbering and exit the wizard.

### Renumber Automatic

---

Selected [pipes and/or junctions](#) can be renumbered starting at the lowest selected number or at a specified number. If a pipe or junction which is not selected to be renumbered already has the next number in sequence, it will only be changed if the Force Sequential checkbox is checked.

xStream will start at the beginning of the flow path containing the lowest numbered selected pipe and will attempt to march down the pipes in the direction of flow. It is recommended that renumbering be performed after the model has been completed so that the junctions and pipes are completely connected.

---

**Tip:** If you want certain sections of the model to be numbered in sequence, then use the Renumber feature on the selected sections only. It is also preferable in the case of large or complicated models to renumber small sections at a time.

---

### Renumber Increment

---

Renumber Increment is on the Edit menu and allows you to increment or decrement all selected pipes and junctions by a specified value.

### Scale/Flip Workspace

The [Workspace](#) can be scaled horizontally and/or vertically by choosing Scale/Flip Workspace on the [Arrange menu](#). Scaling will move the junctions and will stretch or shrink the pipes so that the area the model covers is changed by the percentage specified.

Flipping the Workspace horizontally and/or vertically will move the pipes and junctions to their mirror image locations. This is useful when you are building a system which has symmetrical or mirrored sections. For example, you could select the section, duplicate it, and then flip it horizontally, vertically, or both with Scale/Flip Selected Objects Only checked.

### Magnifier Tool

The Magnifier tool can be used to provide a magnified view of any area on the Workspace. This is useful for navigating and viewing details of a large model without needing to zoom in on that area. The Magnifier tool can be accessed from the Workspace Toolbar, as can be seen in Figure 1 below. The magnifier tool can be enabled/disabled by clicking the button on the Toolbar.

There are several options available to adjust the Magnifier view:

- **Zoom Level** - Adjusts the level of magnification used
- **Magnifier Size** - Adjusts the size of the magnifier circle

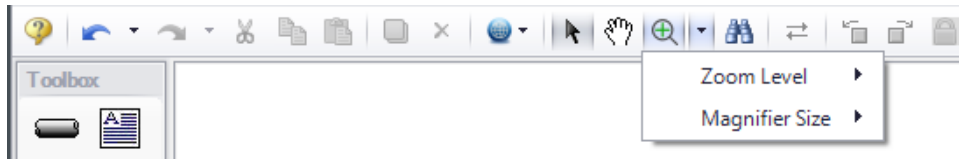


Figure 1: Magnifier tool on the Toolbar

The Magnifier tool can also be used in the [Visual Report window](#).

## Related Topics

Visual Report Window.....	331
Toolbars.....	277

## Annotations

The [Toolbox](#) tool to the right of the Pipe Drawing Tool is the Annotation Tool. With this tool, you can draw a box that will contain an Annotation displayed on the [Workspace](#).

The Annotation Tool allows you to create a text message, show an outline around the message, insert pictures in the message box, and draw lines with pointers to certain areas of the model. Different style, color and font options are provided.

Annotations can be "attached" to the Workspace, a pipe, or a junction. When attached to the Workspace, the annotation always remains in the same location until you move it. When attached to a pipe or junction, the annotation will move when the pipe or junction is moved. The Scenario Options tab allows you to specify in which [scenarios](#) the annotation appears.

Annotations can be layered to be either "Always in Front" or "Always in Back".

Annotations print out with the model.

Annotations can be displayed or hidden among different scenarios using the Annotation Manager from the [Tools Menu](#).

## Extended Model Check

The connections and settings of each pipe and junction are exhaustively checked using the Extended Model Check item on the [View menu](#). This is also helpful if the model seems to not be connected correctly.



## Keyboard Shortcuts for Workspace

### Selection Box

---

- Select all objects completely inside box by dragging cursor from left to right
- Select all object completely or partially inside box by dragging cursor from right to left

### Pipe Drawing Tool

---

- If you double-click the pipe drawing tool, it will allow you to draw multiple pipes at a time. Clicking Esc or clicking the pipe drawing tool again will deactivate the tool.
- When [orthogonal drawing](#) is off and you hold down the SHIFT key while drawing pipes on the Workspace, you can [draw pipes](#) that are perfectly horizontal or vertical. When the orthogonal drawing is on and you hold down the SHIFT key down while drawing new pipes on the Workspace the pipe vertex will flip. If you move an existing pipe and hold the shift key while selecting a handle at one of the pipe endpoints a new vertex will be inserted at that point endpoint.
- If you hold down the CTRL key when completing the pipe drawing (just before releasing the mouse button), the Pipe Drawing Tool remains active, and you can draw a series of pipes without returning to the Toolbox each time.
- To add [pipe segments](#), use CTRL+SHIFT+Y
- To remove all [pipe segments](#), use CTRL+SHIFT+N

### Junctions

---

- If you double-click any junction in the Toolbox, you will be able to place multiple of that junction on the Workspace by clicking the Workspace where you would like to place each junction. Clicking Esc or clicking the junction icon in the Toolbox again will deactivate the tool.
- If you hold down CTRL key and select a [junction](#) from the Toolbox and drop it onto an existing Workspace junction, the old junction will "[morph](#)" into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.
- If you hold down the CTRL key then drag a junction on the Workspace which has pipes connected to it, the junction will move but the pipes will not move with it
- If you hold down the SHIFT key while selecting a junction from the Toolbox and then drop the junction onto an existing pipe, the pipe will split into two pipes. You will specify the length of the first pipe and the second pipe will be automatically set to the remaining length from the original pipe. Thus the sum of the two pipe lengths will equal the length of the original pipe. You can also choose which pipe contains any Fittings & Losses that were specified within the original pipe.

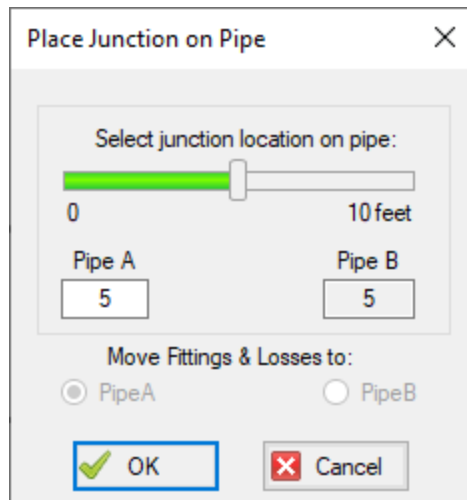


Figure 1: Split pipe keyboard shortcut

---

## Pipe and junction Properties window

- When in a pipe or junction Properties Window you can toggle the highlight feature by pressing the F2 function key. You can also toggle this from the View menu.
- When in a [Pipe Properties Window](#), if you press the F5 function key, you will jump to the next highest numbered pipe. If you press the CTRL key while pressing the F5 function key, you will jump to the next lowest numbered pipe. The F5 function key also behaves this way for junction Properties Windows.

---

## Pan and Zoom

- The Workspace position can be moved with CTRL + arrow keys.
- The Zoom level can be adjusted with CTRL + (+/-)
- The Zoom level can be reset to default with CTRL + 0
- Mouses with scroll wheels can be used to zoom by scrolling, or to pan by clicking and holding the scroll wheel

---

## Switching Primary Windows

- Go to the Workspace window using CTRL + W
- Go to the Model Data window using CTRL + M
- Go to the Output window using CTRL + U
- Go to the Graph Results window using CTRL + G
- Go to the Visual Report window using CTRL + I

## Other

---

- Many commands use common windows shortcuts. Some of the most common are listed here:
- Run the model: CTRL + R
- Save the model: CTRL + S
- Open a file: CTRL + O
- Undo the last action: CTRL + Z
- Cut: CTRL + X
- Copy: CTRL + C
- Paste: CTRL + V
- Other available shortcuts can be seen listed by their command on the menus

## Related Topics

---

Add/Remove Pipe Segment .....	189
Joining or Splitting Pipes .....	157
Creating Junctions .....	158
Defining Objects .....	161
Editing Objects .....	160
Morphing Junctions .....	160
Moving Objects .....	159
Creating Pipes - the Pipe Drawing Tool .....	155
Workspace Pan .....	164

## Undefined Objects Window

This window is launched from the Workspace toolbar or the [View menu](#).

This window displays all undefined pipes or junctions in the model. Clicking on the undefined object will display the undefined properties in the list below. All undefined data for all objects can be saved to the file or copied to the clipboard.

Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.

## Related Topics

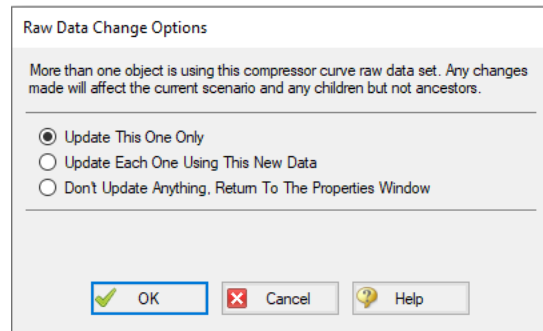
---

Defining Objects .....	161
------------------------	-----

## Raw Data Changes

When other objects in the model use the same base raw data as the current object and the data is changed, you can decide whether to use the new data on the other model objects.

This frequently happens when you have multiple junctions based on the same x-y data for compressor/fan curves or junctions resistance curves, and later change that x-y data. The question is whether you want to have the current junction you are working on updated, or all junctions that use that same base data.



**Figure 1: Raw Data Change Options**

## Duplicate Special

Duplicate Special allows the model to be copy & pasted, while simultaneously applying other helpful settings as explained below.

- **Increment All Pipe and Junction Numbers By** - Adds the specified number to all pipe/junction numbers for the created object duplicates. Useful for maintaining consistent numbering
- **Create Group** - Adds the duplicates to a [group](#).
- **Hide in Output** - Automatically hides the pipe and junction duplicates in the Output window so only the original pipes and junctions will be seen.

## Model Data Window

The Model Data window offers a text-based perspective on all engineering information in the pipe flow model. This window is immensely useful for obtaining a global view of the model and rapidly checking the input.

The Model Data window does not offer any tools to build or add to a model; all model assembly must happen in the [Workspace](#).

The Model Data window allows you to manipulate existing elements from the Workspace. Therefore, a complete model can be assembled in the Workspace window without ever opening the Model Data window. In fact, for small models this may be the preferred approach.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to re-size them. Each section can also be hidden by clicking the arrow in the upper left of each section.

You can use the zoom feature in the lower right hand corner of the Model Data window to re-size the text displayed on screen.

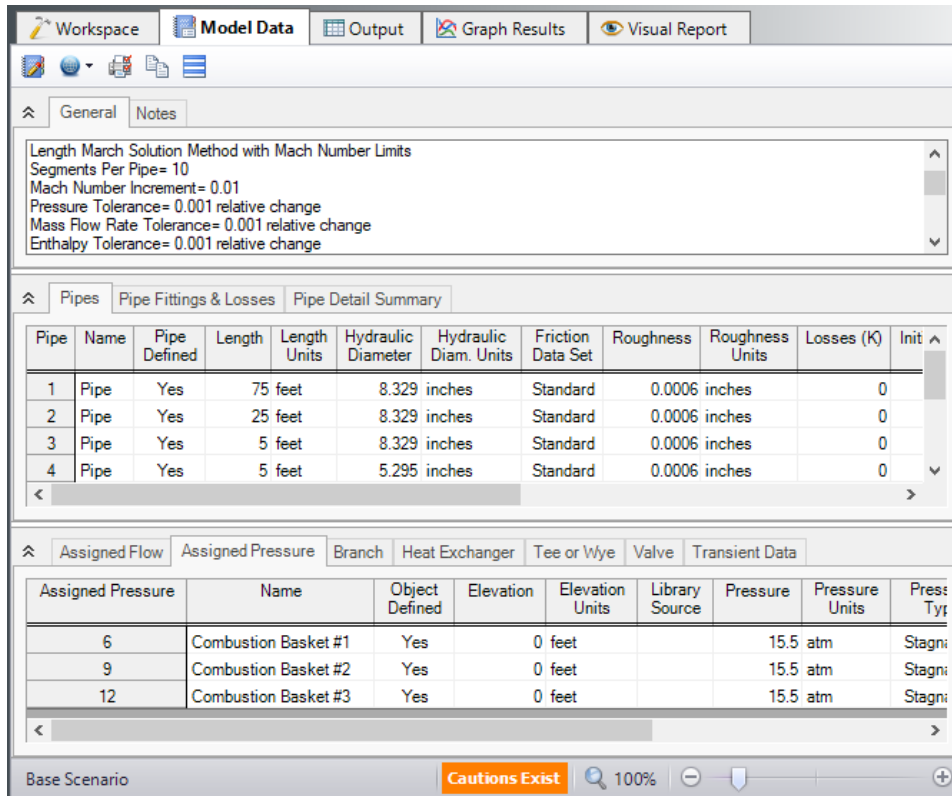


Figure 1: The Model Data window displays input in text form

## Editing Features

The Pipe table allows display of all input data for all pipes in the model. Details of [Fittings & Losses](#) are displayed in the Pipe Fittings & Losses table, accessed with an adjacent tab. The Pipe Fittings & Losses table displays all fittings with K factors grouped into columns.

The other display tab in the pipe area is the Pipe Detail Summary, which assembles a list of details about each pipe in another format useful to some engineers.

The junction data area is below the pipe data area. Here the junction data is separated into tables for each junction type. Click the tabs to see the data for any type of junction. All transient data entered for the junctions can be seen in the Transient Data tab on the far right.

To edit information in the Model Data window, you can open the Properties Window for the pipe or junction. This can be done by double-clicking anywhere in the row for the pipe or junction to be edited. Alternatively, you can use the [Global Edit](#) windows opened from the Edit menu.

### Display and Printing Features

---

[Model Data Control](#) lets you specify the combination of pipe, junction, and general information you want to see in the Model Data Window.

The Model Data Print Content window lets you specify the content to include in the printed Model Data report and select the font to be used for printing. This window is opened from the Tools menu (or Toolbar).

The Model Data window contents can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to Excel
- Saved directly to an Adobe PDF file
- Sent to the printer

### Related Blogs

---

[Having model convergence problems? Start here!](#)

### Model Data Control Window

The Model Data Control allows you to customize what data to show for the pipes and for each junction type and also which pipes and junctions to show (see Figure 1). It can be opened from the [Tools menu](#) (or Toolbar).

### Pipe Data Display

---

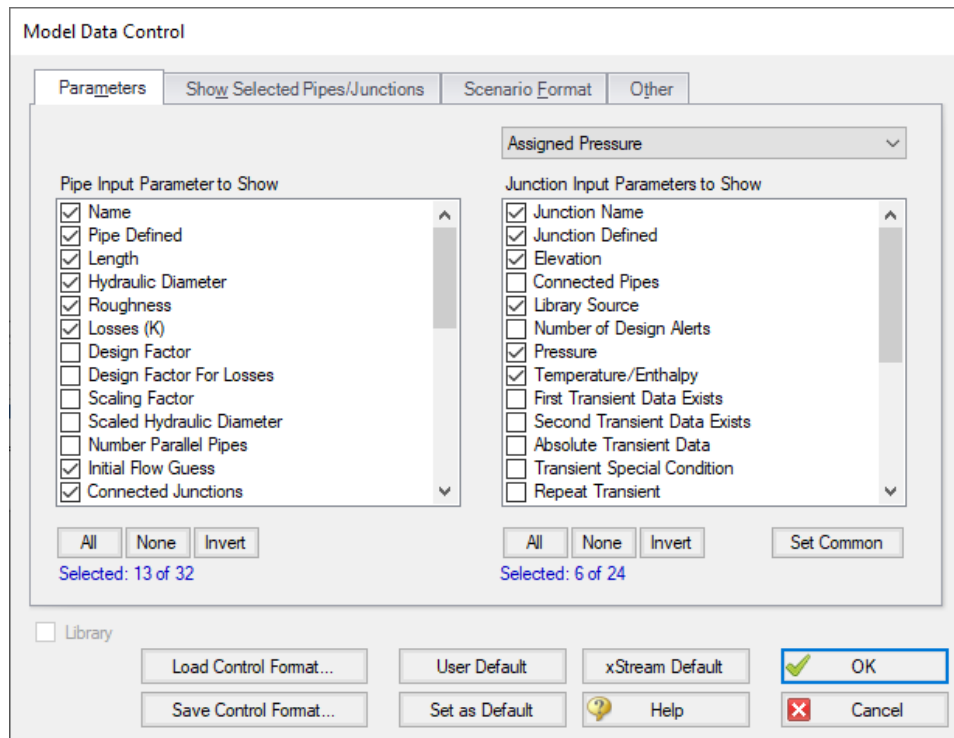
The data to be shown for the pipes is selected on the left side of the Model Data Control window. Only the items with check marks will be shown in the Model Data window. All pipes shown will report the selected items.

### Junction Data Display

---

The data to be shown for the junctions is selected on the right side of the Model Data Control window. The data is grouped by junction type. However, some data is general and applied to each of the junction types. Select the junction type from the list and then select the data for that junction type. Each type also includes the general data which may be selected/deselected.

You can select/deselect the general data for all junction types simultaneously by clicking Set Common.



**Figure 1:** The Model Data Control window allows selection of the parameters to show in the Model Data window

### Model Data Format

The Model Data Control format can be saved to a file so that the format can be quickly applied to other model files. There are multiple ways to do this. The Model Data Control format can be saved to the internal User library by clicking the Set As Default button, then choosing User Default in a new model file to apply the settings.

The Model Data format can also be saved to an external file, which can then be shared with different users. The external file can be generated by clicking the Save Control Format button at the bottom of the window. The format can then be applied to other model files by clicking the Load Control Format button.

### Show Selected Pipes and Junctions

Specific pipes and junctions can be shown in Model Data. Click the Show Selected Pipes/Jct tab, then click the items in the lists to select the pipe(s) and junction(s) to be shown.

### Other Data

---

Here you can specify whether pipe elevation data, junction curve data, [Design Alerts](#), and/or [Force Sets](#) will be displayed in the Model Data window.

### Scenario Format

---

A powerful Model Data feature when using scenarios is to show all direct ancestor data in the Model Data along with data from the current scenario. The Scenario Format features allow you to do this, and to highlight where data changes occur. Note that the scenario name is appended to the pipe and junction number in the left columns. This tab only appears when multiple scenarios exist.

### Library Connections

---

In the lower left of the Model Data Control window there is a Library checkbox. If this box is checked, your Model Data Control parameters are setup as determined by the library to which you are [connected](#). This is referred to as an active library. To make it inactive, uncheck the box or change one of the Model Data Control settings controlled by the library.

If the check box is unchecked, but enabled, you are connected to a library but the settings are not being passed to the Model Data Control window. The library is thus inactive. To make it active, check the box then click the OK button.

If the check box is disabled, there is no connected library.

## Output Window

The Output window is the primary vehicle for communicating the results of an analysis in text form (see Figure 1). The Output window follows the same general window organization as the Model Data window.

When a model is submitted to the Solver, the [Solution Progress window](#) appears. After the solution is obtained, you can view the results in the Output window by clicking the View Output button.

If the information in the Output window is not in the desired format, you can choose [Output Control](#) from the Tools menu (or [Toolbar](#)) to modify the format. The Output Control window lets you specify the parameters, their units, and the order of their display in the Output window.

The Output Print Content window available on the Tools menu (or [Toolbar](#)) lets you specify the content to include in the printed Output report and select the font to be used.

The Sort window opened from the Arrange menu (or [toolbar](#)) offers the ability to sort the Pipe Results or Junction Results tables according to the values in any of the columns.

The [Transfer Results to Initial Guess](#) feature on the Edit menu (and [Toolbar](#)) sets all pipe and junction initial steady-state flows and pressures equal to the converged results, which causes future runs to converge much faster.

You can view the input data for individual pipes or junctions directly from the Output window in one of two ways. First, if you press down the right mouse button on the pipe or junction number in the far left column



of the table you will see the [Inspection window](#). Second, if you double-click the pipe or junction number you will open the Properties window in read-only form where you can review input in detail.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to re-size them. Each section can also be hidden by clicking the arrow in the upper left of each section.

You can use the zoom feature in the lower right hand corner of the Output window to re-size the text displayed on screen

The screenshot shows the Output window with the following data:

**Simulation Summary:**

- Total Number Of Flow Iterations = 11
- Total Number Of Enthalpy Iterations = 11
- Number Of Pipes = 11
- Number Of Junctions = 12
- Matrix Method = Gaussian Elimination
- MOC Transient Execution Time = 5 minutes, 22.718 seconds (322.72 seconds)
- Model Start Time = 0 seconds

**Transient Max/Min Table:**

Pipe	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station	Min. Static Pressure (psia)	Min. Static Pressure Time (seconds)	Min. Static Pressure Station	Max. Mass Flowrate (lbm/sec)	Max. Mass Flowrate Time (seconds)	Max. Mass Flowrate Station	Min. Mass Flowrate (lbm/sec)	Min. Mass Flowrate Time (seconds)	Min. Mass Flowrate Station
1	274.8	2.000	0	247.8	0.000	120	25.519	0.1058	11	17.312	0.04360	120
2	269.2	2.000	0	242.1	0.000	40	25.006	0.01526	5	16.038	0.02586	40
3	268.8	2.000	0	242.0	0.000	8	25.004	0.000	0	15.910	0.02180	8

**All Junctions Table:**

Jct (AFT Arrow Steady)	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. (Btu/lbm)	H Stag. In (Btu/lbm)
1	PD Compressor	25.000	0.00000	249.6	249.6	248.8	248.8	0.000000	70.35	70.35	70.00	70.00	0.00	410.
2	Natural Gas Heater	25.000	5.10365	248.6	243.5	247.8	242.5	-69.377075	70.29	139.67	69.94	139.21	-39.28	410.
3	Supply Valve	25.000	0.05077	243.1	243.0	242.1	242.1	0.002075	139.65	139.65	139.19	139.19	0.00	450.

**Figure 1: The Output window displays output in text form**

## Right-click Context Menu Actions

There are several options available when right-clicking on a cell, row header, column header, or entire table in the Output window:

- **Column Header**
  - **Change Units** - Quickly redisplay the selected column in a different unit.
  - **Sort** - Sort the table in Ascending or Descending order on the selected column.
  - **Add "Parameter" for all Table Objects to Excel Export Manager** - Add this column to the [Excel Export Manager](#).
  - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
  - **Copy Selected Cell Data** - Copies all of the values in the column to the clipboard.
- **Row Header**
  - **Add all Object Parameters to Excel Export Manager** - Add this row to the [Excel Export Manager](#).
  - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).

- **Find on Workspace** - [Quickly locate](#) an object on the Workspace.
- **Copy Selected Cell Data** - Copies all of the values in the row to the clipboard.
- **Open Properties Window** - Opens the properties window for the associated pipe or junction.
- **Table (upper left cell)**
  - **Add Object Table to Excel Export Manager** - Add the entire table to the [Excel Export Manager](#).
  - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
  - **Copy Selected Cell Data** - Copies all of the values in the table to the clipboard.
- **Cell**
  - **Add Object "Parameter" to Excel Export Manager** - Add this cell to the [Excel Export Manager](#).
  - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
  - **Create Design Alert** - Create a [Design Alert](#) related to the selected object and value.
  - **Copy Selected Cell Data** - Copies the value of the cell to the clipboard.

### Output Window Updates

---

Whenever you change the input model, all output windows are erased. This prevents you from changing your input model, forgetting to rerun it, and mistaking the previous Output window for the current model's results.

### Steady Flow Results

---

Figure 2 shows the steady flow results in tables for pipes and junctions as specified in the Output Control window. Details of the steady state results at each pipe calculation station can be viewed on the Axial Points tab of the Pipes section.

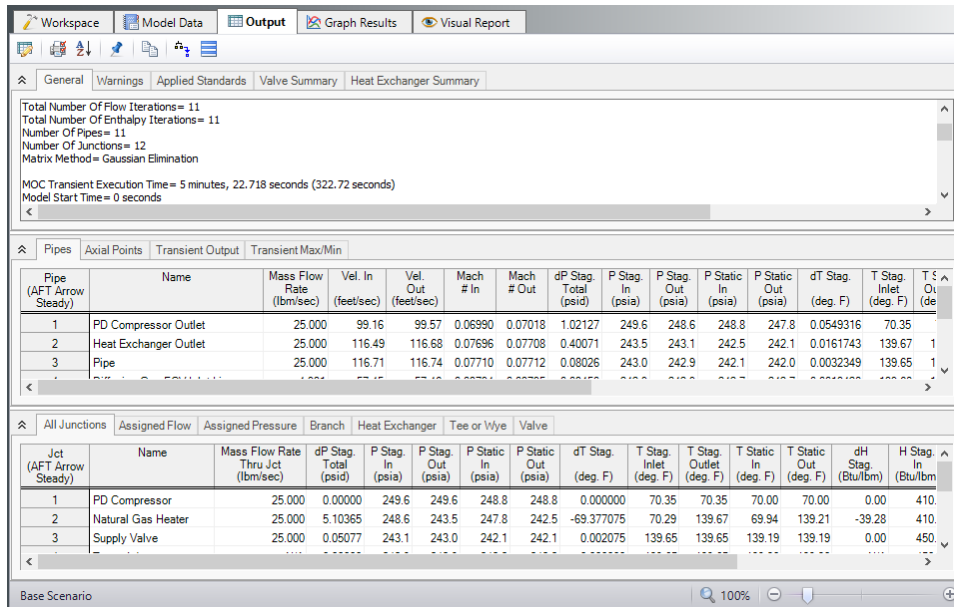


Figure 2: The Output window shows steady flow results in the Pipes and Axial Points tables and the various junction tables. Transient data is shown on the transient tabs.

## Transient Flow Results for Pipes

You can review the transient results for any pipe station for any time which was saved to file as specified in the [Simulation Duration panel](#). These results are shown in the Output window Transient Output table (Figure 3)

Figure 3 shows the transient flow results at the initial time step (which is equal to zero here). The columns of data shown in Figure 3 are specified in the Output Control window. Using the Time Slider will change what time step results are displayed.

Note that the pipe stations shown are those saved to the output file.

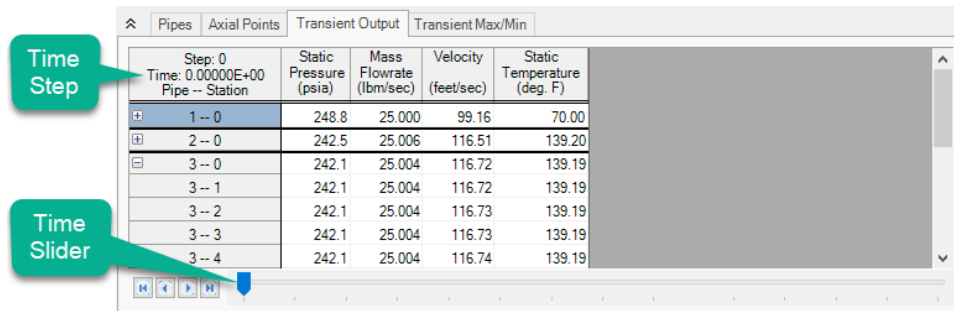


Figure 3: The Transient Output table in the Output window shows output results for pipe stations and at specified times

## Transient Max/Min Results for Pipes

You can review the maximum and minimum transient results for all pipes for all times. The data shown is not dependent on which pipe stations or time steps were saved to file. Figure 4 shows the Transient Max/Min table in the Output window. The parameters that are shown are the same as those specified in Output Control. However, there are four columns for each parameter. The four columns show the maximum and minimum values for the parameter, and the times at which each maximum and minimum occurs in the pipe.

Pipe -- Station	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Min. Static Pressure (psia)	Min. Static Pressure Time (seconds)	Max. Mass Flowrate (lbm/sec)	Max. Mass Flowrate Time (seconds)	Min. Mass Flowrate (lbm/sec)	Min. Mass Flowrate Time (seconds)	Max. Velocity (feet/sec)	Max. Velocity Time (seconds)
1 -- 117	273.9	2.000	247.8	0.000	25.006	0.000	17.3473	0.04518	99.59	0.000
1 -- 118	273.9	2.000	247.8	0.000	25.006	0.000	17.3353	0.04459	99.59	0.000
1 -- 119	273.9	2.000	247.8	0.000	25.006	0.000	17.3234	0.04419	99.59	0.000
1 -- 120	273.9	2.000	247.8	0.000	25.005	1.982E-04	17.3117	0.04360	99.60	0.000
2 -- 0	269.2	2.000	242.5	0.000	25.006	1.982E-04	17.3117	0.04360	116.51	1.982E-04
2 -- 1	269.2	2.000	242.5	0.000	25.005	0.000	17.7107	0.04340	116.52	0.000
2 -- 2	269.2	2.000	242.5	0.000	25.005	0.000	17.9505	0.04261	116.52	0.01684
2 -- 3	269.1	2.000	242.5	0.000	25.005	0.01605	17.9636	0.04162	116.53	0.01625

Figure 4: The Transient Max/Min table details the maximum and minimum values for the entire simulation

## Summary Max/Min Results

There is a summary version of the Transient Max/Min table, which is the default view. The version shown in Figure 4 is what is called the Detailed version. You can switch between detailed and summary views by selecting the option on the Output Control, Format & Action tab.

The summary version shows only maximum and minimum values for the entire pipe, as opposed to each pipe station. The summary also shows the pipe station that is the overall maximum or minimum for the pipe. Therefore, for each Output Control transient parameter there are six columns in the summary version of the Transient Max/Min table. See Figure 5 for an example.

Pipe	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station	Min. Static Pressure (psia)	Min. Static Pressure Time (seconds)	Min. Static Pressure Station	Max. Mass Flowrate (lbm/sec)	Max. Mass Flowrate Time (seconds)	Max. Mass Flowrate Station	Min. Mass Flowrate (lbm/sec)	Min. Mass Flowrate Time (seconds)	Min. Mass Flowrate Station
1	274.8	2.000	0	247.8	0.000	120	25.519	0.1058	11	17.312	0.04360	12
2	269.2	2.000	0	242.1	0.000	40	25.006	0.01526	5	16.038	0.02556	4
3	268.8	2.000	0	242.0	0.000	8	25.004	0.000	0	15.910	0.02180	0
4	269.0	2.000	0	242.7	4.162E-03	8	8.191	2.000	0	4.843	0.02735	0
5	229.5	0.02160	0	225.3	0.02973	0	8.187	2.000	0	4.921	0.000	0

Figure 5: Summary version of the Transient Max/Min table

## Transient Event Messages

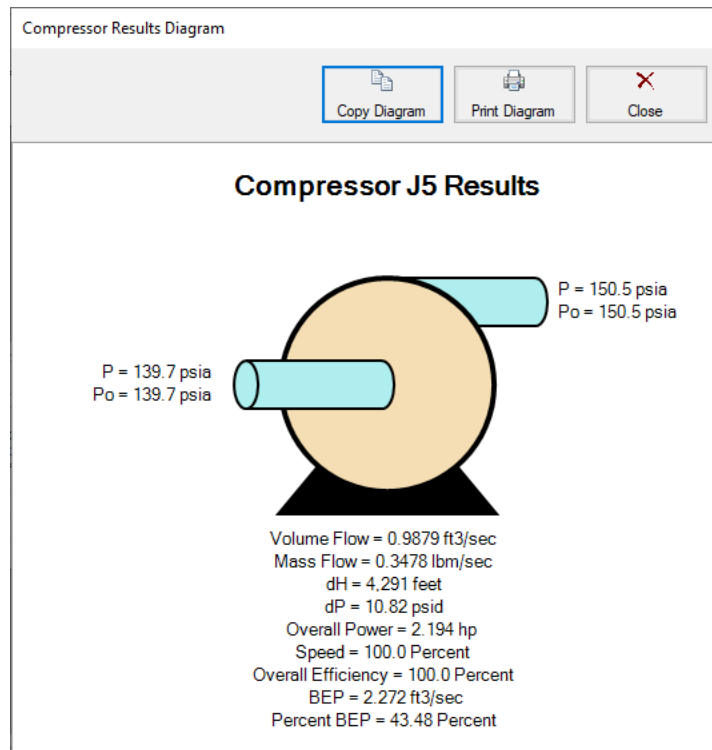
When events occur at junctions during the transient simulation, a log of all events is displayed in the two Event Message lists. In the first list events are sorted by junction number. In the second list they are

sorted by time. The list includes when check valves close (and re-open), relief valves open, control valves lose control (and regain it), as well as all standard [event transients](#).

### Results Diagrams

---

Results diagrams are available for steady state compressor/fan results (see Figure 6). These diagrams allow you display the information in a more intuitive manner, copy the image of the diagram to your clipboard, or print the diagram. The option to show the results diagram can be found in the compressor/fan summary tab.



**Figure 6: Compressor/Fan Results Diagram**

### Printing and Exporting

---

Data in the Output window can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Exported directly to Excel

- Saved directly to an Adobe PDF file
- Sent to the printer

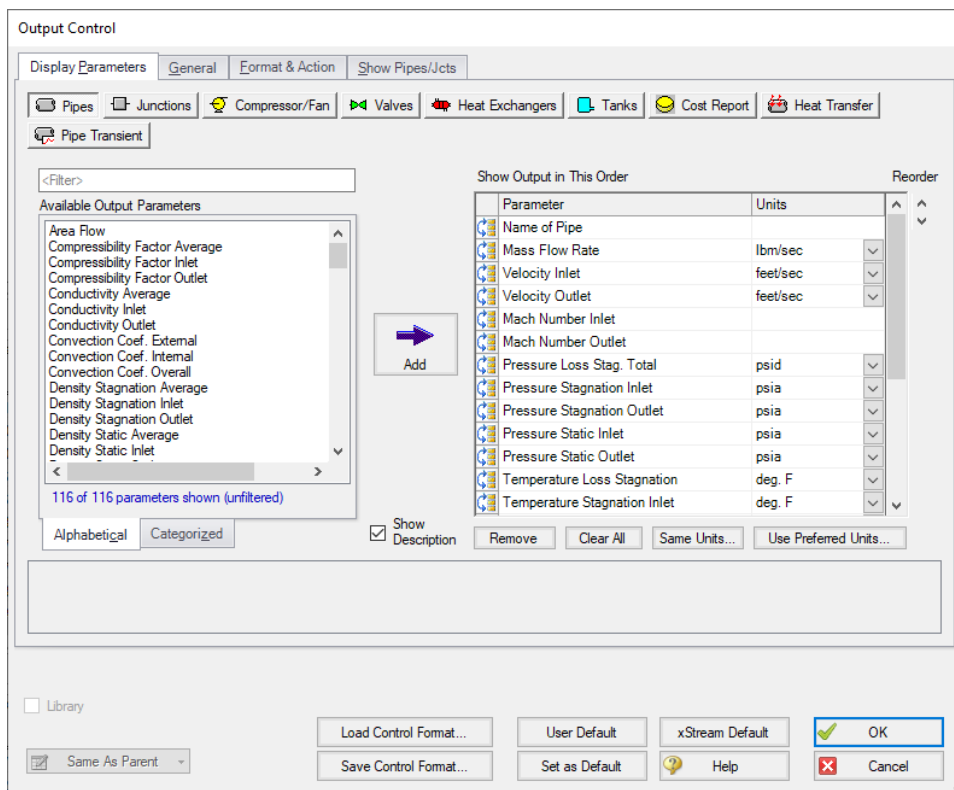
The printed and Adobe PDF output follows the same flexible, organized, and accessible format as the Output window itself.

## Related Topics

Exporting Force Data.....	34
Critical Warnings, Warnings, and Cautions.....	492
Applied Standards.....	311

## Output Control Window

The Output Control window (see Figure 1) allows you to customize the parameters shown on the [Output Window](#).



**Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe Output parameters are shown. The other output parameters function the same way**

### Display Parameters

---

The Display Parameters tab controls the parameters to be shown for the pipes, junctions and summary reports. The output type is selected from the group of buttons at the top. For each output type, the list on the left side shows all available parameters for the selected report. For the pipe and junction output, this list can be viewed in alphabetized or categorized mode, which makes it easier to find a parameter.

Choose a parameter to be shown by selecting it on the left-hand list and clicking the Add button, or double-clicking the parameter name. The selected item will appear in the list on the right either at the bottom or at the previously selected location. The list on the right shows the order of the parameters to be displayed in the output and the units (if any) in which to display the results.

You can change the order of the output by selecting a parameter on the right-hand list, then clicking the Reorder scroll bar on the far right or by clicking and dragging the row using the handle on the left. The list will be reordered accordingly. The output is presented from left to right in the order defined in this list.

The units used for a parameter can be changed by selecting the parameter then choosing the desired units from the adjacent drop-down list. By selecting the units of interest, you can obtain your results in whatever units you find most convenient and meaningful. This means that you can enter all your input parameters in one set of units (or a variety of units, for that matter), and have all your output parameters in a completely different set of units.

To set selected units for both pipes, junctions and summary reports to the preferred units (set in the [User Options](#) window), click the Use Preferred Units button. If you change the units of a parameter and you would like that unit to be used for all similar parameters in the list or entire output, select the Same Units button.

You will note that some output parameters do not offer the ability to specify output units. These parameters either do not have units associated with them (such as Reynolds Number or friction factor), or are in fact echoes of input parameters (such as Pipe Nominal Size).

Several Special Summary Reports can be displayed in the Output window. These include the following:

- **Compressor/Fan Summary**
- **Valve Summary**
- **Heat Exchanger Summary**
- **Tank Summary**
- **Pipe Transient Summary**
- **Forces Summary**

### General

---

The General Section is the text area located at the top of the Output window. The project title, reference information and solution balance display are controlled here.

You can enter a descriptive Analysis Title which will be used in the [Model Data](#), [Output](#) and [Visual Report](#) windows. A title is required and can have up to 100 characters.

You can keep a lengthy explanation or any other documentation about your model in the Reference Information section. Names of projects, individuals, and assumptions can all be kept with the model. This information can then be included in your reports to improve tracking.

There is an option "[Show Solution Balance Summary](#)" that is defaulted to off. This option toggles the display of a general output tab showing the total flow entering and exiting each junction, as well as the sum of those flows. This can be helpful in verifying the results as well as in some troubleshooting cases.

### Format & Action

---

The results can be automatically sent to the printer or saved to a file when the model has reached convergence. Additionally, the results can be automatically saved as initial conditions when the model is run. By saving the results as initial guesses, the iteration time for AFT xStream to arrive at a converged solution is reduced. The initial guess can also be transferred manually by selecting [Transfer Results to Initial Guesses](#) on the Edit menu or Toolbar. The model can also be automatically saved after the [initial conditions are transferred](#). The valve state (open, closed or failed) can also be saved along with the initial conditions.

When enabled, the Wrap Table Column headers option makes the output tables wrap the column headers as an extra line if the header is wider than usual.

There are four formatting methods for the pipe and junction output data:

- **Always Align Decimal Point on Output** - allows magnitude of the data to be easily compared as the data is read down the column. However, a number whose magnitude is small may force larger numbers to show many digits. For example, assume that the minimum number of digits is set to four, and there are two output values: 1234.5 and .09876. These two values will be displayed in the output as:
  - 1234.50000
  - 0.09876
- **Show Only Number of Digits (not aligned)** - forces the data to use exactly the number of digits specified, which may result in the decimal point location changing from value to value. Using this method with the above example and four digits, the output will be displayed as:
  - 1235.
  - 0.09876
- **Align Only When Max/Min < Limit**- aligns the data on the decimal point only if the maximum value in the column divided by the minimum value is less than a limiting order of magnitude that you set. If this ratio is greater than the limit, then each value in the column will be exactly the number of digits specified. Using the above example with a limit set to 4 then  $1234.5/.09876 = 12500$  or an order of magnitude of 5. This is greater than the limit. The entire column will not be aligned and will be displayed as:
  - 1235.
  - 0.09876
- **Use Exponential When Max/Min>Limit** - similar to the above method except that if the ratio is greater than the limit then the entire column will be shown in exponential format. Using the above example, the column would be shown as



- 1.235E+03
- 9.876E-02

These formatting options are for the entire Output window; however, the last two are calculated and applied on a column basis. In other words, if you choose the third option one column may be aligned but another may not be.

The output can be forced to always display in exponential notation by choosing the checkbox.

The Limit Event Options box can be checked to set maximum number of events which will be displayed in the Event Messages tabs that appear for cases such as valve transients. This can be useful in cases such as valve chatter.

For transient results, the maximum and minimum values for all selected parameters are shown in the Output window. The results can show the max/min values for all pipe stations, or only the max/min values for anywhere in the pipe. You can choose which type of report you want by selecting Detailed or Summary in the Max/Min Transient Table Contents area. For models with a large number of pipe sections, the Summary report is generally preferable because of the time required to display the Detailed table.

The Grid Headers and Design Alert highlight colors can be configured to create a visually appealing output for ease of reading.

The box next to Show Results Diagram Column can be unchecked to hide the column which allows easy access to the Compressor Diagram for Steady State compressor output.

### Show Pipes/Jcts

---

You can choose which pipes and junctions are to be displayed in the Output window. Clicking a selected item in the list on the Show Pipes/Jcts tab will deselect it. Clicking a deselected item will select it. This feature is especially useful when you are interested only in a certain area of the model.

You can quickly define the output you want by doing the following:

1. Select the pipes and junctions on the Workspace
2. Open the Output Control window and select the Show Pipes/Jcts tab
3. Click the Workspace button for both the pipes and junctions
4. Click the Show button

### Library Connections

---

In the lower left of the Output Control window there is a Library checkbox. If this box is checked, your Output Control parameters are setup as determined by the library to which you are connected. This is referred to as an active library. To make it inactive, un-check the box or change one of the Output Control settings controlled by the library.

If the check box is unchecked, but enabled, you are connected to a library but the settings are not being passed to the Output Control window. The library is thus inactive. To make it active, check the box then click the OK button.

If the check box is disabled, there is no connected library.

### Command Buttons

---

There are eight buttons at the bottom of the Output Control window. xStream has built-in default parameters, units and settings which you can choose by clicking the xStream Default button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of xStream's defaults). To make your own default, first select the output parameters, units and settings you would like to use then click the Set As Default button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the User Default button. The saved settings are updated only when you click Set As Default.

You can save the output settings (except for the selected pipes/junctions) to a file by pressing the Save Control Format button and entering a file name. These settings are loaded again by pressing the Load Control Format button and choosing the file name. For example, you may have a final report format that is always desired. You can load this format before generating final results. If you have another format you use for reviewing model accuracy, you may want a larger number of parameters in the output.

The format files you create can be placed on a network for sharing among a group or company.

If you have made changes you don't want to keep, click the Cancel button. Click OK to use the setting you have specified

### Related Blogs

---

[Quick Guide to Custom Output](#)

[AFT Responds to 2020 Customer Survey Feedback](#)

## Output Parameters

A variety of parameters are available to display in the Output window depending on the junctions and loss models used in the model.

A description of each parameter may be seen by double-clicking the header in the Output Window table, and is also displayed below the Output Parameters table when that parameter is selected in the [Output Control window](#) (Figure 1).

Additional information can be found below for some of the output parameters.

Output Window Table Header:	<a href="#">Flow Area</a>
Visual Report Abbreviation:	A
Description:	<a href="#">Pipe cross-sectional flow area</a>

**Figure 1: A description is provided at the bottom of Output Control for the selected parameter**

---

## Cv Est./Kv Est.

---

The estimated Cv is only displayed when the valve does not have a defined Cv loss value. The estimated Cv/Kv loss coefficient is estimated using the sizing equations from the [ANSI/ISA-75.01.01-2012](#) standard, as are discussed in the [Valve Kv and Cv Loss Model](#) topic.

---

## Mass Flow Rate Net At Junction

---

Displays the total mass flow rate exiting the system through the junction, which is useful to display the flow exiting through junctions such as a relief valve. In other words, the Mass Flow Rate Net At Junction is equal to the flow rate into the junction from the pipes minus the flow rate out of the junction into the pipes.

---

## Mass Flow Rate Through Junction

---

Displays the total mass flow rate through the junction for junctions connected to only two pipes. If the junction is connected to more than two pipes this parameter will be displayed as N/A. When more than two pipes are connected the user should refer to the output for the pipes connected to the junction in order to view the flow rate through the pipes. The user can add the "Mass Flow Rate Net At Junction" to view flow exiting the system rather than flowing out of the junction into a pipe.

---

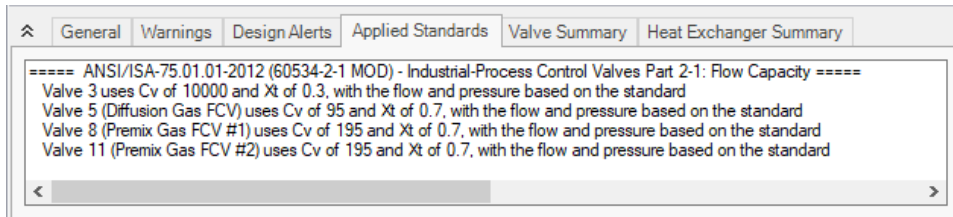
## Related Topics

---

References..... 461

## Applied Standards

When an applicable standard is used in a calculation, an additional tab appears in the general area of the [Output Window](#). This tab indicates what standard(s) were used for what calculations in the model.



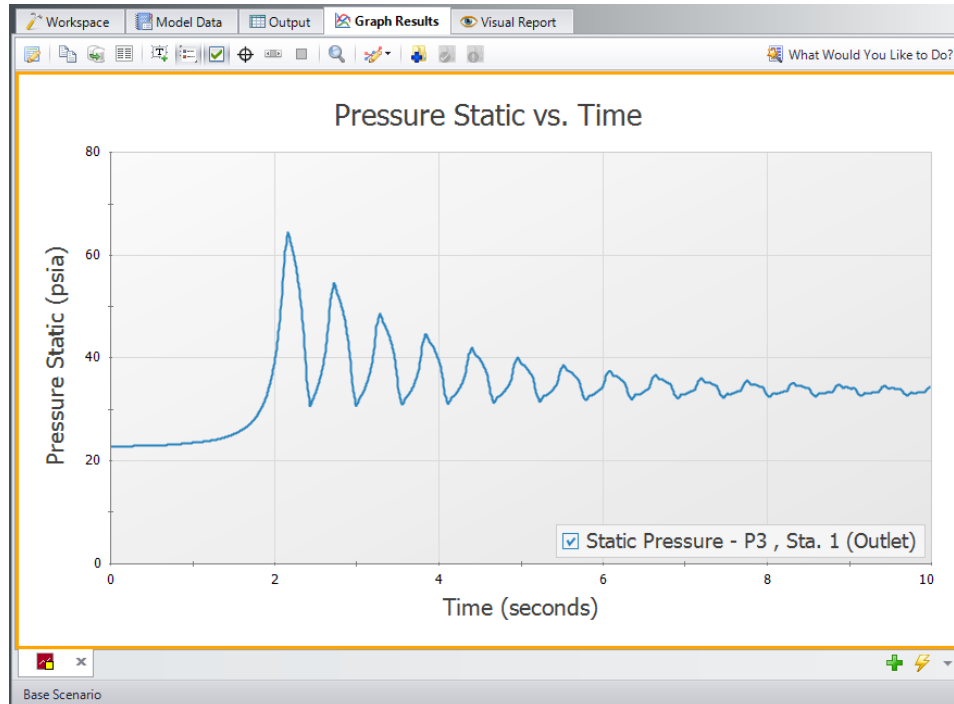
*Figure 1: Applied Standards when applied for Cv/Xt loss at a valve*

## Graph Results Window

The Graph Results window, shown in Figure 1, allows users to generate high quality printed graphs.

The Graphing Panel offers [several types of standard graphs](#) for transient pipe or junction, profile, and force information. Users can also control graph content, plotting information for multiple parameters, junctions, pipes, paths, or scenarios simultaneously.

Results can be graphed as linear or log versus linear.



**Figure 1: The Graph Results window offers full-featured plot generation**

## Graph List Manager

Allows the user to create, update, and even replace unique graphs that can be regenerated with new data. Multiple graphs can be displayed at the same time and organized into separate Folders. The default folder is named "My Graphs" and more folders can be created using the "New Folder..." icon on the Graph List Manager Toolbar or by right clicking on "My Graphs" and selecting "New Folder..."

Once a folder is created and selected, a Graph can be added to its list by selecting the "Add Graph to List" icon found on the Graph Results Toolbar. This can also be done by right-clicking anywhere on the graph. Once Graphs are in Folder individual Graphs can be generated in unique tabs or each Graph in a Folder can be displayed in a single tab by right clicking on the Folder and selecting "Load Current Folder Graphs in New Tabs" or "Load Current Folder Graphs in Same Tabs". These options are available in the Graph List Manager toolbar as well. Other options that can be found by right clicking on a Folder or Graph in the Graph List Manager or on the toolbar are:

- **Load Graph in Current Tab** – Regenerates the graph in the currently active tab.
- **Load Graph in New Tab** – Creates a new tab and regenerates the graph there.

- **Load Current Folder Graphs in New Tabs** – Regenerates all the Graphs in a Folder in separate tabs.
- **Load Current Folder Graphs in Same Tab** – Regenerates all the Graphs in a Folder in a single tab.
- **Keep Folder Display Layout** – Saves the current layout in the Graph Results window if multiple graphs have been loaded into one tab from a folder.
- **Reset Folder Display Layout** – Resets the layout in the Graph Results window if multiple graphs have been loaded into one tab from a folder.
- **Print Preview Selected Item** - Opens graph item directly in the Print Preview window.
- **Move Graph Up** – Moves the selected Graph up one level.
- **Move Graph Down** – Moves the selected Graph down one level.
- **Rename Graph...** – Changes the name of the Graph.
- **Duplicate Graph** – Creates a copy of the selected Graph with a new name.
- **New Folder** – Creates a new Folder where more Graphs can be saved to.
- **Delete Graph** – Removes Graph from the Graph List Manager.
- **Move To Folder** – Moves the selected Graph to a previously existing Folder or a new Folder.

### Graph Results Toolbar

---

The Graph Results Toolbar also has many options that are available by right clicking on the Graph Results space at any point. These options are:

- **Select Graph Parameters** – Select the type of Graph to create and its parameters.
- **Add Graph To List** – Saves the Graph to the currently active list allowing it to be regenerated in the future.
- **Update Current Graph In List** – Saves any changes to the current graph that have been made. (e.g. Changing the Graph Title or Line Style). Note that it is preferred to use the Replace Graph in List option if changes to the parameters or objects/paths being shown have been made.
- **Replace Graph In List** – Replaces the graph highlighted in the Graph List with the current graph that is shown.
- **Copy Graph Image** – Copies a .jpeg of the Graph to the clipboard
- **Copy X-Y Data** – Copies the X-Y used by the Graph to the clipboard
- **Show Data** – Displays the X-Y data used to generate the graph in a column to the right of the graph
- **Create Annotation** – Add an annotation to the graph that can be formatted and saved to be reloaded with the Graph in the future.
- **Show Legend** – Toggles the legend off and on.
- **Show Check Boxes in Legend** – Adds a check box in the legend that allows the user to selectively turn off and on certain curves in a Graph without having to recreate a new Graph.
- **Show Cross Hair** – Displays and cross hair that shows the cursors X-Y values along the X and Y axis. This also displays the nearest data point X-Y values in a tooltip. If this feature is used at the

same time as "Show Data" the Cross Hair will focus on that highlighted data point and display its values in a tooltip. The user can also change what data point is highlighted with the arrow keys and the Cross Hair and tooltip will move to that data point.

- **Show Range Finder** – Adds a toolbar at the bottom of the graph that displays the entire X axis, but allows the user to zoom in to a specific range by clicking and dragging the handles on either side of the range finder.
- **Show Junction Locations** – Add a vertical line with the number of the junction labeled at the location of the junction along the flow length.
- **Manage Junction Locations** – Allows junction labels on the graph to be shown/hidden individually.
- **Zoom to 100%** – Resets the focus of the graph to 100%
- **Format** – Allows the user to change the Graph's format. Including
  - Header, Axis, and Legend names and fonts.
  - X and Y scaling
  - Line color and thickness

### Graph Outline Colors

---

The active graph will always have an outline which indicates what state it is in.

- **Orange** – Graph not added to list
- **Dark Green** – Graph added to list
- **Light Green** – Graph added to list, but changes have occurred. These can be saved and make they graph dark green by selecting "Update Current Graph in List".

### Graph Tab Icons

---

Each Graph Type has a specific color associated with it. This color will display in the box on the tab for the Graph. The colors are below:

- **Transient Pipe** – Dark Red
- **Transient Junction** – Dark Gray
- **Profile** – Purple
- **Forces** – Yellow
- **Frequency** – Light Brown
- **Blank Graph** – Light Gray

These tabs also display if the Graph in the tab is not saved or has been changed from the original Graph List item by showing a small yellow square over the tab's icon. If the current graph has been saved, there will be no yellow square.

### Graph Tab Actions

---

- **New Tab** – Creates a blank tab in which a new type of Graph can be generate and added to the Graph List Manager
- **Tab Actions**
  - Close Active Tab
  - Close All Tabs
  - Close All But The Active Tab
  - Two Vertical Tab Groups
  - Two Horizontal Tab Groups
  - Reset to Single Tab Group
  - Tabs on.... – Selects the location of tabs (defaults to the bottom of the Graph)
- **Graph Tab Selection** – Selects a Graph for the Graphs currently in the tabs based on name.

### Printing and Exporting

---

The Graph Results image can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The x-y data in the Graph Results graph can be:

- Copied to the clipboard
- Exported to a delimited file suitable for direct importing into spreadsheet software

[Animations](#) of graphs can be saved as .wmv or .mp4 files.

### Data Point Limitations

---

By default, the maximum number of data points xStream will plot for an individual graph is 500,000, though users can change this limit under the General -> Other section of the [User Options window](#). This limit is in place to help ensure users do not encounter memory issues related to plotting large numbers of data points. The allowed range of data points plotted is 10,000 - 1,000,000. The data point limit is applied to an individual graph. Two parameters plotted as stacked graphs count as two separate graphs. Two parameters plotted with one as a secondary y-axis graph count as one graph. One parameter plotted for two scenarios on the same graph counts as two graphs (i.e. the data point limit is essentially doubled from a single-scenario graph to a two-scenario plot graph).

When a user attempts to generate a graph that exceeds the maximum number of data points, xStream will skip data points in order to remain under that limit. The frequency of skipped data points is proportional to the amount by which the generated graph exceeds the maximum number of data points. When data points are skipped, there is the potential for important data points to be excluded from the graph such as maximum or minimum values.

Users can take the following steps to avoid running into this issue, and to double-check that the graph gives an accurate representation of the results and does not skip any maximum or minimum data points:

- Increase the limit in the [User Options window](#) (the time required to load a graph will increase)
- Plot fewer curves in the graph
- Graph data for limited amounts of time rather than the entire simulation time
- Save fewer output data points from the [Simulation Duration panel](#) in the Analysis Setup window
- Double check results shown in the graph with transient output shown in the Output window. Users can either reference the Transient Output, showing output values for every time step and pipe section data is saved for, or examine the Transient Min/Max values shown for the parameters selected in the [Output Control window](#).

### Related Topics

---

Animating Graph Results.....39

### Related Blogs

---

[Quick Guide to Custom Output](#)

#### **SSL Module**

#### **Settling Slurry Module**

An optional add-on module to AFT Impulse which allows users to simulate settling slurries.

#### **PFA Module**

#### **Pulsation Frequency Analysis Module**

An optional add-on module to AFT Impulse which allows users to determine resonant frequencies in their system.

### **Quick Graph Formatting Window**

The Quick Graph Formatting window allows you to customize the plot by giving you control over the many aspects including:



- Axis titles and font sizes
- Legend text and font size
- Line thicknesses and basic color sets
- Axis min and max values

You can save the settings you set up by clicking the Save button. If you make changes to the settings, and want to get back to previous settings, click the Load button.

## Graph Types

AFT xStream offers several types of graphs to help users better understand and interpret their data. Each type of graph is given a brief description below.

### Transient Pipe

Transient pipe results can be plotted against time for any pipe station that is saved to the output file. By default, all stations for each pipe will be saved to the output file. The amount of pipe data saved can be changed in the [Simulation Duration panel](#).

The user can plot the data for the entire simulation, or for a user specified time frame.

One or more parameters can be graphed by using the Add button with the green plus sign. The Remove button can be used to eliminate extra parameters. Parameters can either be plotted in stacked graphs, or two parameters can be plotted on the same graph using the secondary axis. To move a parameter to the secondary axis, add it below the parameter that will be on the primary axis, then click the right blue arrow button, or double-click on the gray square to the left of the parameter. This will cause the parameter to be indented, as shown in Figure 2.

Further, if [Design Alerts](#) are specified for the pipe (e.g., max/min operating pressures), these can be cross-plotted against the transient data by selecting the check box under "DA".

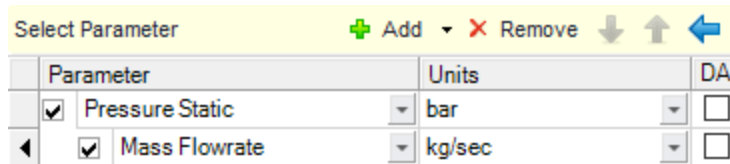


Figure 1: Creating a graph with a parameter on the secondary axis

### Transient Junction

Similar to transient pipe results, transient junction results can be plotted vs. time for any junction that is saved to the output file. Junction data includes items such as compressor speed and valve Cv.

### Profile

---

This graph category allows users to plot the selected parameter(s) along a flow path. In order to generate this graph, the specified pipes in the Select Graph Parameters window that comprise the path must form a single, continuous path. If they do not, an error message will appear listing the pipes which are preventing the path from being continuous. The independent variable in these graphs is the length along the flowpath.

A path of pipes can be specified in one of two ways. First, the user can select the pipes in the displayed list. Additionally, you can select the path on the workspace, and then use the Workspace button on the Select Graph Parameters window to automatically select all pipes currently selected on the Workspace. These methods permit only a single path to be specified.

One or more flow paths can be specified using [groups](#). Only those groups that consist of a continuous pipe sequence will be displayed as an option in the Select Graph Parameters window.

The results can be the values at a particular point in time, or the overall maximum and/or minimum values. You can also cross-plot [Design Alerts](#), the max/min values, and the steady-state results. This is done clicking the check boxes underneath "Mx" for maximum, "Mn" for minimum, "SS" for steady-state, and "DA" for design alerts.

All profile graph types support animation. Animating graphs is discussed further in [Animating Transient Results](#).

### Forces

---

If transient force sets have been defined in the [Forces panel](#), the resulting force data can be plotted vs. time. Force sets selected from the list of defined force sets will be plotted. The unbalanced transient force data can be plotted for the entire transient simulation, or the user has the option of selecting a specific time frame to display.

This force data can also be [exported](#) for analysis in pipe stress analysis software.

### Frequency

---

([PFA module](#) only) As part of the pulsation frequency analysis, the tab will appear to plot the frequencies identified for the system.

### Multi-Scenario Graphing Features

---

Users can plot information from multiple scenarios in the same plot. In the Select Graph Parameters window, the button for 'Multiple Scenarios' allows users to select the scenarios in the model they wish to plot for a particular graph. The scenarios being plotted in the graph must have the same pipe layout for all scenarios, but can have different sectioning in AFT xStream 2.

When creating multi-scenario profile graphs, the current scenario will determine start/stop time, time step, junction label locations, and design alerts that are shown on the graph. For multi-scenario graphs with different time steps/sectioning groups for multiple paths will not be allowed.

The active scenario must be included in a multi-scenario graph. When creating a new graph, users are not able to deselect the current scenario. When a user loads a saved Graph List Item that does not include the active scenario, xStream will add the active scenario into the Graph List Item, but will not save the active scenario into it.

Scenarios can be selected using the Multi-scenario... button as shown in Figure 2 below.

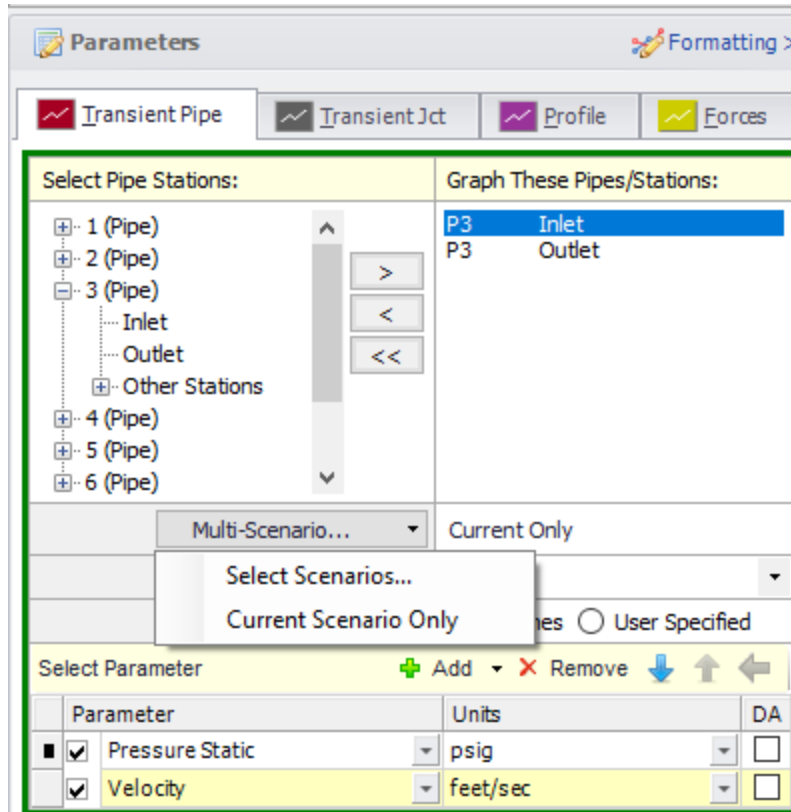


Figure 2: Graph Parameters window showing Multi-scenario selection options

## PFA Module Pulsation Frequency Analysis Module

An optional add-on module to AFT Impulse which allows users to determine resonant frequencies in their system.

## Basic Graph Tutorial

Generating graphs is a powerful tool for analysis. Below are step by step instructions about how to make and manage graphs from one of the example models. Many of the graphs created in this example will be multi-scenario graphs. However, the information discussed within is the same regardless of which scenarios are selected.

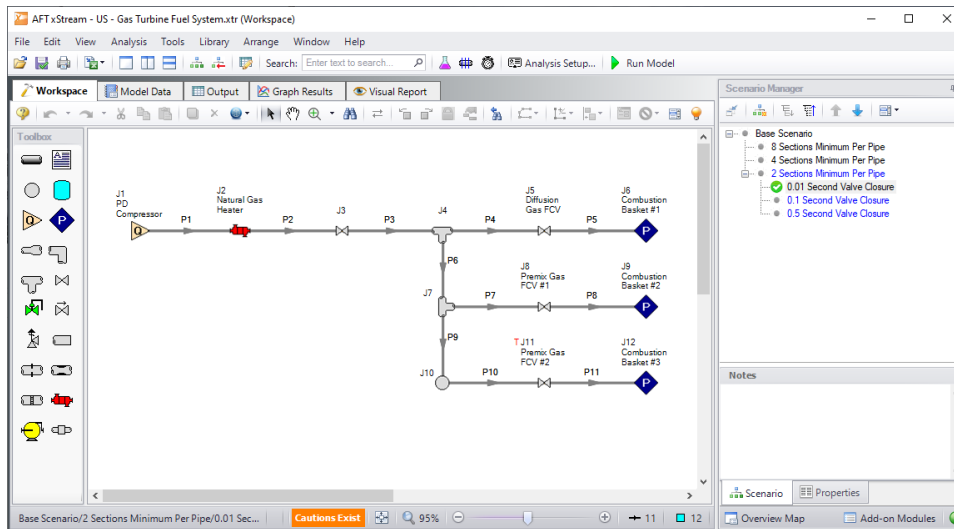
For this example, the US - Gas Turbine Fuel System model file was used. A copy of the completed model can be downloaded from the [Gas Turbine Fuel System example](#), or can be found in the AFT xStream installation folder.

Open the model and create three children of the 2 Sections Minimum Per Pipe scenario. The three children scenarios evaluate three different closure times of the Premix Gas FCV #2 valve (J11): 0.01 seconds, 0.1 seconds, and 0.5 seconds. Run the three scenarios and return to the 0.01 Second Valve Closure scenario.

To skip to different sections of this tutorial, use the links below:

- [Transient Junctions and Saving Graphs](#)
- [Transient Profiles and Animations](#)
- [Displaying multiple graphs simultaneously](#)
- [Creating graphs from the Workspace](#)

## Transient Junctions and Saving Graphs [\(return to top\)](#)



**Figure 1: High Pressure Steam Example**

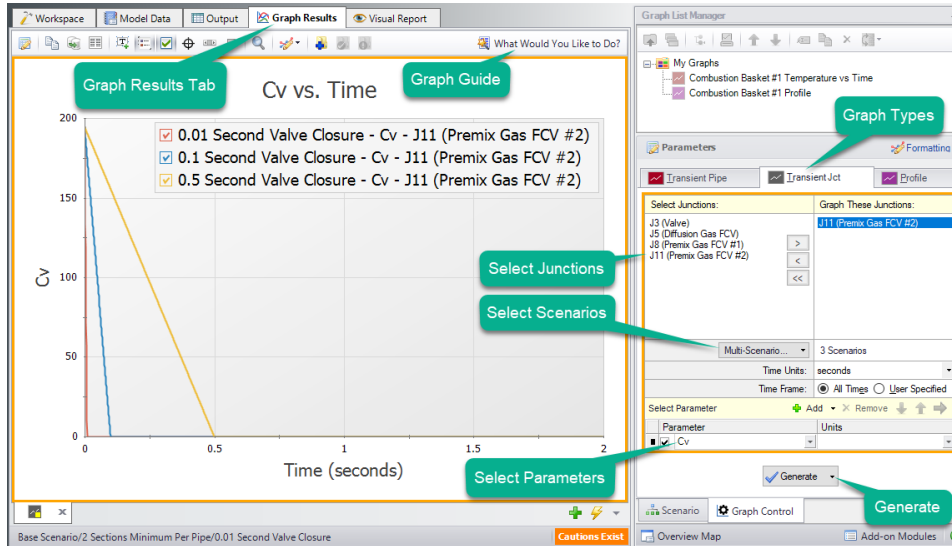
Go to the Graph Results Primary Window and determine what type of graph you want to generate. The first graph created in this example will be a Transient Junction Graph. The Transient Junction is one of several different types of Graphs that can be made. This graph is created by first selecting the Transient Jct tab and selecting the desired parameters. The only junction with saved transient data will be the valve J11. Add this junction to the “Graph These Junctions:” column by double-clicking the junction ID number, or selecting the “>” symbol. Select “Cv” as the parameter. Select all three valve opening children

## Interface and User Options

scenarios using the Multi-scenario... button, and then select "Generate". This creates a plot of "Cv vs. Time" for all three scenarios on the left.

Note that the Graph Guide, accessed by clicking on the "What Would You Like to Do?" icon on the upper right of the graph, will guide you through the process of creating a graph

**Note:** The model file used in this tutorial does not have force sets applied, so the Forces graph tab is not available in the Graph Parameters area. The Forces tab will be available if [force sets are defined](#) and applied in the Analysis Setup window.



**Figure 2: Valve Cv vs. Time**

The Graph has been created, but has not yet been added to the list under "My Graphs". To add this Graph to the list, first select the "Add Graph to List" icon on the Graph Results toolbar (this option and many others can be accessed by right-clicking on the Graph at any time). Name the Graph "Transient Junction - J11 Cv". The Graph has now been saved and will be easily regenerated with the same format if the output changes. The name of the saved Graph can be seen in the Graph List Manager, with two small squares in the icon associated with it indicating it is a multi-scenario graph. The border of the Graph has also changed to dark green along with borders of the parameters and the name of the Graph in the Graph List Manager, indicating that the Graph is current, saved, and that the displayed parameters are the ones used to generate the Graph.

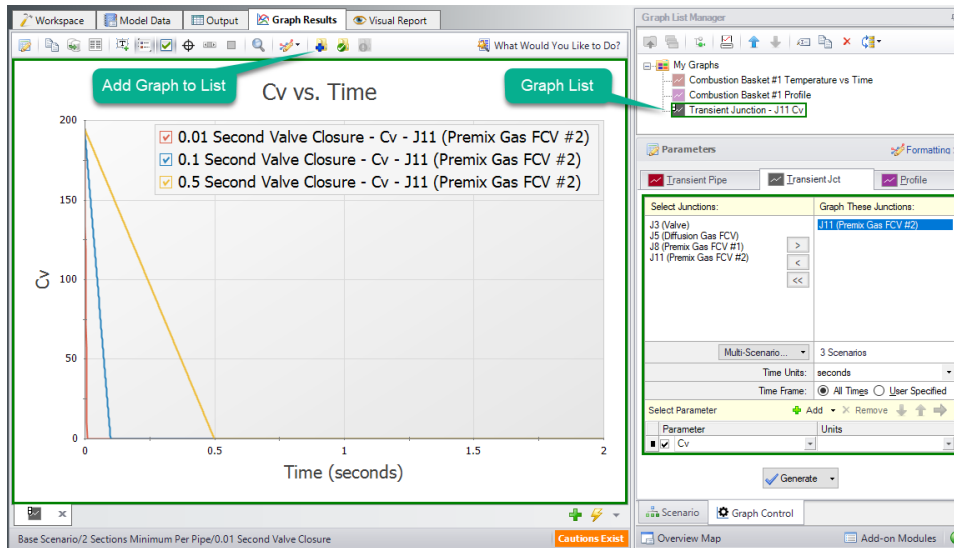


Figure 3: Adding a Graph to My Graphs

Formatting options are available by selecting “Formatting” above the different graph types. When formatting changes occur the border of a Graph will change from dark green to light green, indicating a change has been made to a graph in the Graph List. Another indicator that the Graph is not saved is the small yellow square over the icon on the tab. Here, the title has been changed from “Cv vs. Time” to “Valve J11 Cv vs. Time”. Once the formatting settings have been changed, this change and other formatting changes are not retained to the Graph unless “Update Current Graph in List” is selected.

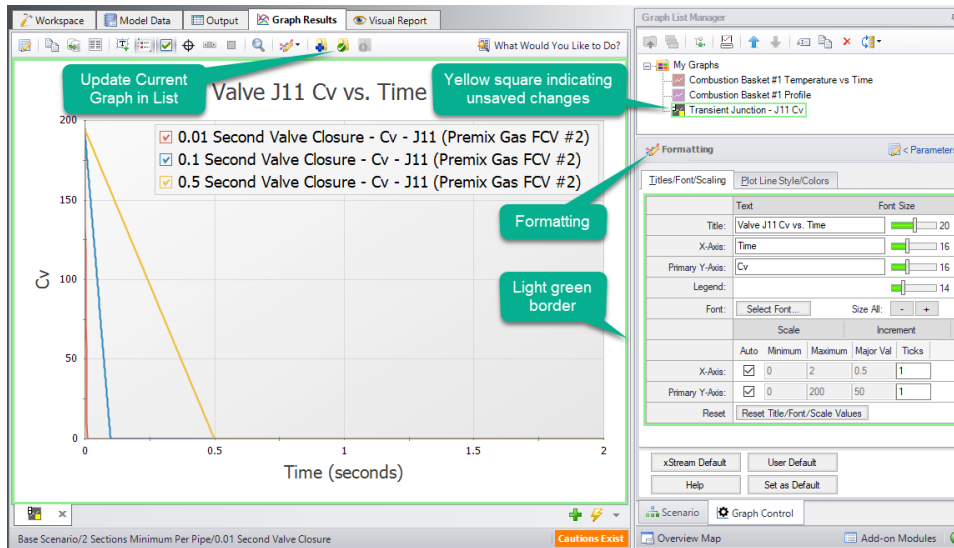


Figure 4: Update Formatting Changes to Graphs

After selecting “Update Current Graph in List” the border has changed back to dark green and the yellow square has disappeared. This indicates that this is the current formatting of this graph, and that regenerating this graph will generate the same graph (according to the current model output).

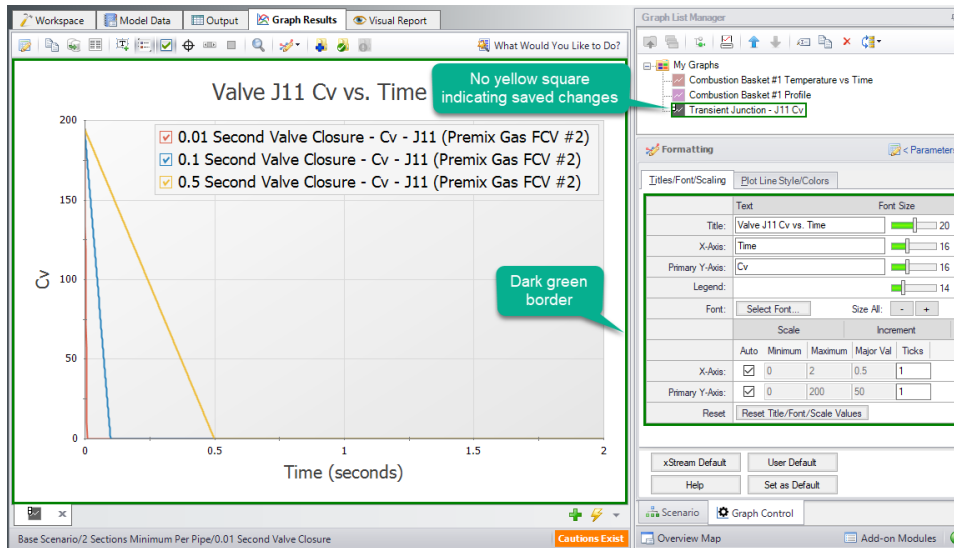
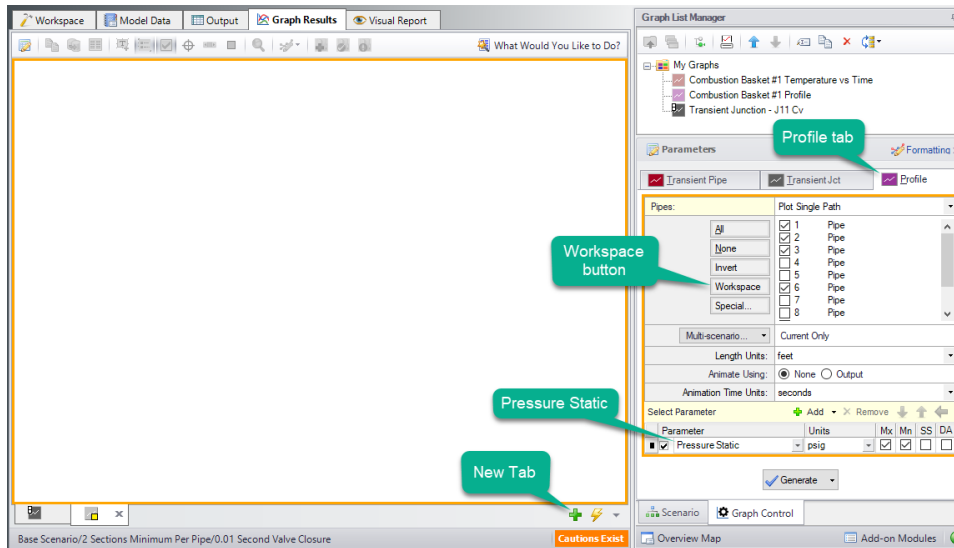


Figure 5: Valve Cv graph with updated formatting changes

## Transient Profiles and Animations [\(return to top\)](#)

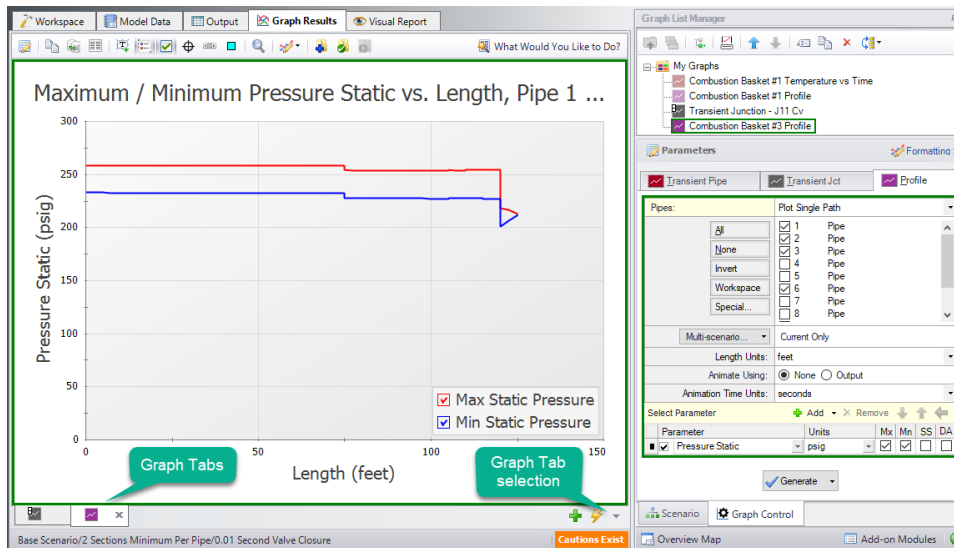
The next step in the example is to make a new Graph in a separate tab. This is done by selecting the "New Tab" icon in the lower right. This Graph will be a "Profile" Graph that displays Static Pressure along the length of the flow path from the compressor through J11 and to Combustion Basket #3. Select pipes 1, 2, 3, 6, 9, 10, and 11 in the system and change the parameter to "Pressure Static" with units of length in feet and units of pressure in psig. Make sure that the Mx and Mn boxes are checked to display the maximum and minimum static pressures. Make sure "Animate Using" is set to none.

**Note:** An easy way to select the pipes in a longer path would be to select them in the Workspace, then click the Workspace button under Pipes to automatically select the desired path.



**Figure 6: Generating a Profile plot with Min and Max Pressure Static**

The graph shows the minimum and maximum static pressures during the transient simulation for each pipe station along the flowpath. Save this new graph as a new Graph List item. The two graphs we have made can be switched quickly between by either selecting the graph from the list, by clicking on the corresponding tab, or using Graph Tab Selection.



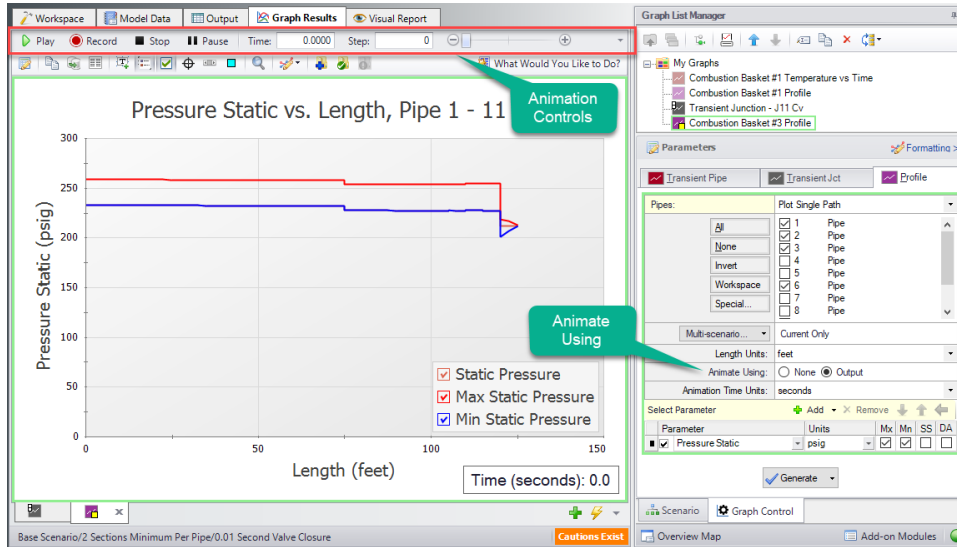
**Figure 7: Pressure Static vs. Flow Length Profile added to Graph List**

This is a useful graph, but animating the pressure profile over time can provide even more information. Animating from the Output file requires that all data for the pipe stations being animated to be saved to the Transient Output File in the [Pipe Sectioning and Output panel](#), which is done by default.

Select the Animate Using Output option, then click generate.



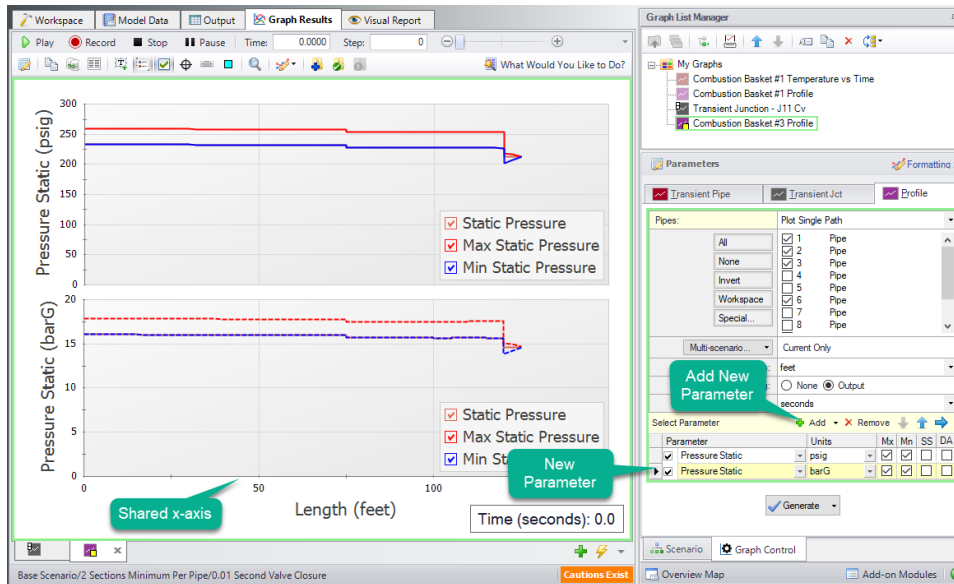
After selecting generate the Graph will now allow for animation with options to Play, Record, Stop, and manipulate the animation in other ways. The recorded animation can be exported to a video file for convenience.



**Figure 8: Max/Min Profile Graph with Animation**

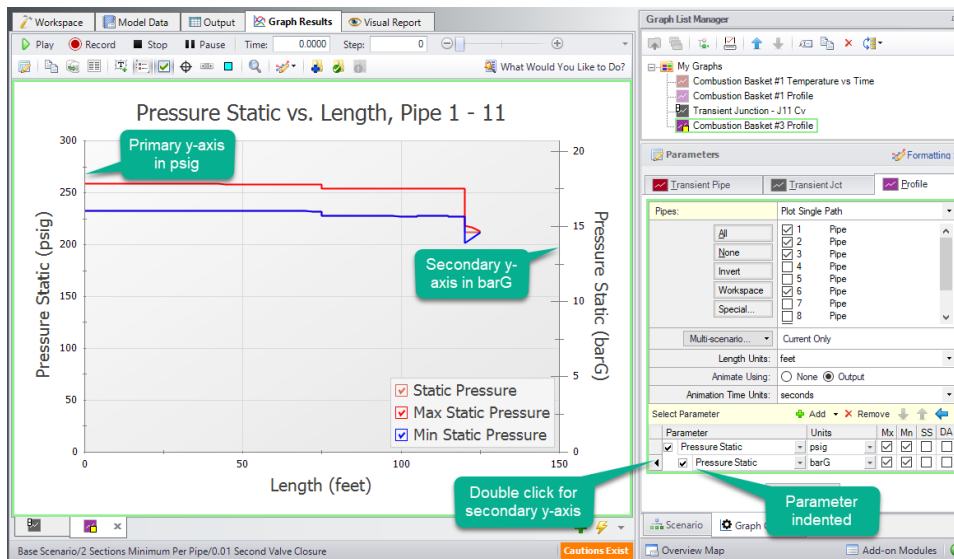
## Displaying Multiple Graphs Simultaneously [\(return to top\)](#)

Another option that is available is to add another parameter to the graph. This is called generating stacked graphs. To do this select "Add" next to "Select Parameter". This will graph an additional parameter that shares the same X-Axis as the original parameter. For this example, select "Pressure Static" as the parameter, but with units in barG rather than psig.



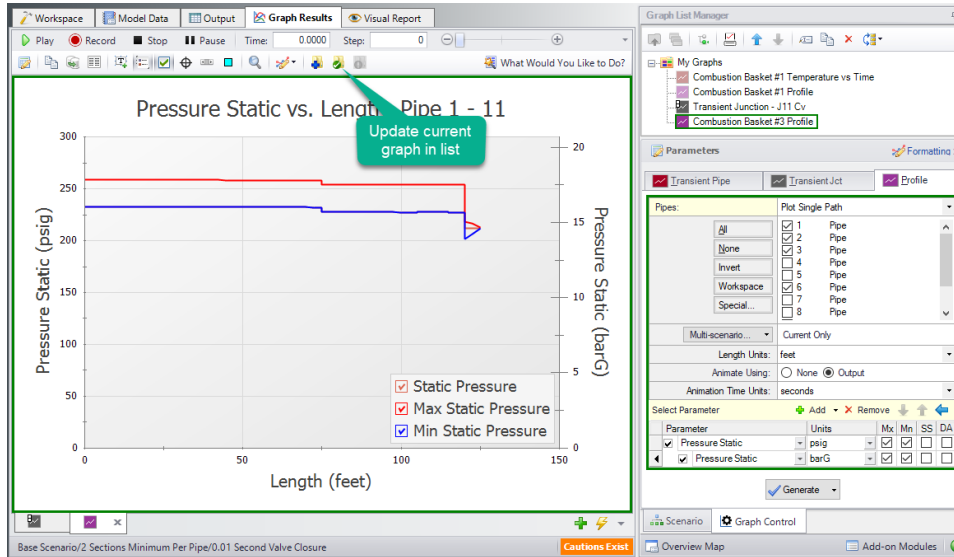
**Figure 9: Stacked Pressure Profile with different units**

We now have the same plot twice, which is not the most efficient way to view this data. A better way would be to have the secondary parameter (which could be any of the available options for a profile plot) on a secondary y-axis. This can be done by double clicking on the arrow to the left of “Pressure Static: barG” or by selecting the blue right arrow on the “Select Parameter Row”. This will visibly indent the secondary parameter. When the Graph is regenerated it will now only be one plot of Pressure Static vs. Length, but it will have units displayed in psig on the primary y-axis and units of barG on the secondary y-axis.



**Figure 10: A Single Graph with Alternate Units on a Secondary Y-Axis**

Update the Current Graph in List using the toolbar button. The Graph is now complete and can be easily regenerated if input data is changed on the Workspace and the model reran. It can also be used to generate the same graph for data from other scenarios.

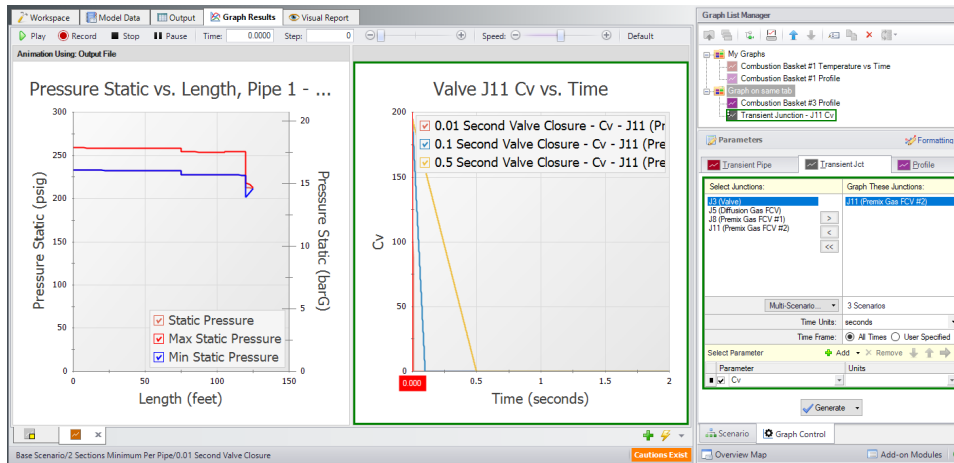


**Figure 11: The Resulting Graph when it is Updated**

It is now possible to close the Graphs and regenerate them either in their own tabs or all graphs in a folder in the same tab. To test this out, close both Graph tabs and then create a new "Graph on same tab" folder from the right click menu. Right click each of the graphs made in this tutorial, and choose to move the graph list items to the "Graph on same tab" folder.

Right-click on the "Graph on same tab" folder and then choose "Load Folder Items" and then "Same Tab". After the Graphs have been loaded in the same tab, the tab icon changes to an orange square, indicating that multiple Graphs are being displayed on one tab. All graphs will now be animated, with a vertical red line appearing on the graphs with a time axis. This red line will move over the animation time to compare values more readily.

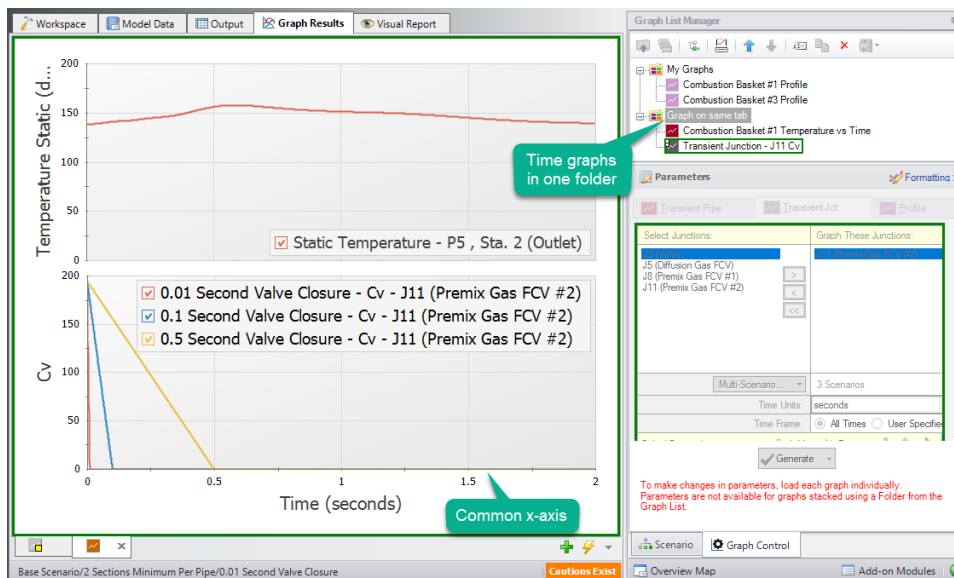
Depending on the monitor display being used, or the number of graphs being displayed, this view may be easy to view or difficult to see data. For the sake of this example, we've kept the display size relatively small, meaning it is more difficult to view the data on all the graphs. However, with larger monitors available, it is likely easier to view this data.



**Figure 12: All graphs loaded into the same tab**

Other ways to view multiple graphs simultaneously are to plot them as stacked graphs, as we did previously for our pressure profile in psig and barG, or to plot saved graph items in two tabs either stacked or side-by-side. Right-clicking on a Graph-Tab at the bottom of the window shows the options for displaying two separate tabs.

If we have two Time graphs with the same x-axis, we can display them in a single, stacked graph. This approach to viewing graphs can be especially helpful in examining how a junction transient is affecting output parameters nearby. To create a stacked graph of multiple Graph List Items, first make sure both items have the same time scale. Then, move them into their own folder. Here, we've moved the Combustion Basket #3 Profile graph back to the My Graphs folder and moved the example's Combustion Basket #1 Temperature vs Time graph to the "Graph on same tab" folder. Finally, right-click the "Graph on same tab" folder, select "Load Folder Items", and then the "Stack Common X-Axis in Same Tab" option.

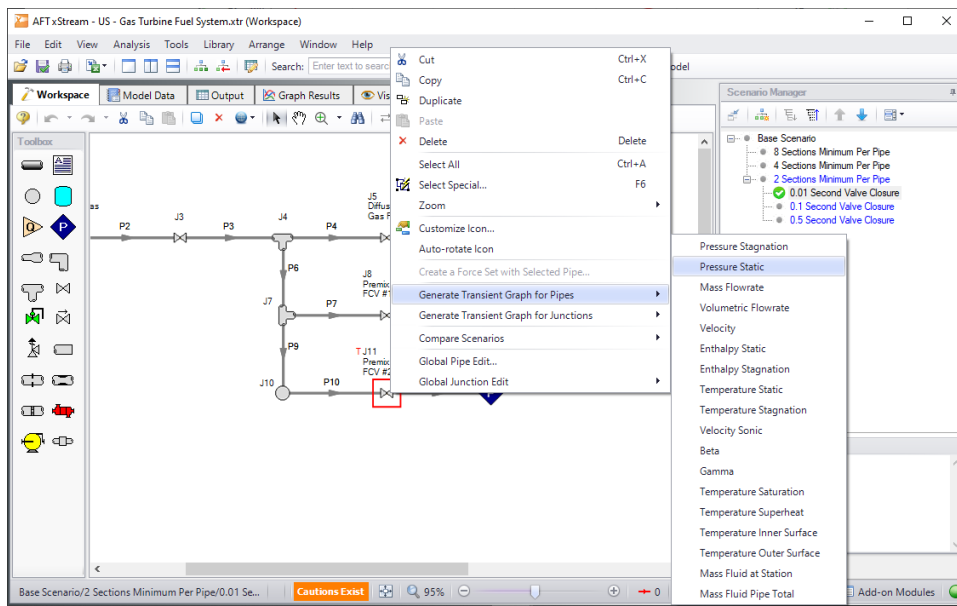


**Figure 13: Load Current Folder Graphs in Same Tab - Stack Same x-Axis**

As we just saw with the Profile Graph Folder, it is possible to create more Graph Folders other than “My Graphs”. Saved Graph List Items can then be organized into different folders and can independently be generated into their own tabs or a whole folder of Graphs can be generated into one tab. Using multiple folders will allow the user to create many unique variations of animations or other Graphs that can be used to convey transient output in a meaningful and intuitive manner.

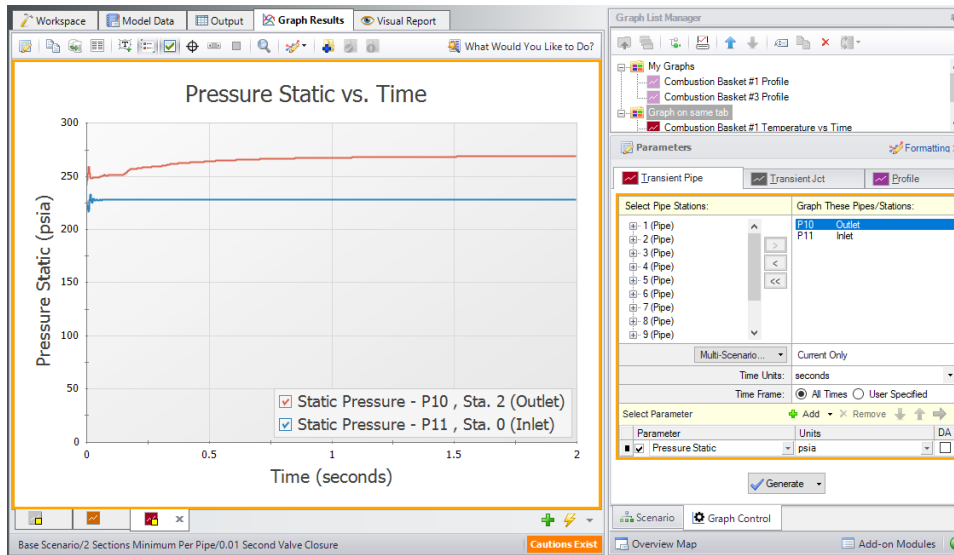
### Creating Graphs from the Workspace [\(return to top\)](#)

Some graphs can also be generated directly from the Workspace after running the model. To create a graph for the valve inlet and outlet static pressures from the Workspace right click on the valve J11, choose “Create Transient Graph for Pipes”, and select “Pressure Static”.



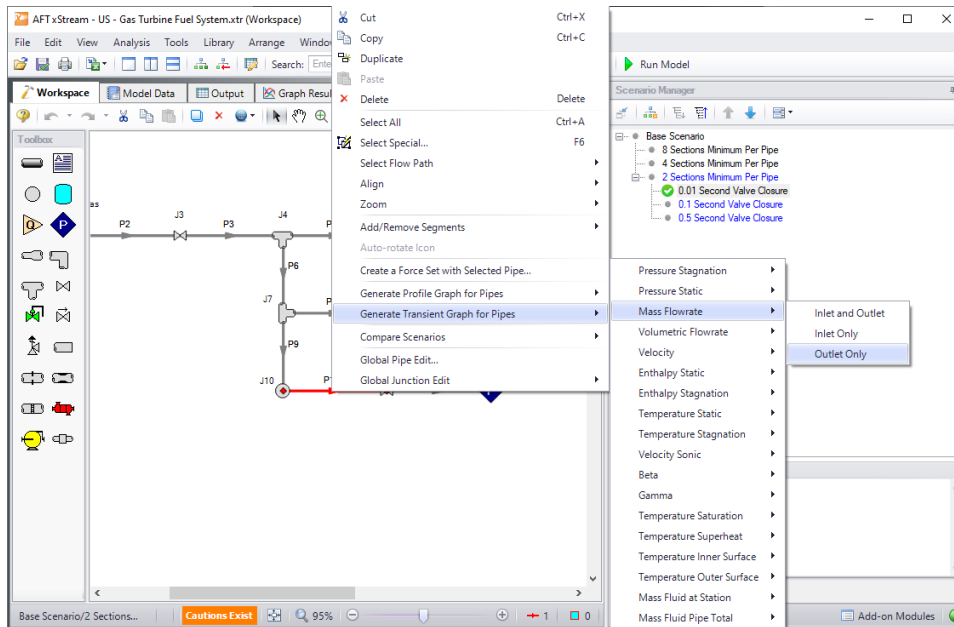
**Figure 14: Creating Transient Graphs of the Inlet/Outlet Pipes Connected to a Junction**

This will automatically generate the requested Graph with default units. This is a quick and easy way to see the pressure difference across the valve. Notice that this graph will be a Transient Pipe graph, since the pipe inlet/outlet stations are being used to graph the equivalent valve inlet/outlet values.



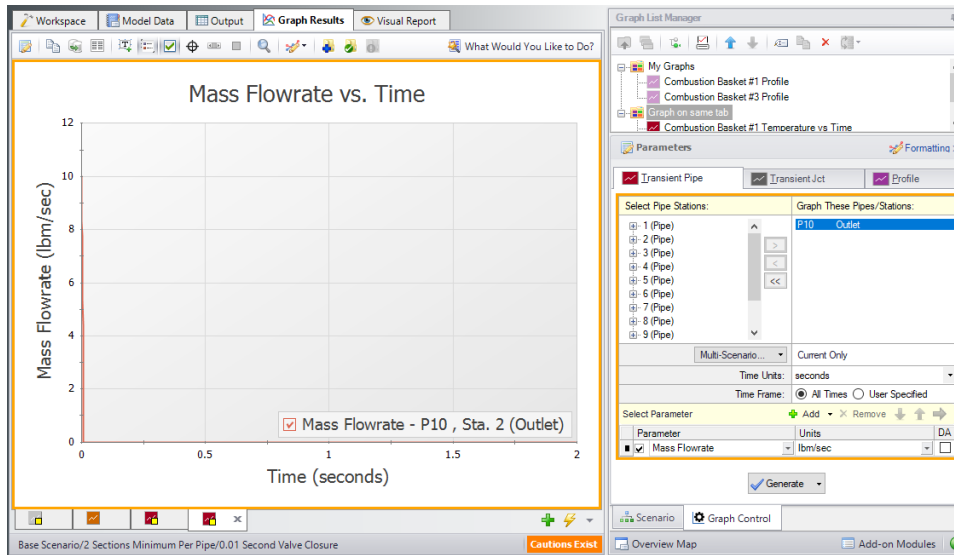
**Figure 15: The Pressure Static vs. Time of the Two Pipes Connected to Valve J11**

Make one more Graph from the Workspace by right-clicking pipe 10, selecting “Create Transient Graph for Pipes”, “Mass Flowrate”, and then “Outlet Only”.



**Figure 16: Graph the Changes to Mass Flowrate vs. Time from the Workspace**

This will add a Graph of the Mass Flowrate at the Outlet Station of Pipe 10 as is shown below in Figure 17.



**Figure 17: Mass Flowrate vs. Time at the Outlet of Pipe P10**

## Visual Report Window

The Visual Report will help you visualize the model data and flow solution in relationship to the model layout. The Visual Report window displays the pipe system layout from the [Workspace](#) window with information in text form (see Figure 1). Additionally, magnitudes of parameters may also be shown with color variations using color maps, or animated using [Visual Report Animation](#).

The values to be shown may come from the input data (when in input mode), the steady-state output or transient output results. You can change between these by selecting the option at the top of the [Visual Report Control](#) window.

In the Visual Report window, the positions of the junctions and pipes are fixed based on their locations on the Workspace. While the pipe system layout cannot be edited in the Visual Report window, the integrated text results for each pipe or junction can be rearranged by dragging and dropping to improve the final appearance of the window.

The contents of the Visual Report window can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

Visual Report animations can be recorded to a .wmv or mp4 file.

The controls in the lower-right corner allows you to zoom in and out, as well fit the image to the size of the screen. The printed size is consistent with the zoom state. That is, if you zoom out to 50%, the Visual Report screen prints at 50% of its normal size.

The Visual Report window is always erased when you make changes to model input. If viewing output results, rerun the model to view the new results.

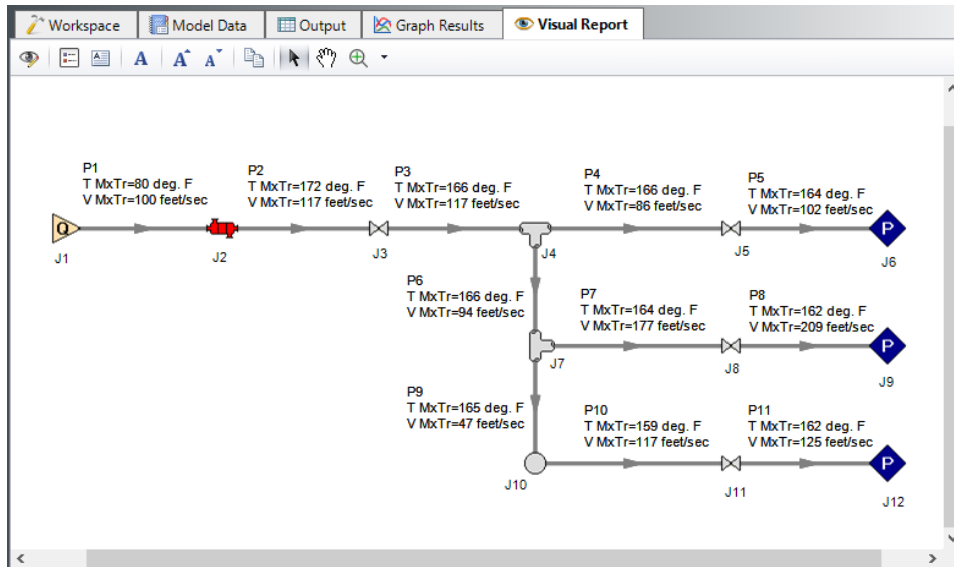


Figure 1: Visual Report window integrates input data and output results with model layout

## Related Topics

Animating Visual Report Results..... 43

## Related Blogs

[Quick Guide to Custom Output](#)

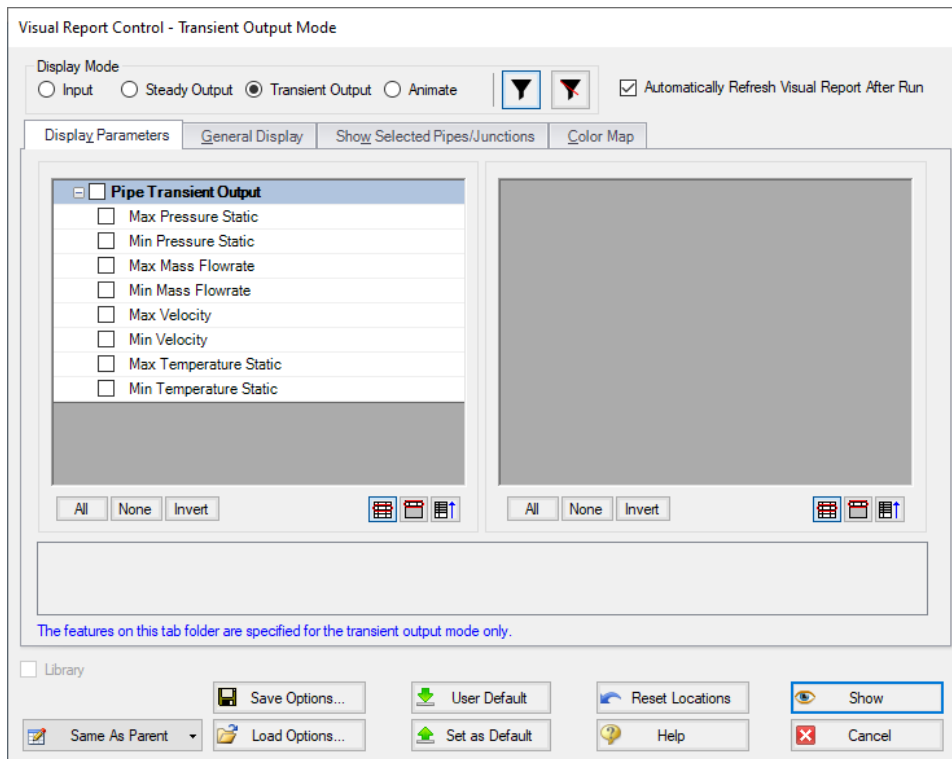
[AFT Responds to 2020 Customer Survey Feedback](#)

## Visual Report Control

The Visual Report Control window lets you control the content and appearance of the [Visual Report window](#) (Figure 1). You can also save custom Visual Report window layouts to disk for later recall, so you don't have to reformat this window repeatedly.

When the Visual Report window is opened and no output results are available, Visual Report Control will be in the Input Display Mode. If transient output results are available, it will be opened in the Transient Output Display Mode. Parameters available for display vary depending upon the mode, i.e. input, steady output, transient output, or animate. If output results exist the display mode may be changed at any time.





**Figure 1:** The Visual Report Control window allows you to configure the Visual Report display

## Display Parameters

The parameters to be shown in the [Visual Report window](#) are selected from the Display Parameters tab. The left list selects the pipe parameters and the right list selects the junction parameters. In these lists you can show only the parameters you have selected in the [Output Control window](#), or all output parameters. You can select more than one parameter; however, the Visual Report window may get crowded if too many are selected.

## General Display

The General Display tab allows you to configure several items.

The title, defined in the [Output Control window](#), file name and scenario name can be shown or not shown depending on your preference.

The units for all parameters can be shown in a legend instead of next to the parameter value. This will reduce the amount of text and crowding shown in the [Visual Report window](#). The legend can be moved to any location in the Visual Report window. If this option is chosen while in input only mode, all values will be converted to a consistent set of units.

You can change various font sizes by clicking the Change button beside any one of the three types of text-containing categories (Data/Text, Title, and Legend) under the Fonts header and choosing the font, font style, and font size desired.

---

**Tip:** The font and font size in the Visual Report can also be changed outside of the General Display in Visual Report Control by clicking on the Fonts button and the Font Size Up or Font Size Down buttons, respectively, on the [Toolbar](#).

---

The number of digits to show in the numerical display can be configured using the provided drop-down list.

Normally on the [Workspace](#), closed objects are displayed using special graphics (dashed lines, X's, etc.). The number of a junction which has special conditions set, by default, has an X preceding the J (e.g. XJ56). If you want these symbols to be used in the Visual Report window also, choose the Show Closed Symbols checkbox. Normally, pipes in closed sections are shown as dashed lines, and junctions outlined with a dashed line. If you want these symbols to be used in the Visual Report window also, choose the Use Closed Object Style checkbox.

### Show Selected Pipes/Junctions

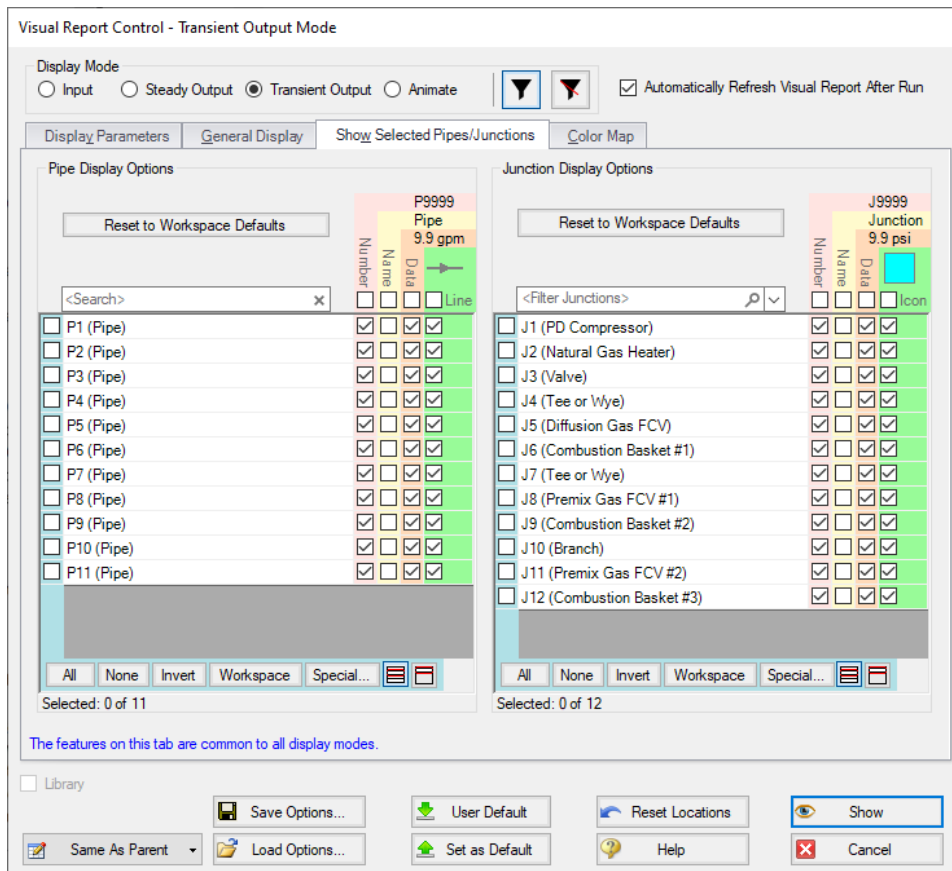
---

You can choose on an individual pipe and junction basis what you would like to show in the Visual Report window. This display customization feature is useful for large models where only some of the data points are of interest.

Each pipe and junction is listed in their respective Display Options frames. Every object can display four pieces of information on the Visual Report - Number, Name, Data, and the Object itself. The display of these items is indicated by the existence of a check mark in the corresponding box. To toggle the display of a particular item for a particular object, simply click the checkbox.

To restore the settings to what is shown in the Workspace and the data, click the Reset to Workspace Defaults button.

Multiple objects can have items toggled on or off at once by selecting the desired objects with the left-most checkboxes and clicking the corresponding checkbox at the top of each column. Selections can also be made with the standard selection buttons at the bottom of these panels.



**Figure 2: Show Selected Pipes/Junctions**

## Color Map

The Color Map feature allows you to visualize magnitudes of various parameters by coloring the pipes. There is one Color Map for the steady output mode, one for the transient output mode, and another for the input only mode. A Color Map is useful to show graphically where, for example, high velocities or reverse flows are located or to visualize the pipe diameters or lengths in a model.

To create a Color Map, first choose a parameter and units for which all pipes will be categorized. Then choose a color, select the operator (i.e. >, >=, =, <=, <) and enter a value, then press Add to Map. Values may be categorized on an absolute value basis by selecting this option.

The order of entry is not important. xStream will sort the entries first by value and then by the operator, with the > being above the <. The pipes will be colored using the first match found in the map, starting from the top of the map. If no match is found, the pipe will be the Workspace pipe color. Be careful to avoid overlapping operators and values. If, for example, the map had the following four colors,

- 1) >=60
- 2) <60
- 3) >=20

4) <20

then there is no way colors 3 and 4 will be used because any pipe less than 20 is also less than 60. Since <60 is before <20 in the map, the pipe will be colored using 2.

To avoid overlapping colors, only use less than (<) as the last entry. A better map would be to use

1) >=60

2) >=20

3) <20

You can choose to categorize the pipes based on their absolute values by choosing the Display Absolute Value checkbox. To delete a map value, click the color you want to delete then click the Remove Setting button.

Instead of coloring the pipes, you could choose to color just the pipe numbers. This is done by choosing the Color Pipe Labels and clearing the Color Pipes checkboxes in the Color Options area. You could also apply the colorization to both. Choose the Print Black & White checkbox if you don't have a color printer. The pipes will be printed in black.

The color map values can be displayed in a legend by checking the Display in Legend checkbox. The color map will appear in the legend after the pipe and junction units, if that option was selected. The legend may be dragged and dropped to any location in the [Visual Report window](#).

### Auto Color Map

---

An automatic color map can be generated for you by selecting "Auto Color Map". This feature will generate a color map using greater than (>) conditions to divide the difference between your max and min value into the number of conditions. These conditions will have their own unique color that is automatically set on a rainbow spectrum. These colors can be manually changed by selecting the "..." button.

### Saving Visual Report Layouts

---

The Visual Report window content is specified in the Visual Report Control window. To open the Visual Report Control window, choose it from the View menu or Toolbar. Through the Visual Report Control window, you can choose the parameters you want to see in the Visual Report window.

The parameters and text locations are automatically saved with the model. However, you may want to save the settings for use with a different model. To save your preferred layouts to an AFT xStream visual layout file, click the Save Options button and assign a file name. The default file extension is .rvs. Later you can load your customized layouts for rapid preparation of the Visual Report window.

### Related Blogs

---

[AFT Responds to 2020 Customer Survey Feedback](#)

### User Options Overview

AFT xStream provides an array of User Options in a single window that can be found in the [Tools menu](#).

The General section of the User Options window allows you to adjust:

- Enabling the Startup Options Window
- Enabling the Tip of the Day feature
- Enabling orthogonal drawing tips
- Displaying [Fluid Accuracy Settings](#) in the Analysis Setup window Fluid panel
- Whether to confirm before clearing output results
- Enabling Highlighting Required Data on Property windows
- Options for opening models with scenarios
- Visual Themes and Styles
- The default location of the Quick Access Panel
- Number of data points to show in input tables
- Maximum Number of Data Points per Graph
- Setting parameters for "button flashing" for curve fit buttons on Junction Property windows

### User Option Defaults

---

There are six buttons at the bottom of the User Options window. xStream has built-in default parameters, units and settings which you can choose by clicking the Reset to AFT xStream Defaults button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of xStream's defaults). To make your own default, first select the parameters, units and settings you would like to use then click the Save As New User Defaults button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the Load User Defaults button. The default settings are updated only when you click Set As New User Defaults.

If you have made changes which you don't want to keep, click the Cancel button. Click Save and Close to use the settings you have defined.

### Unit System

You can set the units you prefer to use in AFT xStream in the Unit System section. Once selected as the preferred units, these units appear as the default units in all instances in which empty fields exist in the model input. Any model input existing prior to changing the Preferred Units will maintain the units used when the data was input, regardless of changes made to unit preferences. After saving your preferred units, a window appears asking if you would like the output parameters to be displayed in your preferred units. Choose Yes if you do want the output parameters to be displayed in the preferred units, or No if you want the output parameters to be displayed in the units previously used.

### Preferred Units

---

To customize the units used by AFT xStream, first select Preferred Units under the Unit System section in the User Options window under the Tools menu.

AFT xStream allows you to specify certain units from a large selection to be displayed in the main interface.

The primary selection is the Unit System:

- **US** - Only US units will be available
- **Metric** - Only Metric units will be available
- **Both with US Defaults** - All Units will be available, but US units will be chosen by default
- **Both with Metric Defaults** - All Units will be available, but Metric units will be chosen by default

Additionally, a filter can be applied to limit the selection:

- **All Units** - all units for the Unit System will be displayed
- **Common Only** - only "common" units will be displayed
- **Common Plus Selected Industries** - Units particular to certain industries can be included with the common units

If the selection has been limited to Common Only, there may units common for certain industries no longer present. The Additional Units for Industry section allows the user to append additional sets of units to the Common unit set. For example, barrels/day is a common unit in the petroleum industry but not many others. Checking the Petroleum box adds these units to the main interface.

Preferred Units allows fine grain control over what unit in the above selection is displayed as the default when defining model input. For example, it may be desired to have a particular parameter such as pipe diameter default to inches, while keeping other parameters metric. This area allows the user to define such defaults.

### Show/Hide Unit Items

---

For additional control of the available units, select Show/Hide Unit Items under the Unit System section. From here, you can select which units for each Unit Family Type you would like to be available for the model input and output by checking the box next to each unit you would like to be used. Removing units that you do not wish to use as input or view as output can simplify your modeling and reduce errors associated with incorrect units.

### Edit Custom Unit Names

---

A custom unit name can be assigned to display different names in the interface for units. For example "feet/min2" could be changed to display as "ft/min2."

To assign a custom unit name choose the appropriate Unit Type from the drop-down and enter the Custom Unit Name in the table.

To speed up the process of entering custom unit names the units can be exported to csv and re-imported as follows:

1. Click "Save Units to CSV" and save the file where it will be easy to locate
2. Open the CSV file in a spreadsheet editing software like Excel, and fill in the desired custom unit names in the Custom Name column
3. Save the CSV file
4. In the User Options window click "Load Units from CSV" and select the edited CSV file

---

**Note:** To use units with superscripts, make sure the csv file is saved with UTF-8 encoding. This can be done in Excel by choosing save as and selecting "CSV UTF-8" as the file type.

---

### Related Blogs

---

[AFT Responds to 2020 Customer Survey Feedback](#)

## Workspace

In the [Workspace](#) section, you can change the symbols used, text content, and location of the text on the Workspace. Note that all display changes are made only for the user's convenience/preference and do not affect the model calculations. Additionally, many display options available in the User Options window are also available in individual pipes and junctions.

### Grid

---

A [traditional 2-D grid](#) or an [Isometric grid](#) can be enabled on the workspace for modeling convenience by choosing the appropriate Pipe Drawing Mode. The number of lines in the grid can be adjusted to create a grid that is fine, medium or coarse. The grid can be hidden or shown by checking or unchecking the Show Grid box accordingly. If the grid is shown, the grid line pattern can be solid or dashed. The pipes and junctions can also be snapped to the grid, which moves junctions and pipe end points to the closest intersection of grid lines.

[2D Orthogonal](#) drawing mode can be enabled in the drawing area. This provides automatic Workspace drawing guides in which vertices are automatically inserted into pipes to enforce drawing only along the horizontal and vertical. When junctions are moved with connected pipes, then pipes may also have vertices inserted to maintain orthogonality. The various pipe drawing modes can also be toggled on the [Arrange menu](#).

### Pipes

---

The display of pipe lines, closed pipes, pipe endpoints, and pipe direction arrows can be adjusted according to your preferences here.

Pipe line thickness can be adjusted from 1 to 10 pixels by selecting the desired pixel value in the box next to Pipe Line Thickness (pixels). You may choose to show all pipes as having the same thickness, or you can vary the thickness with an increasing pipe diameter. If varying the thickness according to the pipe diameter, you can allow xStream to vary the thickness in a gradual fashion with larger diameters shown as having increasingly larger thicknesses, or you can customize two diameters between which the thickness will vary. In the latter case, any pipes with diameters below this minimum value will have the pipe thickness of the minimum diameter pipe, and any pipes with diameters above this maximum value will have the pipe thickness of the maximum diameter pipe. To customize the pipe thickness variation, input the minimum and maximum diameter between which you wish the pipe thickness to vary, as well as the pipe thickness that will represent the minimum diameter pipe and the pipe thickness that will represent the maximum diameter pipe. The pipe thickness will vary in a linear fashion between the two diameters specified.

From the User Options window, you can select how you would like a closed pipe to be displayed in the workspace. The options include a solid line, dashed line, a dotted line, and dashed-dotted combinations.

When building a model, xStream automatically adjusts pipe endpoints to the centers of junctions for user convenience. If you would like to disable this feature, uncheck the box next to Automatically Adjust Pipe Endpoints to Junction Centers in the Pipes window.

The display of flow direction arrows on the pipes can also be adjusted. Under the Pipe Direction Arrows section of the Pipes window, you can select the size of arrow you would like to be displayed as a percentage of the pipe thickness. Additionally, you can opt to not show a pipe direction arrow by checking the option Do Not Show Arrow.

---

**Note:** The pipe direction arrow is used as a reference direction for flow and does not change direction on the workspace if the calculated flow is in the opposite direction indicated by the arrow. Instead, the flow is reported as a negative flow in the output.

---

### Junctions

---

The junction icon style displayed on the Workspace can be chosen in Junctions underneath the Workspace section in the User Options window. The two available icon styles include classic icons and transparent icon. Classic icons are the icons used in earlier versions of AFT products in which each junction is surrounded by a box, and transparent icons are updated icons not surrounded by boxes.

You can also select the icon size and label location for junctions. To adjust the size, click on the box below Icon Size in the Junctions window and choose the percentage of the default size that you would like the junction icons to be sized. For the junction label location, you can select from any of the eight locations shown in the window by clicking on the selection circle that corresponds to the desired location.

---

**Note:** Junction and pipe labels can be moved individually on the Workspace by using the mouse to drag and drop the label according to your preferences.

---



The final option in the Junctions window of the User Options is to enable or disable a dotted outline around a closed junction. If this option is enabled, a dotted box will appear around the junction. If this option is disabled, no dotted box will appear.

### Labels

---

Display settings for pipe and junction labels on the Workspace can be customized under Labels in the Workspace section of the User Options window. When a pipe or junction is inserted onto the Workspace, the next whole number available at the increment specified in this window is automatically assigned to the new pipe and/or junction. xStream's default is to increment pipe and junction numbers by one. Therefore, the first pipe you place on the Workspace will automatically be labeled as Pipe 1 (P1 for brevity), the second will automatically be labeled P2, etc. Junctions are labeled in a similar fashion with a "J" in front of the number to indicate it is a junction. However, if you choose a different increment in the User Options window for pipes and/or junctions, the automatic numbering will increment pipes and/or junctions according to the specified number. You can renumber any pipe or junction individually as long as the number is not already used in any scenario of the model. If a number at the increment is already in use, xStream will skip to the next available whole number. Pipe and Junction label numbers are limited to integers between 0 and 99,999.

---

**Note:** A unique label must exist for each pipe and junction on the Workspace. Therefore, there cannot be more than one P1 in the model, but a P1 and J1 can exist simultaneously because the leading letter is different.

---

Additionally in this section in the User Options window, you can select if you would like the pipe and/or junction numbers only to be displayed, if you would like the names only to be displayed, or if you would like both the name and number to be displayed. To choose which display option you would like, select the radio button next to the desired display option under Pipe/Junction Default Display.

---

**Note:** If you would like a pipe and/or junction display label type to vary from the default option selected in Labels in the User Options window, you may choose the display label type for individual pipes and junctions in the Optional tab of the individual pipe or junction properties window under Display on Workspace.

---

You can also choose whether the pipe and junction labels should be moveable at all times, or if they should only be moveable when the user is holding down the shift key. Select which option you prefer by clicking on the radio button next to your preference. Restricting the movement of labels can be especially helpful to users with complicated or dense models, as it is possible to inadvertently grab a label when attempting to reposition or select an icon on the Workspace.

### Colors and Fonts

---

The Colors and Fonts window located under the Workspace section in the User Options window allows you to change the colors of the following parameters seen on the Workspace:

- Pipe and Junction Symbols
- Junction Numbers

- Pipe Numbers
- Pipes
- Selected Items
- Undefined Numbers
- Workspace Background Base Scenario
- Workspace Background Children
- Workspace Background of Grandchildren
- Workspace Background 3rd Generation
- Workspace Background 4th Generation +
- Grid

The currently selected color for each parameter is shown to the right of the parameter. To change the color for any parameter, either select the box containing the parameter's color and then click on the Set Color box under the list, double click on the box containing the color you would like to change, or click the "... " button next to the color of the parameter you would like to change. A color window pops up that allows you to choose from 48 basic colors, or to create a custom color by clicking anywhere in the color box and adjusting the slider at the right of the box to create the color of your choice. If you would like to save the custom color for use in other areas of the model, click on any empty box underneath Custom colors: and then click on the Add to Custom Colors box below the color box. When finished with selecting a color, click OK on the bottom left of the Color window.

If you would like the background color of the children and grandchildren scenarios to maintain the same color as the base scenario, click on the Set All Background Colors to Base Color button under the Parameter/Color list.

You may also choose to change the font of all junction and pipe names and numbers by clicking on the Change Font button to the right of the Font for Junction and Pipe Numbers on the Colors and Fonts box in the Workspace section. A preview of the current font is given underneath the Font for Junction and Pipe Numbers title.

### Other

---

Symbols representing pipe or junction special conditions and inflow or outflow are listed in the Other window under the Workspace section in the User Options window. These symbols can be changed by the user. To change a symbol, click on the box containing the symbol you want to change, and type in the new symbol.

If you choose to include a background picture in your model, you can adjust the picture size in this section, as well. To scale a background picture, enter a Scale Multiplier in the box underneath Background Picture Scaling.

### Parameters

The Parameters section of the User Options window allows you to specify some default parameters and model input when a new pipe or junction is added to the Workspace. Changing any of the parameters in this window adjusts the specified default pipe and/or junction input parameters and/or values to simplify the modeling process and to prevent the user from having to enter repetitive data.

### Pipes and Junctions

---

Pipe and Junctions, located within the Parameters section of the User Options window, allows you to specify default pipe and junction input information.

Under the Default Pipe Data box in this window, you can enable or disable the option Use Most Recent Pipe Size Data. If you enable Use Most Recent Pipe Size Data, the pipe property windows of new pipes added to the Workspace will automatically contain pipe diameter, material, and friction information from the most recently entered pipe. This prevents you from having to re-enter the pipe diameter, material, and friction model information in the case that several pipes require this same input. Disabling this feature leaves the pipe diameter, material, and friction model for each new pipe added to the Workspace as the default specified in the User Options window.

The Default Junction Elevation box allows you to select the default junction elevation you would like for all new junctions added to the Workspace. If you select Unspecified in this box, the junction elevation field in the [Junction Properties window](#) remains empty until you specify each junction elevation individually. If you select Use Most Recent, the junction elevation of a new junction added to the Workspace is shown as the elevation of the most recently specified junction. If you select Specified, you are able to input an elevation of your choice that will automatically be input into each new junction that is added to the Workspace.

The Default Design Factors box allows you to add a default Design Factor to each pipe shown in the list on the [Pipes](#) and [Junctions](#) window. Setting a default design factor for pipes can be especially useful when applying safety factors on given equipment.

## Files

The Files section of the User Options window allows you to specify file locations for saving models, database folders, user settings, and model backup files. You can also customize the model backup file settings here.

### File Locations

---

Under File Locations in the Files section of the User Options window, you have the option to select the default folder in which to save your model. If a default model folder is specified, this location will be the default location to save the model. To specify the default location, click Add below the Default Model Folder box and browse to the location you want to set as the default folder. Once you have selected the folder, click OK. The location of the default model folder will then show in the Default Model Folder box. You can add more than one default folder, but only the folder selected in blue will be the active default folder.

To specify locations for xStream to search for [library](#) files, click Add under the Library Search Locations box. Browse to the file location that contains the library, select the location, and click OK. You can add multiple folders to be searched. To add additional folders, repeat the process for each folder.

The User File consists of all saved settings and preferences. In order to backup these user settings in a \*.ini file, click Add under User File Backup Locations. Browse to the file location in which you would like to save these settings. Once you have selected the desired file location, click OK. Note that the User File

can be saved in multiple locations. To add additional locations, repeat the process for each folder that you would like to save the settings.

### Backup Settings

---

A backup file is a copy of a previously saved version of your \*.xtr file that can function as a complete, standalone model. These files are used as a safety measure to provide you with one or more previous versions of the model in the case that the current model is deleted or damaged.

To customize the amount of backup files (i.e. the number of previously saved versions of the model) and the frequency that the model is saved, open the Backup Settings window, which is located under the Files section of the User Options window.

From the Backup Settings window, choose the total number of backup files you want to save by clicking on the drop-down box next to Number of Backup Files Created and specify the total number of files. After the model has been saved initially, subsequent instances of saving the model will create a new backup file each instance until the number of backup files saved equals the number of backup files specified in Backup Settings. The most recent backup file will always contain the file extension \*.bak001. The file extension of earlier versions is numbered in chronological order with earlier versions receiving larger numbers in the file extension. Therefore, given any model file, the backup file with a file extension of \*.bak002 will always contain the model file saved after (more recently than) the backup file containing the file extension of \*.bak003. Likewise, for the same model, the backup file with the file extension of \*.bak003 will always contain the model file saved after (more recently than) the backup file containing the file extension of \*.bak004. This extension naming convention applies to any number of backup files.

If you want xStream to automatically save the model file periodically when the file is open, tick the box next to Automatically Save Model and select the frequency in the drop-down box.

If you need to use a backup file as a standalone model, change the extension to .xtr for the backup file, and open the file in xStream.

You can opt to save no backup files by choosing "0" in the Number of Backup Files Created box.

### Encoding

---

The File Encoding option can be set to "Unicode" in order to use characters outside of the standard english keyboard such as diacritics, cyrillic, etc for text fields such as the pipe/junction name, notes, etc.

## File Import and Export

This section focuses on the various file types that can be imported and exported from AFT xStream. Topics included discuss importing piping layouts from various sources, importing and exporting between AFT products, and importing and exporting data with Excel.

### Contents

---

<a href="#">Importing Piping Component Files</a> .....	345
<a href="#">Importing GIS Shapefiles</a> .....	348
<a href="#">Importing Caesar II Netral Files</a> .....	350
<a href="#">Importing and Exporting Epanet Files</a> .....	352
<a href="#">Importing and Exporting from Other AFT Products</a> .....	354
<a href="#">Exporting to Excel</a> .....	355
<a href="#">Importing Excel Data</a> .....	359

### Related Blogs

---

[Meet the Import Piping Layout Wizard](#)

[Now it's Super Easy! Changing Input Data in AFT Models using Excel Change Data](#)

[That was EASY! Quickly Change Pipe and Junction Input Data into AFT models using Excel Change Data](#)

## Importing Piping Component Files

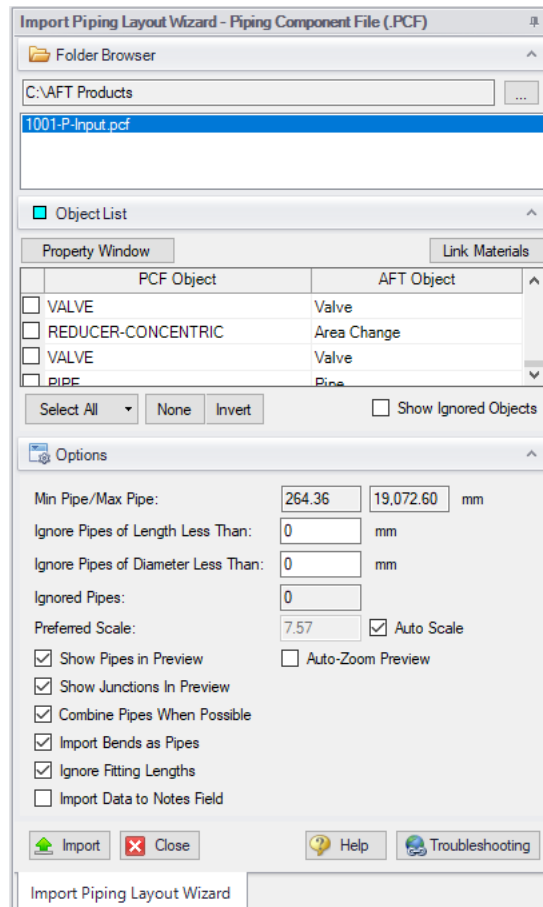
AFT xStream includes the ability to import Piping Component Files (.pcf), which can be generated by a wide range of software, and provide a quick way to get started with the transient modeling portion of a project. Files can be imported into existing models.

Selecting File > Import Piping Layout From > Piping Component File (.pcf) brings up the Import Piping Layout Wizard in the [Quick Access Panel](#), as shown below in Figure 1.

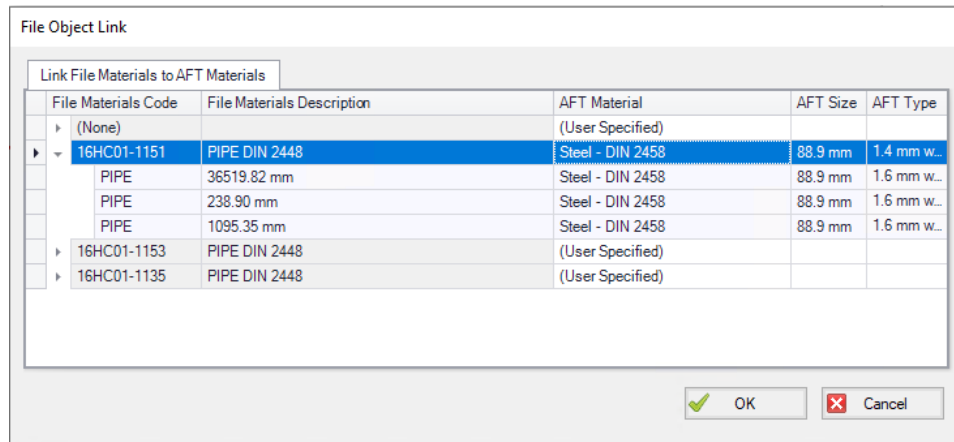
The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .pcf files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.
  - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
  - **Link Materials** - Opens the File Object Link window, where the Materials Codes from the pcf file can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the pcf file. Selections made for a pcf material will be applied to all pipes that reference that pcf material. Pipes associated with each material can be seen by clicking the arrow on the left expand the group (Figure 2).
  - **Show Ignored Objects** - Certain types of objects are ignored on import - these can be objects such as flanges, or additional file components such as materials or bolt specifications.
- **Options**
  - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
  - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
  - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
  - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for gas transient analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
  - **Preferred Scale** - The wizard will Auto Scale the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
  - **Show Pipes in Preview** - Display the pipes in the Workspace preview.
  - **Show Junctions in Preview** - Display the junctions in the Workspace preview.
  - **Combine Pipes When Possible** - It is required that all pipes in the Workspace be connected by two junctions. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be manually combined, before the model could be solved.
  - **Import Bends as Pipes** - Pipe bends can be considered as junctions or simply a visual change in direction of the pipe. This is identical to using [segments](#) to show piping layout. Note that these bends are visual only and have no losses associated with them - to account for losses, either junctions or [fittings & losses](#) must be used. This option is helpful to reduce unnecessary model clutter.

- **Ignore Fitting Lengths** - Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.
- **Import Data to Pipe or Junction Notes Field** - Add the raw data from the originating file to the [Notes](#) tab of the corresponding object.



**Figure 1: Piping Component File Import**



**Figure 2: File Object Link window for Piping Component File Import**

## Related Blogs

[Meet the Import Piping Layout Wizard](#)

## Importing GIS Shapefiles

### Import GIS Shapefile

GIS Shapefiles that adhere to the ESRI standard can be imported into xStream for ease of model creation. These files have the extension .shp, but there are two other supporting file types that need to be in the same folder as the .shp in order for the import to work correctly. These are .shx and .dbf file types. The option is located under File > Import Piping Layout From > GIS Shapefile (.shp). This opens the "Import Piping Layout Wizard" in the Quick Access Panel that allows the importing of one line .shp that is of pipes and multiple point .shp files for branches, valves, compressors/fans, and tanks. Once the .shp files have been added to this window the object type, scaling, and mapping can be selected.

### Type

Any .shp files that are of lines are automatically set to the object type of "Pipes". Only one .shp file containing lines can be imported at a time. Any .shp file that is of points can be set to Branches, Valves, Compressors/Fans, or Tanks.



### Scaling

---

The default scaling is 1 grid point to 1 foot. This can be adjusted by selecting the "..." button in the Scaling column. This button opens a new window in which a new scale can be committed to the shapefile. This window also allows the viewing the coordinates of specific points and parts of different shapes within the shapefile.

### Mapping

---

The Data Mapping window allows the user to select what data fields have relevant data for xStream to import.

- Pipes can map:
  - **Labels** - Names
  - **Length** - If no length is imported, the pipe lengths will be calculated based upon the shapefiles scaling.
  - **Friction** - Absolute Roughness, Relative Roughness, and Explicit Friction Factor friction models are available.
  - **Inner Diameter**
- Branches can map:
  - **Labels** - Names
  - **Elevation**
- Compressors/Fans can map:
  - **Assigned Flow** - Imports as a Fixed Flowrate compressor
  - **Labels** - Names
  - **Elevation**
- Tanks can map:
  - **Labels** - Names
  - **Elevation**
- Valves can map:
  - **Labels** - Names
  - **Elevation**

### Related Blogs

---

[Meet the Import Piping Layout Wizard](#)

## Importing Caesar II Neutral Files

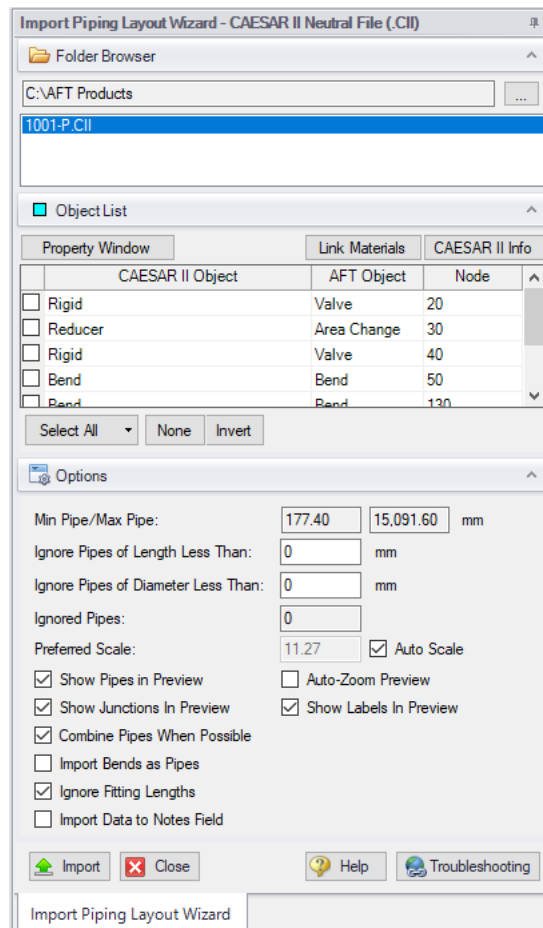
AFT xStream includes the ability to import CAESAR II Neutral Files (.cii), providing a quick way to get started with the transient modeling portion of a project. Files can be imported into existing models.

Selecting File > Import Piping Layout From > CAESAR II Neutral File (.cii) brings up the Import Piping Layout Wizard in the [Quick Access Panel](#), as shown below in Figure 1.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .cii files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.
  - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
  - **Link Materials** - Opens the File Object Link window, where the pipes can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the CII file, and will be applied to all of the pipes.
  - **Show Ignored Objects** - Certain types of objects are ignored on import - these can be objects such as flanges, or additional file components such as materials or bolt specifications.
- **Options**
  - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
  - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
  - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
  - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for gas transient analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
  - **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
  - **Show Pipes in Preview** - Display the pipes in the Workspace preview.
  - **Show Junctions in Preview** - Display the junctions in the Workspace preview.
  - **Show Labels In Preview** - Show the corresponding label from the .cii file in the Workspace preview. This is only shown in the preview and will not be imported into the model itself.

- **Combine Pipes When Possible** - It is required that all pipes in the Workspace be connected by two junctions. However, this is not a requirement in all software. Therefore, some .cii files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be manually combined, before the model could be solved.
- **Import Bends as Pipes** - Pipe bends can be considered as junctions or simply a visual change in direction of the pipe. This is identical to using [segments](#) to show piping layout. Note that these bends are visual only and have no losses associated with them - to account for losses, either junctions or [fittings & losses](#) must be used. This option is helpful to reduce unnecessary model clutter.
- **Ignore Fitting Lengths** - Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.
- **Import Data to Pipe or Junction Notes Field** - Add the raw data from the originating file to the [Notes](#) tab of the corresponding object.



**Figure 1: CAESAR II Neutral File Import**

## Related Blogs

---

[Meet the Import Piping Layout Wizard](#)

### Importing and Exporting Epanet Files

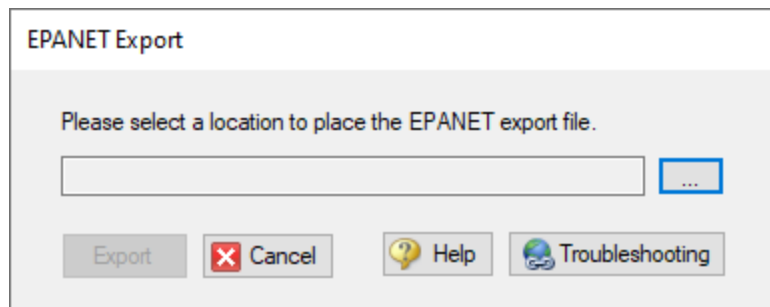
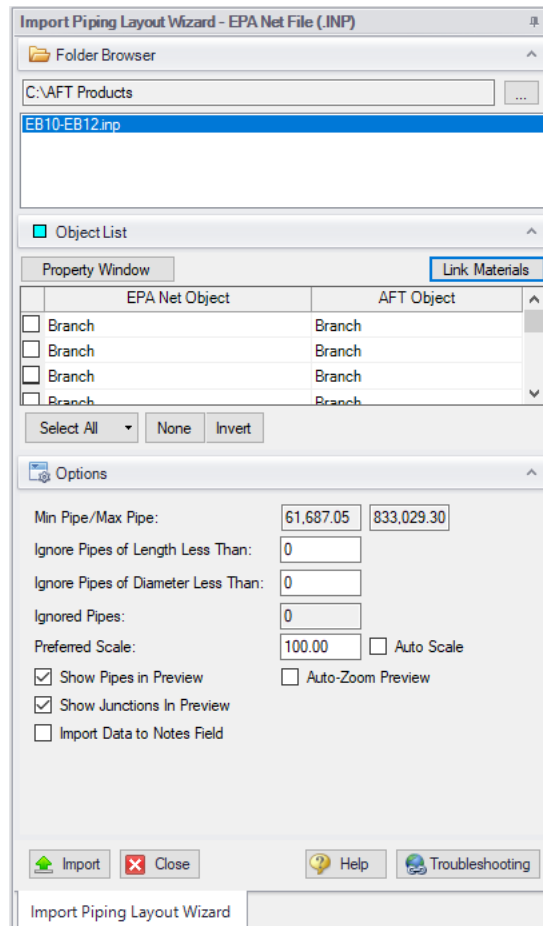
AFT xStream includes the ability to import and export EPANET Files (\*.INP). EPANET is public domain modeling software developed by the US Environmental Protection Agency. Many different applications support the file format, so it can therefore provide a quick way to get started with the transient modeling portion of a project.

Selecting File > Import Piping Layout From > EPANET File (.inp) brings up the Import Piping Layout Wizard in the [Quick Access Panel](#), as shown below in Figure 1. Note that importing from EPANET requires that the Workspace be empty.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .inp files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Clicking on an object in the workspace highlights it in the wizard.
  - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
  - **Link Materials** - Opens the File Object Link window, where the pipes can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the EPANET file, and will be applied to all of the pipes.
- **Options**
  - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
  - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
  - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
  - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for gas transient analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
  - **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
  - **Show Pipes in Preview** - Display the pipes in the Workspace preview.
  - **Show Junctions in Preview** - Display the junctions in the Workspace preview.

- **Import Data to Notes Field** - Add the raw data from the originating file to the Notes tab of the corresponding object.
- **Auto-Zoom Preview** - This option can be enabled to automatically zoom out so that the whole model can be seen in the workspace, which can be convenient for large systems.



**Figure 1: EPANET Import/Export Dialog**

## Related Blogs

[Meet the Import Piping Layout Wizard](#)

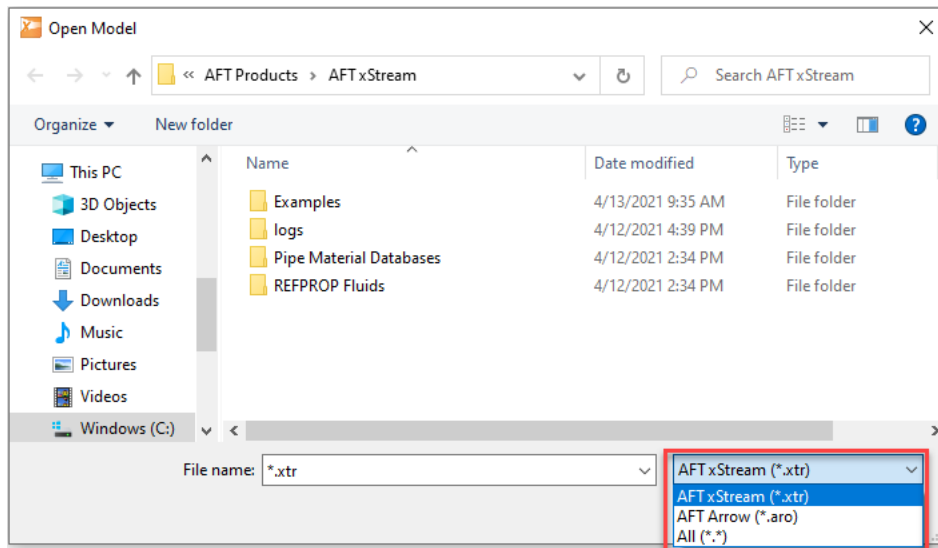
### Importing and Exporting from Other AFT Products

Models made in other AFT products such as AFT Arrow can be opened directly in AFT xStream and vice versa. This can be a useful starting point when it is desired to perform additional analysis on a model that has already been built in another AFT product, such as if it is desired to perform transient analysis on a system previously built in AFT Arrow.

To convert a model from one AFT product to another just browse to the model you would like to convert and open it in the software you are converting the model to. This can be done by changing the file extension selected, as is shown below in Figure 1. Note that though AFT Fathom and AFT Impulse are not explicitly listed, models from Fathom and Impulse can also be converted by selecting "All" and browsing to the model file.

When the model is opened in AFT xStream a backup copy of the original model will be created from which the original model can be restored if needed. See the [Backup Settings](#) explanation for more information on backup files.

The conversion process will automatically change junction types not included in AFT xStream, such as the Spray Discharge junction, to either a General Component or Branch junction. Be aware that some features available for use in other AFT products are not available for use in AFT xStream. If a junction is defined using a feature not available in AFT xStream, the non-applicable information will be removed, and the user may be required to input additional information to fully define the model. It is advised to fully review the model after import.



**Figure 1: File extensions options for Open File window**

## Exporting to Excel

AFT xStream includes a powerful ability to export any [Output](#) data to any specific cell(s) in an Excel spreadsheet. Specific values from the model can be exported, as well as columns, rows, or entire output tables as defined in [Output Control](#). These items are controlled with the Excel Export Manager, available from the File menu.

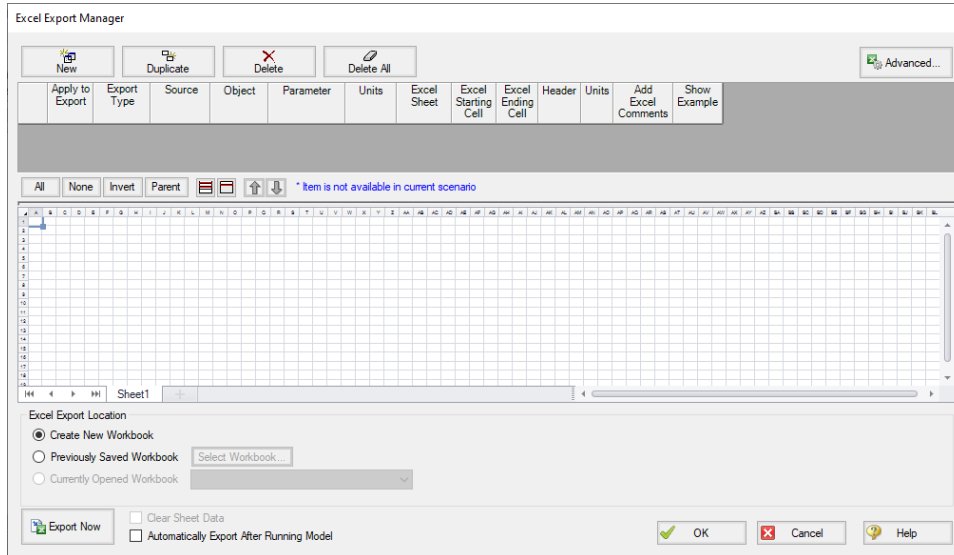


Figure 1: The Excel Export Manager

## Adding Items to the Excel Export Manager

Data from the Output can be added to the Excel Export by clicking New, or by selecting an existing item and clicking Duplicate. Items can also be created directly from the [Output Window](#) by right-clicking on a cell or header.

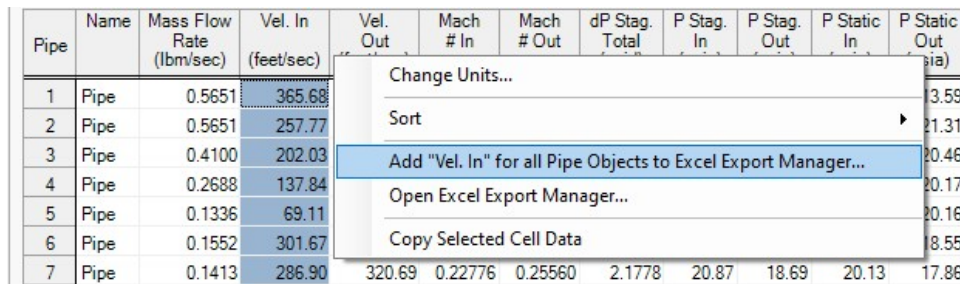


Figure 2: Adding items directly from the Output

Each row in the Excel Export Manager corresponds to a cell, column, row, or table of data to be added to the spreadsheet. Each column helps define what data will be transferred where. The grid below the entries shows a preview of where the data will be added to help prevent intersecting data by showing

where there are overlapping regions. Note that graph items can vary in size and do not show a complete region here.

- **Apply to Export** - Items can be optionally included or excluded from the export.
  - **Export Type** - What type of data is being exported. This affects other settings, and will affect how much room is required on the target Excel sheet.
    - **Single Value** - The value of a single parameter for a single pipe or junction object.
    - **Column** - Values of a single parameter for all of the [displayed](#) objects.
    - **Row** - Values of all [displayed](#) parameters for a single object.
    - **Table** - Values of all [displayed](#) parameters for all displayed objects.
    - **Graph** - The graph points from a specified [Graph List Item](#).
    - **Other** - Additional information not available in a standard Output table, such as Fluid definition, Model Title, or Scenario Names.
  - **Source** - Where the data is coming from.
    - **Pipe** - From the steady-state Pipe section of the output.
    - **Axial Points** - From the Axial Points tab in the pipe output section
    - **Junction** - From the Junction section of the output.
    - **General Output Summary** - From one of the available output Summary sections. For example, if there is a compressor junction in the model, Compressor/Fan Summary is available.
    - **Transient Max/Min** - From the Transient Max/Min tab in the Pipes section of the output.
    - **Forces** - From the Forces tab in the pipe output section
    - **Graph** - From a specified [Graph List Item](#).
  - **Object** - The object in particular that is desired. If Column or Table are selected for Export Type, this selection will be fixed to ALL.
  - **Parameter** - The parameter in particular that is desired. If Row, Table or Graph are selected for Export Type, this selection will be fixed to ALL.
  - **Units** - The units of the exported value. This is available only for Single Value and Column Export Types. Some parameters do not have units and this will display N/A.
  - **Excel Sheet** - Which Excel Sheet to export to. This value can be edited by the user - if it is desired to export to a named sheet this can be entered here.
  - **Excel Starting Cell** - The upper-left cell to export the data to.
  - **Excel Ending Cell** - The automatically calculated lower-right cell of the data, which depends on the Export Type, export options, and the number of parameters or objects in the model. It is important to recognize that the Excel Starting Cells can be defined in a way that data gets **overwritten**. The Ending Cell column is a handy reference to help avoid this.
  - **Header** - Optionally export the name of the Parameter(s) related to the item.
  - **Units** - Optionally export the name of the Unit(s) related to the item.
  - **Add Excel Comments** - Optionally adds an Excel comment to the value cell that contains the Parameter name and unit.
  - **Show Example** - Displays an example of how the exported cells will appear.
-



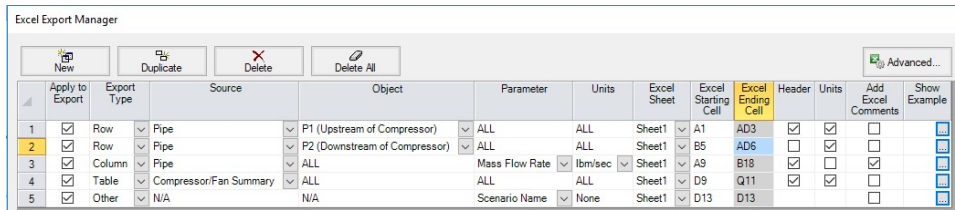


Figure 3: Example Export

	Name	Mass Flow Rate	Vel. In	Vel. Out	Mach # In	Mach # Out	dp Stag, Total	P Stag, In	P Stag, Out	P Static In	P Static Out
3	P1 (Upstream of Compressor)	Upstream of Compressor	0.5651	365.63	368.96	0.31565	0.31838	0.1193	14.7	14.58	13.72
6		Downstream of Compressor	0.5651	257.76	281.8	0.20289	0.22271	2.1147	24.17	22.06	23.49

	Name	Mass Flow	Vol. Flow	dp Stag	dp Static	dh Static	Overall Efficiency	Speed
11	P1 (Upstream of Compressor)	0.5651						
12	P2 (Downstream of Compressor)	0.5651						
13	J2 (Compressor)	Compressor	0.5651	8.5977	9.5922	9.8832	17,987.5957	80, 100

Figure 4: Result of the above example. Color added for emphasis.

## Excel Export Location

After all of the desired data has been added to the Excel Export Manager, the next step is to actually export the data. There are three options available for the export:

- **Create New Workbook** - A new Excel Workbook will be opened with the exported data.
- **Previously Saved Workbook** - Browse to a Workbook saved to file. The Workbook will be opened and the modifications made to the sheets as specified.
- **Currently Opened Workbook** - Select a Workbook that is currently open in Excel. The already open Workbook will be modified.

**Note:** The Excel Export Manager **does not** save Excel files for non-batch run exports. Any changes made need to be manually saved by the user in Excel.

There are two options available for exporting:

- **Clear Sheet Data** - All of the existing data on the sheets specified in the Excel Sheet column will be cleared. This is especially useful for setting up a new export when many changes are being made.
- **Automatically Export After Running Model** - After the model is run, the Export will be carried out automatically, per the settings defined by the user in the Excel Export Manager.

### Advanced Options

---

Items can be added directly from the Output instead of adding custom items with the New button. There are two behavioral options that can be modified for this action:

- **Open Excel Export Manager** - If enabled, the Excel Export Manager will be opened every time an item is added from the Output. This is useful for double checking the setup.
- **For Existing Items** -
  - **Add New Item** - The item selected will be added to the Excel Export Manager even if it already exists.
  - **Select Existing** - If the item already exists, do not add a duplicate item and instead select the existing item.
  - **Ask** - If the item already exists, prompt the user on the action to take.
- **Create an Export Guide** - When exporting, an Export Guide can optionally be included in the export. The Export Guide contains information about the model and the generation of the export, to help keep the connection between the data and the model clear for future reference.

### Exporting Data From Multiple Scenarios

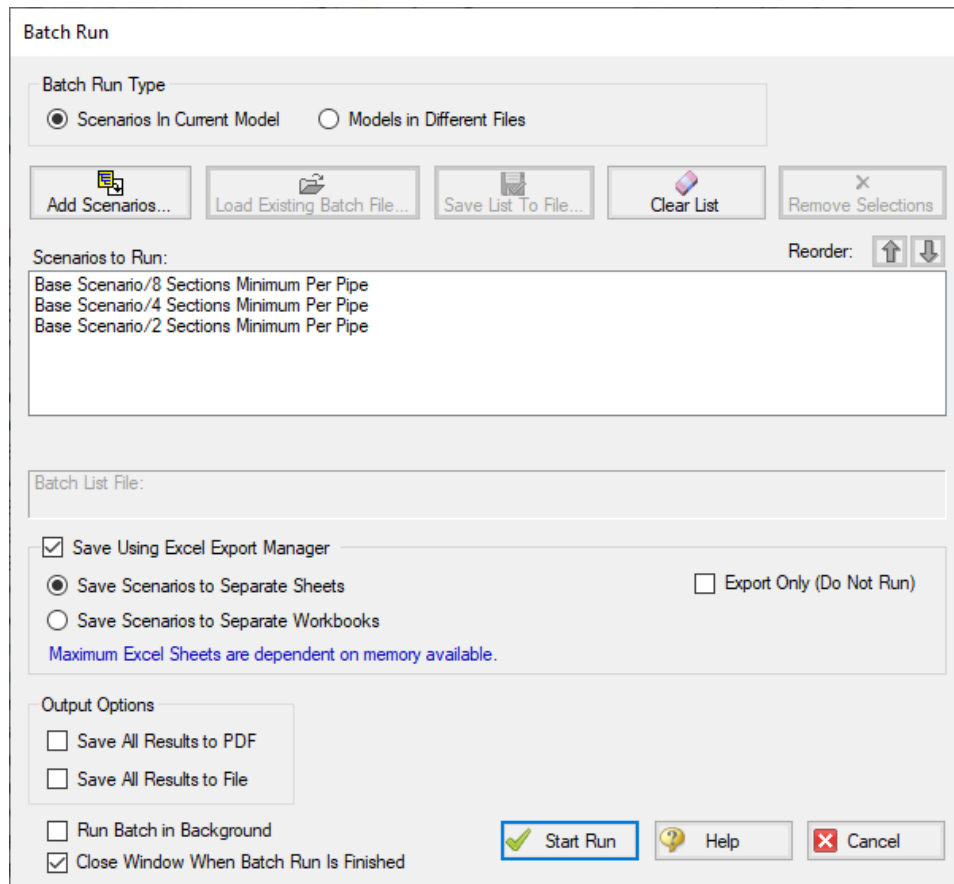
---

The Excel Export Manager can be employed with [Batch Runs](#) with a couple of important distinctions.

First, if multiple sheets are specified in the Excel Export Manager, the only way to export multiple scenarios is to export **every scenario to its own Workbook**. To keep data from multiple scenarios in one Workbook, there can only be one sheet specified in the Excel Export Manager.

Second, **the Excel File will be saved after each scenario**. In normal single-scenario exports the file is never saved, but in batch mode it is required.

In the Batch Run tool, users are also able to use Excel Export without having to run the scenarios again. Note that if any of the scenarios in the Batch Run do not have output, the batch Export Only will not export any data for that scenario, as it will not run those scenarios to get output.



**Figure 5: Saving Results From Multiple Scenarios**

## Related Blogs

[Quick Guide to Custom Output](#)

[Paint a Clear Picture Faster Than Ever with AFT's New Excel Export Manager](#)

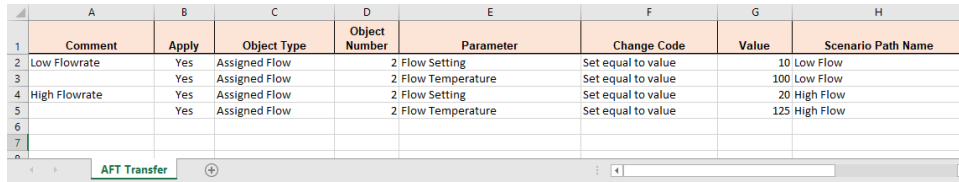
## Importing Excel Data

A Microsoft Excel spreadsheet can be used to make changes to objects in an AFT xStream model. The values in the Excel file are located in specific locations as described in the Setting Up Excel section.

## Setting Up Excel

When reading an Excel file, AFT xStream will search for a worksheet named **AFT Transfer**. There can be other worksheets in the workbook but xStream will only look for a sheet with this name. The file

structure looks like the following:



	A	B	C	D	E	F	G	H
1	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	Low Flowrate	Yes	Assigned Flow	2	Flow Setting	Set equal to value	10	Low Flow
3		Yes	Assigned Flow	2	Flow Temperature	Set equal to value	100	Low Flow
4	High Flowrate	Yes	Assigned Flow	2	Flow Setting	Set equal to value	20	High Flow
5		Yes	Assigned Flow	2	Flow Temperature	Set equal to value	125	High Flow
6								
7								
8								

**Figure 1: Sample AFT Transfer Excel Sheet**

Row 1 and column A are not read by AFT xStream - these cells can have any value in them, though it is recommended to enter the values above for the header row.

Each column has certain accepted values, which can depend on the values in neighboring columns.

---

**Note:** A template spreadsheet containing validation rules for the required layout is included in the AFT xStream installation. By default the file path is: C:\AFT Products\AFT xStream\AFT xStream Excel Import Workbook.xlsx

---

## Apply Column

---

This column accepts Yes or No. Only changes marked as Yes will be imported into the model.

## Object Type

---

This column must be a junction name or Pipe. Valid choices can be seen below in Table 1.

## Object Number

---

This corresponds to the junction or pipe number.

## Parameter

---

The specific parameter to be changed. These parameters vary based on the Object Name selected. Each parameter has a type (C/S/I) associated with it, as is explained in the next section.

**Table 1: Object Types, Associated Parameters, and Change Codes**

<b>Pipe</b> S Design Factor - Additional Losses S Design Factor - Friction S Fixed K Loss S Initial Guess Flowrate S Inner Diameter (for unspecified, cylindrical pipes) S Length I Nominal Size Increment C Nominal Size Set (must match text in Pipe Property Window) C Nominal Type Set (must match text in Pipe Property Window) C Pipe Name S Roughness Value (for unspecified friction models) S Scaling Factor	<b>Area Change</b> S CdA (for sonic choking) S Elevation C Junction Name S Loss Value	<b>Assigned Flow</b> S Flow Setting S Flow Temperature S Elevation C Junction Name I Special Condition	<b>Assigned Pressure</b> S Elevation C Junction Name S Pressure Setting S Temperature	<b>Branch</b> S Elevation C Junction Name
<b>Check Valve</b> S CdA (for sonic choking) S Elevation S Forward Velocity to Close C Junction Name S Loss Value S Pressure/Head to Reopen I Special Condition S Xt Value	<b>Compressor/Fan</b> S Assigned Flow S Elevation C Junction Name S Nominal Efficiency I Special Condition S Speed (%)	<b>Control Valve</b> S Control Setting S Elevation S Full Open Loss C Junction Name I Special Condition S Xt Value	<b>Dead End</b> S Elevation C Junction Name	<b>General Component</b> S CdA (for sonic choking) S Elevation C Junction Name S Loss Value I Special Condition
<b>Orifice</b> S Area/Diameter S CdA (for sonic choking) S Elevation S Exit Pressure S Exit Temperature C Junction Name S Loss Value	<b>Relief Valve</b> S CdA (for sonic choking) S Elevation S Exit Pressure S Exit Temperature C Junction Name S Loss Value S Set Pressure I Special Condition	<b>Tank</b> S Elevation C Junction Name S Pressure S Temperature	<b>Tee or Wye</b> S Elevation C Junction Name	<b>Valve</b> S Exit Pressure S Exit Temperature S Elevation C Junction Name S Loss Value I Special Condition S Xt Value

## Change Code

Each Parameter has restrictions on how it can be changed. All of the Parameters are decimal numbers, integers, or strings.

**Table 2: Valid Change Codes for each type of Parameter**

Decimal Number (S)	Integer Number (I)	String (C)
Set equal to value	Set equal to value	Set equal to value
Change by value (+/-)	Change by value (+/-)	Add string to list
Change by percent (+/-)		Delete string from list

### Value

---

The actual value to change the Parameter to or by.

For special conditions there are specific values assigned to each special condition option, as are listed in the table below.

**Table 3: Values used to import special conditions**

Junction	Special Condition	Value
Assigned Flow	None	0
	Closed	1
Check Valve	None	0
	Closed	1
	Open	2
Compressor	None	0
	Compressor Off No Flow	1
	Compressor Off Flow Through	2
Control Valve	None	0
	Closed	1
	Fully Open - No Control	2
General Component	None	0
	Closed	1
Relief Valve	None	0
	Failed Open	2
	Ignore Relief Valve	5
Valve	None	0
	Closed	1

### Scenario Path Name

---

The Scenario that the changes are intended to apply to. If no scenario is present, the changes will be applied to the current scenario.

It is important that the Scenario name is uniquely qualified - if there are multiple scenarios with the same name, the full Scenario Path Name can be used. Scenario Path Names can easily be copied from AFT xStream by right-clicking the scenario name and selecting "Copy Scenario Path Name."

### Importing the File

---

When the sheet has been completely filled out, it can be imported from File -> Import Excel Change Data. When importing the data, the Object Change Log will appear, listing the changes that were made and any errors that occurred during the import.

**Table 4: Excel Importing Error Messages**

Error	Description
Scenario name is not unique in the model (use full scenario path name)	There are multiple scenarios with this name - use the fully qualified path name (e.g. Base Scenario/US Units/Pump A)
Invalid change type	The Change Code is not valid for the Parameter. For example, trying to change a string type with Change by Value (+/-)
Invalid parameter ID	The parameter given does not match the object type (e.g. Length for a junction)
Invalid parameter value	The value cannot be changed because the value is not valid. For example, a pump speed cannot be negative
Object not found	A pipe or junction with the given ID was not found
Parameter conflict	The value cannot be changed because it conflicts with another setting. For example, the loss value cannot be set when the junction is using a resistance curve
Scenario not found in the model	No scenario by the given name was found in the model

---

**Note:** In order for the junction to accept the changes, some fields must already contain data. For example, if the control setpoint for a control valve is being changed, there must already be a setpoint and units entered into the control valve junction object. This is to ensure that the correct set of units are being used. These values can be set using [Global Edit](#) to an arbitrary value before the changes are made from Excel.

---

### Related Blogs

---

[Now it's Super Easy! Changing Input Data in AFT Models using Excel Change Data](#)

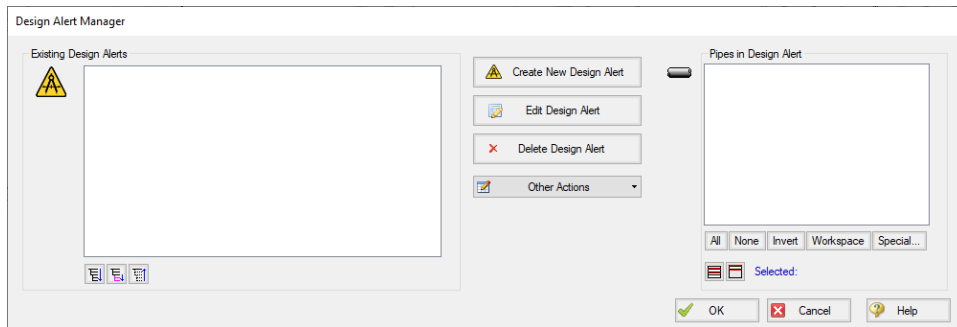
[That was EASY! Quickly Change Pipe and Junction Input Data into AFT models using Excel Change Data](#)

## Design Alerts

Design Alerts are a powerful feature that can help speed up the modeling process and help verify that the model is behaving as expected. The creation of a Design Alert does not constrain or affect the operation of the model in any way. However, it automatically creates an output alert any time the design limit is violated. One common use case might be defining a maximum allowable pressure for the pipes in the model. While the output can be reviewed manually for these violations, this quickly becomes tedious as the model grows in size. Creating a Design Alert instead allows the engineer to spend less time reviewing output tables as any violation will be immediately identified.

The Design Alert Manager allows the user to create or modify Design Alerts. It can be accessed from the Tools Menu or in the Design Alerts tab in pipe and junction properties windows.

Design Alerts can also be created [directly from the output](#), or can be [imported/exported](#) for use in other models.



**Figure 1: Design Alert Manager**

## Creating and Applying Design Alerts

Click Create New Design Alert to add a new Design Alert to the model. The Design Alert Manager expands to allow the creation of the new alert.

A Name, Object Type, and at least one Parameter are required. Depending on the Object Type, a different selection of Parameters is available, as shown below. To add a Parameter:

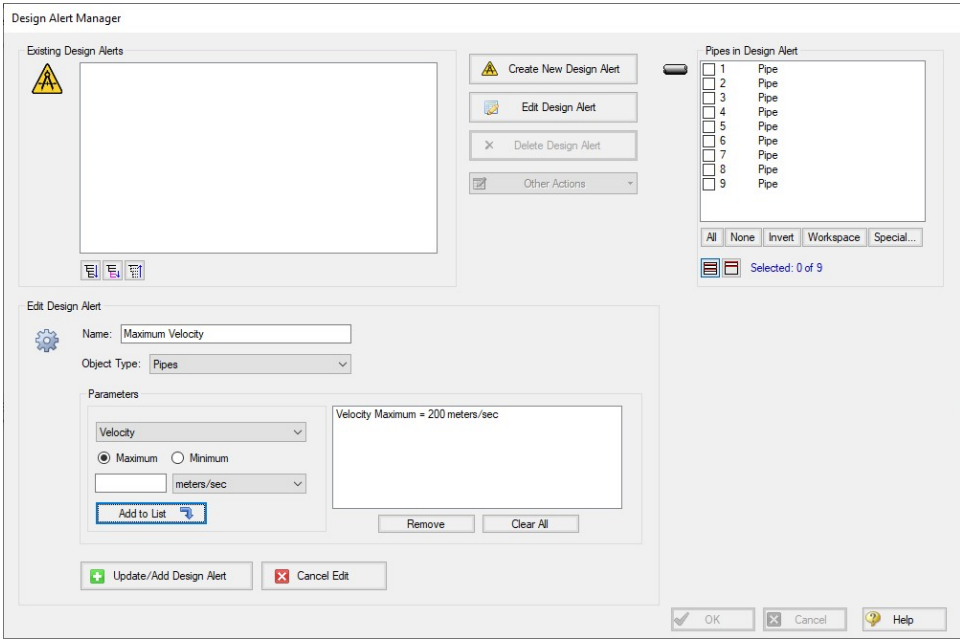
1. Select the desired parameter
2. Select whether this is the maximum or minimum allowable value
3. Enter the value with the appropriate units (if applicable)
4. Click Add to List.

Several Parameter limits can be added to one Design Alert. For example, an alert can be created that requires a pressure within a range by adding both the maximum and minimum pressure limits.

To update a Parameter, simply create it again and click Add to List - the old value will be replaced with the new one.



# Design Alerts

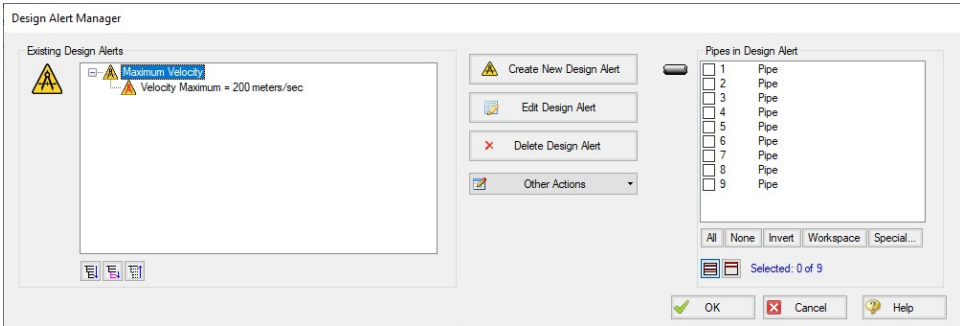


**Figure 2: Creating a Design Alert**

Pipes	General Junctions	Compressors/Fans	Valves	Heat Exchangers	Forces
Enthalpy Stagnation	Pressure Loss Stagnation Total	Mass Flow Rate	Cv	Mass Flow Rate	Maximum Absolute Force
Enthalpy Static	Pressure Loss Static Total	Pressure Discharge Stagnation	Flow Area	Pressure Drop Static	Maximum Negative Force
Mach Number	Pressure Stagnation Inlet	Pressure Discharge Static	K	Temperature Inlet Static	Maximum Positive Force
Mass Flow Rate	Pressure Stagnation Outlet	Pressure Rise Static	Kv	Temperature Outlet Static	
Pressure Stagnation	Pressure Static Inlet	Pressure Suction Stagnation	Mass Flow Rate		
Pressure Static	Pressure Static Outlet	Pressure Suction Static	Open Percentage		
Temperature Stagnation		Speed	Pressure Drop Stagnation		
Temperature Static			Pressure Static Inlet		
Velocity			Pressure Static Outlet		
Vol. Flow Rate					

**Figure 3: Available Design Alerts**

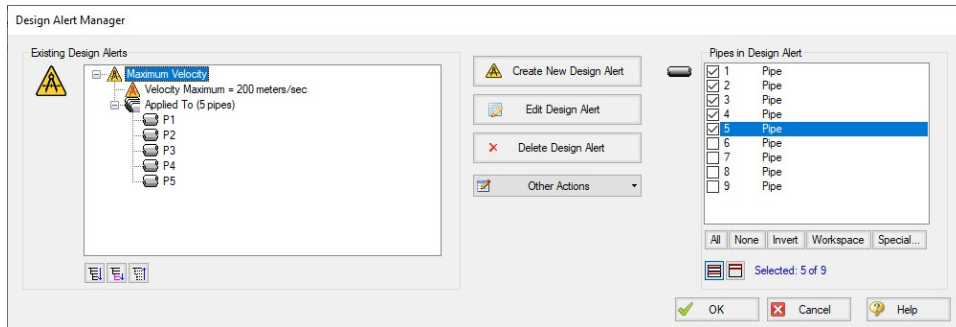
At this point, the Design Alert still has not been created. To finalize it, click "Update/Add Design Alert."



**Figure 4: An Existing Design Alert**

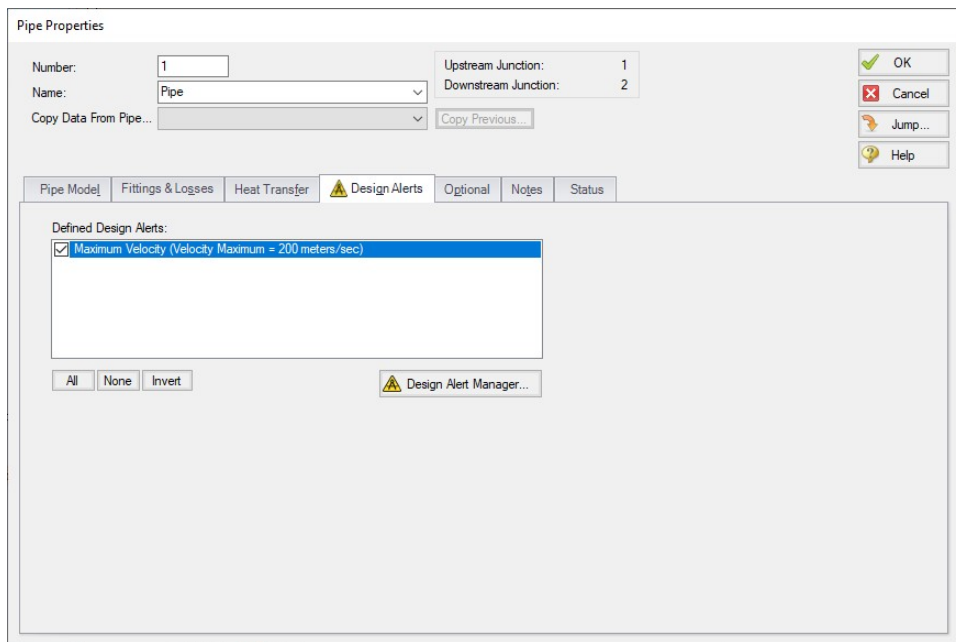
The Design Alert now exists, but has not yet been applied to any objects in the model. To include objects in the model, select them in the pane on the right.

## Design Alerts



**Figure 5: Fully Defined Design Alert**

Note that the application of Design Alerts can also be managed for individual objects in their Properties Window's Design Alerts tab.



**Figure 6: Apply Desired Design Alerts to this Object**

When running the model, the selected object's output values will be checked against the limits that have been defined. If any of the Design Alerts are violated, the user will receive a Design Alert [warning](#) and the violating parameters will be highlighted in the [output](#) tables.

# Design Alerts

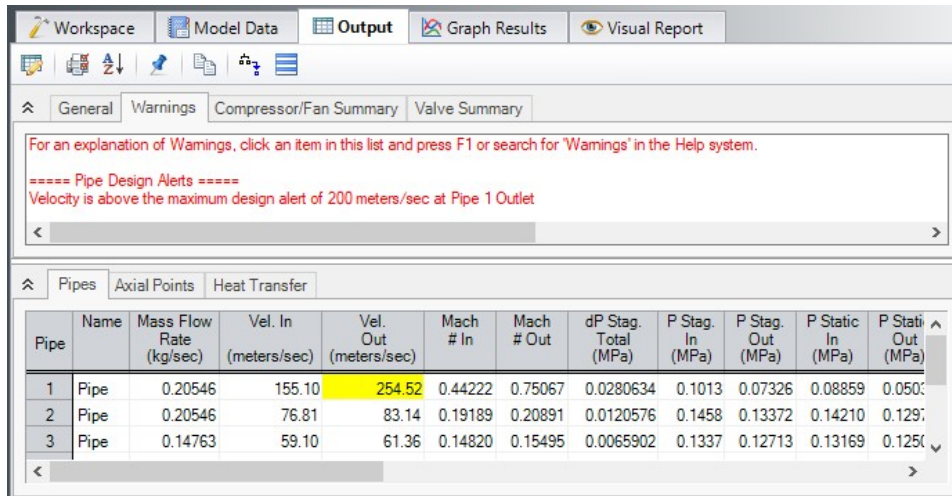


Figure 7: Design Alert Violations in Output

Design Alerts can be cross-plotted in the [Graph Results window](#) vs. the actual results. One good example where this is useful is for cross-plots of maximum and minimum allowed operating pressures.

Design alerts are visible in every scenario in a model, but can be turned off or on for specific scenarios. When creating a design alert for the first time it will be active in that scenario and all its child scenarios. For more information about scenarios, please read the [Scenario Manager](#) help topic.

## Design Alerts From Output

Right-clicking on a cell in the Output Window brings up a context menu. One of these options is Create Design Alert...

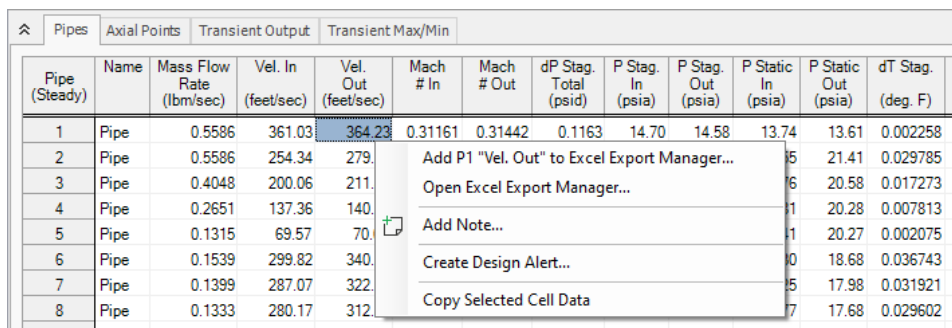
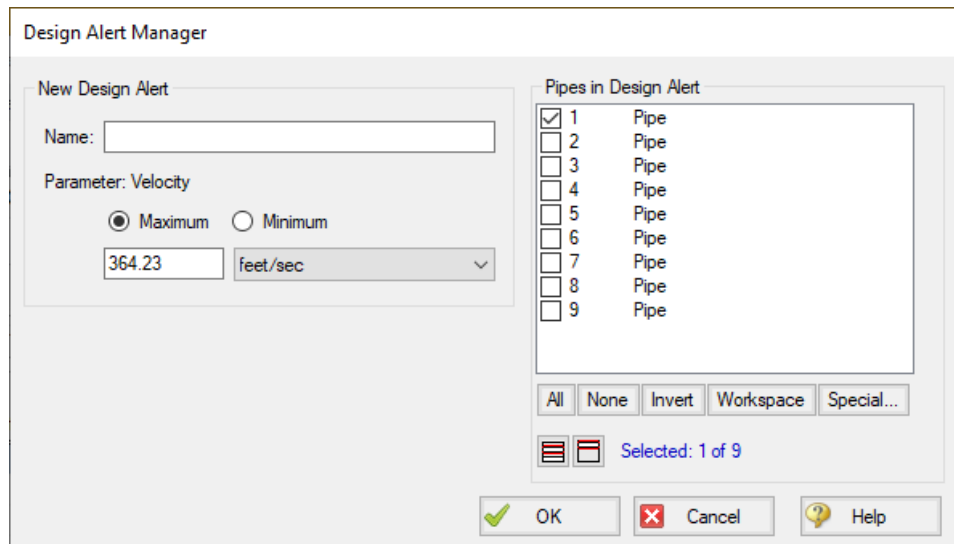


Figure 1: Output Context Menu

Create Design Alert automatically defines a Design Alert based on the selected value and quantity:



**Figure 2: Automatically defined Design Alert**

The Design Alert needs to have a name defined to be added. The specific value of the alert can be modified, and whether it is a maximum or minimum can be selected. The associated object is automatically selected as part of the Design Alert, but this is not required and additional objects can be selected now. After the Design Alert has been created it can be further modified in the [Design Alert Manager](#).

## Design Alert Importing & Exporting

Design Alerts can be exported to a text file to be imported into other model files. This allows the user to quickly load commonly used design alerts into multiple model files.

### Export a Design Alert

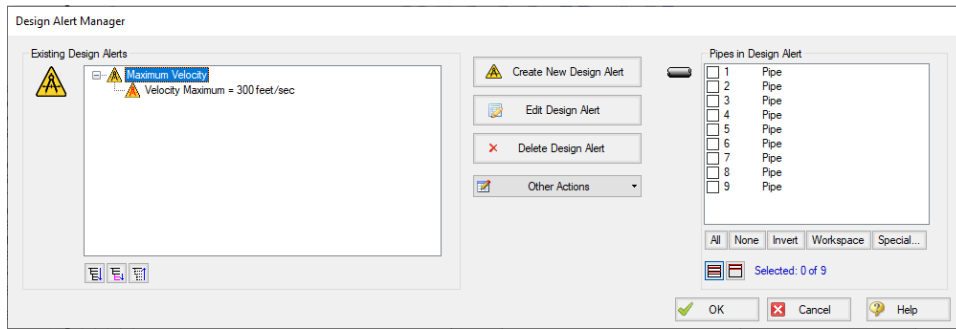
---

To export a Design Alert:

1. Fully [define a Design Alert](#) for any parameter in the Design Alert Manager. For this example, we are using a velocity design alert (Figure 1).
2. From the Other Actions menu, select Save, then save the file with a descriptive name.

# Design Alerts

---



**Figure 1: Defined Design Alert window**

## Import a Design Alert

---

1. Open the Design Alert Manager from the Tools menu.
2. From the Other Actions menu, select Load, then browse to the saved design alert file.
3. The Design Alert should now be added to the Existing Design Alerts list, and can be applied to objects in the model.

## Scenario Manager

The Scenario Manager (in the Quick Access Panel) is a powerful tool for managing variations of a model, referred to as scenarios. It lets you to quickly test the same model with selected differences (such as compressor size, pipe diameter, system location, units, etc.) to find the system which works best. The Scenario Manager allows you to:

- Create, name, and organize scenarios
- Select the scenario to appear in the [Workspace](#) (the 'current' scenario)
- Delete, copy, and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario's properties
- Pass changes from a scenario to its variants
- See what scenarios have output by automatically changing the text color to blue if they have output

All scenarios are saved within the model file.

---

### Scenario

A variant case of a model that is created in the Scenario Manager.

---

### Base Scenario

The Base Scenario is the root scenario at the top of the scenario tree. As the highest level ancestor, It does not inherit from any other scenario. A new scenario can be inserted above the base scenario, at which point the new scenario would behave as the base scenario, and the original base scenario would behave as a child scenario. "Base Scenario" is the default name given in a new, single scenario model.

---

### The Scenario Manager

Illustrated in Figure 1, the top of the Scenario Manager displays the scenario tree, showing the names of the currently defined scenarios, their relationship to other scenarios, and which is the current Workspace scenario (the current scenario is identified both as the scenario with a green check mark next to it and in AFT xStream's [Status Bar](#)). At the bottom, there is a place where you can record notes related to that scenario (propagates to children).

Those attributes whose value comes from a different scenario are linked, while those values coming from the selected scenario are not linked. Linked attributes will inherit changes made in the same object and attribute of the linked ancestor. This inheritance will occur over as many generations as the link exists.

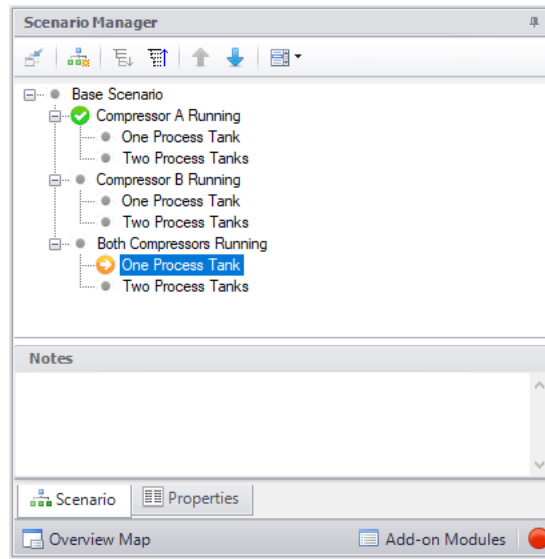


Figure 1: Scenario Manager on Quick Access Panel

## Related Examples

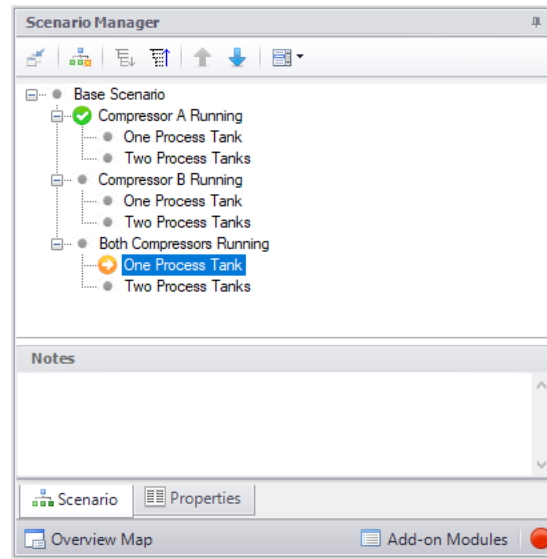
Gas Turbine Fuel System Example .....575

## Creating, Organizing, and Editing Scenarios

Scenarios are referred to as either a parent or child, plus the [Base Scenario](#), which may be thought of as the root scenario. Upon opening a new model file, only a Base Scenario will be present. To create a new scenario below the Base Scenario, right-click the Base Scenario and select Create Child.

Any scenario can be made current by double-clicking on it.

Figure 1 illustrates a model with a Base Scenario with three children: "Compressor A Running", "Compressor B Running", and "Both Compressors Running". Each of these have two children: "One Process Tank" and "Two Process Tanks", representing the different operating cases where the process fluid could be fed to either one or two storage tanks.



**Figure 1: Scenario Manager Window**

The "Compressor A Running", "Compressor B Running", and "Both Compressors Running" scenarios were created by first selecting the Base Scenario and clicking 'Create Child'. Each of the "One Process Tank" and "Two Process Tanks" scenarios were created by choosing 'Create Child' for each of the three different compressor configuration scenarios.

---

**Note:** Alternatively, one could have made the "Compressor A Running" scenario current and created the "One Process Tank" and "Two Process Tanks" scenarios below it. Then, with "Compressor A Running" still selected, select 'Clone With Children' twice, then rename the new clones "Compressor B Running" and "Both Compressors Running".

---

The "One Process Tank" and "Two Process Tanks" scenarios may be thought of as grand children of the Base Scenario. This concept of scenario ancestry is helpful in keeping track of the differences and similarities between scenarios discussed later.

A box with a plus sign next to a scenario indicates it may be expanded to reveal children scenarios while a box with a minus sign indicates the level is fully expanded with all children scenarios shown, if any. No box indicates there are no children. Levels may be expanded by clicking on the plus sign and condensed by clicking on the minus sign.

Right-clicking a scenario provides the following choices:

- **Create Child with No Output** - create child scenarios without results saving duplication and load time
- **Rename** - renames the selected scenario
- **Delete Scenario** - deletes the selected scenario
- **Delete All Children** - deletes all children of the selected scenario
- **Clone Without Children** - creates a copy of the selected scenario at the same level (a sibling). Cloning a scenario prompts the user for a name for the new scenario



- **Clone With Children** - creates a copy of the selected along with its children. Cloning a scenario prompts the user for a name for the new scenario
- **Insert Scenario Above Selected...** - creates a new scenario where the selected scenario is and shifts the selected scenario and its children down a level. This can be used to create a new base scenario. If this is done the user will be prompted to rename the previous base scenario.
- **Promote** - raises a scenario one level, i.e. from a child to a sibling, active only for scenarios more than one level below the base scenario
- **Copy/Save Scenario Names**
  - **Copy Scenario Path Name** - copy the fully-qualified name of the scenario to the Windows clipboard ("Base Scenario/Case 1/Child 3")
  - **Copy Scenario Path Name With File Name** - copy the fully-qualified scenario name along with the model name ("Model.xsm :: Base Scenario/Case 1/Child 3")
  - **Save Scenario Names Only to File...** - copy all scenario names on separate lines, with indents to show child/parent relations
  - **Save Full Scenario Paths to File...** - copy all scenario names on separate lines, with the full path name used for each scenario
- **Save Scenario to File Without Children** - saves the selected to scenario to a separate file without its children
- **Save Scenario to File With Children** - saves the selected scenario and its children to a separate file
- **Compare to Active Scenario** - compares all parameters for all objects between the selected scenario and active scenario and displays the [Comparison Grid](#) showing the results
- **Scenario Comparison Tool** - opens the [Scenario Comparison tool](#) to create a custom scenario comparison

---

**Note:** It is important to remember that AFT xStream's Edit menu Undo does not apply to Scenario Manager operations. To undo a Rename scenario operation, you can rename the scenario again to the old name. To undo a Clone or Save Scenario operation, you can delete the clone or file created. Once a scenario is deleted or promoted, there is no way to un-delete or un-promote the scenario.

---


### Viewing Scenario Differences

---

A scenario's data and that for all of its direct ancestors can be viewed in the [Model Data Window](#). This functionality can be enabled in the Model Data Control window.

### Fast Scenario Changes

---

AFT xStream remembers the most recent scenario you had open, and allows you to quickly change back to that scenario by using the Last Scenario feature. Last Scenario is found on the View menu and Toolbar .

## Modifying Individual Scenarios

When first created, a scenario is identical to its parent. To modify or run a scenario, it must first be made in the current Workspace scenario by selecting it in the Scenario Tree on the Quick Access Panel, then double clicking it or choosing the Load Scenario option on the Right Click menu. (The name of the current scenario is displayed in AFT xStream's [Status Bar](#) and is indicated in the Quick Access Panel with a green check mark.) Once a scenario is the current scenario, changes may be made to differentiate it from its parent using any of AFT xStream's editing tools and functions. When an attribute is changed within a scenario it breaks the link for this attribute. Subsequent changes in the scenario's ancestors will no longer affect this attribute.

Properties of a scenario that may differ from its ancestors include:

- Presence and location of objects - one may add or delete pipes and junctions within a scenario and the workspace locations may vary from scenario to scenario
- [Pipe](#) and [junction](#) properties - all values set in the Properties windows
- [Fluid](#) - fluid selected
- [AFT Arrow Steady Solution](#) - solution method, tolerances, relaxation
- [Output Control](#) - Output window content and formatting
- [MOC Transient](#) - The transient settings can be varied between scenarios
- [Library Manager](#) - The connected database content can be varied between scenarios
- [Visual Report Control](#) - Visual Report content and color maps

[Model Data Control](#) and [User Options](#) apply to all scenarios.

## Scenario Inheritance

Scenario Manager will pass changes made in a scenario to linked attributes of its descendants, allowing multiple scenarios to be modified in one operation and maintain similarity between scenarios where desired.

Any attribute whose value is the same as that of the scenario's parent is linked to the parent. In turn, if the parent's attribute value is the same as that of its parent, then the child's attribute is linked to its grandparent. This ancestral linking of attributes can exist across any number of generations. A change made to a scenario will be passed downward to its children, grandchildren, etc., as far as the link exists. Changes are NOT passed upward from child to parent.

Once an attribute is [changed within a scenario](#), the link is broken to that individual attribute. Subsequent changes in the scenario's ancestors, parent, grandparent, etc., will no longer affect that attribute, but all other attributes within the scenario will remain linked. The [Model Data window](#) with Ancestral Data enabled shows how the data changes between scenarios.

## Scenario Logic Examples

---

For many users, it is easiest to grasp Scenario Manager when it is explained how the coding logic is actually implemented. Blank fields for children, grandchildren, etc., mean to look to the parent for the data.

---

The Base Scenario never has blank fields (Table 1). Data only passes downwards, never upwards.

**Table 1: Blank fields for Child #1 and Grandchild #1 mean that the data is to come from the parent. If the Base scenario data is changed, all descendants are changed.**

	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

If a child scenario does not have a blank field, then data for that property is initiated at that scenario level (see Table 2).

**Table 2: Child #1 does not have a blank field, so its Diameter would be 2, not 3, as would Grandchild #1.**

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--

If a scenario is changed, and its child has different data, then the change will not pass downwards (see Table 3).

**Table 3: Changing the Base Scenario Diameter from 3 to 6 would not impact Child #1 or any descendants in that line. Changing the Length from 25 to 40 would also change the length in Child #1, Grandchild #1, and any descendants of Grandchild #1.**

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--



	Diameter	Length
Base	6	40
Child #1	2	--
Grandchild #1	--	--

If a child scenario has data that is different than the parent, its children cannot relink to the parent (see Table 4).

**Table 4: Even if the Grandchild #1 has the same Diameter as the Base, it is not linked to the Base because it and its parent are not blank.**

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	3	--

If a child scenario's data, which was previously changed and is thus different from the parent, is changed back to the same value as the parent, the inheritance link is re-established (see Table 5). Its descendant's link is also re-established.

**Table 5: If the Diameter in Child #1 is changed to be the same as the Base, it will be "blanked out" the next time the scenario is loaded and the link re-established. And so will Grandchild #1, if its Diameter is also the same.**

	Diameter	Length
Base	3	25
Child #1	3	--
Grandchild #1	3	--



	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

Table 6 shows data for two pipe properties across three scenarios. The data that would be used when each scenario is loaded is as follows:

- Base scenario
  - Diameter = 3
  - Length = 25
  - Changes to Diameter will not pass downwards
  - Changes to Length will affect only Child #1.
- Child #1 scenario
  - Diameter = 2
  - Length = 25
  - Changes to Base Diameter will not affect Diameter
  - Changes to Base Length will affect Length
- Grandchild #1 scenario
  - Diameter = 2
  - Length = 15

- Changes to Base Diameter will not affect Diameter
- Changes to Child #1 Diameter will affect Diameter
- Changes to Base Length or Child #1 Length will not affect Length

*Table 6: Input data for these properties explained in the text.*

	Diameter	Length
Base	3	25
Child #1	2	-
Grandchild #1	-	15

---

## Re-Establishing Broken Scenario Links

---

A link may be re-established by returning the attribute to the same value as that of its parent. This can be done manually by entering the value or selecting the Copy Data From Pipe list and selecting the Parent Pipe Data option. Within other data windows, it is done by selecting the Same As Parent option in Output Control, Analysis Setup, Cost Settings, Database Manager or Visual Report Control.

Since links are identified by comparing attribute values of pipes or junctions with the same Workspace ID number, renumbering a scenario will break the links of all pipes and junctions renumbered. Since numbers must be unique, once a link has been broken by renumbering, it may not be re-established.

## Running Models in Batch Mode

AFT xStream [scenarios](#) or models can be run in batch mode, a feature that is especially useful for running a group of scenarios or models during lunch or overnight.

To run models in batch mode, follow these steps:

1. Click "Start Batch Run" on the File menu.
2. Select the Batch Run Type. This is where you select if you want to run scenarios from the current model or different model files. If the current model does not have any scenarios, then only the second option will be available.
3. If the Batch Run Type is "Scenarios in Current Model", then choose the Add Scenarios button and select the scenarios you wish to run.
4. If the Batch Run Type is "Models from Different Files", then the models can be selected individually using the "Add Model Files" button, or loaded from a Batch File (by Choosing the "Load Existing Batch File" button). A Batch File is a text file listing all models you would like to run. At any time, a selected model file can be removed by clicking the "Remove Selections". The current list of models can be saved to a Batch File for use in the future by clicking the "Save List to File" button.
5. Use the Output Options in the lower left to save the output data to a file and/or send it to a printer or Adobe PDF file. All reports will use the specified font.

6. Alternatively, the option can be chosen to Save Using Excel Export Manager by either saving to separate sheets, or saving the scenarios to separate workbooks. The Excel Export option can only be chosen if the batch run type is for Scenarios in Current Model. Additionally, the desired output must first be configured in the [Excel Export Manager](#) under the File menu for any data to be exported.
7. Check the box to Run Batch in Background if you wish for all selected scenarios to run without showing the Solution Progress window. AFT xStream will run the simulations in the background and will not interrupt any work being done in other programs.
8. Click the Start Run button to begin the batch run.

AFT xStream then opens each scenario or model file in sequence, automatically runs each scenario/model that has a completed [Analysis Setup Window](#), and sends results to the specified destination. All pertinent information that is generated during the batch run (such as error messages) are automatically displayed when the final scenario/model is finished. Output files generated from each scenario or model run is automatically saved before moving to the next run.

The entire batch run can be canceled at any time by clicking the Cancel button in the Solution Progress window. Output from scenarios that have already been run is still retained when the Batch Run is canceled.

---

**Note:** You can plot output from multiple scenarios using [Multi-Scenario Graphs](#).

---

### Open Model Using Last Scenario

When the model is saved, the current [scenario](#) is remembered, and by default, reloaded when the model is opened again. Your preference of whether to always open the model to the most recent scenario or to the base scenario can be specified in the [User Options window](#).

### Scenario Comparison Tool Overview

Scenario Comparison Tool (accessed via the Tools menu) is a powerful tool that allows users to compare model parameters between scenarios in a given model. It lets users quickly identify differences between two or more scenarios without needing to search through the model data for each scenario or inspect each element of the model. This tool can be especially powerful for users reviewing others' models, or for users returning to models after a period of time. It can also be powerful for troubleshooting purposes. The Scenario Comparison Tool lets you:

- Compare any two or more scenarios
- Compare selected Pipes and Junctions between selected scenarios
- Compare selected Pipe and Junction parameters between selected scenarios
- Compare selected General Properties between selected scenarios

### Related Examples

---

---

High Pressure Steam - Forces Example ..... 606

---

## Related Blogs

---

[New Feature Highlight - Scenario Comparison Tool](#)

### Accessing Scenario Comparison Tool

The Scenario Comparison tool can be accessed from the Scenario Manager, the Workspace, and the Tools menu.

---

#### From the Scenario Manager

---

To set up a comparison from the Scenario Manager, first [select](#) the pipes and junctions to compare in the workspace. Then, select the desired scenario to compare the current scenario to in the Scenario Manager. Note the current scenario is indicated by the green check-mark symbol. Right-click the desired scenario, and select Compare Selected Scenarios.... This will bring users to the Scenario Comparison window and display a [grid](#) of the differences between the selected objects in the two scenarios.

To modify the comparison use the [Modify Comparison](#) button at the bottom of the window.

---

#### From the Workspace

---

From the workspace, users can quickly select desired pipes and junctions in the scenario and compare them to those found in any other scenario.

First, select the desired pipe(s) and/or junction(s) in the workspace. Then, right-click the Workspace, choose Compare Scenarios, and select the desired comparison. This will bring users to the Scenario Comparison window and display a [grid](#) of the differences between the selected objects in the two scenarios.

To modify the comparison use the Modify Comparison button at the bottom of the window.

---

#### From the Tools Menu

---

From the [Tools](#) menu, there are multiple options to generate the Scenario Comparison Report.

A comparison grid can be directly generated by selecting the desired objects to compare in the Workspace, then selecting the scenarios to be compared from the Compare Scenarios menu option.

Alternatively, the Scenario Comparison Tool menu option can be selected to build a custom scenario comparison by directly selecting the scenarios, objects, and parameters to be included in the comparison. See [Modify Comparison](#) for more information on the available options.

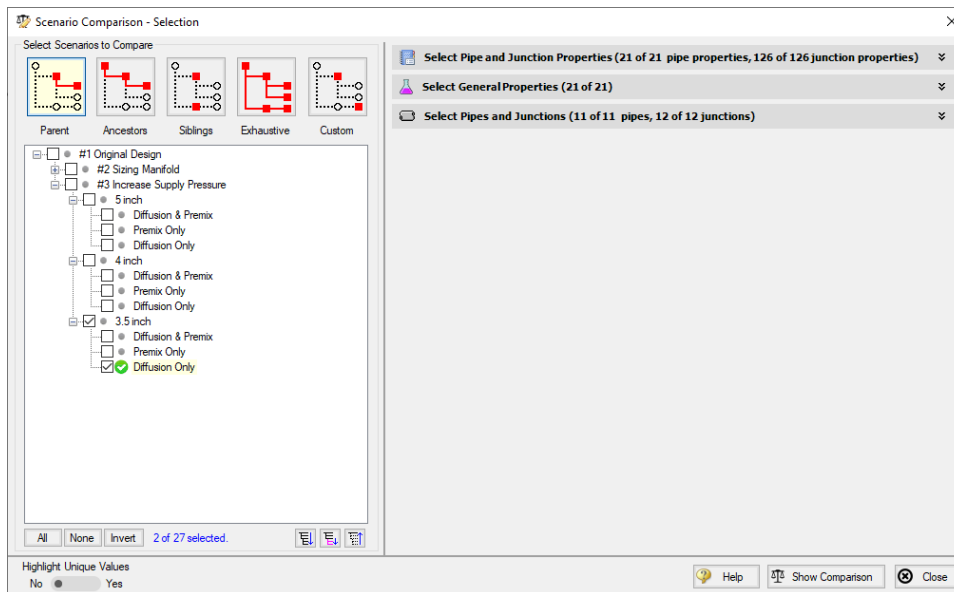
Click the Show Comparison button at the bottom of the window to view the comparison. To make additional changes, use the Modify Comparison button at the bottom of the window.

## Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

## Modify Comparison

Several Options are available to customize the [Scenario Comparison Grid](#). These options are available from the Modify Comparison button at the bottom of the scenario comparison report, or by selecting Scenario Comparison Tool from the Tools menu. The Modify Comparison window is shown in Figure 1 below.



**Figure 1: Scenario Comparison window options to Modify Comparison**

## Select Scenarios to Compare

The Scenario Comparison Tool has four pre-set templates for comparing scenarios, plus the ability to set up a custom comparison. These are:

- **Parent:** This template compares the current scenario with that scenario's parent.
- **Ancestors:** This template compares the current scenario with all of that scenario's direct ancestors. This setting would be similar to the comparison available in the Model Data window, though it does provide a more comprehensive comparison.
- **Siblings:** This template compares the current scenario with all of that scenario's siblings.
- **Exhaustive:** This template compares all of the scenarios in the model with each other.
- **Custom:** This option allows users to compare any two or more scenarios throughout the model.



## Select Pipe and Junction Properties

---

This section sets what parameters will be compared such as length, diameter, roughness, etc.

Users can choose to compare a set of specified properties, or all pipe and junction properties by selecting the desired properties under Select Pipe and Junction Properties. Only properties selected in this section will be considered when the comparison is run.

A helpful option to filter the parameters to be displayed in the comparison is to display the parameters shown in the Model Data window. This can simplify the comparison, but might hide some differences between scenarios. Use the [Model Data Control](#) window to further define what parameters are shown with this option.

## Select General Properties

---

This section sets what General Properties will be compared such as fluid properties, tolerances, etc.

Users can choose to compare all or a select set of general properties for the scenarios including options that are located in the Analysis Setup window such as the [AFT Arrow™ Steady Solution](#), [Fluid](#), [Sectioning](#), and [MOC Transient](#) panels.

## Select Pipes and Junctions

---

This section sets which pipes and junctions will be compared for the model.

Users can choose to compare all objects in the model, or a selected set of pipes and junctions. Users can manually select the desired objects, select those highlighted in the workspace, or use the [Select Special](#) tool to select desired groups or other sets of objects.

## Related Blogs

---

[New Feature Highlight - Scenario Comparison Tool](#)

### Scenario Comparison Grid

Choosing the Compare Selected Scenarios menu item from the right-click or Tools menu will bring the user directly to the Scenario Comparison Grid based on any selected Workspace objects and scenarios. The settings for the scenario comparison can be seen/modified by clicking [Modify Comparison](#) at the bottom of the window.

The tabular data from the report can be copied to and pasted to a separate file or worksheet using the Copy button at the bottom of the window.

Item Show	Parameter Show	Scenario		
		3.5 inch	4 inch	Diffusion Only
P4	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P6		No Differences		
P7	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P9		No Differences		
P10	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
<b>Junctions (12 of 12 compared)</b>				
J1 (Booster Compressor Discharge)		No Differences		
J2 (Natural Gas Heater)		No Differences		
J3 (Supply Valve)		No Differences		
J4 (Tee or Wye)		No Differences		
J5 (Diffusion Gas FCV)	Control Setting	5 lbm/sec	5 lbm/sec	22 lbm/sec
J6 (Combustion Baskets)		No Differences		
J7 (Tee or Wye)		No Differences		
J8 (Premix Gas FCV 1)	Special Condition	None	None	Closed (stays closed)
J9 (Combustion Baskets)		No Differences		
J10 (Bend)		No Differences		
J11 (Premix Gas FCV 2)	Special Condition	None	None	Closed (stays closed)

Only parameters with differences will be shown. If all chosen parameters for an item are the same, then the item will say 'No Differences'. Parameters containing tabular data (e.g. polynomial constants) can be viewed by right-clicking the parameter's row.

Item	Parameter	Highlight Unique Values
All Parameters Same Value	Same Value Across Scenarios	No <input type="radio"/> Yes <input type="radio"/>
At Least 1 Parameter Difference	Different Values Across Scenarios	No <input type="radio"/> Yes <input type="radio"/>

Figure 1: Default Scenario Comparison Report view

## Filtering Options

When a large comparison is run, typically many of the objects/parameters in the model will be unchanged across scenarios, which may result in a large number of items with no differences. To help condense the Scenario Comparison Grid, several toggles are available at the top of the Item and Parameter columns which allow the user to show/hide rows in the report. Switching the toggle to All will show all items or parameters being compared. If the toggle is set to differences, then only Items/Parameters that have a different value in at least one scenario will be shown. Thus, if both toggles are set to All, then all parameters will be displayed for all items. If both toggles are set to Differences, then only items that have parameters with different values will be shown, and only the parameters that differ will be displayed.

## Color Coding

The cells in the Item and Parameter columns will be colored either yellow or green to indicate whether that item/parameter has differences between the scenarios being compared. Yellow indicates differences, while green indicates that no differences exist. Parameters will be colored yellow if the value of that parameter is different in at least one scenario that is being compared. Items will be colored yellow if at least one parameter that is being compared for that item has a different value in one or more scenarios.

The Highlight Unique Values toggle is used to color the values to visually link identical values. All values under the currently active scenario will remain grey, while values that have differences will be colored. This is powerful to quickly see differences when more than two scenarios are being compared. If two scenarios have the same value for a parameter, the values will have the same shading. See Figure 2 below for an example.

# Scenario Manager

Item Show		Parameter Show		Scenario		
Differences <input checked="" type="checkbox"/> All		Differences <input checked="" type="checkbox"/> All		3.5 inch	4 inch	Diffusion Only
P4	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
P6		No Differences				
P7	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
P9		No Differences				
P10	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch	4 inch	3-1/2 inch
<b>Junctions (12 of 12 compared)</b>						
J1 (Booster Compressor Discharge)		No Differences				
J2 (Natural Gas Heater)		No Differences				
J3 (Supply Valve)		No Differences				
J4 (Tee or Wye)		No Differences				
J5 (Diffusion Gas FCV)	Control Setting	5 lbm/sec	5 lbm/sec	5 lbm/sec	22 lbm/sec	22 lbm/sec
J6 (Combustion Baskets)		No Differences				
J7 (Tee or Wye)		No Differences				
J8 (Premix Gas FCV 1)	Special Condition	None	None	None	Closed (stays closed)	Closed (stays closed)
J9 (Combustion Baskets)		No Differences				
J10 (Bend)		No Differences				
J11 (Premix Gas FCV 2)	Special Condition	None	None	None	Closed (stays closed)	Closed (stays closed)

Only parameters with differences will be shown. If all chosen parameters for an item are the same, then the item will say 'No Differences'. Parameters containing tabular data (e.g. polynomial constants) can be viewed by right-clicking the parameter's row.

Item	Parameter	Highlight Unique Values
All Parameters Same Value	Same Value Across Scenarios	No <input checked="" type="checkbox"/> Yes
At Least 1 Parameter Difference	Different Values Across Scenarios	No <input type="checkbox"/> Yes

Copy Help Modify Comparison Close

**Figure 2: Scenario comparison showing differences only. Each unique value is highlighted with a different color to indicate which scenarios have differences from the currently active scenario, which is the "3.5 inch" scenario for this example.**

## Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

## Add-on Modules

AFT xStream has one optional add-on module which extends its modeling capabilities into other areas. The module allows pulsation frequency analysis.

- [Introduction to the PFA Module](#) (Pulsation Frequency Analysis)

## Introduction to the PFA Module

### What is the PFA Module?

---

Steady state pressure and flow dynamics, henceforth referred to simply as “pulsation”, is a common problem, which can cause high vibration, fatigue failures, frequent maintenance outages, and flow uniformity or product quality problems.

The PFA module helps engineers identify and avoid resonant frequencies in systems. Problems can arise when the frequencies that are excited by pulsation are at or near the acoustic resonant frequencies of a fluid system. The module is used to calculate the acoustic resonance frequencies that could damage system equipment.

The PFA module can also be used to check compliance with the pressure limits defined in API-618 for reciprocal compressors.

### How Does the PFA Module Work?

---

The Pulsation Frequency Analysis (PFA) module is used to determine resonant frequencies that could damage system equipment by calculating the acoustic resonance.

The general steps of pulsation analysis involve the following:

1. The piping and process equipment system is first modeled.
2. At run-time a pulse is generated to determine excitation frequencies, and generate the forcing function. This forcing function is then placed in the transient data of the junction at which the pulse is applied.
3. The model is run with the forcing function to generate the Acoustical Frequency Analysis graph, providing information on the resonant frequencies in the system.
4. Replace the pulse transient with a transient at the frequencies of interest in the reciprocating compressor and turn on the API-618 evaluation option
5. When the model is run xStream will show warnings if API-618 is violated

### Using the PFA Module

---

The user has the option of activating or not activating the PFA module when AFT xStream first loads. After AFT xStream is loaded, the PFA module can be activated or deactivated for use from the [Tools menu](#). Whether or not PFA is activated impacts the Analysis menu and Graph Results window.

If the PFA module is active, the user can still run models without pulsation. This is selected under Pulsation Frequency Analysis on the Analysis menu. Hence there are three possibilities for the PFA module.

- PFA is not active
- PFA is active and pulsation is disabled
- PFA is active and pulsation frequency analysis is performed

## Related Examples

Frequency Analysis - PFA Example ..... 623

## Defining Pulsation Settings

The Pulse Setup panel (Figure 1) allows you to specify required input for the [PFA module](#), including the junction at which the pulse will be applied, as well as the pulse properties. This panel will not be visible unless the PFA module has been activated from the [Modules panel](#).

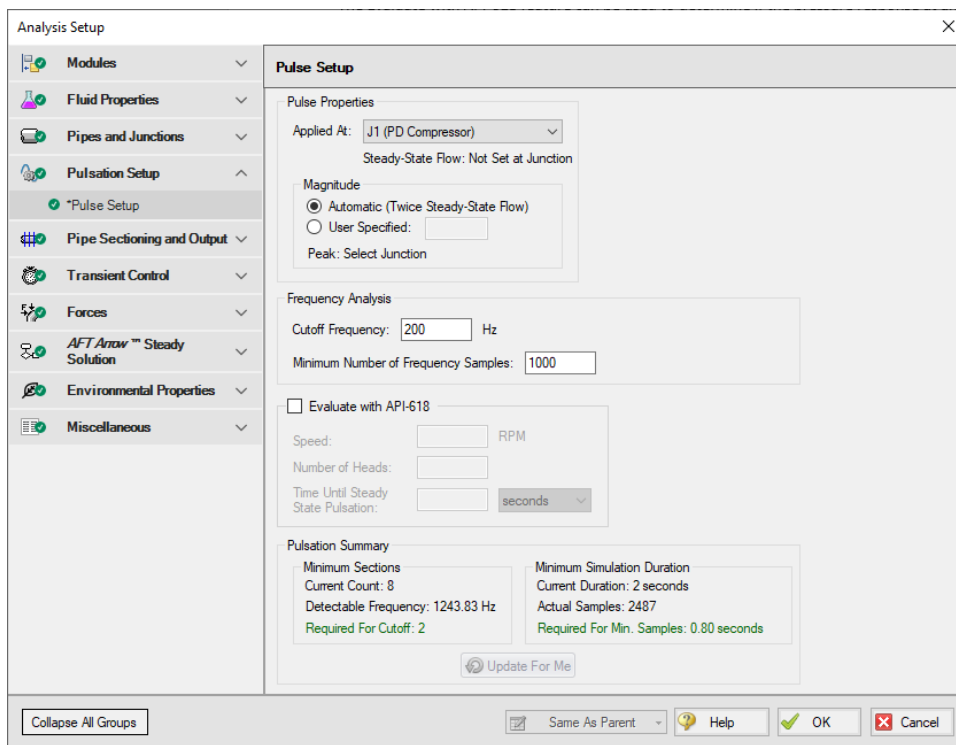


Figure 1: Pulse Setup panel in the Analysis Setup window

## Pulse Properties

At the top of the window the Pulse Properties can be specified, including the junction that will act as the source of the pulse and the magnitude of the pulse. By default, the magnitude of the pulse is twice the

steady-state flow, but you have the option to define this magnitude as “User Specified”. This pulse will be assumed to last for two time steps and then return to steady-state flow. The pulse can only be applied to an assigned flow junction.

## Cutoff Frequency

---

The cutoff frequency in xStream is the maximum frequency for displayed output. The defined cutoff frequency must be less than or equal to the detectable frequency for the analysis, which is calculated using the equation for maximum frequency as follows

$$\textit{Max Frequency} = \frac{1}{2 dt}$$

where dt is the time step. The time step for the model is not known until run-time; However, the time step can be estimated based on the inputs on the [Pipe Sectioning panel](#). The Pulsation Summary at the bottom of the Pulse Setup panel shows the Detectable Frequency based on the current settings on the Pipe Sectioning panel, as well as the minimum number of sections required to capture the specified cutoff frequency. If the current Cutoff Frequency and sectioning are not valid for the analysis, then either the Cutoff Frequency should be decreased, or the minimum number of sections should be increased.

When the Analysis is run it is possible to account for all frequencies up to the Detectable Frequency calculated based on the time step. However, the results in the output will only display frequencies up to the user defined Cutoff Frequency. Therefore it is important to define a large enough Cutoff Frequency to be able to analyze all relevant frequency responses.

## Minimum Number of Frequency Samples

---

The minimum number of frequency samples used to generate the forcing function is specified in the Frequency Analysis section of the Pulse Setup panel. The default of 1000 should be sufficient for most cases. The Minimum Number of Frequency Samples will impact the required simulation duration - a larger number of samples will require a longer simulation duration. The required simulation duration can be calculated by dividing the minimum number of frequency samples by the detectable frequency. If the current simulation duration is not long enough to accommodate the minimum number of frequency samples, then the simulation duration should be lengthened on the [Simulation Mode/Duration panel](#), or the minimum number of frequency samples should be decreased.

## Evaluate with API-618

---

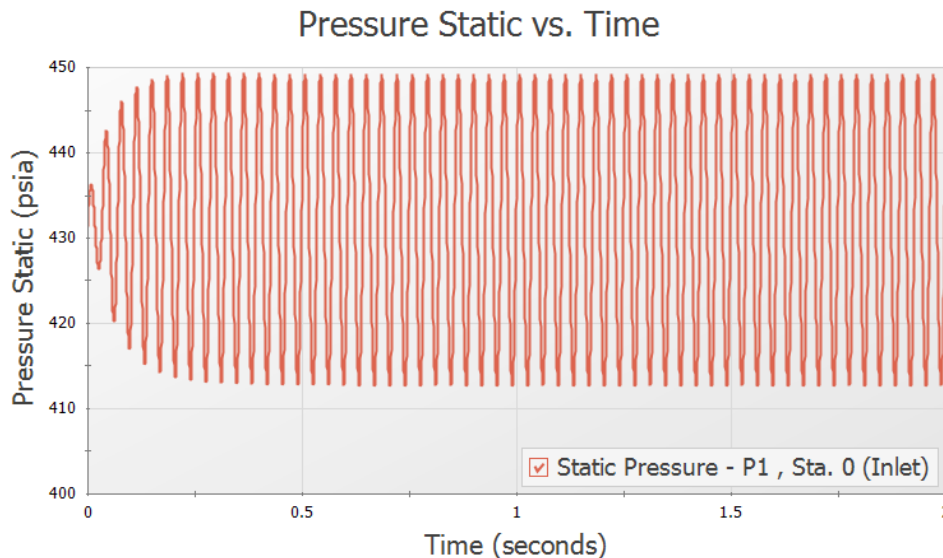
The Evaluate with API-618 feature can be used to determine if the pressure response at any of the acoustic natural frequencies of the system exceeds the limits defined in API-618 for reciprocating compressors. To evaluate API-618 the model should be first be run with PFA enabled and the API-618 feature disabled to determine the acoustic frequencies. Once the excitation frequencies are known, child scenarios can be created by the user to evaluate the pressure results when the compressor operates at the excitation frequencies, as is demonstrated in the last step of the [Frequency Analysis Example](#). The

API-618 evaluation should be enabled in the excitation frequency child scenarios to check if the pressure results exceed the limits in API-618.

The required inputs to check API-618 are as follows:

- **Speed** - The speed the reciprocating compressor is operating at.
- **Number of Heads** - The number of pistons in the reciprocating compressor.
- **Time Until Steady State Pulsation** - The time required for the dissipation of artificial transients caused by the introduction of a flow rate wave to the system. This time should be determined by running the scenario with the "Time Until Steady State Pulsation" set to 0, then plotting the pressure results at the outlet of the compressor. The Time Until Steady State Pulsation should be set as the time at which the pressure waves have consistent amplitudes. For example, in Figure 2 below the waves begin to reach constant amplitude at about 0.25 seconds. It is best to choose a slightly longer time interval than the estimated time interval for increased accuracy when checking API-618.

Note the Time Until Steady State Pulsation input only changes which pressure results are included in the API-618 evaluation, the transient results are not impacted by this time.



**Figure 2: Example pressure results at the outlet of the reciprocating compressor with an oscillating flow transient specified.**

The equations for the API 618 peak to peak pressure level limits are as follows ([American Petroleum Institute 1995](#), pg 40):

$$P_1 (\%) = \left( \frac{300}{(P_L I D f)^{0.5}} \right) \quad US Units$$

$$P_1 (\%) = \left( \frac{397.1}{(P_L ID f)^{0.5}} \right) \quad \text{SI Units}$$

where  $P_1(\%)$  is the maximum allowable peak to peak pressure level of individual pulsation components at fundamental and harmonic frequencies, as a percentage of absolute line pressure,  $P_L$  is the average absolute line pressure in psi (bar for SI),  $ID$  is the inside diameter of the pipe in inches (mm), and  $f$  is the pulsation frequency in hertz.

## Pulsation Summary

The Pulsation Summary section at the bottom of the Pulse Setup panel provides information on the current transient settings in the model to check that the sectioning and simulation duration are sufficient based on the user-specified Cutoff Frequency and Minimum Number of Frequency Samples. Therefore the Pulsation Summary will only be able to display results if the [Pipe Sectioning panel](#) and [Simulation Mode/Duration panel](#) have already been fully defined.

The Update For Me button under the Pulsation Summary can be used to update the Minimum Number of Sections and/or the Stop Time for the simulation if they are not sufficient based on the user-specified Cutoff Frequency and Minimum Number of Frequency Samples.

A short description of each of the Pulsation Summary items is given here.

- Minimum Sections
  - **Current Count** - Displays the "Minimum Number of Sections Per Pipe" defined on the Pipe Sectioning panel
  - **Detectable Frequency** - Displays the Max Frequency calculated as discussed above in the Cutoff Frequency section
  - **Required for Cutoff** - Displays the required Minimum Number of Sections Per Pipe which allow the Detectable Frequency to be greater than or equal to the Cutoff Frequency
- Minimum Simulation Duration
  - **Current Duration** - Displays the current simulation duration defined on the Simulation Mode/Duration panel (Stop Time minus Start Time)
  - **Actual Samples** - Displays the Actual Samples used for the frequency analysis, calculated by multiplying the Current Duration by the Detectable Frequency
  - **Required For Min. Samples** - Displays the minimum required simulation duration based on the user-specified Minimum Number of Frequency Samples and the calculated Detectable Frequency

## Related Examples

Frequency Analysis - PFA Example .....623



## Interpreting Pulsation Results

The accuracy of results when using the PFA module in xStream will be affected by the sources of error described in the [Transient Result Accuracy](#) topic, but there are a few additional considerations specific to the PFA module.

The methodology the PFA module uses assumes that the fluid parameters change linearly in the system, similar to the assumptions made for the [Method of Characteristics](#) in the transient solver. The assumption of linear behavior primarily affects the magnitude of the frequency response results, rather than affecting the locations of the largest amplitude responses. However, if the error for the analysis is large it is possible that the predicted resonant frequencies may be shifted. To minimize error the considerations in the Transient Result Accuracy topic should be taken into account, in particular running the model with higher sectioning counts. It may also be prudent to analyze the system response not only at the predicted resonant frequencies, but also at frequencies near to the predicted frequencies to ensure that all frequencies of concern have been accounted for in the system.

### Related Topics

Method of Characteristics.....	435
--------------------------------	-----

### Related Examples

Frequency Analysis - PFA Example .....	623
--	-----

## PFA Module Glossary

Much of the terminology associated with pulsation analysis is specific to time-frequency analysis and/or to vibrational analysis. The following words have been defined for ease of comprehension when working with the PFA module. Note that most of the defined words include page numbers and, occasionally, equation numbers. All of these references can be found in the document referenced at the end of this page.

- **Sample Rate** - The inverse of the time step (Page 14)
- **Harmonic Frequencies** - Multiples of the base frequencies that excite the system
- **Forcing Function** - The function that will be used to “ring” the system. This function can be the pulse in a non-low-pass-filtered system, or can be determined by the cutoff frequency that is determined by using Fast Fourier Transform (Page 7)
- **Pulse** - The near-instantaneous increase, then normalization, of flow
- **Fast Fourier Transform (FFT)** - A method used to transpose the time-domain (flow vs. time) into the frequency-domain (magnitude vs. frequency)
- **Cutoff frequency (fc)** - This is the maximum frequency that will be shown in the frequency analysis results.
- **Detectable Frequency** - This is the maximum frequency which will be recognized by xStream during the transient solution. If frequencies exist above this frequency they will be aliased to lower frequencies in the final solution.

## References

---

- 1.) Blanding, James M., Ph.D., and Trey Walters, PE. Pulsation Analysis in Positive Displacement Pump Systems Using Waterhammer, Modal, and Animation Software. Houston, TX: 45th Turbomachinery & 32nd Pump Symposia, Sept. 2016. <http://www.aft.com/documents/TechnicalPapers/2016/TPS2016-PulsationAnalysis.pdf>

## Libraries

One of the more powerful features of AFT xStream is that it provides extensive customization capabilities. AFT xStream provides the tools to build custom libraries for fluid properties, pipe materials, and all junction types except Volume Balance. AFT xStream incorporates these libraries internally and offers them to you through list boxes.

There are three categories of libraries: Local User, External, and AFT Internal.

The **Local User Library** is located in the Xtr\_USER2.DAT file in the local user directory, and stores user-specified data for most component types for local use. It cannot be transferred to other users.

The **AFT Internal Library** is located in the xStream2.DAT file in the local user directory and stores AFT-provided libraries, as well as user-specified fittings & losses. It cannot be transferred to other users.

**External** libraries are created by the user and can contain data for any of the library types. In AFT Impulse you can create multiple network or local custom libraries using the [Library Manager](#) on the [Library menu](#). External libraries can be shared to multiple users from a network location as is discussed in [Creating an Enterprise-wide Network Library System](#).

<a href="#">Library Manager</a> .....	391
<a href="#">Network Library Overview</a> .....	403
<a href="#">Library Verification</a> .....	407
<a href="#">Library Locations in General Preferences</a> .....	408

## Related Blogs

---

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Library Manager

Accessed from the [Library Menu](#), the Library Manager is a new feature to AFT xStream 2 and replaces the Database features used in previous versions. The Library Manager allows you to:

- Connect to and disconnect from libraries
- Create and delete libraries
- Add content to libraries
- Review the content of libraries
- Move contents between libraries

## Related Blogs

---

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Library Browser

The first panel of the [Library Manager](#) is the Library Browser. Use the Library Browser to create a new library, add an existing library, view contents of an added library, and connect or disconnect libraries to the model.

## Adding and Removing Existing Libraries

---

To work with a library you must first make it available to AFT Impulse. Referring to Figure 1, clicking on the Add Existing Library button will open a standard file dialog box with which you may browse to and select a library. A library may be removed from the list of available libraries by right-clicking on the library and selecting Remove Library from List. A summary of the library contents can be seen in the section on the right when a library, library section, or component is selected in the list of available libraries.

The AFT DEFAULT INTERNAL LIBRARY and the AFT XSTREAM [LOCAL USER LIBRARY](#) may not be removed and will always appear in the list of available libraries. External shared libraries will also always appear in the available library list.

*Figure 1: Library Browser in the Library Manager*

## Connecting and Disconnecting Libraries

---

In order to use the components from a library in the model the library must be **connected**. A library is connected to the model when it is shown with a check mark in the Library Browser. All libraries are automatically connected when they are first added to the Library Manager.

Sections or components in a library can also be connected/disconnected on an individual basis if the library as a whole is connected.

It may be desired to disconnect a library if you do not want the components from that library to be available for use in the current model, but you do want the library to be available for use in other models.

Clicking on Set As Default will establish the list of connected libraries as the default list, while clicking on User Default will set the connections to the default list.

AFT Internal Pipe Material libraries can be connected or disconnected as a group from the Other Actions button under the available libraries list.

### Editing Library Contents

---

Existing content can be moved to/from, copied to/from, or deleted from available libraries in the Library Browser.

To edit an item right-click on the item in the Library Browser, then click the desired action from the right-click menu. Note that moving, copying, and deleting can be done to a library as a whole, or can be done for just a section or single component in a library. Individual components can also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.

*Figure 2: Library contents can be transferred or deleted from the right-click menu*

### Creating External Libraries

---

By default library items are stored in the AFT Default Internal Library or Local User Library, both of which cannot be shared with other users. Library items can be moved or copied to an external library which can be shared with multiple users.

To create an external library click the Create New Library button underneath the available libraries list. You will be prompted to provide a name for the file, and a library description (Figure 3). The provided description will be displayed as the library's name in the Library Manager.

To add content to an external library copy or move content from an existing library as described in the Editing Library Contents section. Junction components can also be directly added to external libraries as is described in the [Junction Libraries](#) topic.

For more details on creating and sharing external libraries see [Creating an Enterprise-wide Network Library](#).

*Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected*

### Related Blogs

---

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Junction Library

---

### Adding Custom Junction Data to the Component Library

---

To simplify modeling and eliminate repetitive data entry, AFT xStream allows you to save frequently-used junction data to a custom library. Once saved, the custom equipment can be selected from the Library List in the [Properties window](#) at any later time for immediate recall of the data. Using this tool, you can easily build your own library of compressors, valves, or other components supplied by specific manufacturers.

**To save a Custom junction to the library** - open the junction Properties Window and enter all required data. Close the window and choose Add Component to Library from the [Library menu](#). You will be prompted to enter a name for the custom component. Enter a descriptive name and close the window. If you open the junction again, the custom name will be selected in the Library List.

---

### Editing the Component Library

---

You can delete or rename any existing library component in the Edit Junctions panel of the Library Manager, accessed from the Library menu. To edit an existing library entry, select the item in the list and click the Edit Data button. This opens the data into a Properties Window where the data can be modified.

---

### Related Blogs

---

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Fluid Library

AFT xStream provides a [fluid library](#) for numerous common fluids. If the working fluid you wish to model is not in this library, you may elect to enter the fluid properties in a custom library for later recall using the Edit Fluids panel in the [Library Manager](#).

You can open the Edit Fluids panel from the Library menu or from the Edit Fluid list button in the [Fluid panel](#) in [Analysis Setup](#). The Edit Fluids panel allows you to review currently defined AFT Standard and custom fluids, to change the data for custom fluids, and to enter new fluids. All AFT Standard fluid properties are assumed to be a function of temperature. Custom fluids can be a function of temperature or solids concentration.

Once you have entered a custom fluid, it will appear in the Fluids Available in Library list of the Fluid panel when AFT Standard is selected.

You can add new fluids in the Library Manager Edit Fluids panel by clicking the Add New Fluid button. A number of fluid properties are required for a compressible flow analysis. There are also several optional properties. The required properties are dynamic viscosity, specific heat at constant pressure, enthalpy, and molecular weight. Other entries for critical property data are used for some of the equation of state and enthalpy models. A constant compressibility factor can be assigned. The thermal conductivity is optional if pipe heat transfer is being modeled as adiabatic or isothermal.

Except for the constants, density, and enthalpy, all properties are assumed to be functions of temperature only. The assumption can break down in systems with very large temperature changes, and should be checked by the user. These properties are modeled with fourth order polynomials, and features for creating curve fits from raw data are provided.

The enthalpy is a strong function of temperature but a weaker function of pressure. Frequently the pressure dependence of enthalpy can be ignored. The three enthalpy models offered by AFT Arrow allow you to choose different ways of handling the pressure dependence.

The Fluid Library window also allows you to open the Change Fluid Data window which is essentially the same window as the Add New Fluid window. For easy editing, data for the fluid you wish to change is entered for you in the appropriate areas. The original curve fit data you entered is also displayed.

The Add New Fluid/Change Fluid Data windows offer cross-plotting of your custom curves against the curve fit data points. You can also import data from text files into these windows.

---

## Related Topics

Curve Fitting.....	66
--------------------	----

---

## Related Blogs

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Pipe Material Library

---

### Adding Custom Pipe Materials to the Pipe Materials Library

You can open the Edit Pipe Materials panel from the [Library Menu](#). Using this panel, you can easily add to the user library the geometric information for any pipe material in the length units of your choice. Custom pipe materials can be selected for use in the [Pipe Properties Window](#), minimizing the data entry required by the user.

Pipe materials typically have a range of nominal sizes and a specific type, class, or schedule that describes the actual dimension. AFT xStream works with the inner diameter of the pipe, since that is the parameter that directly affects the fluid mechanics.

**To add custom pipe materials to the user library**, choose the entry location you desire in the list on the left (Pipe Materials, Sizes and Types), then click the New button below. For example, to add an entirely new material, click the Pipe Materials item at the top of the list then click the New Material button. To add a new size for an existing material (say Steel), click *Steel* in the list then click the New Size button. To add a new type to the Steel 1 inch size, click *1 inch* under the *Steel* material in the list then click the New Type button.

When adding a new material, you also will be asked to specify a geometry, enter a nominal size, type of material, diameter, wall thickness, friction data set, and friction value to define the first data. Optionally you can add in a wall thickness. From then on you can enter additional nominal sizes and types. The name descriptors you choose for the material, size, and type are at your discretion. They can contain text mixed with relevant numbers if you desire.

You can also add multiple Friction Data Sets for each pipe material. This allows you to keep, for example, different friction values that may apply to the same pipe under different conditions. These conditions could be related to the age of the pipe or the type of fluid being carried. See the Friction Data Sets section below for more information.

In addition to manual entry of the custom pipe materials, you also can import pipe material data from a file. When you click the Import From File button, a dialog box will appear explaining the necessary import file format. By choosing to import from a file, you will be able to have a separate copy of the data for later access.

---

## Friction Data Sets

---

Friction Data Sets are pipe friction values for a particular pipe condition.

To create a new Friction Data Set select a Pipe Material, Size, or Schedule to associate the data set with, then click New Data Set from the Friction Data Set tab on the right (Figure 1). Modify the Friction Data Set description to provide a name, then enter the Friction Model and Value.

A Friction Data Set can be associated with the entire pipe material, or with specific sizes or types. For instance, a friction data set associated with Steel (i.e., the top level) might be called "25 year usage" and have a roughness value of 0.005 inches. This data set is then available to all sizes and types of steel. However, if "25 year usage" is associated only with the 1 inch of Steel, it will apply only to the pipe schedules that are 1 inch.

You can also associate friction data sets with a specific type (i.e., schedule). For instance, you can make a "25 year usage" and associate it only with 1 inch schedule 40 Steel pipe and assign its roughness value as 0.006 inches. Then all Steel pipe would use 0.005 inches for "25 year usage" friction while the different value for 1 inch schedule 40 of 0.006 inches would supersede the value of 0.005 inches associated with all Steel pipe.

Finally, among multiple friction data sets you can specify a default data set.

Multiple friction data sets can be created for each type of pipe material. For example, there may be separate friction data sets for Steel pipe such as: New Pipe, 10 Year Use, 20 Year Use and 30 Year Use. Typically, the older pipes will have higher friction.



*Figure 1: Friction Data Set tab on the Edit Pipe Materials panel*

## Pipe Physical Properties

---

The Pipe Material Library window allows you to assign physical properties of the pipe. The density and Poisson ratio of the pipe can be entered as constants, and the modulus of elasticity as a function of temperature.

## Related Topics

---

Curve Fitting .....	66
Pipe Material Library References.....	456

## Related Blogs

---

- [Databases 101 - Get to Know Them](#)
- [Databases 201 - Name That Fluids](#)
- [Databases 301 - Those Are Some Nice Pipes!](#)
- [Databases 401 - Managing Databases](#)

## Insulation Library

The Edit Insulations panel is accessed from the [Library menu](#). This panel allows you to view the insulation data provided with AFT xStream, or to add, edit or delete user defined insulation materials. Insulation library items are used for [heat transfer calculations in pipes](#).

To add a new insulation material click Add New Insulation, then enter the data in the Insulation Physical Properties window. The insulation thermal conductivity is entered as a function of temperature, and density as a constant.

Insulation data can be applied directly to pipes, and can also be associated with the [Pipe Material Library](#) as insulation data sets.

## Related Blogs

---

- [Databases 101 - Get to Know Them](#)
- [Databases 201 - Name That Fluids](#)
- [Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

## Fittings & Losses Library

The Edit Fittings & Losses panel can be accessed from the [Library Menu](#) and allows the user to review the [Fittings & Losses](#) data provided by default in xStream, and to add, edit or delete user defined Fitting & Losses in the AFT Default Internal Library.

Fittings & Losses can either be added by using the entry form in the window, or by importing a text file as described below. To enter a new Fitting/Loss item in the Edit Fittings & Losses panel click Create New Loss Item. Define the applicable category, type, geometry and loss model information then click Accept. The user-defined item will appear in the Fittings & Losses on the left under the relevant category/type with the source shown in parentheses as "U".

Custom fittings & losses are saved to the [local user library](#). From there the user can move them to other libraries (see [Library Manager](#)).

### Import From File

Fittings & Losses can be imported from a \*.CSV type file, which can be created from any spreadsheet-type software, such as Excel.

The entries in the file should be in the order as shown in Figure 1, but with no headers in the workbook. If a header row or a blank row is used, xStream will present a warning, but give the user the option to still import the valid rows.

	A	B	C	D	E	F	G	H
1	Category	Type	Sub Type (Optional)	Geometry (Optional)	Geometry Value (Optional)	Loss Model	Loss Value	Reference/Notes (Optional)
2	Elbow/Bend	Mitre	General	Angle (degrees)	45	K (constant)	5	

**Figure 1: Example of import file setup for a custom elbow/bend. Headers should not be present in the file to import.**

Valid options for each of the columns are listed below, with detailed options given in Table 1.

- **Category** - Must match one of the values from Table 1. "Other" may be used if none of the specific fitting/loss categories fit.
- **Type** - This field is required, any name/value may be entered. Types already existing in the library are shown in Table 1, and may be used if desired.
- **Sub Type** - This field is optional. A sub-type may be chosen from Table 1, or any other name/value may be entered.
- **Geometry** - This field is optional. A geometry type from Table 1 must be used if a geometry is entered. If the Geometry field is left blank, the Geometry Value will be ignored.
- **Geometry Value** - This field is only required if a Geometry type has been entered. Any positive numerical value may be entered.
- **Loss Model** - This field is required. The Loss Model must be either "K (constant)" or "K / fT (Crane)".
- **Loss Value** - This field is required. Any positive numerical value may be entered for the loss value.
- **Reference/Notes** - This field is optional. Any information may be entered here as needed.

**Table 1: Valid Entries for each field. The Geometry Value and Loss Value fields will accept any numerical entry. The Reference/Notes field will accept any entry.**

Category	Type	Sub Type	Geometry	Loss Model
Elbow/Bend	Standard Threaded	General	Angle (degrees) Pipe Size (nominal) None	K (constant) k / FT (Crane)
	Mitre	General		
	Smooth Flanged	r/D=1		
		r/D=1.5		
		r/D=2		
		r/D=3		
		r/D=4		
		r/D=6		
		r/D=8		
		r/D=10		
3-K (Darby)	90 deg			
	45 deg 180 deg			
Valve	Angle	Type 1	Angle (degrees) Open Percentage L/D Pipe Size (nominal) None	K (constant) k / FT (Crane)
		Type 2		
		Direct flow		
	General			
	Ball	None		
Butterfly	Plane disk - AR=0.92			
	Plane disk			

Libraries

		t/D=0.1 t/D=0.15 t/D=0.2 t/D=0.25 t/D=0.3 t/D=0.35 Type A Type B Type C		
	Cylindrical	General		
	Gate	Rect. Tube		
		Wedge/Plug		
		Wedge		
		Disk w/o recess		
		Disk with recess		
	Globe	Disk		
		Wedge - Full Open 10 deg.		
		Wedge - Full Open 20 deg.		
		Sluice		
	Plug	Disk with recess		
		90 degree wall		
		45 degree wall		
		Standard 45 deg. wall		
	Poppet	Standard 90 deg. wall		
		Standard		
		Y Type		
		Straight-way		

Libraries

	Three-way	W/D=.035 W/D=.07 No Seat Width		
	3-K (Darby)	Through flow Branch flow		
		Angle Globe Plug Gate Ball Diaphragm		
Check Valve	Swing	90 deg. seat 45 deg. seat	Angle (degrees)  None	K (constant)  k / FT (Crane)
	Lift	90 deg. wall 45 deg. wall		
	Tilting disc	General		
	Stop-check	Globe 90 deg. Globe 45 deg. Globe thru flow		
	3-K (Darby)	Swing Check Lift Check		
Orifice	Sharp	None	Area Ratio	K (constant)
	Long	None	None	k / FT (Crane)
Entrance/Exit	Entrance	Sharp Reentrant Rounded Flush	R/D None	
	Exit	None		
Other	Diff Flowmtr	Orifice Plate	Area Ratio	K (constant)
		Nozzle	None	k / FT (Crane)

## Libraries

		Venturi 7 deg. Venturi 3 deg. Dall Tube		
	Honeycomb cell	L/D=2 L/D=4 L/D=6 L/D=8 L/D=10 L/D=12		
	Screen	Perforated Plate Circular wire Silk thread Round wire or netting		
	Tee	Through flow (dead branch) Branch flow (dead run)		
Flange	Steel	150 lb. 300 lb.	None	N/A

### Related Topics

Fittings & Losses.....	169
Specifying Losses.....	198

### Related Blogs

- [Databases 101 - Get to Know Them](#)
- [Databases 201 - Name That Fluids](#)
- [Databases 301 - Those Are Some Nice Pipes!](#)
- [Databases 401 - Managing Databases](#)

## Network Library Overview

Custom libraries contain custom data generated by the user, including fluid properties, pipe materials, and data for junctions types such as pumps and valves.

All custom data created by the user is kept on the local computer in a file called XSM\_USER2.DAT in the local user directory. This data is not shared and is always available to the local user for access or for editing.

In addition to the local user library, you can create multiple network or local custom libraries. The [Library Manager](#) window on the Library menu simplifies this process. See [Creating an Enterprise-wide Network Library System](#) for details on how to do this.

## Benefits of Shared Libraries

When the xStream libraries are placed on a network, powerful data management procedures can be implemented. Specific, quality-checked data can be entered into libraries located on a local or wide area network of PC's. This data is not accessible to the end users through AFT xStream itself, so it is guarded data.

Once the data is placed on a network, users can connect to any library listed in the [Library Manager](#). Users can connect to as many libraries as they want.

If the user connects to a library with these files and selects to be connected to any of these control files, the model will inherit the control formatting specified in the control files. This means that special report layouts, special graphical representations and colors, and other special controls can be decided upon by the library constructor, and those control features can be loaded into local models by end users. This is highly useful for creating customized reports for certain customers or project requirements.

By way of example, assume the [Output Control](#) content was specified as coming from the "Woodland Park Office Library" library.

If you uncheck the Library checkbox or change certain data in the Output Control window, the library will remain connected but change to an inactive state. When the library is inactive, none of its data flows through to the model. To bring the Output Control data back into sync with the connected library, recheck the Library check box in Output Control.

This entire custom library concept lends itself naturally to the idea of specific project libraries. Each project has differing requirements that depend on the nature of the project and the end customer. Data for each project can be assembled into separate libraries, allowing faster and more accurate transmission of the controlled data and desired reporting formats to engineers.

Maintaining a network library allows commonly-used data to be controlled, which may be helpful in instances where quality control is strict.

## Creating an enterprise-wide, network library system

### Overview

---

Users of AFT xStream on a network can access common libraries to simplify model verification, improve analysis and reporting consistency, and eliminate redundant work.

To accomplish this there is some initial setup that must be done. First, the data for components, materials, formats, etc., needs to be created and saved to various data files. Secondly, these files must be made available to the users of AFT xStream by creating a file named DATABASE.LIB in the xStream folder. Thirdly, the users need to connect to the new libraries listed and select the sections to use. Each of these broad steps is discussed below. To maintain control and integrity of the data, it would be advised that one person is given the responsibility to create and maintain the network libraries. The discussion below will be given from the viewpoint of this person.

Table 1 shows the relationship between Internal Libraries and External Libraries as well as the library file listing that AFT xStream uses to find these libraries. An External Library is any library file that is created by the user and listed in the XTR\_DBUSER.LIB residing in the user's local user directory. The user can make these available and unavailable, and connect and disconnect from these libraries using [Library Manager](#). The library files themselves may be located on any local or mapped drive. These library files are also distinguished by being under the local user's complete control. External Shared Libraries, on the other hand, are common to all AFT xStream users on the network, therefore the local user cannot remove them from the list of available libraries. However, the user can choose to connect or disconnect from these libraries using Library Manager. Finally, to complete the discussion, there are two special libraries, the Default Internal Library and the [Local User Library](#). These files are AFT xStream Internal Libraries which cannot be disconnected or made unavailable.



**Table 1: A summary of the types of libraries used for the example below. Note the difference between External and External Shared libraries is just which \*.LIB file their file path is listed in.**

Library Groups & Filenames	Library Titles	Library Locations
<b>xStream Internal Libraries</b>		
XSTREAM2.DAT	AFT Default Internal Library	AFT xStream 2 folder
XTR_USER2.DAT	AFT xStream Local User Library	AppData folder
<b>External Shared Libraries (DATABASE.LIB)</b>		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in DATABASE.LIB
<b>External Libraries (XTR_DBUSER.LIB)</b>		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in XTR_DBUSER.LIB

## Creating Library Files

The first thing to do is create the files that contain the information to be shared. These files will be referred to as library files and can contain information about pipe materials, components (e.g. pumps, valves, etc.), fluids, etc. The following are several ways to create library files:

- Use the Create New Library button in the Library Manager, as described previously. Data can be taken from the current model or other libraries.
- Copy and paste sections from one file into another file using a text editor.

This discussion will focus on the first and second methods. Let's assume that the person responsible for the network library has already created a new pipe material. Also, assume that the goal is to create a network-wide library to contain this material.

The information for the material is copied from the [Local User Library](#) (XTR\_USER2.DAT) to a new file using the [Library Browser](#) in the Library Manager. For the source library choose Local User Library and then select Pipe Materials. Then click Create New Library and enter a file name, VailMaterials.dat, for this example. Enter a description of the library - this description is how AFT xStream users will refer to the library and should concisely describe what is in the library (for this example we'll use "Vail Resort Materials"). Right-click the pipe materials section from the Local User Library and choose Copy Content to

copy the pipe materials to this new library file. The pipe material library file has now been created and is ready to be used.

### Sharing Library Files Using DATABASE.LIB

---

Now that the library file for the Pipe Materials has been created, it needs to be made available to the users of AFT xStream. Depending on the scope of use, library files can be placed into one of two groups. If the library is to be used locally by one user, it is put in the External Library group. On the other hand, if the library is to be common among many users and shared over a network it is put into the External Shared Library group.

Continuing the above example, the goal is to share the library file, so it will need to be an External Shared Library listed in the DATABASE.LIB file in the xStream directory on the network. However, since xStream does not write directly to the network files for security reasons, it is easiest to temporarily treat these files as local External Library files which will be listed in the XTR\_DBUSER.LIB file located in the local user directory of the local machine.

It is recommended that External Shared Library files be copied to the network server and placed in a subfolder of the AFT xStream folder (e.g. AFT Products\AFT xStream\Libraries). The files may, however, be located anywhere on the network that is accessible to the users. For our example, we are going to put the pipe material library on the network in a subfolder called Libraries. It is easier to copy the file to the Libraries folder before they are made available in AFT xStream. This will help establish the correct paths to the files as the files are made available using the Library Manager.

Once the library file is in the network location, open the [Library Manager](#) in AFT xStream to the Library Browser. Each of the new library files we have created needs to be added to the Available Libraries list by clicking Add Existing Library and browsing to the file. Once you click OK, the description of the library you just added appears in the Available Libraries list. Once you have added the new library, click Close and the External Library listing file (XTR\_DBUSER.LIB) in your local user directory is automatically updated.

Now we need to discuss how the paths should be setup. When you add a library, AFT xStream will write the full path of the library to the External Library listing (XTR\_DBUSER.LIB). That means that if you browsed to the library file using a mapped drive letter then the path will include that drive letter. However, if other users try to access the libraries but have the AFT xStream folder mapped to a different drive letter they will not be able to locate the library files. Other options include using either relative paths or the computer name in the path (e.g. \\Network Server\Eng Apps\AFT Products\AFT xStream\Libraries\filename.dat). You can automatically have AFT xStream use the computer name when you add the file by choosing Network Neighborhood, then the computer and then browse to the library file. If you would rather use relative paths, you can manually edit them in the library listing file. All library files must be relative to the AFT xStream folder where the executable is located.

When the pipe materials library was added it became an External Library listed in XTR\_DBUSER.LIB. Now we need to move this listing from the local machine to the AFT xStream folder on the network so that all users can access the library files. Once the listing file is moved to the network, it must be renamed DATABASE.LIB. Once this is done, all users can now access the library file we have created.

### Connecting to the External Shared Libraries

---

Now that the libraries have been placed on the network and made available to all AFT xStream users, each user must now connect to the libraries and make the sections active as needed. When a user opens

the Library Manager, the External Shared Libraries are listed in the Library Browser. Make sure the checkbox is checked to make those libraries connected for use. Also ensure that the sections within the library are checked for use. A library can be connected without having any sections selected and active. Finally, to have these connections in place every time you start AFT xStream, click Set As Default. To apply these settings to a model made previously, open the model, go to Library Manager, click User Default and, if you have set the default, the connections and setting will now be applied to the model.

### Sharing Model Formats/Preferences

---

Note that the Library Manager used to provide the option to include formats/preference settings such as the Output Control or Visual Report Control layouts into shared libraries. These type of files can still be shared, but are no longer shared through the Library Manager. Instead these type of format files must be loaded directly through their control windows, such as loading the Output Control format being loaded in the Output Control window.

## Library Verification

The Library Verification window is displayed when you open a model or new scenario which contains connected library items that have some inconsistency.

### 1. Library Item Not Found

---

You may have a library item that is not found in any currently connected library. For example, a compressor may be connected to a library, and derive its data from that library. However, when the model is loaded, the source library is not available. This could be for several reasons that will be discussed shortly.

The compressor in the model does in fact have all of the required data needed to run the model. But when it is not found in the library, you will be offered several options:

- **Disconnect it from the source library** – this means that the model will forget the library the compressor came from and make it a standalone compressor not connected to any library.
- **Add it to a library** – this means that the compressor can be added to a custom library
- **Leave the library connection** – this means that the compressor will remember what library to which it was originally connected and connect to that library again when it is made available to AFT xStream. If, however, you try to open the Compressor Properties window, AFT xStream will disconnect the compressor from the library and warn you.

The compressor might have lost its library connection for one of these reasons:

- The library is no longer available because it was removed either through the Library Manager or because your libraries have been changed on the network.
- The model is being run on a different PC than where it was created, and the original libraries are no longer available. This could be, for example, because you have moved the model onto a laptop

computer to take it on travel. Or it could happen because the model was developed by another company which had developed libraries of its own but which are not available to you.

Whatever choice you make, the model will retain all its engineering data and yield the same results as it did originally.

## 2. Item in Model is Different from Library

---

You may have a library item in the model that is connected to a external library, but for some reason the data is not the same. For example, you may have a compressor curve for the Rocky Mountain Compressor Company's Model 100A in your model, but it does not match the compressor curve for this model in your library. Here you have three choices:

- **Update the items in the model to match the library** – this will actually change the model's input data, and it might yield different results than originally
- **Update the library (if it is local) to match the model's items** – this will change the library, but will leave the model's input data as it was. It will yield the same results as it did previously.
- **Disconnect it from the source library** – this will leave the model as it was, and it will yield the same results.

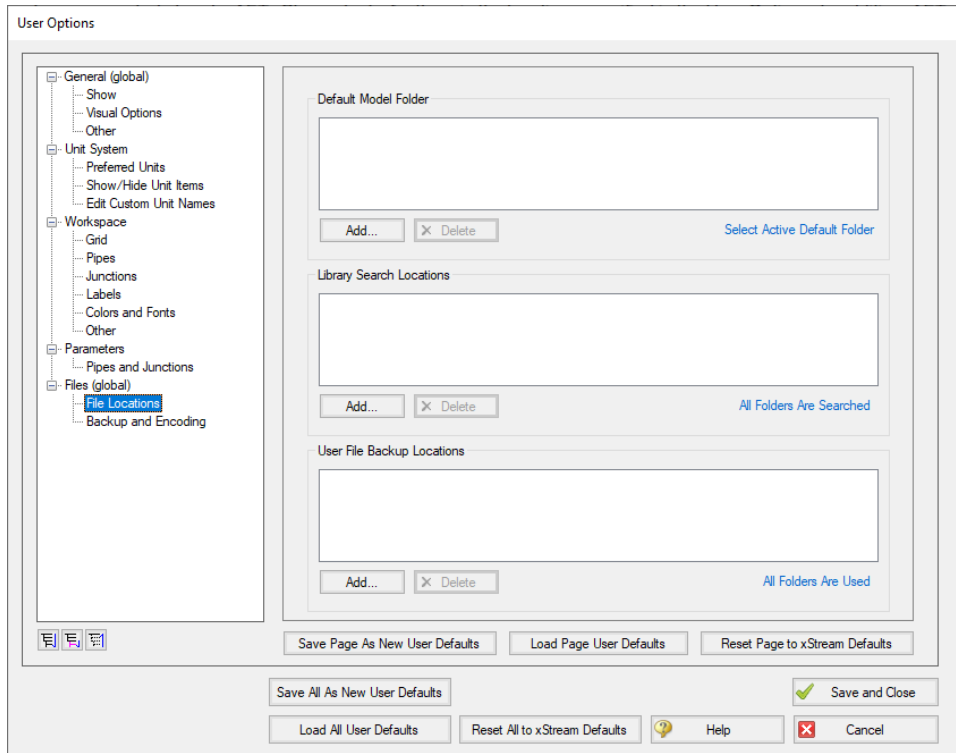
The compressor in the model might have a different compressor curve than the one in the library for one of these reasons:

- When the model was built, the compressor was connected to the Rocky Mountain Compressor Company (i.e., RMCC) Model 100A. But since that time, the RMCC library has changed. Perhaps there was an input error in the original data, or better data has since become available and the library was updated. If this were the case, it would be best to update all your compressors to the library values.
- The library may have been changed by an unauthorized individual, and while it was accurate originally, now the data is no longer valid. In this case, it would be best to **not update** the compressor in your model, and to inquire into the cause of the inaccurate data within your organization.

## Library Locations in General Preferences

When performing an engineering analysis with or even without cost estimating, there may be many engineering libraries that underlie the model. These libraries need to be connected in the [Library Browser](#) in order to be accessible by the model. However, to know what libraries are actually needed, one needs to first load the model. This creates a dilemma.

To help alleviate the dilemma, users can create "Library Search Locations" in the User Options window. Figure 1 shows an example. Once specified, you can set these as your default locations by clicking the Set As Default button. When AFT xStream tries to open a model that requires certain libraries, and those libraries have not been connected already, AFT xStream looks for them in the locations specified in the User Options. In addition, AFT xStream looks in the model's folder. If AFT xStream finds the libraries in one of these locations, it automatically adds it to the list of connected libraries in the Library Manager. If it cannot find a library, it informs you once the model is loaded.



**Figure 1: Library Search Locations can be defined in the User Options window**

## Theory and References

This section focuses on the fundamental theories that are used in AFT xStream's solution method. Additionally, this chapter gives information on the source material for the theories used. Topics include discussions on the theories that comprise the network solution, compressible flow theory, sonic choking, transient flow theory, and information on references used in the development of AFT xStream.

### Content

<a href="#">Nomenclature</a> .....	410
<a href="#">Review of Compressible Flow Theory</a> .....	412

### Nomenclature

$a$	sonic speed
$A$	cross-sectional flow area of a pipe
$B$	<a href="#">compatibility equations</a> parameter for method of characteristics
$C$	<a href="#">compatibility equations</a> parameter for method of characteristics
$C_d$	discharge coefficient
$c_p$	specific heat, constant pressure
$c_v$	specific heat, constant volume
$C_V$	valve coefficient
$D$	diameter of a pipe
$e$	specific energy
$F$	Force
$f$	friction factor
$f_T$	turbulent friction factor (used by Crane, 1988)
$f_L$	laminar friction factor
$f_{Tr}$	transition friction factor
$F$	error value in mass balance
$g$	gravitational constant
$h$	internal convection coefficient
$h$	enthalpy, static

$h_O$	enthalpy, stagnation
$H$	head
$J$	Jacobian matrix
$k$	thermal conductivity
$K$	loss factor
$L$	length of a pipe
$\dot{m}$	mass flow rate
$M$	Mach number
$n$	general constant
$N$	compressor synchronous speed (rpm)
$P$	pressure, static
$P_O$	pressure, stagnation
$P_w$	wetted perimeter
$P$	power
$q$	heat rate to a pipe
$Q$	volumetric flow rate
$r$	radius
$r$	relaxation
$R, R'$	resistance or gas constant
$R_c$	coefficient of rigidity
$Re$	Reynolds Number
$s$	fan speed
$s$	entropy
$t$	time
$T$	temperature, static
$T_O$	temperature, stagnation
$u$	internal energy
$v$	specific volume
$V$	volume
$V$	velocity

$w$	work
$x$	distance along pipe centerline
$z$	elevation
$Z$	compressibility factor
$\alpha, \mu, \theta$	angle
$\beta$	diameter ratio
$\beta$	coefficient of volume expansion
$\lambda$	method of characteristics multiplier
$\eta$	efficiency
$\nu$	kinematic viscosity
$\varepsilon$	roughness
$\gamma$	specific heat ratio
$\rho$	density
$\tau$	shear stress
$\omega$	rotational velocity
$\mu$	dynamic viscosity

---

## Subscripts

---

$i$	junction or pipe station at which solution is sought
$j$	junctions with pipes connecting to junction $i$
$R$	rated conditions for a compressor, usually BEP
$o$	stagnation
$\infty$	infinity, far away, ambient
$new$	current time for computation
$old$	most recent time for computation

## Review of Compressible Flow Theory

This topic gives an overview of internal compressible flow theory. For a more complete development of compressible flow theory, please consult one of the many textbooks available ([Anderson, 1982](#), [Saad, 1993](#), and [Shapiro, 1953](#)).



## What is Compressible Flow?

---

Compressible flow is a specialized branch of fluid mechanics that describes the flow of gases. This field of study is sometimes called gas dynamics, which is a synonymous term.

Compressible flow theory draws heavily from the fields of fluid mechanics and thermodynamics. This important branch of fluid mechanics has been developed almost completely since the year 1900. Its development has been largely driven by the extraordinary progress in the applied fields of aerodynamics and aircraft technology.

The field of compressible flow breaks down naturally into the two related areas of internal and external flow. Among the important applications of external compressible flow are the design of aircraft and airfoil design. Among the important applications of internal compressible flow are pipe flow, nozzles and gas turbines. In AFT xStream, the interest is entirely in internal compressible flow.

The basic feature that distinguishes compressible from incompressible flow is the changing density of the gas. The density of the gas is related to the temperature and pressure of the gas through an equation of state. The variable density makes traditional incompressible analysis techniques of limited usefulness, and in general makes pipe flow analysis much more complicated.

The variable gas density has several important ramifications. First, the velocity of the gas is related to density through the continuity equation, and thus velocity also becomes a variable even in constant area pipes. Second, the density dependence on temperature means that the energy equation is coupled to the equation of state and must be solved simultaneously with the other governing equations. And last, the velocity of the gas is constrained by the sonic gas speed. Thus unusual effects like sonic choking can occur which further complicates analysis.

It should be noted that not all gas flows are compressible. In some applications the gas velocities are low enough or the pressure drops are small enough that compressible effects are insignificant. In such situations, systems can be modeled with traditional incompressible techniques. AFT xStream is capable of modeling gas flows that are either compressible or incompressible.

## Equation of State

---

The equation of state relates the density, pressure and temperature of a gas. In principle, knowledge of any two of these parameters makes it possible to determine the third.

The ideal gas equation of state is given by the following

$$P = \left( \frac{R}{\widehat{M}} \right) \rho T$$

where  $R$  is the universal gas constant and  $\widehat{M}$  is the molecular weight of the gas. These two terms are frequently combined into a constant  $R_g$

(63)

$$P = R_g \rho T$$

where  $R_g$  is the gas constant for the particular gas. The above equation is known as the ideal gas equation, because it offers an idealized model to relate the pressure, density and temperature of a gas. For simplicity, the “g” subscript will be dropped and it is to be understood that  $R$  is not the universal constant, but that for a particular gas.

Experiments have shown that [Equation 63](#) can be applied to a more general selection of gases by introducing the compressibility factor,  $Z$ , which compensates for departure from ideal gas behavior.

$$P = Z R \rho T \tag{64}$$

There are many methods of determining  $Z$ . These methods are found in textbooks and handbooks, each having strengths and weaknesses.

### First Law and Isentropic Relationships

---

From the First Law of Thermodynamics we know that

$$\delta q + \delta w = \delta e \tag{65}$$

For a reversible process, the work term can be represented by the following

$$\delta w = -P dv \tag{66}$$

where  $v$  is the gas specific volume (inverse of density). In addition, the incremental addition of heat is given by

$$\delta q = T ds \tag{67}$$

where  $s$  is the entropy. Substituting Equations 66 and 67 into 65 and rearranging yields

$$T ds = de + P dv \tag{68}$$

The enthalpy is related to the internal energy by the following identity

$$h = e + P v \quad (69)$$

Differentiating yields

$$dh = de + v dP + P dv$$

so that Equation 68 becomes

$$T ds = dh - v dP \quad (70)$$

By using a suitably average specific heat constant, the enthalpy can be related to temperature

$$dh = c_p dT \quad (71)$$

and can be substituted into Equation 70 to obtain

$$ds = c_p \frac{dT}{T} - \frac{v dP}{T}$$

The specific volume is the inverse of density, thus

$$ds = c_p \frac{dT}{T} - \frac{dP}{\rho T} \quad (72)$$

To simplify the following concepts, a perfect gas will be assumed and extensions to real gases will be made without proof. Later applications will describe the effects of real gas behavior in more detail.

For a perfect gas, the third term in Equation 72 can be modified to obtain

$$ds = c_p \frac{dT}{T} - R \frac{dP}{P} \quad (73)$$

Integrating Equation 73 results in

$$s_2 - s_1 = c_p \ln \frac{T_2}{T_1} - R \ln \frac{P_2}{P_1} \quad (74)$$

In many applications the local gas flow is isentropic, which causes the left side of Equation 74 to go to zero yielding

$$c_p \ln \frac{T_2}{T_1} - R \ln \frac{P_2}{P_1} = 0 \quad (75)$$

For an ideal gas it can be shown that

$$R = c_p - c_v \quad (76)$$

or rearranging

$$\frac{c_p}{R} = \frac{\gamma}{\gamma - 1} \quad (77)$$

where  $\gamma$  is the ratio of specific heats.

For a real gas, Equations 76 and 77 become

$$Z R = c_p - c_v \quad (78)$$

$$\frac{c_p}{Z R} = \frac{\gamma}{\gamma - 1} \quad (79)$$

The ratio of specific heats can also be calculated by

$$\gamma = \frac{c_p}{c_v} = \frac{c_p}{c_p - Z R} \quad (80)$$

The parameter  $\gamma$  is very important in determining the thermodynamics in isentropic processes. When considering a real gas, as the pressure increases Equation 80 takes on a modified form (Bejan, 1988, pp. 178)

$$\gamma = -\frac{v}{P} \left( \frac{\partial P}{\partial v} \right)_T \frac{c_p}{c_v} \quad (81)$$

By algebra Equation 75 becomes

$$\frac{P_2}{P_1} = \left( \frac{T_2}{T_1} \right)^{\gamma/(\gamma-1)} \quad (82)$$

By similar methods it can be shown that

$$\frac{\rho_2}{\rho_1} = \left( \frac{T_2}{T_1} \right)^{1/(\gamma-1)} \quad (83)$$

To summarize, Equations 82 and 83 are isentropic relationships that are valid for real gases if the appropriate value for  $\gamma$  is used.

## Sonic Speed and Mach Number

---

The sonic speed of a gas is the velocity at which disturbances propagate within a gas. Unless a system is specially designed to produce supersonic flow, the sonic flow speed is the maximum speed at which the gas can travel.

It can be shown that a sound wave is an isentropic phenomenon involving virtually no energy loss, and that the speed of sound is given by the following equation (based on mass and momentum conservation)

$$a^2 = \left( \frac{\partial P}{\partial \rho} \right)_s \quad (84)$$

where  $a$  is the speed of sound and the subscript "s" refers to isentropic conditions. Another more useful form of Equation 84 is obtained by combining Equations 82 and 83 to relate pressure and density

$$\frac{\rho_2}{\rho_1} = \left( \frac{P_2}{P_1} \right)^{1/\gamma} \quad (85)$$

Using Equation 85 in Equation 84 and carrying through the differentiation

$$a^2 = \frac{\gamma P}{\rho}$$

or using the ideal gas equation (Equation 63)

$$a = \sqrt{\gamma R T} \quad (86)$$

Equation 86 is the classic equation for speed of sound, and since the specific heat ratio varies only slightly, the speed of sound effectively depends only on temperature.

For real gases, Equation 86 requires using the real gas equation (Equation 64) to obtain

$$a = \sqrt{Z \gamma R T} \quad (87)$$

As an example of sonic velocity, consider air at 70° F (530° R). The specific heat ratio for air is about 1.4, and assuming ideal gas ( $Z = 1$ ) then

$$a_{air} = \sqrt{1.4 \left( 1716.3 \frac{ft^2}{s^2 \cdot R} \right) 530^\circ R} = \frac{1128 \text{ ft}}{s} = \frac{344 \text{ m}}{s}$$

The relationship between the gas velocity and sonic speed is known as the Mach number.

$$M = \frac{V}{a} = \frac{V}{\sqrt{Z \gamma R T}} \quad (88)$$

The Mach number is a dimensionless quantity that is the most important parameter in compressible fluid flow. Relating all parameters to the Mach number greatly simplifies compressible flow calculations.

In general, the sonic gas velocity is the maximum that can be attained in a system, thus the largest Mach number in a pipe system is 1. This is known as sonic flow. Flows less than Mach 1 are referred to as

subsonic flows. It is in fact possible to attain Mach numbers greater than 1 by using a specially designed convergent-divergent nozzle. Flows greater than Mach 1 are referred to as supersonic flows. There are special applications for supersonic flows in internal flow systems, however these applications are not generally of interest to the piping engineer and thus no attempt is made to address supersonic flows in AFT xStream. AFT xStream strictly assumes that Mach 1 is the maximum in the system.

### Stagnation Properties in Compressible Flow

---

In application of Equation 65 to compressible pipe flow, use of the fluid enthalpy simplifies energy balance calculations. The enthalpy is a convenient parameter that combines the internal energy changes with the work performed by volume changes in the flow. The energy equation is:

$$q = \dot{m} d\left(h + \frac{V^2}{2} + gz\right) \quad (89)$$

A new parameter called the stagnation enthalpy is introduced which is defined as:

$$h_o = h + \frac{V^2}{2} \quad (90)$$

Introducing Equation 71 into Equation 90 results in

$$T_o = T + \frac{V^2}{2c_p} \quad (91)$$

where  $T_o$  is the stagnation temperature. The stagnation properties are sometimes referred to as “total” properties, which is a synonymous term. In contrast, the temperature,  $T$ , is the thermodynamic temperature that we are accustomed to, and takes on the more descriptive name of static temperature.

Using Equation 79 to replace  $c_p$  results in

$$\frac{T_o}{T} = 1 + \frac{\gamma - 1}{Z\gamma} \frac{V^2}{2RT} \quad (92)$$

and using the Mach number, Equation 88, this becomes

$$\frac{T_o}{T} = 1 + \frac{\gamma - 1}{2} M^2 \quad (93)$$

Equation 93 relates the stagnation and static temperatures through the single flow parameter Mach number and  $\gamma$ , which is frequently constant.

Following from Equation 82 and 83

$$\frac{P_o}{P} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma/(\gamma-1)} \quad (94)$$

$$\frac{\rho_o}{\rho} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma/(\gamma-1)} \quad (95)$$

Between pressure, temperature, enthalpy and density, it is easiest to grasp the physical difference between static and stagnation properties when considering pressure. The classic example is a pitot tube that is normal to the flow or pointed directly into the flow. Figure 1 depicts the distinction.

The ratio of stagnation to static conditions for pressure and temperature are shown in Figure 2.



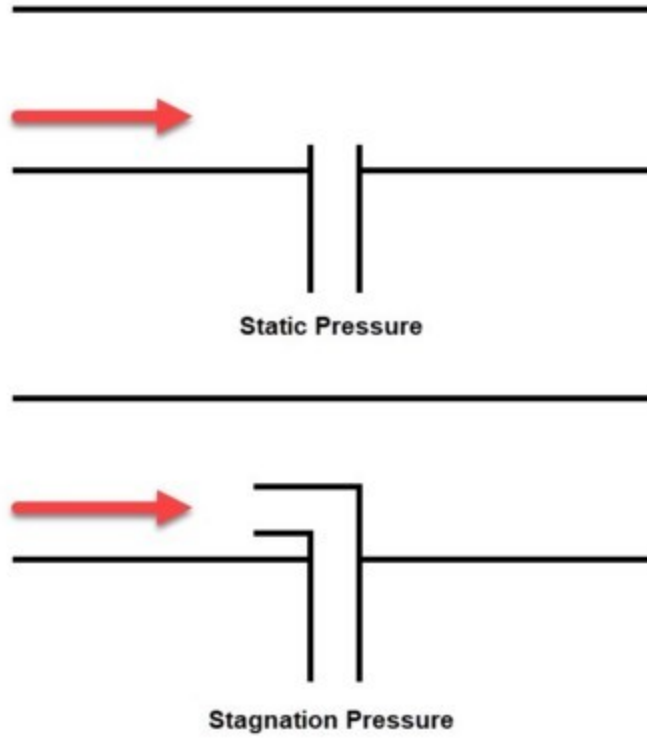


Figure 1: Comparison of static and stagnation pressure measurement at a pitot tube.

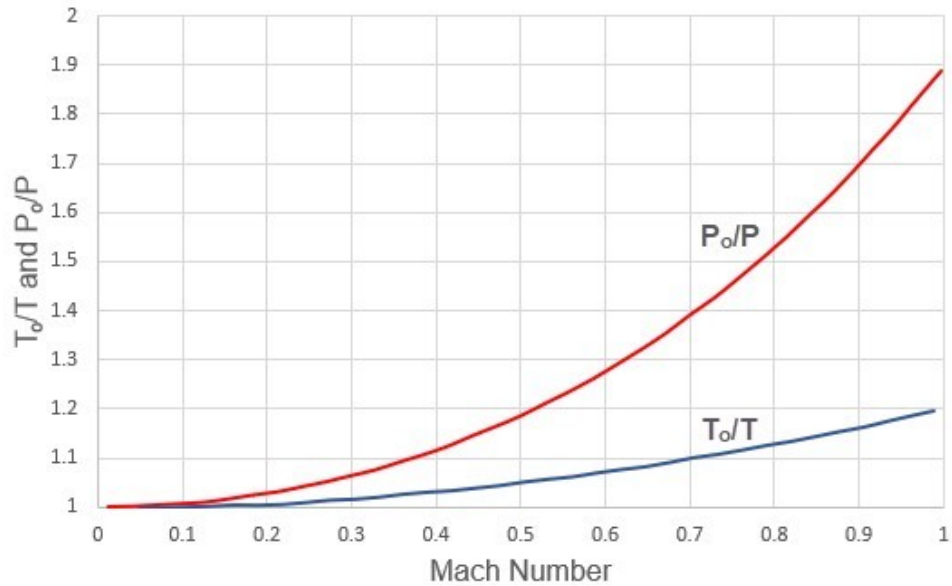


Figure 2: Variation of stagnation to static temperature ratio and stagnation to static pressure ratio with Mach number. Gamma = 1.4.

The stagnation temperature, stagnation enthalpy and stagnation pressure (and to a lesser extent, stagnation density) are of great importance in compressible flow systems. The stagnation property is made up of the “static” portion (which is the thermodynamic portion) and the “dynamic” portion (due to the fluid motion). The stagnation property is that property that would exist if the flow was brought to rest. In a fluid at rest (such as a large gas tank) the stagnation and static properties are equal.

The stagnation properties take on significance because they combine the effect of fluid velocity. Thus, a frictionless pipe will have constant stagnation pressures even if the area (and hence velocity) changes. The stagnation enthalpy will remain constant as long as no heat transfer and no elevation change occurs (see Equation 89). If the stagnation enthalpy is constant, the stagnation temperature remains constant insofar as Equation 71 remains valid. Departures from Equation 71 mean that the stagnation temperature can change in an adiabatic pipe, but this change is usually small. For a perfect gas, Equation 71 is always valid and thus stagnation temperature is always constant in adiabatic flow.

The convenience of stagnation properties is that frictional pressure loss and heat transfer calculations can be performed by using the stagnation property, thus avoiding the need to separately calculate dynamic (velocity) effects.

Also of importance is the issue of pipe networks. When a branch exists in a compressible flow network, a change in flow area is almost certain. The change in flow area means a change in velocity, and hence a change in the static pressure and static temperature. However, the stagnation pressure and temperature must always match at branching boundaries. The result is that the stagnation pressure and temperature are the key variables to solve for in a network pipe system.

## The Continuity Equation

---

The continuity equation is the equation that applies the Law of Mass Conservation. For steady flow, the vector integral form of the continuity equation is given by

$$\int_{cs} \rho V dA = 0 \quad (96)$$

Carrying out the integration of Equation 96 and assuming one-dimensional flow in a pipe results in

$$\dot{m} = \rho A V = \text{constant} \quad (97)$$

If the pipe is of constant area, Equation 97 can be differentiated to obtain the following

$$\frac{d\rho}{\rho} + \frac{dV}{V} = 0 \quad (98)$$

Equation 98 states that any increase in gas density must be accompanied by a decrease in velocity.

---

## The Momentum Equation

---

The momentum equation is an expression of Newton's Second Law of Motion, which states that the change in momentum is proportional to the sum of the acting forces, or

$$\sum F = - \int_{cs} V(\rho V dA) = 0 \quad (99)$$

For gas flow in a pipe, the acting forces are the pressure forces and friction forces. Assuming one-dimensional flow in a constant area pipe, the differential form of Equation 99 is

$$dP + \frac{4}{D} \tau_w dx + \rho g dz = -\rho V dv \quad (100)$$

where  $\tau_w$  is the wall shear stress. The wall shear stress can be expressed in terms of a Moody friction factor,  $f$  (one-fourth of the sometimes used Fanning friction factor), and defined as

$$\tau_w = \frac{1}{2} \rho V^2 \frac{f}{4} \quad (101)$$

Substituting Equation 101 into Equation 100 results in

$$dP + \frac{1}{2} \rho V^2 \frac{f}{D} dx + \rho V dV + \rho g dz = 0 \quad (102)$$

Equation 102 relates the change in fluid momentum (the third term) to the pressure forces (first term), friction forces (second term) and body forces (fourth term). The four terms balance to zero.

---

## The Energy Equation

---

The First Law of Thermodynamics was introduced earlier in Equation 89,

$$q = \dot{m} d\left(h + \frac{V^2}{2} + gz\right) \quad (89)$$

or, using the stagnation enthalpy (Equation 90), the energy equation becomes

$$q = \dot{m} (dh_o + dz) \quad (103)$$

In the case of constant elevation,  $dz$  is zero and Equation 103 states that any heat transfer in a pipe is accompanied by a proportional change in stagnation enthalpy. If no heat transfer occurs (adiabatic flow), the stagnation enthalpy is constant.

For a perfect gas, the enthalpy is related to temperature by Equation 71. In general, enthalpy is a function of static temperature and pressure

$$h = f(T, P)$$

While it is tempting to relate stagnation conditions on a similar basis because of the simplifications that result, in general only static conditions should be used. Thus

$$h_o \neq f(T_o, P_o)$$

### Frictional Effects in Pipes

To clearly understand the effect of friction in a pipe, it is convenient to look at the case of isothermal flow. The momentum equation, Equation 102, will be used as the starting point. Neglecting gravity effects,

$$dP + \frac{1}{2}\rho V^2 \frac{f}{D} dx + \rho V dV = 0$$

From the continuity equation (98)

$$dV = -V \frac{d\rho}{\rho}$$

Substituting this into the momentum equation

$$dP + \frac{1}{2}\rho V^2 \frac{f}{D} dx - V^2 d\rho = 0$$

For ideal gas, isothermal flow,

$$d\rho = \frac{dP}{RT} \quad (104)$$

Therefore

$$dP + \frac{1}{2}\rho V^2 \frac{f}{D} dx - V^2 \frac{dP}{RT} = 0$$

or using the Mach number

$$dP + \frac{1}{2}\rho V^2 \frac{f}{D} dx - \gamma M^2 dP = 0$$

or

$$dP = -\frac{\rho V^2 \frac{f}{D} dx}{2(1 - \gamma M^2)} \quad (105)$$

For subsonic flow, the quantity on the right hand side will always be negative. Therefore, dP will always be negative when there is isothermal flow with friction, which is what one would expect. If dP is negative

$$dP < 0$$

then from the equation of state (104)

$$d\rho < 0$$

Therefore, from the continuity equation (98)

$$dV = -V \frac{d\rho}{\rho} > 0$$

The conclusion therefore is that in isothermal frictional pipe flow, the velocity will always increase along the pipe. Simply put, the gas will accelerate. It can also be shown that the gas will accelerate if the flow is adiabatic. This further means that the Mach number will increase along the pipe length, eventually reaching Mach 1 sonic conditions if the pipe is long enough.

Once sonic conditions are reached, no additional acceleration is possible. Thus the flow will experience sonic choking. When a flow experiences sonic choking, lowering the downstream pressure to increase the differential pressure will not increase the flow. This phenomenon is frequently encountered in long gas transfer lines, and is addressed by placing compressors in the pipeline at regular intervals.

It should be noted for completeness that the previous development assumed isothermal flow.

If pipe conditions are such that a gas is cooled along its length, then it is possible to decelerate a compressible flow. The reason decelerating flow can occur when the gas is cooled is seen from the equation of state. The ideal gas equation, in differential form, is

$$\frac{dP}{P} = \frac{d\rho}{\rho} + \frac{dT}{T}$$

and eliminating density using the continuity equation

$$\frac{dV}{V} = \frac{dT}{T} - \frac{dP}{P}$$

If cooling occurs, then the temperature change is negative, and because the pressure change is also negative, the velocity change can be either negative or positive. Thus, accelerating or decelerating flow is possible. In most applications, the pressure changes more rapidly than the temperature, making the velocity change positive resulting in accelerating flow.

### **Isothermal and Adiabatic Flow**

---

Since the majority of gas flow applications operate somewhere between isothermal and adiabatic flow, these two idealized cases are frequently taken as the extremes to consider. As discussed in the previous section, this is not strictly true.

The case of adiabatic flow is frequently encountered in short pipelines or in cases where the pipes are well insulated. When the flow is adiabatic, the static temperature will decrease along the pipe. This can be seen from Equation 93. For perfect gases, the stagnation temperature is constant in adiabatic flow. As the Mach number,  $M$ , increases, the static temperature must decrease.

But what happens when the pipe is cooled? When cooled, the stagnation temperature will decrease and the static temperature will actually decrease more than it would naturally. Thus, we can see that adiabatic flow cannot always be relied upon to be an extreme case.

The case of isothermal flow is frequently encountered in long gas transfer pipelines. When the flow is isothermal, then by definition the temperature remains constant. This can only occur if the gas is heated from its tendency to cool down. In a long transfer line exposed to ambient conditions, the tendency is for the gas pipe material to come to equilibrium with the surroundings, and the gas will tend toward equilibrium with the ambient conditions as well. The gas is heated by the ambient conditions outside the pipe.

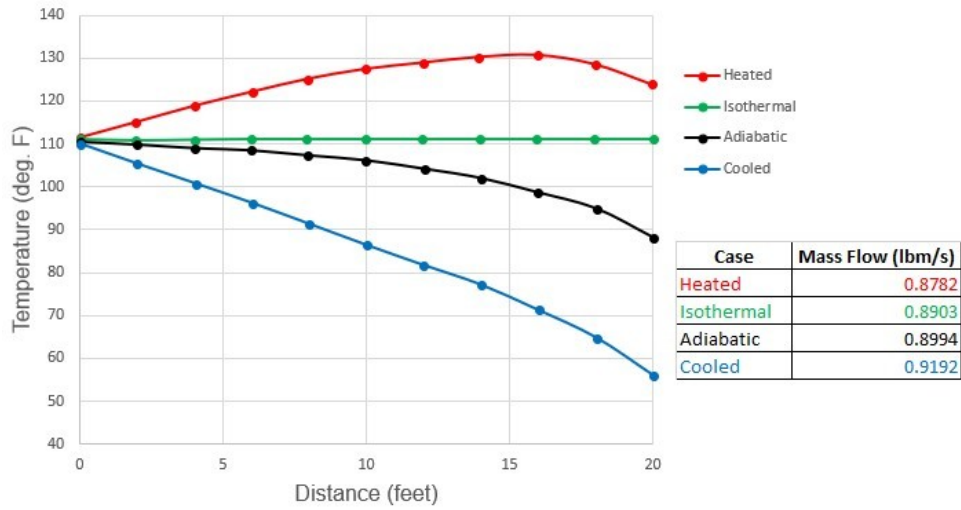
In a more general case of gas heating, the heating may be strong enough to actually increase the static temperature. This means that the isothermal condition is not the extreme limit either. There is, in fact, no limit to gas heating effects until conditions reach ionization or dissociation.

Figure 3 shows an example of air flow in a pipe where the adiabatic and isothermal flow do not bracket the range of possible flow conditions.

This discussion points out a frequent misconception among engineers that isothermal and adiabatic flow are the extreme cases to consider in gas flows. A more accurate statement is that the conditions under which the majority of gas pipelines operate have heat transfer boundary conditions that maintain the gas between adiabatic and isothermal behavior. For most practical applications, adiabatic and isothermal flow can be considered as the extreme cases. But it has been shown that this is not a completely general rule, and may be incorrect in some applications.

One last comment of practical importance. Whether a pipe operates adiabatically or isothermally or somewhere in between or outside, the primary impact of these assumptions is on the gas temperature.

Frequently, for analysis simplification purposes, a gas is assumed to operate either adiabatically or isothermally, and the result is that the flow rates and gas pressures in the system do not change significantly between these two idealized cases. Thus, if the purpose of the analysis is to determine the system flow rate or flow distribution in a network, the heat transfer assumption used for the pipe will frequently have only a second order effect on the results.



**Figure 3: Comparison of temperature profiles and flow rate predictions for adiabatic, isothermal, heated and cooled air flow in a 20 ft. long 1 inch steel pipe. Inlet conditions are 100 psia, 111 °F and outlet pressure is 60 psia. Cooled flow has ambient temperature of 30 °F. Heated flow has ambient temperature of 220 °F. Both heated and cooled cases have pipe wall and insulation heat transfer coefficients which total to 100 Btu/hr-ft<sup>2</sup>-F. Note how adiabatic and isothermal cases do not bracket conditions. See [Walters, 2000](#).**

### Related Topics

Stagnation vs. Static Pressure Boundaries.....	68
Stagnation Properties.....	455
Nomenclature.....	410
References.....	461



## Transient Theory

AFT xStream simulates transient systems using fundamental equations based on conservation of mass, momentum, and energy. These fundamental equations are solved using the Method of Characteristics.

The math and calculations needed to support AFT xStream's transient simulations can be complex at times. The topics in this section attempt to clearly detail the theory behind AFT xStream. Please reach out to AFT Support if you have further questions.

## Transient Flow Equations

Any rigorous fluid dynamics solution must start with the most fundamental laws of physics. For fluid flow there are three core equations - conservation of mass, Newton's second law of motion, and conservation of energy. We will consider the below control volume in a pipe. The control surfaces 1 and 2 are permitted to move relative to one another, and the unlabeled control surface on the pipe wall must remain fixed to the wall. For the purposes of analyzing a fluid transient, it is safe to consider the system one-dimensional in the  $x$  direction. Note that there is one radial effect that is important to consider - the change in cross-sectional area of the pipe.

The resulting equations are nonlinear hyperbolic partial differential equations, requiring a numeric solution such as the [Method of Characteristics](#).

For a complete development of the equations please see one of the many excellent [references](#) (Moody 1990, Wylie 1993, Chaudry 1987, and Swaffield 1993).

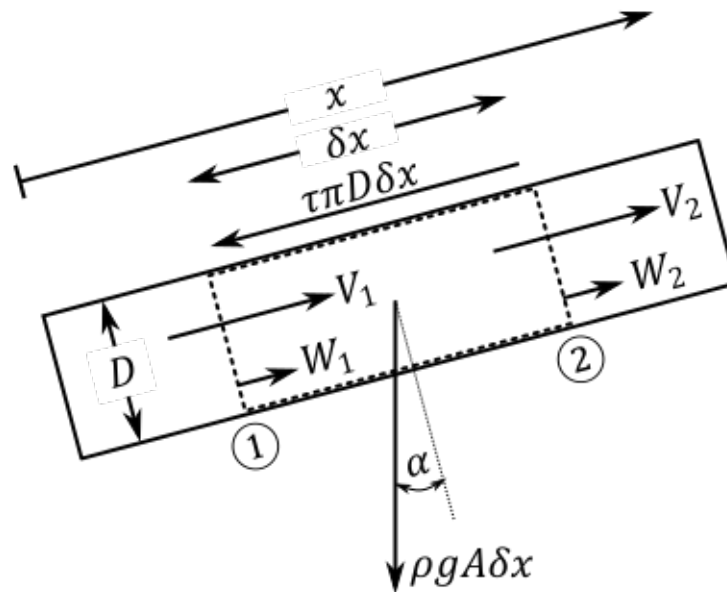


Figure 1: Control volume

## Continuity Equation

First we apply conservation of mass to the control volume via Reynold's Transport Theorem:

$$\frac{d}{dt} \int_{CV} \rho dV - \oint_{CS} \rho (\vec{V} \cdot \hat{n}) dA = 0$$

Simplifying to one dimension:

$$\frac{d}{dt} \int_{x_1}^{x_2} \rho A dx - [\rho_1 A_1 (V_1 - W_1) - \rho_2 A_2 (V_2 - W_2)] = 0$$

According to the Leibniz integral rule:

$$\frac{d}{dt} \int_{x_1}^{x_2} \rho A dx = \int_{x_1}^{x_2} \frac{\partial}{\partial t} (\rho A) dx + \underbrace{\rho_2 A_2 \frac{dx_2}{dt}}_{W_2} - \rho_1 A_1 \underbrace{\frac{dx_1}{dt}}_{W_1}$$

Applying this relationship and canceling terms:

$$\int_{x_1}^{x_2} \frac{\partial}{\partial t} (\rho A) dx + \rho_2 A_2 V_2 - \rho_1 A_1 V_1 = 0$$

Considering a mean value of the integral:

$$\frac{\partial}{\partial t} (\rho A) \delta x + \Delta(\rho A V) = 0$$

Divide by  $\delta x$

$$\frac{\partial}{\partial t} (\rho A) + \frac{\partial}{\partial x} (\rho A V) = 0$$

Expand and group terms to find the continuity equation:

$$\frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} + \rho \frac{\partial V}{\partial x} + \frac{\rho}{A} \left( \frac{\partial A}{\partial t} + V \frac{\partial A}{\partial x} \right) = 0$$

### Momentum Equation

The application of Newton's second law follows from the same control volume above. Again applying Reynold's Transport Theorem we begin with the following relationship:

$$\sum F = \frac{d}{dt} \int_{CV} \rho V dV - \oint_{CS} \rho V (\vec{V} \cdot \hat{n}) dA$$

Very similar to the continuity equation, we can apply a one dimensional simplification, the Leibniz integral rule, and consider the resulting integral as a mean over the interval. These steps are extremely similar to the above and are not shown for brevity.

$$\sum F = \frac{\partial}{\partial t} (\rho V A) \delta x + \Delta(\rho A V^2)$$

Summing the forces on the body:

$$\sum F = P_1 A_1 - P_2 A_2 - \tau \pi D \delta x - \rho g A \sin(\alpha) \delta x$$

To find the shear term, we consider a *steady* force balance with only two terms - the pressure across a section, and the shear term. We can find the pressure across the section with the Darcy-Weisbach relationship. Note that this step assumes that the transient shear term can be calculated with an equivalent steady state velocity.

$$\sum F_{steady} = \frac{\pi D^2}{4} dP - \tau \pi D \delta x = 0$$

$$dP = \frac{f \rho V |V| \delta x}{2 D}$$

$$\tau = \frac{f \rho V |V|}{8}$$

Incorporating this, dividing through by  $\delta x$ , and assuming that the areas on the pressure terms can be considered constant, we arrive at the following combined equation:

$$\frac{\partial}{\partial t}(\rho V A) + \frac{\partial}{\partial x}(\rho A V^2) = -A \frac{\partial P}{\partial x} - \frac{f \rho V |V|}{8} \pi D - \rho g A \sin(\alpha)$$

Expanding the partial derivatives, and dividing through by  $\rho A$ , we find the continuity equation and can drop those terms.

$$\frac{V}{\rho A} \underbrace{\left( \frac{\partial}{\partial t}(\rho A) + \frac{\partial}{\partial x}(\rho A V) \right)}_{\text{Continuity Eqn} = 0} + \frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{f V |V|}{2 D} - g \sin(\alpha)$$

Gathering the remaining terms, we arrive at the momentum equation:

$$\boxed{\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{f V |V|}{2 D} + g \sin(\alpha) = 0}$$

## Energy Equation

First apply Reynolds Transport Theorem to Newton's first law:

$$\frac{dE}{dt} = \frac{\partial}{\partial t} \int_{CV} \rho e dV + \int_{CS} \rho e (\vec{V} \cdot \hat{n}) dA$$

From the First Law of Thermodynamics:

$$\frac{dE}{dt} = \frac{dQ}{dt} - \frac{dW}{dt}$$

Assuming that the shaft work and viscous work are zero, only the pressure work will contribute as follows:

$$\frac{dW}{dt} = \int_{CS} P (\vec{V} \cdot \hat{n}) dA$$

Combining the three equations above, the surface integrals can be simplified as follows

$$\frac{dQ}{dt} = \frac{\partial}{\partial t} \int_{CV} \rho e dV + \int_{CS} \rho \left( e + \frac{P}{\rho} \right) (\vec{V} \cdot \hat{n}) dA$$

Simplifying the surface integrals into one dimension, the equation becomes

$$\frac{dQ}{dt} = \frac{\partial (\rho e A)}{\partial t} dx + d \left[ \rho V A \left( e + \frac{P}{\rho} \right) \right]$$

Dividing through by  $\partial x$ , expanding the derivatives and grouping the derivatives results in the following.

$$\frac{\dot{Q}}{dx} = e \underbrace{\left( \frac{\partial (\rho A)}{\partial t} + \frac{\partial (\rho V A)}{\partial x} \right)}_{\text{Continuity Eqn} = 0} + \rho A \frac{\partial e}{\partial t} + \frac{P}{\rho} \frac{\partial (\rho V A)}{\partial x} + \rho V A \frac{\partial \left( e + \frac{P}{\rho} \right)}{\partial x}$$

To further simplify the above, the continuity equation indicated above can be eliminated. To eliminate the internal energy term use the definition of internal energy and take the partial derivative as follows.

$$\frac{de}{dt} = \frac{\partial h_o}{\partial t} - \frac{\partial \left( \frac{P}{\rho} \right)}{\partial t} + \frac{\partial gz}{\partial t}$$

Assuming that gravity and elevation are constant with time, the partial derivative of  $gz$  can be dropped. The internal energy derivative can then be substituted into the overall equation.

$$\frac{\dot{Q}}{dx} = \rho A \left( \frac{\partial h_o}{\partial t} - \frac{\partial \frac{P}{\rho}}{\partial t} \right) + \frac{P}{\rho} \frac{\partial (\rho V A)}{\partial x} + \rho V A \frac{\partial (h_o + gz)}{\partial x}$$

After expanding the partial derivatives and grouping the common terms results in the following energy equation.

$$\frac{\partial h_o}{\partial t} + V \frac{\partial (h_o + gz)}{\partial x} - \frac{1}{\rho} \frac{\partial P}{\partial t} - \frac{1}{\rho} \frac{\dot{Q}}{A \partial x} - \frac{P}{\rho A} \frac{\partial A}{\partial t} = 0$$

## Modified Conservation Equations

The continuity, momentum and energy equations derived above contain a large number of variables which make them unwieldy.

The stagnation and static enthalpy terms in the energy equation can be eliminated using the definition of [stagnation enthalpy](#), the first law of thermodynamics, and various thermodynamic relations and identities. The rearranged energy equation is as follows.

$$\begin{aligned} & \frac{c_p}{\beta \rho a^2} \left( \frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} \right) - \frac{c_p}{\beta \rho} \left( \frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} \right) \\ &= -V \left( \frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} + g \frac{\partial z}{\partial x} \right) + \frac{1}{\rho} \frac{\dot{Q}}{A \partial x} + \frac{P}{\rho} \left( \frac{1}{A} \frac{\partial A}{\partial t} \right) \end{aligned}$$

In this form, part of the momentum equation can be inserted into the energy equation. Rearrange the momentum equation as follows. Note that the  $\sin(\alpha)$  term in the momentum equation shown previously is equivalent to  $dz/dx$ .

$$\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} + g \frac{dz}{dx} = -\frac{f V |V|}{2 D}$$

The rearranged momentum equation can now be inserted into the energy equation to get the Modified Energy Equation:

$$\frac{1}{a^2} \left( \frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} \right) - \left( \frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} \right) = \frac{\beta}{c_p} \left[ V \left( \frac{f V |V|}{2 D} \right) + \frac{\dot{Q}}{A \partial x} + \frac{P}{A} \left( \frac{\partial A}{\partial t} \right) \right]$$

The continuity equation can be rearranged to insert into the Modified Energy Equation as follows

$$-\left( \frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} \right) = \rho \frac{\partial V}{\partial x} + \frac{\rho}{A} \left( \frac{\partial A}{\partial t} + V \frac{\partial A}{\partial x} \right)$$

After inserting the above continuity equation into the Modified Energy equation, multiply through by  $a^2$  to find the Modified Continuity Equation as follows

$$\left(\frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x}\right) + \rho a^2 \frac{\partial V}{\partial x} + \frac{\rho a^2}{A} \left(\frac{\partial A}{\partial t} + V \frac{\partial A}{\partial x}\right) = \frac{\beta a^2}{c_p} \left[ V \left(\frac{fV|V|}{2D}\right) + \frac{\dot{Q}}{A \partial x} + \frac{P}{A} \left(\frac{\partial A}{\partial t}\right) \right]$$

The Modified Continuity and Energy equations can be further simplified by defining the variable  $\mathbb{F}$ . The final modified equations for  $\mathbb{F}$ , continuity, energy, and momentum can be seen below in order:

$$\mathbb{F} = V \left(\frac{fV|V|}{2D}\right) + \frac{\dot{Q}}{A \partial x} + \frac{P}{A} \left(\frac{\partial A}{\partial t}\right)$$

$$\left(\frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x}\right) + \rho a^2 \frac{\partial V}{\partial x} + \frac{\rho a^2}{A} \left(\frac{\partial A}{\partial t} + V \frac{\partial A}{\partial x}\right) = \frac{\beta a^2}{c_p} \mathbb{F}$$

$$\frac{1}{a^2} \left(\frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x}\right) - \left(\frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x}\right) = \frac{\beta}{c_p} \mathbb{F}$$

$$\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} = -g \frac{dz}{dx} - \frac{fV|V|}{2D}$$

## Related Topics

Transient Result Accuracy.....	49
Nomenclature.....	410
Method of Characteristics.....	435

## Method of Characteristics

The Method of Characteristics is a general approach for solving a partial differential equation by reducing it to a system of ordinary differential equations.

We see here how it is applied to the [transient flow equations](#). The three partial differential equations have three unknowns  $P, V$ , and  $\rho$ , and two independent variables  $x$  and  $t$ . Application of the Method of Characteristics will generate six ordinary differential equations.

## Characteristic Equations

---

First linearly combine the equations with the introduction of constant factors represented by  $\lambda$ :

$$\begin{aligned} \lambda_1 \left[ \frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} + \rho a^2 \frac{\partial V}{\partial x} \right] + \lambda_2 \left[ \frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} \right] + \lambda_3 \left[ \frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} - \frac{1}{a^2} \left( \frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} \right) \right] \\ = \lambda_1 \left[ \frac{\beta a^2}{c_p} \mathbb{F} \right] + \lambda_2 \left[ -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] + \lambda_3 \left[ -\frac{\beta}{c_p} \mathbb{F} \right] \end{aligned}$$

Note that for the above equation, the area was assumed to be constant, so all partial derivatives with respect to  $A$  were dropped to simplify the equation.

Rearranging terms from the linear combination above

$$\begin{aligned} \left( \lambda_1 - \frac{\lambda_3}{a^2} \right) \left[ \frac{\partial P}{\partial t} + \left( V + \frac{\lambda_2/\rho}{\lambda_1 - \frac{\lambda_3}{a^2}} \right) \frac{\partial P}{\partial x} \right] + \lambda_2 \left[ \frac{\partial V}{\partial t} + \left( V + \frac{\lambda_1 \rho a^2}{\lambda_2} \right) \frac{\partial V}{\partial x} \right] + \lambda_3 \left[ \frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} \right] \\ = \lambda_1 \left[ \frac{\beta a^2}{c_p} \mathbb{F} \right] + \lambda_2 \left[ -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] + \lambda_3 \left[ -\frac{\beta}{c_p} \mathbb{F} \right] \end{aligned}$$

We define relationships for  $\lambda_1, \lambda_2$ , and  $\lambda_3$ :

$$\frac{dx}{dt} = V + \frac{\lambda_2/\rho}{\lambda_1 - \frac{\lambda_3}{a^2}}$$

$$\frac{dx}{dt} = V + \frac{\lambda_1 \rho a^2}{\lambda_2} = V$$

$$\lambda_3 \frac{dx}{dt} = \lambda_3 V$$

By asserting that  $\lambda_3 = 1$ , it is found that



$$V - \frac{dx}{dt} = 0$$

and

$$\frac{\lambda_2}{\lambda_1} = \pm \rho a$$

Substituting into the above equation:

$$\left(\lambda_1 - \frac{\lambda_3}{a^2}\right) \frac{dP}{dt} + \lambda_2 \frac{dV}{dt} + \lambda_3 \frac{d\rho}{dt} = \lambda_1 \left[\frac{\beta a^2}{c_p} \mathbb{F}\right] + \lambda_2 \left[-g \frac{dz}{dx} - \frac{fV|V|}{2D}\right] + \lambda_3 \left[-\frac{\beta}{c_p} \mathbb{F}\right]$$

We have three solutions for  $\lambda$ , each valid along a certain characteristic line, as will be described further below. The solutions are as follows

$$\begin{aligned} \frac{dx}{dt} &= V + a, & \frac{\lambda_2}{\lambda_1} &= + \rho a, & \lambda_3 &= 0 \\ \frac{dx}{dt} &= V - a, & \frac{\lambda_2}{\lambda_1} &= - \rho a, & \lambda_3 &= 0 \\ \frac{dx}{dt} &= V, & \lambda_3 &= 1, & \lambda_1 = \lambda_2 &= 0 \end{aligned}$$

This represents the six characteristic equations that we are searching for, paired as the right running (R), left running (L), and particle path (P) sets of equations:

$$\begin{aligned} R \left\{ \frac{dP}{dt} + \rho a \frac{dV}{dt} = \frac{\beta a^2}{c_p} \mathbb{F} + \rho a \left[ -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] \right. & \quad \text{on } \frac{dx}{dt} = V + a \\ L \left\{ \frac{dP}{dt} - \rho a \frac{dV}{dt} = \frac{\beta a^2}{c_p} \mathbb{F} - \rho a \left[ -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] \right. & \quad \text{on } \frac{dx}{dt} = V - a \\ P \left\{ \frac{d\rho}{dt} - \frac{1}{a^2} \frac{dP}{dt} = -\frac{\beta}{c_p} \mathbb{F} \right. & \quad \text{on } \frac{dx}{dt} = V \end{aligned}$$

where

$$\mathbb{F} = \left( \frac{f \rho |V^3|}{2 D} \right) + \frac{\dot{Q}}{A \partial x}$$

The relationship between the time step, length step, sonic velocity and velocity from the characteristic equations shown here is crucial for effectively sectioning the pipes, which is discussed further in [Pipe Sectioning - Introduction to Method of Characteristics](#).

---

**Note:** This set of ordinary differential equations represent an exact solution to the governing equations, given the minor assumptions that flow is one dimensional, gravity does not change in time and space, and the pipe is rigid and constant area.

---

## Numerical Method

---

We can simplify the equations further into a form that is convenient for simulation.

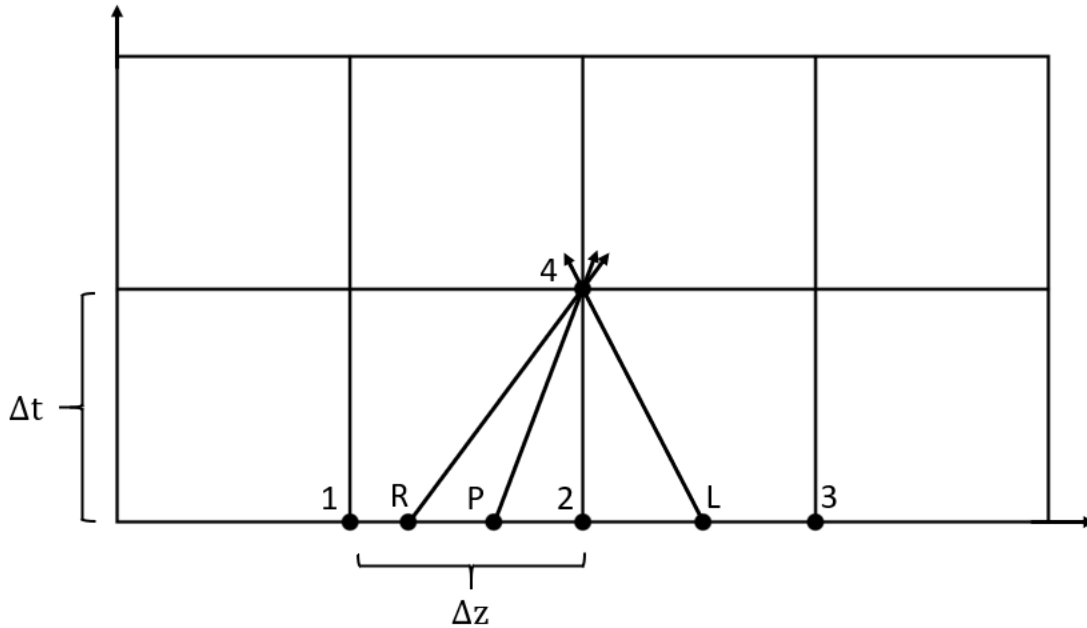
For each of the equations multiply by  $dt$  as is shown below for the *right running characteristic*

$$dP + \rho a dV = \frac{\beta a^2}{c_p} \left[ \left( \frac{f \rho |V^3|}{2 D} \right) + \frac{\dot{Q}}{A \partial x} \right] dt + \rho a \left[ -g \frac{dz}{dx} - \frac{f V |V|}{2 D} \right] dt$$

This equation can be integrated along the right running characteristic.

It is useful at this stage to consider a graphical representation of the characteristic grid.

On the characteristic grid point 4 will be affected by the right running characteristic from point R, the left running characteristic from point L, and the particle path characteristic from point P.



**Figure 1: Characteristic grid. The point 4 sees effects from points R, L, and P after some time  $dt$ , which has equal magnitude of  $dx/a$**

Setting up the integral along the right running characteristic line:

$$\int_R^4 dP + \int_R^4 \rho a dV = \int_R^4 \left[ \frac{\beta a^2}{c_p} \left( \frac{f \rho |V|^3}{2D} + \frac{\dot{Q}}{A \partial x} \right) + \rho a \left( -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right) \right] dt$$

This integrand represents the exact solution. However, in order to evaluate the integral, several major assumptions will be made:

1. The values are only known at the grid points (points 1, 2, and 3 in Figure 1). Therefore, interpolation is required to find the values at points R, L, and P. In literature this is often cited as the largest source of error. Linear interpolation is used to determine the position,  $z$ , and the properties at each of the starting points R, L, and P. Though compressible flow properties do not vary linearly along a pipe, it has been found that more sophisticated interpolation is less stable and requires a much larger calculation time, which is why linear interpolation is used for xStream.
2. The variables in each integrand cannot all be related, so an analytical solution cannot be found for each of the integrals. Instead, the integrands are assumed to be constant, using the values from the starting point. In some cases this may be a good approximation, as the properties are relatively constant. However, in other cases, such as at high Mach numbers, the properties would change rapidly in reality. Adding additional sections helps reduce error caused by this assumption.
3. Related to the constant integrand and interpolation assumptions, the characteristic lines are assumed to be straight. In reality the slope would constantly change as both the velocity and sonic

velocity are changing over time. Assuming the characteristic lines are straight allows the starting points R, L, and P to be interpolated as discussed above.

Using the 3 assumptions above, the integral is trivial, and the below solution can be found for the right running characteristic:

$$\boxed{(P_4 - P_R) + (\rho a)_R (V_4 - V_R) = F_R \Delta t}$$

where

$$F = \frac{\beta a^2}{c_p} \left( \frac{f \rho |V^3|}{2D} + \frac{\dot{Q}}{A \partial x} \right) + \rho a \left( -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right)$$

We can repeat the above process for the *left running characteristic* and *particle path characteristic* to arrive at:

$$\boxed{(P_4 - P_L) - (\rho a)_L (V_4 - V_L) = G_L \Delta t}$$

$$\boxed{(\rho_4 - \rho_P) - \frac{1}{a_P^2} (P_4 - P_P) = H_P \Delta t}$$

where

$$G = \frac{\beta a^2}{c_p} \left( \frac{f \rho |V^3|}{2D} + \frac{\dot{Q}}{A \partial x} \right) - \rho a \left( -g \frac{dz}{dx} - \frac{fV|V|}{2D} \right)$$

$$H = -\frac{\beta}{c_p} \left( \frac{f \rho |V^3|}{2D} + \frac{\dot{Q}}{A \partial x} \right)$$

We have finally arrived a system of three equations with only three unknowns - pressure, density and velocity at point 4. These equations can be further arranged into a convenient form known as the [compatibility equations](#).

## Related Topics

Transient Result Accuracy.....	49
Nomenclature.....	410
Cv/Kv and XT Loss Model.....	205

## Compatibility Equations

From the [characteristic equations](#) (repeated below), we can construct three convenient equations known as the compatibility equations.

$$(P_4 - P_R) + (\rho a)_R (V_4 - V_R) = F_R \Delta t$$

$$F = \frac{\beta a^2}{c_p} \left( \frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right) + \rho a \left( -g \frac{dz}{dx} - \frac{f V |V|}{2D} \right) \quad \begin{array}{l} \text{(Positive Wave} \\ \text{Characteristic)} \end{array}$$

$$(P_4 - P_L) - (\rho a)_L (V_4 - V_L) = G_L \Delta t$$

$$G = \frac{\beta a^2}{c_p} \left( \frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right) - \rho a \left( -g \frac{dz}{dx} - \frac{f V |V|}{2D} \right) \quad \begin{array}{l} \text{(Negative Wave} \\ \text{Characteristic)} \end{array}$$

$$(\rho_4 - \rho_P) - \frac{1}{a_P^2} (P_4 - P_P) = H_P \Delta t$$

$$H = \frac{\beta}{c_p} \left( \frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right) \quad \begin{array}{l} \text{(Particle Path} \\ \text{Characteristic)} \end{array}$$

The only unknown terms in these equations are the pressure, velocity and density at point 4. The other terms are found from a previous time step, or from the initial conditions as determined from the [steady state MOC simulation](#).

Rearranging terms:

$$P_4 = P_R + F_R \Delta t + \rho_R a_R V_R - \rho_R a_R V_4$$

$$P_4 = P_L + G_L \Delta t - \rho_L a_L V_L + \rho_L a_L V_4$$

$$\rho_4 = \rho_P + H_P \Delta t - \frac{1}{a_P^2} P_P + \frac{1}{a_P^2} P_4$$

and defining six new constants (constant for this time step):

$$C_{wave}^+ = P_R + F_R \Delta t + \rho_R a_R V_R$$

$$C_{wave}^- = P_L + G_L \Delta t - \rho_L a_L V_L$$

$$C_{path}^+ = \rho_P + H_P \Delta t - \frac{1}{a_P^2} P_P$$

$$B_{wave}^+ = -\rho_R a_R$$

$$B_{wave}^- = \rho_L a_L$$

$$B_{path}^+ = \frac{1}{a_P^2}$$

We have our simplest equations yet:

---

$$P_4 = C_{wave}^+ + B_{wave}^+ V_4$$

$$P_4 = C_{wave}^- + B_{wave}^- V_4$$

$$\rho_4 = C_{path}^+ + B_{path}^+ P_4$$

Which can be solved directly for pressure and velocity, then used to directly calculate density.

**Note:** The compatibility equations allow a solution of all interior points in a pipe. However, the pipe end-points are solved by applying specific boundary conditions. The boundary condition relationships build on the previous methods, and offer the final needed information to generate a complete solution.

## Related Topics

Method of Characteristics.....435

## Transient Sonic Choking

During the transient solution sonic choking may begin at any of the geometries described in the [Sonic Choking Detailed Description](#) topic, including all junctions except for the dead end junctions. In addition to the geometries previously discussed, sonic choking may occur at a junction at the inlet of a pipe in response to rapid system changes. Choking at the inlet of a pipe is not sustainable, and may result in shock waves moving through the system, as is discussed further below.

AFT xStream evaluates every boundary calculation (i.e. junction inlet/outlets) at every time step to determine if sonic choking would occur. If sonic choking is detected at a junction, xStream will adjust the solution method at that junction accordingly.

Under sonic conditions, the defined boundary condition at the choke point is no longer meaningful, as it cannot be reached. However, because the flow is sonic the velocity must be equal to the sonic velocity,  $a$ . Sonic velocity depends on the local state, which is unknown. Therefore, iteration is required to determine the velocity.

Iterations will be performed on the velocity using the appropriate [wave compatibility equation](#) to calculate pressure. To calculate the density for an inflow the pressure and the specified temperature/enthalpy will be used. For an outflow boundary the density is calculated using the [particle path compatibility equation](#).

## Shock Waves

---

A shock wave is a compression wave across which velocity transitions from supersonic to subsonic, and also causes discontinuities in other properties such as pressure, temperature, and density. ([Saad, 1993](#)) It is simplest to discuss shock waves by discussing how compression waves are formed. Compression waves may or may not become shock waves, depending on the conditions.

A compression wave is generated in a pipe or duct due to a series of incremental, instantaneous velocity changes in the flow. Each velocity change generates a sound wave that begins traveling through the gas. As the gas compresses each subsequent wave will have a higher sonic velocity caused by the increasing temperature. Eventually the faster waves will catch up to the initial wave, causing the compression wave to steepen and the pressure drop across the wave to increase. If an abrupt enough velocity change occurs, then the compression wave will become a shock wave.

AFT xStream does not employ any shock wave solution methods in the solver. However, the [method of characteristics](#), which is frequently used to model wave movement, can approximate the movement of shock waves through gas systems.

## Related Topics

---

K Factor Loss Model.....	202
Method of Characteristics.....	435
Sonic Choking Detailed Description.....	450
Sonic Choking.....	449

## Related Blogs

---

[Making Sense of xT – Valve Loss in Compressible Flow](#)

### Compressible Flow Theory in Single Pipes

In order to provide a robust method to solve compressible flow in pipe networks, numerous fundamental equations must be solved simultaneously. The AFT Arrow steady state solver which AFT xStream uses, solves the continuity (i.e., mass balance) equation, the one-dimensional momentum equation, the energy equation, the equation of state, and the Mach number equation which ties together temperature and velocity.

If you are unfamiliar with the field of compressible flow, it is suggested that you first read the [compressible flow review topic](#) and/or consult a textbook for a thorough description of the subject.

### Extension of Single-Pipe Methods to Networks

Methods to solve the single pipe equations are available in the literature ([Anderson, 1982](#), [Saad, 1993](#), and [Shapiro, 1953](#)), but their extension to pipe networks involves significant difficulties. In a network system, the flow and energy must balance at each branch point. Traditional solution methods in incompressible flow systems break down because the relationships between pressure and flow cannot be



expressed in a single convenient equation. Thus the calculation speed and reliable convergence benefits of incompressible flow matrix methods are much more difficult to obtain in compressible flow calculations. Heat transfer and the resulting temperature effects further complicate the analysis.

To address these issues, AFT xStream's primary steady solver incorporates unique methods of relating the various thermodynamic and fluid dynamic parameters to each other resulting in a solution methodology that is fast and offers good convergence characteristics. AFT xStream's primary steady methods maintain the integrity of single pipe methods while preserving the required mass and energy balances at all branch points.

The specific techniques used by AFT xStream's primary steady solver to obtain solutions to the equations of compressible flow are proprietary and probably of little interest to the user. What the user will be most interested in is what equations are solved (so that results can be cross-checked) and what balances are calculated at branch points. Therefore, sufficient information is provided to enable users to verify results, while the proprietary aspects of AFT xStream's solution methods are not described.

## Fundamental Equations

The continuity equation applied to a constant area pipe yields:

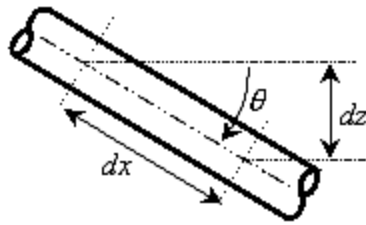
$$\frac{d\rho}{\rho} + \frac{dV}{V} = 0 \quad (1)$$

The one-dimensional momentum equation applied to a constant area pipe yields:

$$dP + \frac{\rho V^2}{2} \frac{f dx}{D} + \rho V dV + \rho g dz = 0 \quad (2)$$

Assuming the pipe is straight over the computing section, Figure 1 shows the relationship between dx and dz:

$$dz = (\sin \theta) dx \quad (3)$$



**Figure 1: Relationship between distance along pipe ( $dx$ ) and vertical elevation change ( $dz$ ).**

The fourth term on the left of Equation 2 accounts for pressure changes due to body forces. AFT xStream models body forces due to gravity. The default value of  $g$  in Equation 2 is the standard earth acceleration (i.e., 32.174 ft/sec<sup>2</sup> or 9.81 m/sec<sup>2</sup>).

From the equation of state for a real gas:

$$P = Z \rho R T \tag{4}$$

**Related Topics**

Nomenclature .....410

**Continuity Equation at a Branch**

By applying the continuity equation at branch  $i$ , we loop over all connecting pipes and get

$$\sum_{j=1}^n \dot{m}_{ij} = 0 \tag{1}$$

where  $\dot{m}_{ij}$  is the flow rate in the pipe connecting junctions  $i$  and  $j$ .

**Related Topics**

Nomenclature .....410

**Energy Equation at a Branch**

By applying the energy equation at a branch  $i$ , we loop over all connecting pipes and get

$$\sum_{j=1}^{n_{pipes}} \dot{m}_{ij} \left( h_{ij} + \frac{V_{ij}^2}{2} \right) = 0$$

or, in terms of stagnation enthalpy,

$$\sum_{j=1}^{n_{pipes}} \dot{m}_{ij} h_{o,ij} = 0 \tag{47}$$

where  $h_{o,ij}$  is the stagnation enthalpy at the connection point of pipe and junction.

The elevation term shown in [Equation 30](#) drops out because all pipes at the junction connect at the same elevation.

## Related Topics

Nomenclature .....410

## Network Implementation

To provide converged solutions in the shortest time possible, matrix solution techniques are used. Matrix techniques allow all pipes to be solved simultaneously, which is faster and offers the best convergence characteristics.

To obtain a valid solution in a network, the [stagnation pressure](#) at each branch point will be equal and the mass flow and energy flow will sum to zero at the branch. Although the stagnation pressures at the pipe endpoints will equal that at the connecting junction, this is not true for [stagnation enthalpy](#) (and temperature). The stagnation enthalpy at the end of a pipe will be that enthalpy that results from the heat transfer in the pipe. Once the fluid flows into the junction, the stagnation enthalpy will change to the mixture stagnation enthalpy, or the average stagnation enthalpy of all inflowing pipes.

If only one pipe flows into the branch, the branch stagnation enthalpy will equal the stagnation enthalpy coming out of the pipe. But if more than one pipe flows into the branch, the stagnation enthalpy will reach an averaged condition. It is this averaged stagnation enthalpy that results from mixing two or more streams at different stagnation enthalpies. This stagnation enthalpy is supplied to all outflowing pipes. This is depicted in Figure 1 and follows Equation 49.

$$h_{o,mix} = \frac{\sum_{j=1}^n \dot{m}_j^+ h_{o,j}}{\sum_{j=1}^n \dot{m}_j^+} \tag{49}$$

where the + superscript indicates the summation applies only to positive inflows into the junction.

The concentration balance at a junction is similar to stagnation enthalpy. The junction concentration itself will not match the concentrations for the inflowing pipes, but will be a mixture of the incoming flow streams.

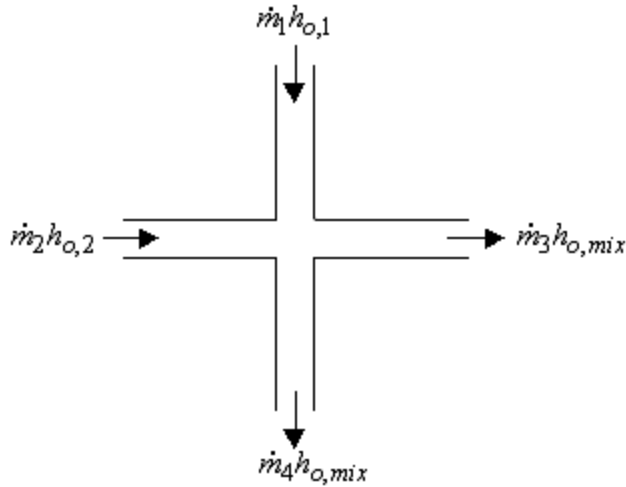


Figure 1: Flow weighted mixing and energy balance

## Related Topics

Nomenclature.....	410
-------------------	-----

## Verifying Network Solutions

AFT xStream's primary steady solver uses a modified Newton-Raphson matrix method to obtain a system level balance of mass and energy. When the Solver is finished, the [continuity](#) and [energy](#) equations are satisfied at all branching sections. Once solved, it is a simple matter to go back and sum up all mass flows and energy at each branch to verify that these equations are satisfied. In fact, AFT xStream does just that.

After the primary steady solver converges, AFT xStream loops over each junction and adds up the inflow and outflow of mass and energy and compares the sum to the [solution tolerances](#). If it appears that any junction is out of tolerance, a warning is given in the output.

In the Solution Balance Summary table AFT xStream provides additional information on the balance at each junction. This table can be displayed by selecting the option in the [Output Control](#) window General Output folder. Figure 1 shows an example of the Solution Balance Summary table.

Jct	Connected Pipes	Mass Flow (lbm/sec)	Flow Energy (Btu/s)
1 (Atmosphere)	1	-0.5980	-106.7
	TOTAL	-0.5980	-106.7
2 (Compressor)	1	0.5980	106.7
	2	-0.5980	-117.8
	TOTAL	0.000	-11.10
3 (Branch)	2	0.5980	117.5
	3	-0.4356	-85.62
	6	-0.1625	-31.93
	TOTAL	-4.109E-05	-8.216E-03
4 (Branch)	3	0.4356	85.44
	4	-0.2859	-56.08
	7	-0.1497	-29.37
	TOTAL	-3.669E-05	-7.275E-03
5 (Branch)	4	0.2859	55.97
	5	-0.1421	-27.82
	8	-0.1437	-28.13
	TOTAL	9.808E-05	0.01917
6 (Branch)	5	0.1421	27.77
	9	-0.1421	-27.77
	TOTAL	0.000	0.000
7 (Tool #1)	6	0.1625	31.89

**Figure 1: Solution Balance Summary table shown in the Output window offers a balance report of all junctions in the window.**

## Sonic Choking

Sonic choking occurs in a pipe system when the gas attempts to accelerate to a velocity faster than the local sonic velocity at some point in a system. Being unable to accelerate any further, the gas reaches its maximum velocity at sonic velocity. Changes to the downstream conditions, such as lowering the pressure, produce no additional flow. In many pipe applications sonic choking dominates the system behavior.

Predicting the flow rates under such sonic choking conditions can be very difficult depending on where the choking occurs. AFT xStream is designed to identify sonic choking and predict the system behavior under such conditions. In addition, xStream can approximate moving shock waves in the system.

## Related Topics

Cv/Kv and XT Loss Model.....	205
Sonic Choking Detailed Description.....	450
Transient Sonic Choking.....	443

## Sonic Choking Detailed Description

Sonic choking occurs when some location in the pipeline experiences a local velocity equal to the sonic velocity of the gas. AFT xStream can account for sonic choking, and approximate moving shock waves in gas systems.

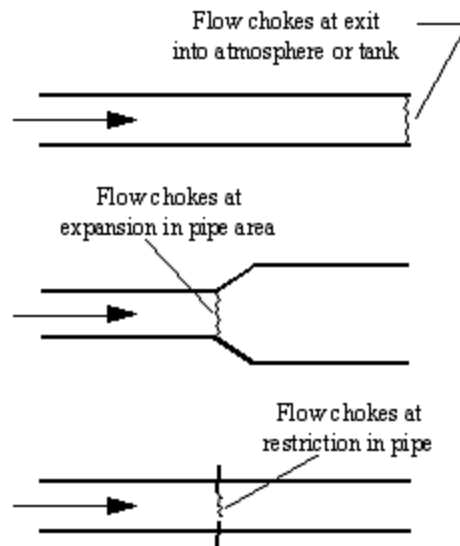
### Sonic Choking Types

Sonic choking can occur in three geometric configurations in a pipe system. Figure 1 depicts the three possibilities.

The first case can be called *endpoint choking* because it occurs at the end of a pipe as it exits into a large vessel or the atmosphere. In this situation, the gas cannot drop its pressure to match that at the discharge without accelerating to sonic velocity. A choke point forms at the end of the pipe, which results in a pressure discontinuity across which the discharge pressure condition is met.

The second case can be called *expansion choking*, and is similar to the first case. Expansion choking can happen when the flow area of the pipe is increased rapidly. For example, this occurs when the pipe is expanded from a two inch pipe to a three inch pipe. Another example is when a pipe enters a flow splitter where the sum of the pipe areas on the splitting side exceeds the pipe area of the supply pipe. A choke point forms at the end of the supply pipe, and a pressure discontinuity is established.

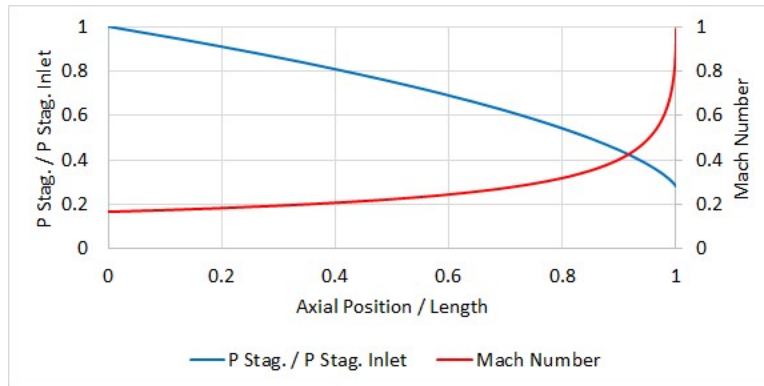
The third case can be called *restriction choking*. This can occur when the gas flows through a restriction in the pipe such as an orifice or valve. In such a case the flow area of the gas is reduced, causing a local increase in velocity which may reach the sonic velocity. A choke point forms at the restriction, with a pressure discontinuity similar to the first two cases.



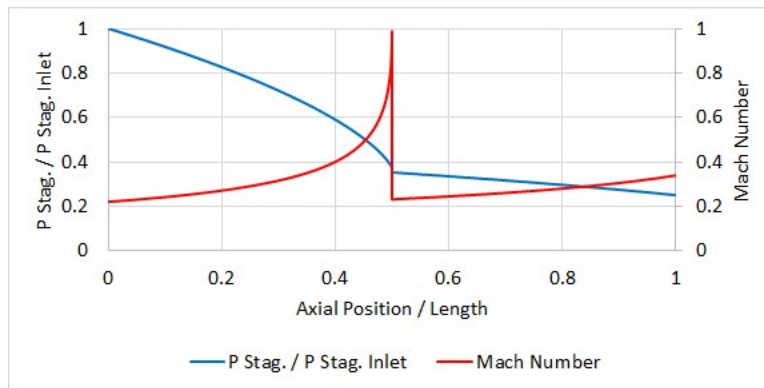
**Figure 1: Three configurations which can result in sonic choking. From top to bottom, endpoint choking, expansion choking, and restriction choking**

Figures 2-4 show example stagnation pressure and Mach number profiles for the three cases. The data for the cases are: Supply air at 100 psia and 1000° R discharging to 30 psia, with Figure 2) 2 inch pipe

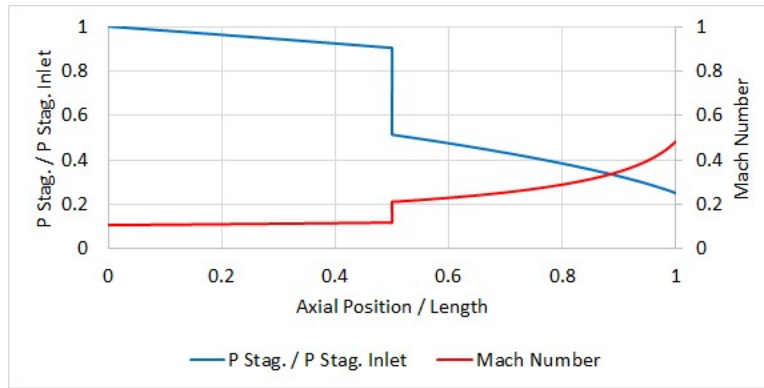
with no restrictions or expansions, Figure 3) 2 inch pipe expanding to 3 inches at the midpoint, Figure 4) all pipe is 2 inch with orifice at midpoint with 0.6 area ratio. Mach number spikes to 1 at the orifice then reduces to 0.29 immediately after. For clarity this spike is not shown.



**Figure 2: Example stagnation pressure and Mach number profiles for endpoint choking.**



**Figure 3: Example stagnation pressure and Mach number profiles for expansion choking.**



**Figure 4: Example stagnation pressure and Mach number profiles for restriction choking. Note that the Mach Number reaches 1 within the restriction at 0.5.**

### Steady State Sonic Choking

During the steady state the choked flow rate can be determined from Equation 50 by inserting a Mach number of 1 in Equation 51. These equations can be derived from the continuity equation ([Saad, 1993](#)):

$$\dot{m} = A \frac{P_0}{\sqrt{T_0}} f(M) \quad (50)$$

where:

$$f(M) = \sqrt{\frac{\gamma}{Z R}} M \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{-(\gamma+1)/2(\gamma-1)} \quad (51)$$

In practice it is difficult to apply Equation 50 to choked conditions because the local stagnation conditions,  $P_0$ , and  $T_0$ , are not known at the point of choking. For instance, to apply these equations to end-point choking (Figure 1, top), one must calculate the stagnation pressure and temperature at the end of the pipe before the choke point. The problem is that the stagnation pressure and temperature before the choke point depend on the flow rate, which is what we are trying to calculate.

The only way to accurately calculate such a problem is to do the following:

1. Assume a flow rate
2. March down the pipe
3. If the Mach number reaches unity before the end of the pipe, lower the assumed flow rate and repeat steps 1-3 until the Mach number reaches sonic right at the pipe endpoint.

Obviously this is not practical without a computer.



The area,  $A$ , in Equation 50 can also be  $C_dA$ , where the  $C_d$  is less than one and accounts for area reductions due to flow separation. This occurs in orifices, valves and other internal restrictions. Note that the  $C_d$  used here is not equivalent to the discharge coefficient used for subsonic flow calculations. See [Subsonic vs. Sonic Pressure Losses](#) for more information.

A helpful output parameter in AFT xStream is the sonic area, which can be compared against actual area to determine what  $C_dA$  would cause choking at a restriction:

$$A_{sonic} = \dot{m} \frac{\sqrt{T_0}}{P_0 f(M)_{M=1}} \quad (52)$$

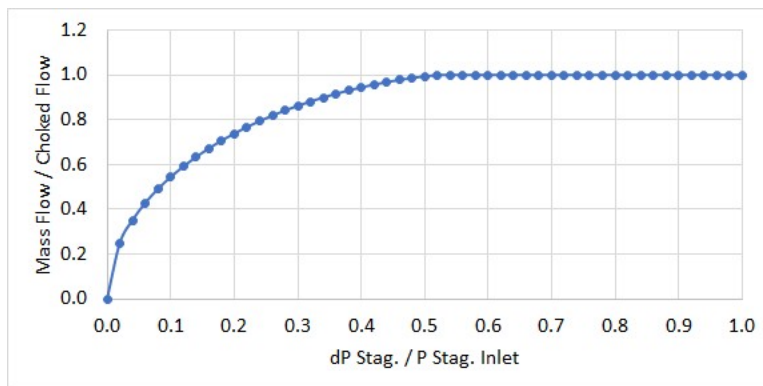
A  $C_dA$  smaller than the sonic area indicates that the flow would be choked. The sonic area is one of the output parameters.

## Transient Sonic Choking

During the transient, the above choking mechanisms will still be applicable, though additional scenarios, such as choking at the inlet of a pipe, may occur. In addition the calculation method must be adjusted as the [Method of Characteristics](#) is used for transient pipe calculations, rather than the marching methods used for the steady state. See [Transient Sonic Choking](#) for further details.

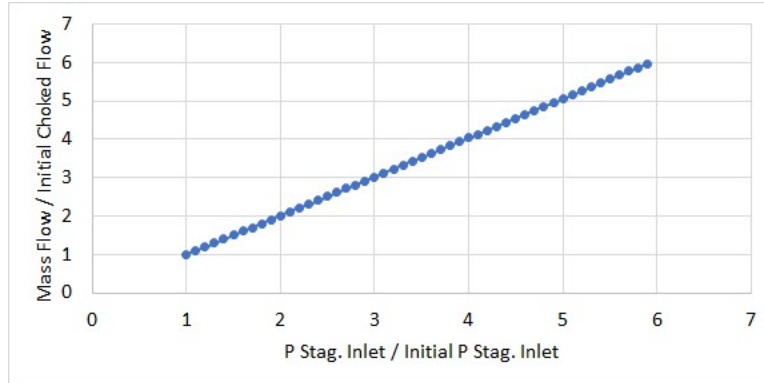
## Sonic Choking Effects

The effect of sonic choking in pipe design or operation is that the maximum physically possible pipe flow rate has been reached for the supply conditions. Lowering the discharge pressure will not produce more flow rate. Figure 5 illustrates this by showing the flow rate as a function of discharge pressure in a two inch pipe carrying air. This pipe does not have any restrictions, and experiences endpoint choking at any discharge pressure below 63.6 psia.



**Figure 5: Example of dimensionless flow rate vs. dimensionless pressure drop. Sonic choking occurs just past the dimensionless pressure drop of ~0.5, after which the flow can no longer increase.**

Some engineers misapply the concept of sonic choking and conclude that the sonic flow rate is the maximum possible through the pipe for all conditions. This is not the case. If the supply conditions can be increased, the flow rate through the pipe can be increased. In fact, the increased choked flow rate appears to increase linearly with increased supply pressure. See Figure 6.



**Figure 6: Example of increasing choked flow rate with supply pressure.**

The pressure drop across the choke point in choked flow cannot be calculated directly. Methods such as "normal shock tables" do not apply. The only way to calculate the pressure drop is to use the choked flow rate as a new boundary condition on the pipe downstream of the choke point. This boundary condition is facilitated by knowing the stagnation enthalpy change across the choke point is constant. For the steady state solution governing equations can then be applied to the remaining pipes to solve for the static conditions. During the transient solution the static conditions can be found from the static conditions at the downstream station at the previous time step.

Finally, a pipe can choke in more than one location. This occurs when the choked flow rate determined by the first choke point is applied to the pipes downstream of the shock wave and the gas at this flow rate cannot reach the end of the pipe without experiencing another shock wave. There is no limit to the number of choke points in a pipe other than the number of possible geometric configurations it contains which may cause sonic choking.

The three mechanisms that cause choking can all occur in the same pipeline in any combination and any order.

See [Walters, 2000](#) and [Walters and Olsen, 1997](#) for more information.

## Related Topics

Nomenclature.....	410
Method of Characteristics.....	435
Transient Sonic Choking.....	443
Sonic Choking.....	449

## Subsonic vs. Sonic Pressure Losses

There are two separate pressure loss mechanisms when gas flows through a restriction, expansion, or end point.

- **Subsonic** losses are standard "form" losses and are dependent on the geometry of the restriction, expansion, or end point. This is the common [K factor](#), or [discharge coefficient](#) for subsonic flow.
- **Sonic** losses result when [sonic choking](#) occurs. Sonic losses are dependent on the minimum effective flow area ( $C_dA$ ) and are unrelated to the K factor.

The two loss mechanisms cannot occur simultaneously in nature or in AFT xStream. During a time step if one mechanism is in use, the input data for the other mechanism is ignored by AFT xStream. The loss mechanism in use can switch during the transient as system operation changes.

It is also important to note that sonic choking at a restriction cannot be calculated if the  $C_dA$  for Sonic Choking is not entered. In this case, AFT xStream will always use subsonic loss calculations if no  $C_dA$  for Sonic Choking has been entered, even if the actual system would experience choked flow at that restriction. The only exception to this is if the restriction occurs at a valve which is defined in terms of [Cv/Kv and XT](#), which can account for both subsonic and sonic loss at the valve.

## Related Topics

Orifice.....	253
Relief Valve.....	256
Discharge Coefficient Loss Model.....	201
Screen.....	257
Venturi.....	265

## Stagnation Properties

Solving compressible flow equations requires equations to relate static and stagnation properties. The following equations are used by AFT xStream to relate stagnation temperature, pressure, enthalpy, and density ( $T_0$ ,  $P_0$ ,  $h_0$ , and  $\rho_0$ , respectively), velocity (or Mach number,  $M$ ), and static temperature, pressure, enthalpy, and density ( $T$ ,  $P$ ,  $h$ , and  $\rho$ , respectively).

Stagnation temperature:

$$\frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2$$

Stagnation pressure:

$$\frac{P_0}{P} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma/(\gamma-1)}$$

Stagnation enthalpy:

$$h_0 = h + \frac{V^2}{2}$$

Stagnation density:

$$\frac{\rho_0}{\rho} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{1/(\gamma-1)}$$

---

## Related Topics

Stagnation vs. Static Pressure Boundaries.....	68
Review of Compressible Flow Theory.....	412
Nomenclature.....	410

## Pipe Material Library References

The pipe material properties in AFT applications were originally compiled from a variety of unofficial sources. Many of those sources are now obsolete. Beginning in 2016, pipe material properties are taken directly from national/international standards such as: ASTM, ANSI, AWWA, and ASME. Several new material standards are also added. For example: DIN and JIS. With the addition of a significant number of new pipe material standards, the increased volume and scope necessitate splitting the pipe material library into several smaller independent libraries. All of the pipe material found in older AFT applications are still available, but are now called "AFT Customary".

Maintaining several independent pipe libraries facilitates including or excluding specific standards. For example: a pipe engineer in Germany may prefer to use only ISO and DIN standards. The Library manager can be used to "Connect" or "Disconnect" each of the pipe material libraries. A list of available pipe libraries can be found below.

Note that while AFT attempts to provide this standard information to make defining pipe information easier for the user, the prudent engineer will always verify this information against manufacturer specifications. ASTM and ANSI standards exist for the thickness and diameter specifications for some of these materials. Furthermore, AFT attempts to source reference information from reputable and authoritative sources. However, data for pipe roughness and thermophysical properties including density, thermal conductivity, modulus of elasticity, and Poisson's ratio have no single standard to follow. Thus, the user should always check that the pipe material inputs match the actual pipe material data for their system.

- **Copper Pipe - ASTM -**
  - Source = ASTM B42-10
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-33
- **Copper Tubing - ASTM -**
  - Source = ASTM B88-09
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-31; [www.cranecopper.com.au/sizes/astmb88.asp](http://www.cranecopper.com.au/sizes/astmb88.asp)
- **Ductile Iron - ANSI -**
  - Source = ANSI-AWWA C150-A21.50-08
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Fiberglass - Green Thread -**
  - Source = Fiber Glass Systems product data, [http://www.nov.com/Segments/Completion\\_and\\_Production\\_Solutions/Fiber\\_Glass\\_Systems/Chemical\\_Industrial/Green\\_Thread.aspx](http://www.nov.com/Segments/Completion_and_Production_Solutions/Fiber_Glass_Systems/Chemical_Industrial/Green_Thread.aspx)
  - Roughness References = Fiber Glass Systems product data
  - Density References = Average value calculated based on ID, OD and Weight in Fiber Glass Systems product data
  - Thermal Conductivity References = Fiber Glass Systems product data
  - Modulus of Elasticity References = Fiber Glass Systems product data
  - Poisson's Ratio References = Fiber Glass Systems product data
  - Additional References = ASTM D2996 - 17
- **HDPE - AFT Customary -**
  - Source = Applied Flow Technology
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **PVC - ASTM -**
  - Source = ASTM D1785-06
  - Additional References = Thermal data from: Materials Science and Engineering, Callister, Table C.4
- **PVDF - AFT Customary -**
  - Source = George Fischer Engineering handbook page 425
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Rectangular Duct - AFT Customary -**
  - Source = Applied Flow Technology
- **Stainless Steel - ANSI -**

- Source = ASME-B36\_19M\_2004
- Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
- Thermal Conduct References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Type 304
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ration Referenecs = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for Type 304 @ 20oC
- **Stainless Tubing - AFT Customary -**
  - Source = Applied Flow Technology
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Steel - AFT Customary -**
  - Source = Applied Flow Technology
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- **Steel - ANSI -**
  - Source = ASME B36.10-2004
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - DIN 2391-1 -**
  - Source = DIN 2391-1-1994
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - DIN 2458 -**
  - Source = DIN 2458-1981
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10216-1 (s1) -**
  - Source = BS EN 10216-1-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10216-2 (s1) -**
  - Source = BS EN 10216-2-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-1 (s1) -**
  - Source = BS EN 10217-1-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn

- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
- Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-2 (s1) -**
  - Source = BS EN 10217-2-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-4 (s1) -**
  - Source = BS EN 10217-4-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-5 (s1) -**
  - Source = BS EN 10217-5-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-6 (s1) -**



- Source = BS EN 10217-6-2002
- Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Commercial Steel
- Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20°C
- Additional References = ANSI 36.10, CRANE Technical Paper 410

## References

1. Anderson, J.D., Jr., *Modern Compressible Flow: With Historical Perspective*, McGraw-Hill, New York, NY, 1982.
2. ANSI/API 526 7th Edition - Flanged Steel Pressure-relief Valves, 2017, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA
3. ANSI/ISA Standard 75.01.01-2012, Industrial-Process Control Valves - Part 2-1: Flow Capacity - Sizing equations for fluid flow under installed conditions, 2012, published by International Society of Automation, Research Triangle Park, North Carolina, USA
4. American Petroleum Institute, *API STD 618: Reciprocating Compressors for Petroleum, Chemical, and Gas Industry Services*, 4th Edition, American Petroleum Institute, Washington, DC, US, 1995.
5. ASME Press, ASME International Steam Tables for Industrial Use (CRTD-Vol. 58), published by the American Society of Mechanical Engineers, Three Park Avenue, New York, NY 10016
6. Ballun, John V., "A Methodology for Predicting Check Valve Slam", *Journal AWWA*, Vol.99, No. 3, pp. 60-65, March 2007.
7. Bejan, A., *Advanced Engineering Thermodynamics*, John Wiley & Sons, New York, NY, 1988.
8. Blanding, James M., Ph.D., and Trey Walters, PE. Pulsation Analysis in Positive Displacement Pump Systems Using Waterhammer, Modal, and Animation Software. Houston, TX: 45th Turbomachinery & 32nd Pump Symposia, Sept. 2016.
9. Carthew, G.A., Goehring, C.A., van Teylingen, J.E., Development of dynamic head loss criteria for raw sludge pumping, *Journal Water Pollution Control Federation (WPCF)*, Volume 55, Number 5, (May 1983).
10. Chaudhry, M. H., *Applied Hydraulic Transients*, 2nd edition, Van Nostrand Reinhold Company, New York, NY, 1987.
11. Crane Co., *Flow of Fluids Through Valves, Fittings, and Pipe*, Technical Paper No. 410, Crane Co., Joliet, IL, 1988.
12. Darby, R., *Chemical Engineering Fluid Mechanics*. 2nd edition. New York, NY, 2001
13. Donsky, B., Complete Pump Characteristics and the Effects of Specific Speeds on Hydraulic Transients, *ASME Journal of Basic Engineering*, pp. 685-699, Dec. 1961.
14. Fox, R. W. and A.T. McDonald, *Introduction to Fluid Mechanics*, 3rd edition, Wiley & Sons, New York, NY, 1985.
15. Idelchik, I. E., *Handbook of Hydraulic Resistance*, 4th edition, CRC Press, Boca Raton, FL, 2007.
16. Incropera, F.P. et. al, *Fundamentals of Heat and Mass Transfer*, John Wiley & Sons, NJ 2007

17. Ingersoll-Dresser Pumps, Cameron Hydraulic Data, Liberty Corner, NJ, 1995.
18. Ito, H., "Friction factors for pipe flow", *Journal of Basic Engineering*, Vol. 81, pp. 123-126, 1959.
19. Jeppson, R. W., *Analysis of Flow in Pipe Networks*, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.
20. Karassik, I.J., J.P. Messina, P. Cooper and C.C. Heald, *Pump Handbook*, 3rd Edition, McGraw Hill, New York, NY, 2001.
21. Karney, B.W., and D. McInnis, Efficient Calculation of Transient Flow in Simple Pipe Networks, *ASCE Journal of Hydraulic Engineering*, Vol. 118, No. 7, July, 1992.
22. Keenan, J.H., *Thermodynamics*, John Wiley & Sons, Inc., New York, NY, 1941.
23. Kittredge, C.P., *Hydraulic Transients in Centrifugal Pump Systems*, *ASME Journal of Basic Engineering*, Vol. 78, No. 6, pp. 1307-1322, Aug. 1956.
24. Lyons, J. L., *Lyons' Valve Designers' Handbook*, Van Nostrand Reinhold, New York, NY, 1982.
25. Miller, D. S., *Internal Flow Systems*, 2nd edition, Gulf Publishing Company, Houston, TX, 1990.
26. Moody, Frederick J., *Introduction to Unsteady Thermofluid Mechanics*, John Wiley & Sons, Inc., New York, NY, 1990.
27. Nayyar, M. L., *Piping Handbook*, 7th edition, McGraw-Hill, New York, NY, 2000.
28. *Pipe Line Rules of Thumb Handbook*, Gulf Publishing, Houston, TX.
29. Saad, M.A., *Compressible Fluid Flow*, 2nd Edition, Prentice-Hall, Englewood Cliffs, NJ, 1993
30. Shapiro, A.H., *The Dynamics and Thermodynamics of Compressible Fluid Flow*, 2 vols., Ronald, New York, NY, 1953.
31. Smith, J.M., van Ness, H.C., and Abbott, M.M., *Introduction to Chemical Engineering Thermodynamics*, 6th edition, McGraw-Hill, New York, NY, 2001.
32. Swaffield, J. A., and Boldy, A. P., *Pressure Surge in Pipe and Duct Systems*, Avebury Technical, Hampshire, England, 1993.
33. TAPPI, Generalized method for determining the pipe friction loss of flowing pulp suspensions, TAPPI, TIS 0410-14, Atlanta, GA, 1988.
34. Tilton, J. N., *Perry's Chemical Engineer's Handbook*, Seventh Edition, Chapter 6, McGraw-Hill, New York, NY, 1997.
35. Thorley, A.R.D., *Fluid Transients in Pipeline Systems*, 2nd Ed., D&L George, Ltd. 2004
36. Thorley, A.R.D., and A. Chaudry, *Pump Characteristics for Transient Flow, Pressure Surges and Fluid Transients*, BHR Group, 1996.
37. Val-Matic Valve and Manufacturing Corp., "Surge Control in Pumping Systems," Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.
38. Walski, T. M., *Analysis of Water Distribution Systems*, Van Nostrand Reinhold Company, New York, NY, 1984.
39. Walters, T.W., "Gas-Flow Calculations: Don't Choke", *Chemical Engineering*, Chemical Week Associates, Jan. 2000, pp. 70-76.
40. Walters, T.W., and J.A. Olsen, "Modeling Highly Compressible Flows in Pipe Networks Using a Graphical User Interface," *ASME International Joint Power Generation Conference*, Denver, CO, 1997.
41. Watters, G.Z, *Modern Analysis and Control of Unsteady Flow in Pipelines*, Ann Arbor Science, Ann Arbor, MI, 1979.
42. White, F. M., *Fluid Mechanics*, McGraw Hill, New York, NY, 1979.

43. Winters, B.A., and T.W. Walters, "X-34 High Pressure Nitrogen Reaction Control System Design and Analysis," NASA Thermal Fluid Analysis Workshop, Houston, TX, 1997.
44. Wylie, E.B., V.L. Streeter & L. Suo, Fluid Transients in Systems, Prentice Hall, Englewood Hills, New Jersey, 1993.

## Chempak Technical Information - Detailed Discussion

© Copyright 1998 Madison Technical Software Inc. Used with permission.

---

**Note:** The information in this Appendix was taken from Section 8 of the Chempak Operating & Reference Manual, Version 4, Windows Edition, Issue: January 1998. Section numbering was left unchanged, and some sections that were judged not relevant were not included so the numbering is not sequential.

---

**Note:** Notation and references in this Appendix apply only to this Appendix, and are referenced at the end.

---

### 8.1 General

This section sets out the data sources, correlations and estimation methods used in the CHEMPAK database. In putting together the database, the methods and sources were selected in the following order of preference.

- Published experimental data
- Published correlations based on experimental data
- Specific category correlations
- General estimation methods

Madison Technical Software has followed the general recommendations in Reid and in Danner and Daubert as far as selection of specific category correlations and general estimation methods are concerned. In selecting specific compound data, a combination of sources has been used wherever possible. Important sources of specific compound data used by CHEMPAK are:

- Reid et al
- Perry et al
- J Chem Eng Data
- Daubert & Danner
- ESDU publications
- API Technical Data Book - Petroleum Refining
- International Critical Tables
- CRC Handbook
- Vargaftik

In many cases, the compound property values are a combination of published data, published correlations and general estimation methods. Several properties in certain compound categories have been

estimated or adjusted by Madison Technical Software. It has been our policy to adopt and maintain a critical approach to available data sources and correlation methods.

The following sections set out details of the correlations and estimation methods used. In certain cases, the user is directed to the original references, particularly where the method is complex.

Data sources for Aqueous Solutions/Heat Transfer Liquids are published experimental data and correlations based on experimental data.

## 8.2 Physical Constants

### 8.2.1 Critical Temperature

The great majority of values are believed to be experimental. Where values had to be estimated, the Joback method was used.

### 8.2.2 Critical Pressure

Most of the values are experimental. In cases where experimental data were not available, the critical pressure was derived from the Joback method.

### 8.2.3 Critical Volume

A majority of the values are experimental. A great majority of the remaining compounds for which experimental values were not available had accurate boiling-point volumes available from which critical volume estimates were derived using the Tyn and Calus correlation. For a few substances, estimates of the critical volume were derived from the Joback method.

### 8.2.4 Normal Boiling Points

All values are believed to be experimental. In some cases, the values were slightly adjusted for vapor pressure.

### 8.2.6 Acentric Factors

The acentric factor is defined as

$$w = -\log_{10}(P_{vr} \text{ at } T_r = 0.7) - 1$$

In all cases the acentric factor was derived from the vapor pressure correlation ( see section 8.8)

### 8.2.7 Joback Group Contribution Method

The Joback method is used to derive values of  $T_c$ ,  $P_c$ ,  $V_c$  and  $T_f$  where no experimental data or other predictive method was available.

$$T_c = T_b / (0.584 + 0.965 \sum(D_t) - \sum(D_t)^2)$$

$$P_c = 1 / (0.113 + 0.0032 n_a - \sum(D_p))^2$$

$$V_c = 17.5 + \sum(D_v)$$

$$T_f = 122 + \sum(D_f)$$

where  $n_a$  is the number of atoms in the molecule and the D contributions are given by Joback and by Reid et al (1987). Error magnitudes for the Joback method are as follows:

- Critical Temperature: average error about 1%
- Critical Pressure: average error about 5%
- Critical Volume: average error about 2%

· Freezing Point: average error about 11%

### 8.2.8 Tyn & Calus Relation

Tyn & Calus showed a close (< 3% error) relation between molar volume at normal boiling point and the critical molar volume of the form,

$$V_b = a V_{cn}$$

$$a = 0.285$$

$$n = 1.048$$

## 8.8 Vapor Pressure

The vapor pressure is expressed in its reduced form

$$P_{vr} = P_v/P_c$$

Reduced vapor pressure varies from very low values at freezing point to unity at the critical point.

### 8.8.1 Published Correlations

The experimental correlations are commonly given in the following formats:

#### **Wagner Equation**

$$\ln(P_{vr}) = (aX + bX^{1.5} + cX^3 + dX^6)/T_r$$

with

$$X = 1 - T_r$$

#### **FKT Equation**

$$\ln(P_v) = a + b/T + c \ln(T) + dP_v/T^2$$

#### **Antoine Equation**

$$\ln(P_v) = a + b/(T + c)$$

### 8.8.2 Gomez-Thodos Vapor Pressure Equation

Gomez-Nieto and Thodos give the following equation:

$$\ln(P_{vr}) = B(1/T_{rm} - 1) + G(T_r - 1)$$

$$G = aH + bB$$

$$a = (1 - 1/T_{br})/(T_{br} - 1)$$

$$b = (1 - 1/T_{brm})/(T_{br} - 1)$$

$$H = T_{br} \ln(P_c/P_b)/(1 - T_{br})$$

For non-polar compounds,

$$B = -4.267 - 221.79/(H^2.5 \exp(0.038 H^2.5)) + 3.8126/\exp(2272.33/H^3) + D$$

$$m = 0.78425 \exp(0.089315 H) - 8.5217/\exp(0.74826 H)$$

$$D = 0$$

except  $D = 0.41815$  for He,  $0.19904$  for H<sub>2</sub>,  $0.02319$  for Ne

For polar non-hydrogen-bonding compounds (e.g. ammonia and acetic acid),

$$m = 0.466 T_c^{0.1667}$$

$$G = 0.08594 \exp(0.0007462 T_c)$$

$$B = (G - aH)/b$$

### 8.8.2 Gomez-Thodos Vapor Pressure Equations

For polar hydrogen-bonding compounds (water, alcohols),

$$m = 0.0052 M^{0.29} T_c^{0.72}$$

$$G = (2.464.M) \exp(0.0000098 M T_c)$$

$$B = (G - aH)/b$$

The advantages of this method are,

- fit guaranteed at  $T = T_b$  and  $T = T_c$
- good performance with polar compounds
- good performance over  $T_r = 0.5$  to 1

In addition, tests carried out by Madison Technical Software show the clear superiority of this method especially at low temperatures over the Lee-Kesler method.

### 8.8.3 Lee-Kesler Vapor Pressure Equation

Lee and Kesler give the following vapor pressure equation:

$$\ln(P_{vr}) = f(0) + wf(1)$$

$w$  = acentric factor

$$f(0) = 5.92714 - 6.09648/T_r - 1.28862 \ln(T_r) + 0.169347 T_r^6$$

$$f(1) = 15.2518 - 15.6875/T_r - 13.4721 \ln(T_r) + 0.43577 T_r^6$$

The characteristics of this equation are,

- guaranteed fit at  $T_r = 1$  and 0.7
- accurate for non-polar compounds

This equation is used in the Lee-Kesler and Wu & Stiel equations of state.

### 8.8.4 Interpolation and Extrapolation

In many cases an accurate empirical equation is known which does not extend to the critical point or to the freezing point. The approach taken here is to fit the Wagner equation by least squares to the empirical equation and use the Wagner equation to extrapolate to the freezing point and to the critical point.

Extrapolation by this method to the critical method is a very accurate procedure. Extrapolation to the freezing point is less accurate but it does provide reasonable values.

In CHEMPAK, the vapor pressure correlations set out in this section are used to provide the basic data. Empirical relations are used wherever possible.

## 8.9 Vapor Viscosity

The methods of Lucas are employed here. The equations take two forms:

- Low Pressure (< 2 atm)
- High Pressure (> 2 atm)

### 8.9.1 Low-Pressure Equation

$$v_0 = a_0 k_0 \{ a_1 + a_2 T_r^n + a_3 \exp(a_4 T_r) + a_5 \exp(a_6 T_r) \} F_{p0}, F_{q0}$$

$v_0$  = low pressure viscosity in kg/ms

$$k_0 = M^{0.5} P_c^{0.667} T_c^{0.1667}$$

with  $P_c$  in Pa and  $T_c$  in K

$$a_0 = 0.0026373$$

$$a_1 = 0.018$$

$$a_2 = 0.807$$

$$a_3 = -0.357$$

$$a_4 = -0.449$$

$$a_5 = 0.340$$

$$a_6 = -4.058$$

$$n = 0.618$$

The low pressure polar correlation is given by,

$$F_{p0} = 1 \quad 0 < m_r < 0.022$$

$$F_{p0} = 1 + f(Z_c) \quad 0.022 < m_r < 0.075$$

$$F_{p0} = 1 + f(Z_c) \cdot g(T_r) \quad 0.075 < m_r$$

$$f(Z_c) = 30.55(0.292 - Z_c)^{1.72}$$

$$g(T_r) = 0.96 + 0.1(T_r - 0.70)$$

$$m_r = 52.46 m^2 P_c / T_c^2$$

$m$  = dipole moment, debye

$P_c$  = critical pressure, bar

$T_c$  = critical temperature, K

### 8.9.1 Low-Pressure Equation

In the above equations, if  $Z_c > 0.292$  the  $F_{p0}$  is taken as unity. There are only a few compounds that are polar with  $Z_c$  sufficiently larger than 0.292 to make  $F_{p0}$  significantly different from unity. The evidence in any case is scanty.

The low-pressure correction for quantum gases (hydrogen, helium and deuterium),  $F_{q0}$ , is given by,

$$F_{q0} = 1.22 Q^{0.15} \{ 1 - 0.00385(12 - T_r)^2 / M \}$$

with

$$T_r < 12$$

$$Q = 1.38 \text{ for helium}$$

$$0.76 \text{ for hydrogen}$$

$$0.52 \text{ for deuterium}$$

### 8.9.2 High-Pressure Equation

Define,

$$Z_1 = v_0 / a_0 k_0$$

For  $Tr \leq 1$ , we define,

$$Z2 = 0.60 + 0.76 Prm + (1 - Tr)(6.99Prm - 0.60)$$

$$m = 3.262 + 14.98 Pr^{5.508}$$

$$n = 1.39 + 5.746 Pr$$

For  $Tr > 1$  define,

$$Z2 = Z1\{1 + aPr^e/(bPr^f + 1/(1 + cPr^d))\}$$

with,

$$a = (a1/Tr)\exp(a2Tr^m)$$

$$b = a(b1Tr - b2)$$

$$c = (c1/Tr)\exp(c2Tr^m)$$

$$d = (d1/Tr)\exp(d2Tr^p)$$

$$e = 1.3088$$

$$f = f1\exp(f2Tr^q)$$

$$a1 = 0.001245$$

$$a2 = 5.1726$$

$$n = -0.3286$$

$$b1 = 1.6553$$

$$b2 = 1.2723$$

$$c1 = 0.4489$$

$$c2 = 3.0578$$

$$m = -37.7332$$

$$d1 = 1.7368$$

$$d2 = 2.2310$$

$$p = -7.6351$$

$$f1 = 0.9425$$

$$f2 = -0.1853$$

$$q = 0.4489$$

then,

$$Y = Z2/Z1$$

$$Fp = (1 + (Fp0 - 1)/Y^3)/Fp0$$

$$Fq = (1 + (Fq0 - 1)(1/Y - 0.007(\ln(Y))^4))/Fq0$$

$$v = v0YFpFq$$



## 8.10 Vapor Conductivity

The method of Ely and Hanley is adopted for both high and low pressure vapor conductivity. The method is based on hydrocarbon data but gives reasonable values for non-polar non-hydrocarbons. Errors are usually less than 10% except for highly polar compounds. There are few experimental data and no satisfactory correlations for high pressure conductivity for polar compounds.

The vapor conductivity is given by,

$$k = k_1 + k_2 + k_3$$

with,

$$k_1 = 1944 v_0 H (1 + 0.042332(C_{v0} - 3R/2))$$

$$k_2 = r_0 H (b_1 + b_2 (b_3 - \ln(T_0/b_4))^2)$$

$$k_3 = k_{30} k_{31} (k_{32} + k_{33} k_{34}) - 1$$

$$k_{30} = H/1000$$

$$k_{31} = \exp(a_1 + a_2/T_0)$$

$$k_{32} = \exp(a_3 + a_4/T_0^{1.5}) r_0^{0.1}$$

$$k_{33} = 6.1843 (r_0 - 1) r_0^{0.5}$$

$$k_{34} = a_5 + a_6/T_0 + a_7/T_0^2$$

The parameters in the above equations are given by:

$$v_0 = \text{low pressure methane viscosity} = \text{Sum}(c_n T_0^{(n-4)/3})$$

$$c_1 = 2.90774E-01$$

$$c_2 = -3.31287E-01$$

$$c_3 = 1.60810E-01$$

$$c_4 = -4.33190E-02$$

$$c_5 = 7.06248E-03$$

$$c_6 = -7.11662E-04$$

$$c_7 = 4.32517E-05$$

$$c_8 = -1.44591E-06$$

$$c_9 = 2.03712E-08$$

$$H = 16.04 f_0^{0.5} / M h_0^{0.667}$$

$$C_{v0} = \text{low pressure specific heat at constant volume in J/mol.K}$$

$$R = \text{gas constant} = 8.314 \text{ J/mol.K}$$

$$M = \text{molecular weight g/mol}$$

$$f = T_c F_1 / 190.4$$

$$h = V_c F_2 / 99.2$$

$T_c$  = critical temperature, K

$V_c$  = critical volume, cc/mol

$$F1 = 1 + (w - 0.011)(0.56553 - 0.86276 \ln(T^*) - 0.69852/T^*)$$

$$F2 = \{1 + (w - 0.011)(0.3949(V^* - 1.02355) - 0.93281(V^* - 0.75464)\ln(T^*)\}(0.288/Z_c)$$

$w$  = acentric factor

$Z_c$  = critical compressibility

$$T^* = T_r \text{ for } T_r \leq 2$$

$$= 2 \text{ otherwise}$$

$$V^* = V/V_c \text{ for } 0.5 < V/V_c < 2$$

$$= 0.50 \text{ for } V/V_c \leq 0.50$$

$$= 2 \text{ otherwise}$$

$$b_1 = -2.5276E-04$$

$$b_2 = 3.3433E-04$$

$$b_3 = 1.1200E+00$$

$$b_4 = 1.6800E+02$$

$$a_1 = -7.1977E+00$$

$$a_2 = 8.5678E+01$$

$$a_3 = 1.2472E+01$$

$$a_4 = -9.8463E+02$$

$$a_5 = 3.5947E-01$$

$$a_6 = 6.9798E+01$$

$$a_7 = -8.7288E+02$$

$$r_0 = 16.04 \text{ h/V}$$

$$T_0 = T/f$$

$V$  = specific volume, cc/mol

$T$  = temperature, K

## 8.11 Ideal Gas Thermodynamic Properties

Ideal gas thermodynamic properties (i.e. at low pressure) are required for the estimation of the thermodynamic properties of liquids and gases.

### 8.11.1 Method of Joback

For those substances not covered by published correlations, the group contribution method of Joback is used to estimate  $C_{p0}$  and the heats of formation.

$$C_{p0} = a + bT + cT^2 + dT^3$$

with,

$$a = \text{Sum}(n_j D_a) - 37.93$$

$$b = \text{Sum}(n_j D_b) + 0.210$$

$$c = \text{Sum}(n_j D_c) - 3.91E-04$$

$$d = \text{Sum}(n_j D_d) + 2.06E-07$$

where  $n_j$  is the number of groups of type  $j$  and the  $D$  contributions are for this type of group.  $T$  is in deg K. The user is referred to Joback for details. Reid et al (1987) gives values of the group contributions. The error associated with this method is usually less than 3%.

### 8.11.2 Low-temperature values of $C_{p0}$

To our knowledge, no general estimation method exists for  $C_{p0}$  below about 260 K. In addition, empirical datapoints and correlations in this region are not generally available. In order to extend the applicability of the estimation methods to temperatures below 260 K, extrapolation of known values and correlations was investigated by Madison Technical Software.

Extrapolation as far as 50 K (-223 C) using monotonic power functions fitted to known values at higher temperatures gave estimates whose error did not exceed 10% at 50 K and which averaged about 2% in the range 100 K to 273 K. This extrapolation was tested against known values for over 80 hydrocarbons and non-hydrocarbons. Thermodynamic properties calculated are limited in CHEMPAK to temperatures above minus 100 C.

### 8.11.3 Derived Ideal Gas Properties

Ideal gas enthalpy and entropy are derived by integration of the ideal gas specific heat:

The ideal gas enthalpy,  $h_0$ , is given by the integral of  $C_{p0}$  from  $T_0$  to  $T$

$$h_0(T) = [C_0T + C_1T^2/2 + C_2T^3/3 + C_3T^4/4]$$

evaluated between  $T_0$  and  $T$ .

The ideal gas entropy is given by the integral of  $C_{p0}/T$  from  $T_0$  to  $T$

$$s_0(T) = [C_0 \ln T + C_1T + C_2T^2/2 + C_3T^3/3]$$

evaluated between  $T_0$  and  $T$ .  $T_0$  is a zero-value reference temperature. The specific heat at constant volume is simply related to the specific heat at constant pressure:

$$C_{v0} = C_{p0} - R$$

## 8.12 The Equations Of State

### 8.12.1 General

The two equations of state employed in CHEMPAK are as follows:

1. The Lee-Kesler equation of state for non-polar compounds with moderate values of acentric factor

2. The Wu & Stiel equation of state for polar compounds or those with extreme values of acentric factor

The equations of state are used to predict the following vapor properties:

- Specific Volume
- Compressibility
- Expansion Coefficient
- Specific Heats
- Enthalpy/Internal Energy
- Entropy
- Heat of Vaporization

### 8.12.2 The Lee-Kesler Equation of State

The Lee-Kesler equation of state is a three-parameter ( $T_c$ ,  $P_c$  and  $w$ ) equation explicit in pressure with general applicability to compounds with low polarity and moderate acentric factors. The Lee-Kesler equation defines two fluids as follows,

- a simple fluid with zero acentric factor
- a reference fluid with acentric factor = 0.3978

The specific volume and other properties for any fluid are determined by interpolation between the simple fluid properties and the reference fluid properties using acentric factor as the interpolating variable. The properties of the simple and reference fluids are determined by a pair of PVT equations with identical form but with different parameters.

We define,

$$P_r = P/P_c$$

$$T_r = T/T_c$$

$$V_r = P_c V / RT_c$$

where  $P$ ,  $P_c$ ,  $T$  and  $T_c$  are the actual and critical pressures and the actual and critical temperatures of the fluid of interest.

First,  $V_r(0)$  is calculated using the constants for the simple fluid. Then  $V_r(r)$  is calculated using the constants for the reference fluid. From these quantities,  $Z(0)$  and  $Z(r)$  are calculated:

$$Z(0) = P_r V_r(0) / T_r \quad Z(r) = P_r V_r(r) / T_r$$

The compressibility of the fluid of interest is calculated using,

$$Z = A(0)Z(0) + A(r)Z(r)$$

$$A(0) = 1 - w/w(r)$$

$$A(r) = w/w(r)$$

$w$  = acentric factor for fluid of interest

$w(r)$  = acentric factor for reference fluid (0.3978)

The expansion coefficient, specific heat departures, enthalpy and, entropy departures are derived from the simple and reference PVT equations using the relations detailed in Lee and Kesler and in Reid and elsewhere. The component simple fluid and reference fluid values are combined by using relations of the same form shown above for compressibility. Danner & Daubert give typical maximum error values 20

kJ/kg for vapor enthalpy and 70 kJ/kg for liquid enthalpy. Typical errors for specific volume of vapors are 1 to 2%.

### 8.12.3 The Wu & Stiel Equation of State

The Wu & Stiel equation of state is a four-parameter equation of state developed as an extension to the Lee-Kesler equation of state to cover polar compounds and compounds with high values of acentric factor. The parameters of this equation of state are,

- Critical Temperature,  $T_c$
- Critical Pressure,  $P_c$
- Acentric Factor,  $w$
- Polarity Factor,  $Y$

The Wu and Stiel equation of state uses three reference fluids as follows:

- simple fluid with zero acentric factor
- reference fluid with acentric factor = 0.3978
- polar fluid (water) with acentric factor = 0.344 and polarity factor = 1.0

The specific volume and other properties for any fluid of interest are determined by interpolation between the properties of the reference fluids using acentric factor and polarity factor as interpolating variables.

The properties of the simple and reference fluids are determined from the Lee-Kesler equation of state. The properties of the polar fluid are determined from the Keenan equation of state for water. With  $P_r$  and  $T_r$  for the fluid of interest,

$$Z(0) = P_r V_r(0) / T_r \text{ from simple fluid equation}$$

$$Z(r) = P_r V_r(r) / T_r \text{ from reference fluid equation}$$

$$Z(p) = P_r V_r(p) / T_r \text{ from polar fluid equation}$$

$$Z = A(0)Z(0) + A(r)Z(r) + A(p)Z(p)$$

$$A(0) = (1 - Y) - (w/w(r) - Yw(p)/w(r))$$

$$A(r) = w/w(r) - Yw(p)/w(r)$$

$$A(p) = Y$$

$$A(0) + A(r) + A(p) = 1$$

When  $Y = 0$  these equations reduce to the non-polar Lee-Kesler formula. When  $Y = 1$  and  $w = w(p)$ , they reduce to the Keenan equation of state for water.

The expansion coefficient, specific heat departures, enthalpy, internal energy and entropy departures are derived from the simple, reference and polar PVT equations. The component simple fluid, reference fluid and polar fluid values are combined using relations of the same form as shown above in the case of compressibility. The polarity factor is determined from empirical density data in accordance with the relations set out by Wu and Stiel. Wu & Stiel report excellent results using this equation of state for polar compounds. Errors in specific volume were less than 1 to 2% for example for the vapor phase.

### 8.12.4 Calculation of Saturation Values

Fluid properties along the saturation line are solved for as follows. For a given value of  $T_r$ ,

1. The simple fluid component properties are solved at  $T_r$ ,  $P_{sr}(O)$  where this latter quantity is the Lee-Kesler reduced saturation pressure at  $T_r$  and  $w = 0$
2. The reference fluid component properties are solved at  $T_r$  and  $P_{sr}(r)$  where this latter quantity is the Lee-Kesler reduced saturation pressure at  $T_r$  and  $w = 0.3978$
3. The polar fluid component properties are solved at  $T_r$  and  $P_{sr}(p)$  which is the reduced water saturation pressure at  $T_r$  with  $P_{sr}(p) = P_s(p) / P_c(p)$

The compressibility of the fluid of interest is found by,

$$Z_s = A(0)Z_s(0) + A(r)Z_s(r) + A(p)Z_s(p)$$

This relation gives the saturation state of the fluid of interest by interpolation between the saturation states of the simple, reference and polar fluids at  $T_r$  rather than by interpolation between the three fluids at  $T_r$  and  $P_{sr}$  where this latter quantity is the reduced vapor pressure of the fluid of interest.

The above relations are presented for the case of the Wu & Stiel equation of state. The relations for the Lee-Kesler equation of state may be obtained by setting the polar contribution equal to zero.

### 8.12.5 Enthalpy and Entropy Scales

All enthalpy and entropy values are quoted relative to a reference zero-value temperature  $T_0$ . This temperature is taken in all cases to be the higher of 273.15 K and the normal melting point.

Two cases can be distinguished:

1. If the critical temperature is greater than or equal to 273.15 K then the enthalpy and entropy of the saturated liquid are taken to be zero at  $T_0$ . In this case the enthalpy and entropy functions are,

$$h(T,P) - h_{Ls}(T_0) = [h(T,P) - h_v(T,P_0)]_{es} + [h_0(T_0,T)]_{ig} - [h_{Ls}(T_0) - h_v(T_0,P_0)]_{es}$$

$$s(T,P) - s_{Ls}(T_0) = [s(T,P) - s_v(T,P_0)]_{es} + [s_0(T_0,T)]_{ig} - [s_{Ls}(T_0) - s_v(T_0,P_0)]_{es}$$

2. If the critical temperature is less than 273.15 then the enthalpy and entropy of the low-pressure vapor are taken to be zero at  $T_0$ . In this case the enthalpy and entropy functions are,

$$h(T,P) - h_v(T_0,P_0) = [h(T,P) - h_v(T,P_0)]_{es} + [h_0(T_0,T)]_{ig}$$

$$s(T,P) - s_v(T_0,P_0) = [s(T,P) - s_v(T,P_0)]_{es} + [s_0(T_0,T)]_{ig}$$

The zero point for water is taken to be 273.16 K in accordance with the Keenan equation of state for water. Liquid enthalpy and entropy are zero at that temperature and saturation conditions.

## 8.13 Properties Of Mixtures

### 8.13.1 Scope of Correlations

CHEMPAK provides the user with the facility of defining mixtures of compounds in the database which then may be stored in the database for future use. The properties of mixtures are calculated using the techniques set out in this section. The user should note the following:

A mixture can be formulated using any components in the database except other mixtures, aqueous solutions, heat transfer liquids and User-Defined Liquids.

No check is carried out by CHEMPAK on the chemical stability, compatibility or miscibility of the selected components or the defined mixture. The user must be satisfied as to the physical reasonableness of the formulation of the mixture.

The accuracy of the calculated properties can be expected to be better when the selected components are chemically similar to each other. The accuracy of the calculated properties of very dissimilar

components is not known but may only be order-of-magnitude in some cases. It is to be expected that the errors involved with mixtures of non-polar compounds will be less than with mixtures of polar compounds.

The range of applicability of the correlations is as follows:

A low-limit temperature is defined corresponding to the maximum of the reduced melting points of the mixture components. Properties cannot be accessed at temperatures lower than this. A high-limit temperature of 1300 K or the pseudocritical temperature (whichever is the greater) is defined for vapor properties. Vapor properties cannot be accessed at temperatures greater than this. The high limit temperature for liquid mixture properties is the pseudocritical temperature of the mixture as defined by the Lee-Kesler rules. Liquid properties cannot be accessed at temperatures higher than this.

### 8.13.2 Mixture Critical Properties

#### Critical Temperature

$$T_c = \text{Sum}(x_i V_{ci} T_{ci}) / \text{Sum}(x_i V_{ci})$$

where  $x_i$ ,  $V_{ci}$  and  $T_{ci}$  are the mole fraction, critical volume and critical temperature of component  $i$ . The reference used here is  $L_i$ . This method is generally recommended by Reid and by Danner and Daubert. Deviations for binary hydrocarbon mixtures can be expected to be less than 4 K. Errors for multicomponent hydrocarbon mixtures average about 11 K. Errors for binary mixtures of non-hydrocarbons would be of this order.

#### Critical Volume

$$V_c = \text{Sum}(a_i V_{ci}) + \text{Sum}(a_{ij} b_{ij}) \text{ (all } i, j)$$

$$a_i = x_i V_{ci}^{2/3} / \text{Sum}(x_i V_{ci})$$

$$b_{ii} = 0 \quad b_{ij} = 2c_{ij}(V_{ci} + V_{cj})$$

$$c_{ij} = A_{ij} + B_{ij}d_{ij} + C_{ij}d_{ij}^2 + D_{ij}d_{ij}^3 + E_{ij}d_{ij}^4$$

$$d_{ij} = \text{ABS}\{(V_{ci}^{2/3} - V_{cj}^{2/3}) / (V_{ci}^{2/3} + V_{cj}^{2/3})\}$$

The binary constants  $A_{ij}$  thru  $E_{ij}$  are given by,

Type	A	B	C	D	E
Aromatic-Aromatic	0	0	0	0	0
Aromatic-Aromatic with a Cycloalkane	0	0	0	0	0
Alkane-Aromatic	0.0753	-3.332	2.22	0	0
Alkane-Aromatic with CO <sub>2</sub> or H <sub>2</sub> S	-0.4957	17.119	-168.56	587.05	-698.89
All others	0.1397	-2.9672	1.8337	-1.536	0

The reference for this method is Schick and Prausnitz. An average error of 10% can be expected for binary mixtures.

#### Critical Pressure

$$w_m = \text{Sum}(x_i w_i)$$

$$T_{cm} = \text{Sum}(Z_i T_{ci})$$

$$Z_i = x_i V_{ci} / \text{Sum}(x_i V_{ci})$$

$$T_{cm}^* = \text{Sum}(x_i T_{ci})$$

$$P_{cm}^* = \text{Sum}(x_i P_{ci})$$

$$P_c = P_{cm}^* [1 + (5.808 + 4.93 w_m)(T_{cm}/T_{cm}^* - 1)]$$

This method is based on a simplification of the method of Kreglenski and Kay as amended by Spencer, Daubert and Danner. Errors in the application of this method average about one bar - see Reid et al for details. This method is not applied when the mixture contains inorganic components. In this case the critical pressure is taken equal to the Lee-Kesler pseudocritical pressure.

### **Acentric Factor**

In accordance with the Lee-Kesler rules,

$$w = \text{Sum}(x_i w_i)$$

$w_i$  = component acentric factor

$x_i$  = component mole fraction

### **8.13.8 Vapor Pressure**

Accurate determination of the vapor pressures of mixtures normally requires a full vapor-liquid equilibrium model with knowledge of interaction parameters. Such a VLE model is not a part of CHEMPAK. In CHEMPAK, however, an approximation is used in order to be able to determine the mixture phase for given input T and P. The Lee-Kesler mixing rules give for the acentric factor,

$$w_m = \text{Sum}(x_i w_i)$$

$$w_i = -\log(P_{vri} \text{ at } T_r = 0.7) - 1$$

$$w_m = -\log(P_{vrm} \text{ at } T_r = 0.7) - 1 = \text{Sum}(-x_i \log(P_{vri} \text{ at } T_r = 0.7) - x_i)$$

hence,

$$\log(P_{vrm} \text{ at } T_r = 0.7) = \text{Sum}(x_i \log(P_{vri} \text{ at } T_{ri} = 0.7))$$

The vapor pressure is represented for all compounds in the CHEMPAK database by the Wagner equation:

$$\ln(P_{ri}) = (a_i X_i + b_i X_i^{1.5} + c_i X_i^3 + d_i X_i^6) / T_{ri}$$

$$X_i = 1 - T_{ri}$$

Hence, for the mixture, the Wagner coefficients are given by,

$$a_m = \text{Sum}(x_i a_i) \quad b_m = \text{Sum}(x_i b_i)$$

$$c_m = \text{Sum}(x_i c_i) \quad d_m = \text{Sum}(x_i d_i)$$

where  $x_i$  is the component mole fraction.

### **8.13.9 Thermodynamic Properties of Mixtures**

The Lee-Kesler and Wu & Stiel equations of state are used to calculate the thermodynamic properties of liquid and vapor phases for all pure compounds in the database. To calculate the thermodynamic properties of mixtures, a set of pseudocritical constants are defined for each mixture and the properties are calculated in the usual way.

CHEMPAK uses the original Lee-Kesler rules as follows:



$$Z_{ci} = 0.2905 - 0.085 w_i$$

$$V_{ci} = Z_{ci} R_0 T_{ci} / P_{ci}$$

$$V_{cm} = \text{Sum}(x_{ij}(V_{ci}^{1/3} + V_{cj}^{1/3})^3) / 8$$

$$T_{cm} = \text{Sum}(x_{ij}(V_{ci}^{1/3} + V_{cj}^{1/3})^3 (T_{ci} T_{cj})^{1/2}) / (8 V_{cm})$$

$$w_m = \text{Sum}(x_i w_i)$$

$$M_m = \text{Sum}(x_i M_i)$$

$$Z_{cm} = 0.2905 - 0.085 w_m$$

$$P_{cm} = R_0 Z_{cm} T_{cm} / V_{cm}$$

$$R_0 = \text{Gas Constant}$$

The Stiel polarity factor for a polar mixture is computed from the mixture liquid specific volume

## 8.13a Thermodynamic And Transport Properties Of Mixtures

### 8.13.10 Vapor Viscosity

The pure component equations are used with the following pseudocritical constants defined for each mixture per the formulation of Prausnitz and Gunn:

$$T_{cm} = \text{Sum}(x_i T_{ci})$$

$$Z_{cm} = \text{Sum}(x_i Z_{ci})$$

$$V_{cm} = \text{Sum}(x_i V_{ci})$$

$$M_m = \text{Sum}(x_i M_i)$$

$$P_{cm} = Z_{cm} R_0 T_{cm} / M_m V_{cm}$$

### 8.13.11 Vapor Conductivity

The pure component equations are used with the following pseudocritical constants from Yorizane et al defined for the mixture:

$$T_{cm} = \text{Sum}(x_{ij}(V_{ci}^{1/3} + V_{cj}^{1/3})^3 (T_{ci} T_{cj})^{1/2}) / (8 V_{cm})$$

$$V_{cm} = \text{Sum}(x_{ij}(V_{ci}^{1/3} + V_{cj}^{1/3})^3)$$

$$w_m = \text{Sum}(x_i w_i)$$

$$Z_{cm} = 0.2905 - 0.085 w_m$$

$$P_{cm} = Z_{cm} R_0 T_{cm} / M_m V_{cm}$$

$$M_m = \text{Sum}(x_i M_i)$$

### 8.13.12 Ideal-Gas Mixture Properties

Ideal-gas mixture properties are taken as the mole fraction of the pure component properties. The pure component properties are defined per mole of substance.

## 8.14 Notation

C Specific Heat

e Expansion Coefficient  
h Enthalpy  
log Logarithm to base 10  
ln Natural Logarithm  
m Dipole Moment  
M Molecular Weight  
P Pressure  
R Gas Constant  
r Riedel Parameter  
s Entropy  
T Temperature  
v Viscosity  
V Specific Volume  
w Acentric Factor  
x Mole Fraction  
Y Wu & Stiel Polarity Factor  
Z Compressibility

***Subscripts***

b Boiling  
c Critical  
es Equation of State  
f Freezing  
ig Ideal Gas  
L Liquid  
m Mixture  
0 Low Pressure  
p Constant Pressure  
ra Rackett  
ref Reference  
r Reduced  
s Saturated  
v Vapor  
v Constant Volume

***Superscripts***

(o) Simple Fluid

(r) Reference Fluid

(p) Polar Fluid

## 8.15 References

- API Technical Data Book - Petroleum Refining, 4 Vols, API, Washington DC, 1988
- CRC Handbook of Chemistry and Physics, CRC Press Boca Raton 1991
- Danner R P and Daubert T E , Data Prediction Manual, Design Institute for Physical Property Data , AIChE, NY 1983
- Daubert T E and Danner R P, Physical and Thermodynamic Properties of Pure Chemicals, Data Compilation, AIChE, Hemisphere NY 1989
- Engineering Sciences Data Units (ESDU), (9 Vols Data Compilation), London, England
- Gomez-Nieto M and Thodos G, Ind Eng Chem Fundam, Vol 17, 45, 1978, Can J Chem Eng, Vol 55, 445, 1977, Ind Eng Chem Fundam, Vol 16, 254, 1977
- Hankinson and Thomson, AIChEJ, Vol 25, 653, 1979
- International Critical Tables, National Research Council, 7 Vols, McGraw-Hill, NY 1926
- Joback K G, SM Thesis, MIT, June 1984
- Keenan et al, Steam Tables, John Wiley, NY 1978
- Knapp et al, Chem Data Ser, Vol VI, DECHEMA 1982
- Kreglewski and Kay, J Phys Chem, Vol 73, 3359, 1969
- Letsou A and Stiel L I, AIChEJ, Vol 19, 409, 1973
- Le Bas G, Molecular Volumes of Liquid Chemical Compounds, Longmans Green, NY 1915
- Lee B I and Kesler M G, AIChEJ, Vol 21, 510, 1975
- Li C C, AIChEJ, Vol 22, 927, 1976
- Lucas K, Chem Ing Tech, Vol 53, 959, 1981
- Missenard F A, Rev Gen Thermodyn, Vol 101, 649, 1970
- Morris P S, MS Thesis, Polytech Inst Brooklyn, NY 1964
- Perry et al , Chemical Engineer's Handbook (various editions), McGraw Hill, NY
- Plocker U J et al, Ind Eng Chem Proc Des Dev, Vol 17, 324, 1978
- Prausnitz and Gunn, AIChEJ, Vol 4, 430 and 494, 1958
- Reid R C et al, Properties of Liquids and Gases, 3rd Ed, McGraw Hill, NY 1977, 4th Ed, McGraw Hill, NY 1987
- Schick and Prausnitz, AIChEJ, Vol 14, 673, 1968
- Spencer and Danner, J Chem Eng Data, Vol 17, 236, 1972
- Spencer, Daubert and Danner, AIChEJ, Vol 19, 522, 1973
- Stiel and Thodos, AIChEJ, Vol 8, 229, 1962

Teja A S et al, Chem Eng Sci, Vol 33, 609, 1978, AIChEJ, Vol 26, 337 & 341, 1980, Chem Eng Sci, Vol 36, 7, 1981, Ind Eng Chem Fundam, Vol 20, 77, 1981, Chem Eng Sci, Vol 37, 790, 1982, J Chem Eng Data, Vol 28, 83, 1983, Ind Eng Chem Proc Des Dev, Vol 22, 672, 1983

Thomson, Brobst, Hankinson, AIChEJ, Vol 28, 671, 1982

Tyn M T and Calus W F, Processing, Vol 21(4), 16, 1975

Van Velzen D et al, Ind Eng Chem Fundam, Vol 11, 20, 1972

Van Velzen et al, Liquid Viscosity etc, Euratom 4735e, Joint Nuclear Research Centre, ISPRA Establishment, Italy 1972

Vargaftik N B, Tables on Therm Props Liq & Gases, 2nd Ed, Hemisphere, Washington DC, 1975

Wu G Z A and Stiel L I, AIChEJ, Vol 31, 1632, 1985

Yorizane et al, Ind Eng Chem Fundam, Vol 22, 454, 1983

## Role of Pressure Junctions - Detailed Discussion (Long)

In modeling a generalized pipe network, it is possible to construct models that do not have a unique solution. A common occurrence of this is when a model contains one or more sections which are completely bounded by known flow rates. A series of examples with accompanying explanation will clarify situations where this occurs. The length of this topic is proportional to the frequency with which engineers get confused over the issue. It became apparent that an exhaustive, lengthy discussion was necessary for some engineers. Do not dismiss this topic too quickly because of its length.

Before beginning it will be helpful to consider an aspect of the philosophy of computer modeling. At the risk of stating the obvious, it must be recognized by the user that a computer model cannot calculate anything that cannot in principle be calculated by hand. All the computer does is accelerate the calculation process. Some users expect the model to generate independent information and become frustrated when the software requests additional information from them. But if the user was doing the calculation by hand that same information would still be required. The difference is that in the case of the hand calculation, the engineer would be forced to think through why the information is needed.

However, when the engineer interacts with AFT xStream, they do not need to think at the same depth as in the hand calculation. So when AFT xStream asks for the additional information, it is not as apparent why it is required. This disconnect in thinking can cause frustration for the user. Fully understanding the concepts in this topic will greatly reduce the frustration for some users, and will improve the quality of models for all users.

## Examples

The simplest example is the system shown in Figure 1. In AFT xStream terms, the system has two assigned flow junctions.



**Figure 1: Model with two assigned flows. This model does not have a unique solution.**

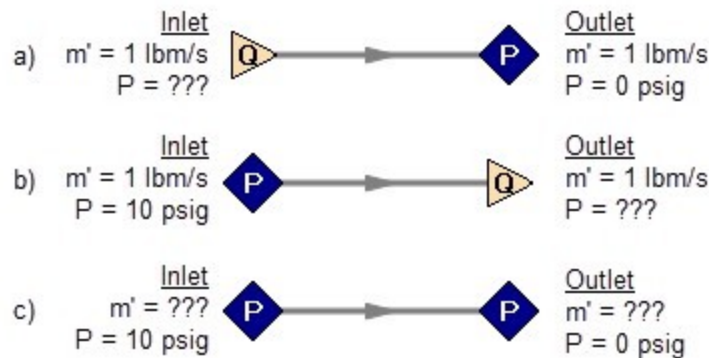
Obviously, the flow in the pipe is known. But what is the pressure at the inlet? At the outlet? It cannot be determined because there is no *reference pressure*. The reference pressure is that pressure from which other pressures in the system are derived. There can be one or more reference pressures, but there always has to be at least one.

The model in Figure 1 can be built with AFT xStream and if you try and run this model, AFT xStream will inform you that it cannot run it because of the lack of a reference pressure. In AFT xStream there are five junctions that can act as a reference pressure: the tank, assigned pressure, control valves with pressure setpoints, and valve and relief valve after cracking (with the exit valve option).

There are a couple other things worth noting about Figure 1. First is that the model in Figure 1 has redundant boundary conditions. When one flow is specified in a single pipe, the other end must have the same flow. Thus the second known flow does not offer any new information. Because the conditions are redundant, there is no unique solution to the model in Figure 1. In any AFT xStream model, there is always at least one unknown for each junction in the model – either pressure or flow rate. Sometimes there are two unknowns such as a compressor/fan. It is possible the user may not know either the flow or the pressure at the compressor/fan. In such cases the user is required to specify the relationship between flow and pressure. This relationship is called a compressor/fan curve, and by specifying the relationship between the two, effectively one of the two unknowns can be eliminated. Thus we always end up with one unknown for each junction.

Second, if the user was allowed to specify two flows as in Figure 1, they could specify them with different flow rates. Clearly they must have the same flow, so an inconsistency occurs. The basic reason the inconsistency is possible is because Figure 1 does not have a unique solution.

Figure 2 a-c shows three other model configuration possibilities. It is not possible to specify inconsistent conditions for any of the Figure 2 models, and they always have a unique solution no matter what input is specified by the user.



**Figure 2: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.**

The four models in Figures 1 and 2 a-c contain all logical possibilities. In all four cases there are four things we want to know. The pressure and flow at the inlet, and the pressure and flow in the outlet. In each case we know two of these four. The problem is that in the Figure 1 model the lack of a reference pressure makes it impossible to determine the inlet and outlet pressures, even though we know the flow. The three models in Figure 2 a-c all have at least one pressure and thus all four of the desired parameters can be determined. Table 1 summarizes this. Also note in Table 1 that in the Figure 2c case the flow can be determined, but it is by iteration.

**Table 1: Four logical possibilities of boundary conditions for single pipe systems**

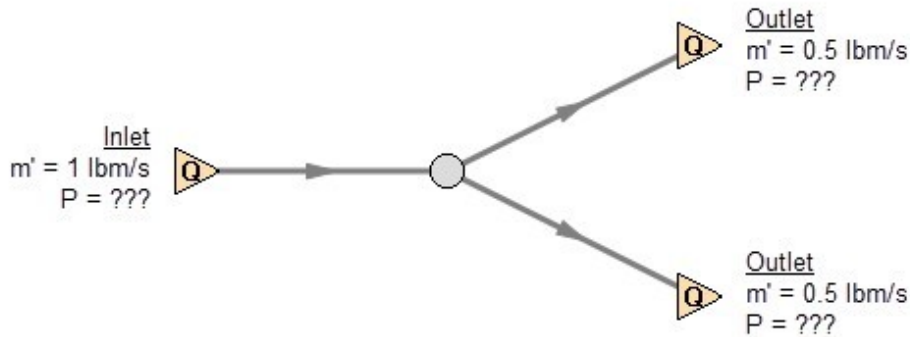
Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 1		X		X	NO	No reference pressure, cannot calculate pressures
Figure 2a		X	X		YES	1 known flow, can calculate everything directly
Figure 2b	X			X	YES	1 known flow, can calculate everything directly
Figure 2c	X		X		YES	2 pressures, can iterate for flow

As we consider multi-pipe systems, there are a host of other possibilities that present themselves. All other configuration possibilities which lack a reference pressure ultimately boil down to the same problem that exists in Figure 1.

The pipe lengths, diameters and friction factors are not included in the remaining models because they do not influence the main point of this topic. It is assumed that these parameters can be obtained for each pipe and the resulting pressure drop calculated with standard relationships.

Consider the model in Figure 3. With three boundaries having a known flow, this clearly has the same problem as the model in Figure 1. No unique pressures at any location can be calculated.

A pressure at any one of the three boundaries in Figure 3 would be sufficient to allow a unique solution of the system. It is also possible that there could be two pressures and one flow, or three pressures. As long as there is one pressure a unique solution exists.

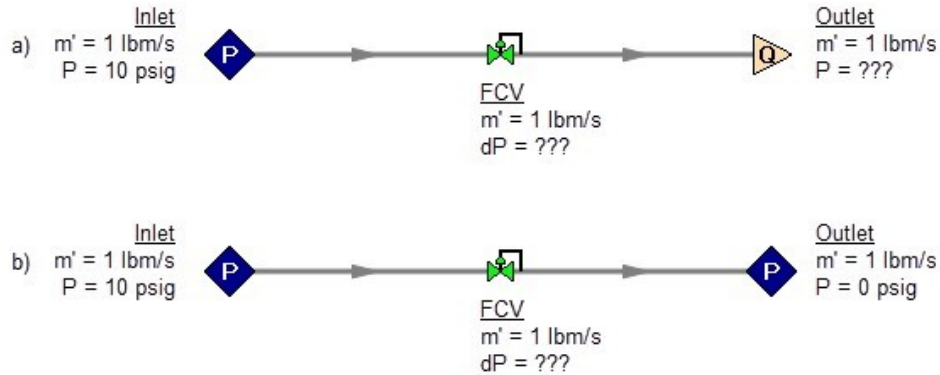


**Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.**

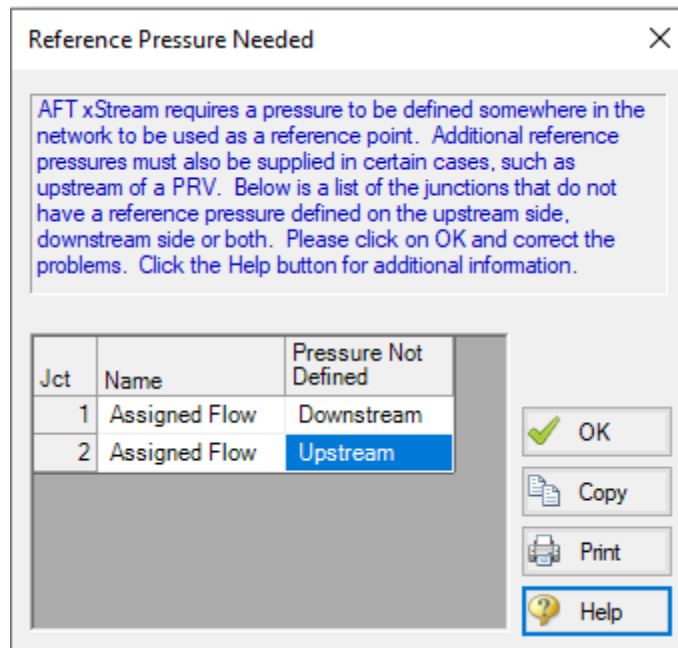
Now consider the models in Figure 4. The Figure 4a model does have one reference pressure, but in this case there is still a problem. The Flow Control Valve (FCV) is controlling the flow to 10 gpm, and the downstream flow is demanding 10 gpm. The section of the system before the FCV can be solved (because there is a pressure upstream), but the section after the FCV does not have a unique solution. Why? In order to obtain a unique solution, the pressure drop across the FCV must be known. But any pressure drop could exist and satisfy the conditions of this model. It is thus not possible to determine the pressure at the outlet flow demand because it depends on the FCV pressure drop which is not known.

The model in 4b does have a pressure downstream of the FCV, and thus there is a unique pressure drop across the FCV and a unique solution exists for the model in Figure 4b.

If you input the remaining data for the Figure 4a model and run it in AFT xStream, you will get the message shown in Figure 5. AFT xStream identifies the part(s) of the model where a known pressure is needed.



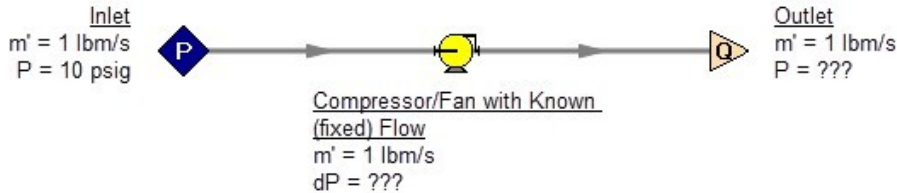
**Figure 4a-b.:** The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists.



**Figure 5:** AFT xStream message when you try to run the model shown in Figure 4a.

## Sizing Compressors or Fans with Flow Control Valves

An analogous situation to Figure 4a is when the user is trying to size a compressor using the Compressor/Fan with a fixed flow. The Compressor/Fan with a fixed flow is really just a reverse FCV. It controls the flow by adding pressure, rather than reducing it. Figure 6 shows this case and is another example of a model without a unique solution, requiring more than one pressure junction. If the user instead entered a compressor or fan curve for this model, it could be solved.



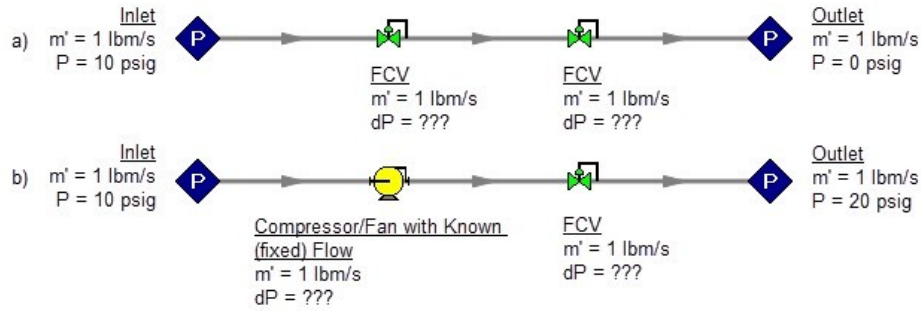
**Figure 6: A compressor/fan modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists**

Increasing in model complexity, Figure 7 shows some additional examples of models without a unique solution. In particular, Figure 7b has a compressor/fan with a fixed flow that can add as much pressure as it wants, and the downstream FCV can then take out as much as it wants. There are an infinite number of possible solutions.

While it would be highly unusual for a system to have two FCV's in series, it is worth considering the model in Figure 7b further. It is common to have an FCV in series with a compressor/fan. How does one size the compressor/fan in such a case? There are several ways to do this, one of which is to change the compressor/fan from a fixed flow to an actual compressor or fan curve. Trying multiple curves will guide you to the best compressor or fan.

But there is a better way. Consider for a moment the pressure drop across the FCV in Figure 7b. In the installed system, is any pressure drop acceptable? Usually not. It is typical to have pressure drop limits based on the system design and the FCV itself. A common requirement is a minimum pressure drop across the FCV. If no minimum pressure drop exists, let's say we choose a compressor or fan that results in the pressure drop across the FCV being 0.01 psid. If the system is built and the compressor or fan slightly underperforms its published curve, the FCV will not be able to control to its flow control point. Even if the compressor or fan does operate exactly on the published curve, eventually fouling in the pipes will result in increased pipe resistance (and pressure drop) and again the FCV will not be able to control to its flow control point.

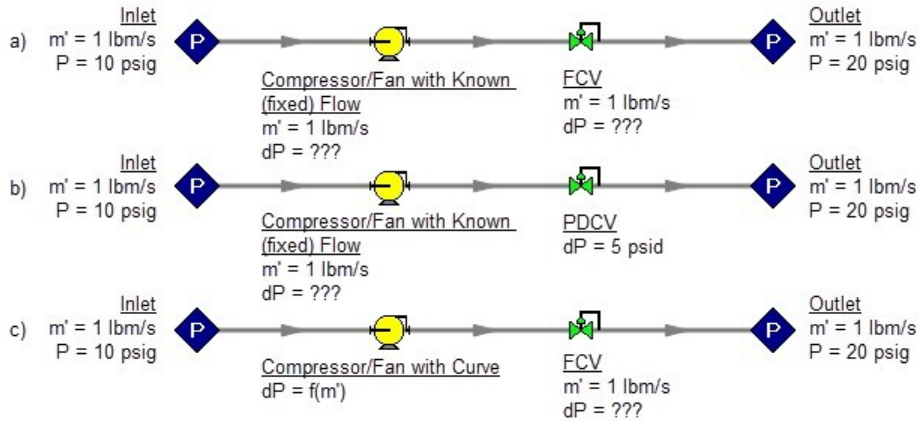




**Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a compressor/fan modeled as a fixed flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions must be changed from a flow controlling device.**

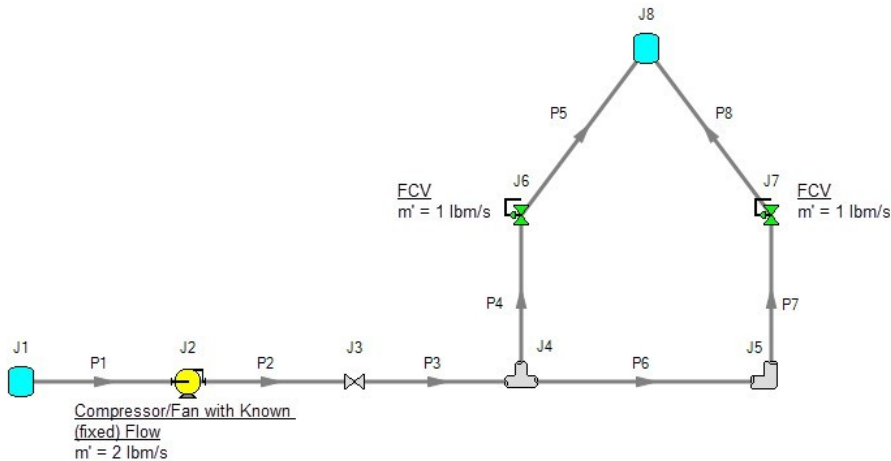
To avoid these kinds of problems in installed systems, a minimum Cv or pressure drop across the FCV is typically required. This minimum pressure drop provides the key to sizing the compressor/fan in Figure 7b. Rather than model the FCV as a flow control valve, instead model it as a pressure drop control valve (PDCV) set to the minimum pressure drop requirement. Then a unique solution exists, the model will run and the compressor/fan can be sized. Once the compressor/fan is sized and an actual curve exists, the curve can be entered and the valve can be switched back to an FCV. Remember that when we use the PDCV, we are still getting the flow we want for the true FCV, because the compressor/fan is now providing the control.

Think for a moment what we just did. We went through a thought process that many engineers have gone through during their hand calculations. Specifically, we size the compressor/fan such that the FCV pressure drop is minimized. If the FCV has a larger pressure drop than the minimum, the compressor/fan must add more pressure to overcome this excess pressure drop. In short, it must use more energy and is thus less efficient. Figure 8 depicts this process.



**Figure 8a-c:** The model at the top is the same as Figure 7b and does not have a unique solution. To size the compressor/fan, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the compressor/fan is sized, a curve exists, and the model at the bottom can be run using an FCV.

Let's make a slight addition in complexity to the model in Figure 8. In this case we have two flow control valves in parallel (Figure 9). To size this compressor/fan, we apply the process described in Figure 8. We choose one of the FCV's, make it a PDCV, size the compressor/fan, choose a compressor/fan with an actual curve, enter the curve into the model, and then return the PDCV to an FCV.



**Figure 9:** A compressor/fan with a fixed flow in series with two parallel FCV's. The model in its current form does not have a unique solution. To size the compressor/fan, first change the FCV at J7 to a PDCV.

This brings up the question, which FCV should one choose to turn into a PDCV? When FCV's are put in parallel, frequently the pipe design has one of the FCV's further away from the compressor/fan than the others. Because of the additional piping leading to this FCV, it will be the weakest link in the chain of parallel FCV's by virtue of having the lowest pressure drop across it. The most remote FCV should be chosen as the PDCV. If the most remote FCV is chosen, when the minimum pressure drop required is

applied to the PDCV all other FCV's in the parallel system will have greater than the minimum pressure drop — thus satisfying the pressure drop requirement for those FCV's as well.

What if you do not know which FCV is the most remote? In this case make your best guess, change it to a PDCV at the minimum required pressure drop, run the model, then verify whether all other FCV's have a pressure drop that meets or exceeds the requirement. If not, then the FCV with the smallest pressure drop as determined by the first run is in reality the weakest FCV. Choose this FCV, change it to a PDCV, and then change the original PDCV back to an FCV since it is not the weak link. If the pipe system leading to each FCV is truly identical and all FCV's have identical pressure drops, any of the FCV's will serve as the PDCV.

The preceding process can be extended to cases where there are three or more FCV's in parallel, and also cases with more than one compressor/fan in parallel supplying the FCV's.

In Figure 9 the most remote valve is J7, the one on the right. If the minimum pressure drop is 5 psid, change J7 to a PDCV with a 5 psi drop. This results in the output shown in Figure 10a. Notice how the pressure drop across the other FCV at J6 is greater than 5 psid because it is closer to the compressor/fan. Also notice how the flow through J7 is still 1 lbm/s even though it is not controlling flow. Once again, the reason for this is that the compressor/fan is controlling the flow to 2 lbm/s, and with the J6 FCV controlling to 1 lbm/s, the excess must go through J7. The compressor/fan is thus sized and its pressure rise is shown in Figure 10b (59.83 psid).

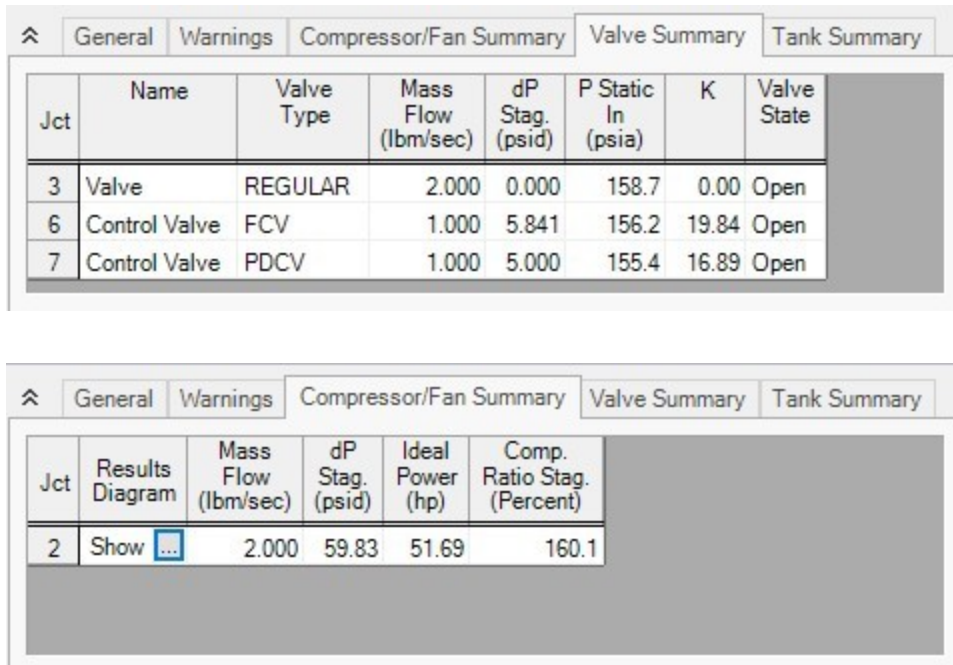


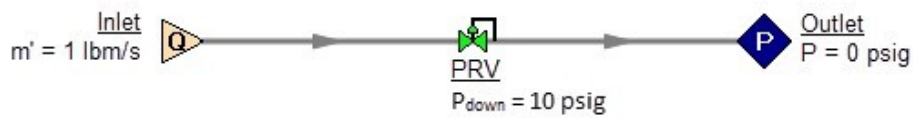
Figure 10a-b: Results from changing J7 in Figure 9 to pressure drop control valve

AFT xStream results for the model in Figure 9 after J8 has been changed to a PDCV. Top results show valve summary and bottom results show compressor/fan size.

## Pressure Control Valves

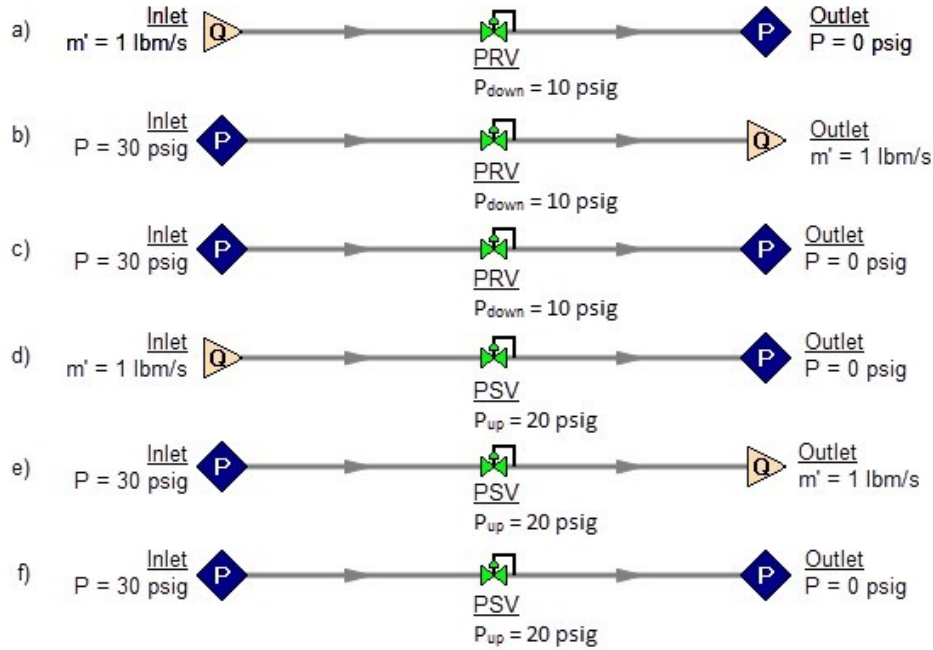
The problem of non-unique solutions can also occur when a Pressure Reducing Valve (PRV) or Pressure Sustaining Valve (PSV) is used. A PRV is used to control pressure downstream of the valve, while a PSV controls pressure at the valve inlet. One thing that PRVs and PSVs have in common with FCVs is that the pressure drop cannot be known ahead of time. The pressure drop depends on the balance of the pipe system.

Consider the simple system shown in Figure 11. With the pressure controlled downstream of the PRV, the pressure upstream is not known. Without knowledge of the PRV's pressure drop, the pressure upstream of the PRV cannot be determined and thus no unique solution exists.



**Figure 11: If a PRV is downstream of a known flow the PRV pressure drop cannot be calculated. This model does not have a unique solution.**

Figure 12 shows the possible cases with PRVs and PSVs, while Table 2 comments on the six cases. In summary, at least one known pressure is always needed on the side of a pressure control valve opposite of the controlled side.



**Figure 12a-f.: Six possibilities for PRV and PSV configurations. Cases A and E do not have unique solutions. See Table 2 for comments.**

**Table 2: Summary of six possibilities for PRV and PSV configurations.**

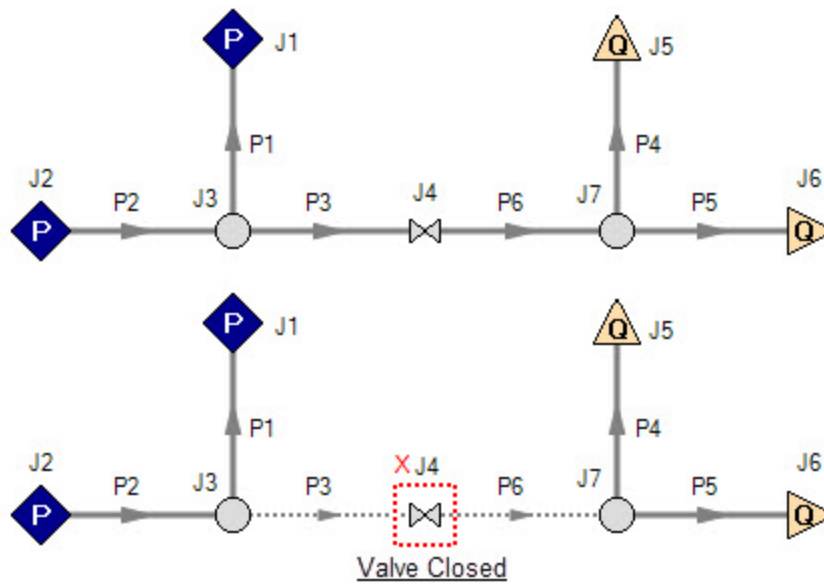
Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 12a - PRV		X	X		NO	No reference pressure upstream of PRV, cannot calculate
Figure 12b - PRV	X			X	YES	1 known flow, can calculate everything directly
Figure 12c - PRV	X		X		YES	2 pressures, can iterate for flow
Figure 12d - PSV	X	X	X		YES	1 known flow, can calculate everything directly
Figure 12e - PSV	X			X	NO	No reference pressure downstream of PSV, cannot calculate
Figure 12f - PSV	X		X		YES	2 pressures, can iterate for flow

If a compressor/fan is being sized in series with a PRV or PSV, modeling the compressor/fan as an assigned flow will not permit a unique solution if the compressor/fan is upstream of a PRV or downstream of a PSV. The reasons for this are the same as those in the previous section on FCVs.

In combination with the previous examples, it should be apparent how the non-unique solution problem can occur in more complicated systems. In all such cases, the reasoning will boil down to the same problems already discussed.

### Closing Parts of a System

A model may be built which has sufficient information to obtain a unique solution. Then a user closes a junction and all of a sudden the problem of no unique solution occurs. This is shown in Figure 13. The top model has a unique solution and will run fine. In the bottom model the valve is turned off and the section of the system downstream of the valve has only known flows. No reference pressure exists in this section because it is isolated from the known pressures in the section on the left.



**Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.**

### Summary

AFT xStream models always require one reference pressure, and may require more if the engineer uses control valves, compressor/fans with assigned flows, or closes pipes or junctions. The actual number of reference pressures needed depends on the model, but AFT xStream always checks to be sure sufficient reference pressures exist and stops to warn the user when they do not.

This capability is a powerful diagnostic feature that will help guide the user in building meaningful models.

**Related Blogs**

---

[Reference Pressure Needed: Where It Comes from and How to Address it](#)

## Critical Warnings, Warnings, and Cautions

A number of warnings may be given in the [output](#) when you run an AFT xStream model. These warnings are intended to alert you about potential problems with the model or solution. There are different warning levels depending on the severity, but all warning types should be reviewed when they are displayed in the output.

1. [CRITICAL WARNING](#) – Very important and reflects a high possibility of incorrect results in the output. Output with critical warnings should be used only for error resolution purposes and never for engineering decisions.
2. [WARNING](#) – A warning offers important information about the converged solution or the model itself. The information should always be reviewed.
3. [CAUTION](#) – A caution message is of less importance than a full warning. The included information may or may not be of interest to the user, but is intended to alert the user to potential issues with the model or results.

Any of the warning message types above can be double-clicked in the Output window to take you directly to the pipe/junction referenced in the message. Whenever any warnings occur that are of the WARNING or CRITICAL WARNING level, the General Results section of the Output Window will be displayed with the text in red.

The above links will take you to a list that includes most of the warnings you may receive, with a brief explanation given for each.

### Related Topics

---

Use the Model Data Window.....	521
Solution Method Panel.....	104
Simulation Mode/Duration Panel.....	98

## Critical Warnings

A Critical Warning message in AFT xStream alerts users to serious issues in the model results. These messages should always be reviewed by the user, and results containing these messages should never be used for anything other than error resolution purposes.

### CRITICAL WARNINGS

---

- [Characteristic speed exceeded maximum determined from...](#)
- [Global energy balance disabled](#)
- [Global mass balance disabled](#)
- [Junction X Had Reverse Flow - Compressor/Fan Head Could Not Be Predicted](#)
- [Maximum transient pipe Mach number X higher than estimate Y](#)
- [Pipe Mach number exceeded 1 during the simulation.](#)
- [Results do not represent a converged solution](#)



- [Required flow to supply the pressure control valve at junction X is above sonic flow](#)
- [Setting relaxation lower than relative pressure/flow tolerance gives unreliable results](#)
- [Specified flow through/out of junction X is above sonic flow and cannot be supplied](#)

## Characteristic speed exceeded maximum determined from...

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*The characteristic speed exceeded the maximum determined from estimated maximum pipe temperature and Mach number at X seconds, Pipe Y, Section Z.*

This critical warning indicates that at some time during the simulation the [characteristic speed](#) exceeded the maximum characteristic speed, which is equivalent to the [MOC grid section length](#) divided by the time step. If this happens the solver in xStream will attempt to interpolate properties outside the MOC characteristic grid, which may introduce instability and inaccuracies to the solution.

The Estimated Maximum Pipe Temperature During Transient and/or the Estimated Maximum Pipe Mach Number During Transient in the [Sectioning panel](#) should be increased to increase the maximum characteristic speed.

For more information about the MOC solution, please refer to the [Pipe Sectioning](#) and [Method of Characteristics](#) pages. This warning message is generated when points R or L are outside of points 1 or 3 in Figure 1 on the Pipe Sectioning page.

## Related Topics

---

Method of Characteristics ..... 435

## Global Energy Balance Disabled

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Global energy balance disabled by user in Solution Control window*

This critical warning message alerts users they have disabled the Global Energy Balance option in the [Advanced Parameters Panel](#). Results shown with this critical warning do not represent a valid converged solution.

## Global Mass Balance Disabled

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Global mass balance disabled by user in Solution Control window*

This critical warning message alerts users they have disabled the Global Mass Balance option in the [Advanced Parameters Panel](#). Results shown with this critical warning do not represent a valid converged solution.

## **Junction X Had Reverse Flow Compressor Head Could Not Be Predicted**

A [Critical Warning message](#) in AFT xStream

Full Message text is:

*The Compressor Junction X had reverse flow during the transient. The head/pressure rise was estimated from the zero flow value.*

This critical warning message displays for [centrifugal compressor](#) when reverse flow occurs at the compressor and the Manufacturer's Curve is being used for calculations. The compressor head rise under such conditions is unknown. The zero flow value is used.

## **Maximum transient pipe mach number X higher than estimate Y**

A [Critical Warning message](#) in AFT xStream

Full Message text is:

*The maximum transient pipe Mach number (X) was higher than the estimate (Y). Results should not be considered accurate until the simulation is re-run with an estimated maximum pipe Mach number higher than the actual transient maximum.*

The Mach number at some location in the pipes exceeded the Estimated Maximum Pipe Mach Number During Transient, which is defined in the [Sectioning panel](#) of Analysis Setup. This can cause instability in the solver and significant error in the results. The model should be re-run with the Estimated Maximum Pipe Mach Number During Transient set to at least 0.1 greater than the maximum pipe mach number encountered during the simulation.

## **Mach Number Exceeded 1 During Simulation**

A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Pipe Mach number exceeded 1 during simulation.*

AFT xStream is limited to sonic and subsonic flows, and cannot model flows exceeding [Mach 1](#). However, the [MOC calculations](#) during the MOC Transient simulation can on occasion calculate

velocities slightly above 1. This critical warning is shown by default when the Mach number in the pipes exceeds 1.0001. Users should examine results with this critical warning carefully, and recognize that results after Mach 1 is exceeded may be in error.

## Results Do Not Represent Converged Solution

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*These results do not represent a converged solution. Use only for error resolution purposes.*

This message alerts users that results are displayed for an unconverged solution. The displayed results should only be used for purposes of determining the reasons for nonconvergence. Under no circumstances should these results be used for engineering purposes.

## Required flow to supply the pressure control valve at junction X is above sonic flow

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Required flow to supply the pressure control valve at Junction X is above sonic flow and cannot be supplied. Please review results and change your model.*

This critical warning message means the user has specified a pressure at a [pressure control valve](#) which requires a flow which is physically impossible because of [sonic choking](#). The set point should be changed to allow physically realistic flows.

## Setting relaxation lower than relative pressure/flow tolerance gives unreliable results

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Setting relaxation lower than relative pressure(flow) tolerance gives unreliable results.*

Setting the relaxation to a value close to or lower than relative tolerance can lead to false convergence and invalid results. AFT xStream alerts the user to this when inputting data in the [AFT Arrow Steady Solution Control panel](#), and also alerts the user once the solution has been obtained.

## Specified flow through/out of junction X is above sonic flow and cannot be supplied

### A [Critical Warning message](#) in AFT xStream

Full Message text is:

*Specified flow through (out of) Junction X is above sonic flow and cannot be supplied. Please review your results and change your model.*

This critical warning message means the user has specified an outflow which is physically impossible because of [sonic choking](#). This message can also occur if [Assigned Flow junctions](#), [Flow Control Valves](#), or [Compressor/Fans](#) modeled as a fixed flowrate are used.

## Warnings

A Warning message in AFT xStream alerts users to issues in the model that will affect the model results. These warnings often offer important information about the converged solution or the model itself, and should always be reviewed before considering the results to be meaningful for the system.

## WARNINGS

---

- [Automatic sectioning did not achieve the desired steady conservation error.](#)
- [Chempak temperature and/or pressure is out of range for the following](#)
- [Control valve open percentage outside valid range](#)
- [The difference in velocity exceeds the specified percent](#)
- [Energy may not be balanced at Junction X](#)
- [FCV could not control flow at Junction X](#)
- [Flow may not be balanced at Junction X](#)
- [Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy](#)
- [Junction X Exceeded End-of-Curve Flow Rate...](#)
- [Junction X had reverse flow during the transient](#)
- [Maximum system pressure exceeded enthalpy table's maximum pressure of X](#)
- [Maximum transient pipe temperature higher than estimate](#)
- [Maximum transient pipe temperature significantly lower than estimate](#)
- [MOC Steady solution did not reach a nominally steady state before transient](#)
- [Negative added pressure by compressor at Junction X](#)
- [Physical flow area is less than the sonic area at junction X...](#)
- [Pipe/Junction X property calculation exceeded maximum local iteration limit](#)
- [Pipe/Junction X property calculation failed to continue towards convergenc](#)
- [Pipe X temperature is outside temperature limits in AFT Arrow Steady Solution Control](#)

- [Pipe X temperature outside valid temperature range](#)
- [PRV could not control pressure at Junction X](#)
- [PSV could not control pressure at Junction X](#)
- [Temperature is below the saturation line for the following. . .](#)
- [Transient choking exists at junction X. The specified flow or pressure could not be met.](#)
- Transient data was extrapolated at Junction X
- [Transient temperature was below fluid minimum temperature](#)
- [Transient temperature was below saturation temperature](#)
- [Unique solution not possible - Relief valve cycles](#)

## Automatic Sectioning Did Not Achieve Steady Conservation

A [Warning message](#) in AFT xStream

Full Message text is:

*Automatic Sectioning did not achieve the desired Steady Conservation Error*

Automatic Sectioning is a tool found in the [Sectioning panel](#) of the [Analysis Setup window](#) when Advanced Options are shown. This tool attempts to add pipe sections until the conservation error between the AFT Arrow Steady and MOC Steady Solutions is below the allowable error. This warning is shown if the Automatic Sectioning is unable to meet the allowable error. The allowable error may be exceeded if the maximum number of sections is reached.

## Chempak Temperature/Pressure Outside Valid Range

A [Warning message](#) in AFT xStream

Full Message text is:

*Chempak temperature and/or pressure is above/below valid range for the following:*

The final solution had temperature and/or pressure conditions in the specified pipes obtained with [Chempak](#) physical property data which were beyond Chempak's range of applicability. The range value is also displayed. This warning indicates the results may not be valid. It also may indicate an input error in the model because of the extreme temperature or pressure state.

## Control Valve Open Percentage Outside Valid Range

A [Warning message](#) in AFT xStream

Full Message text is:

*Control Valve open percentage outside valid range of 0-100% at Junction X*

This message is a warning that a [control valve](#) which includes open percentage data is operating outside the range of 0-100%. In many cases, it is likely that the user has not defined open percentages outside of the valid range, but instead has entered data that might allow xStream to extrapolate values outside that range.

## Difference in Velocity Exceeds Specified Percent

### A [Warning message](#) in AFT xStream

Full Message text is:

*The maximum difference between the AFT Arrow Steady solution and the MOC Steady Solution for velocity is X%*

The [MOC Steady Solution](#) is used to transition from the [AFT Arrow Steady](#) results to the MOC Transient simulation, and some change in system velocities is expected. This difference is checked prior to the [MOC Transient](#) simulation. The user can define an allowable change between the two steady solutions on the MOC Steady Solution panel, with a default value of 10%.

Pipes with initial velocities less than 10 ft/sec are not included in this check as a small absolute change in velocity can be a large relative change in velocity.

## Energy may not be balanced at Junction X

### A [Warning message](#) in AFT xStream

Full Message text is:

*Energy may not be balanced at Junction X (total = Y)*

This warning alerts users that the converged energy balance results may be incorrect for the AFT Arrow Steady Solution. Once a converged solution is obtained, AFT xStream checks all junctions to ensure an energy balance exists. It compares the imbalance to the mass flow weighted enthalpy [tolerance](#), and when the imbalance exceeds the tolerance, this warning is generated.

If the magnitude of the energy imbalance is close to the tolerance specified, it is generally safe to ignore this warning, although it may make sense to reduce tolerances further and rerun the model.

If the magnitude of the energy imbalance is large, the converged solution that may be incorrect. When a large imbalance exists, it may be an indicator the model is not realistic.

These warnings should always be reviewed in detail, and the user should perform hand calculations on the energy balance at the affected junctions to verify whether an acceptable solution exists.

---

**Note:** This warning will usually occur when you assign isothermal boundary conditions to pipes in your model. If pipes are forced to flow at the isothermal temperature specified, it is usually not possible to balance energy at the junctions. The warning can be ignored in such circumstances, because the energy imbalance is a natural result of the isothermal assumption.

---

## FCV could not control flow at Junction X

### A [Warning message](#) in AFT xStream

Full Message text is:

*FCV could not control flow at Junction X*

This warning alerts users that the [Flow Control Valve](#) at junction X was not able to control the flow, and backflow may have occurred.

## Flow may not be balanced at Junction X

### A [Warning message](#) in AFT xStream

Full Message is:

*Flow may not be balanced at Junction X (total = Y)*

This warning alerts users that the converged mass balance results may be incorrect for the AFT Arrow Steady Solution. Once a converged solution is obtained, AFT xStream checks all junctions to ensure a mass balance exists. It compares the imbalance to the flow [tolerance](#), and when the imbalance exceeds the tolerance, this warning is generated.

If the magnitude of the flow imbalance is close to the tolerance specified, it may be safe to ignore this warning, although it likely make sense to reduce tolerances further and rerun the model.

If the magnitude of the energy imbalance is large, the converged solution that may be incorrect. When a large imbalance exists, it may be an indicator the model is not realistic.

These warnings should always be reviewed in detail, and the user should perform hand calculations on the energy balance at the affected junctions to verify whether an acceptable solution exists.

## Fluid is no longer in the vapor phase because the fluid temperature/pressure is...

### A [Warning message](#) in AFT xStream

Full Message is:

*Fluid is no longer in the vapor phase because the fluid pressure is above the Saturation Pressure for the following...*

The solution indicates that the temperature and pressure conditions at the specified pipe are such that one or more gas components has reached saturation conditions. xStream will attempt to extrapolate gas-phase properties beyond saturation conditions because xStream assumes the fluid is single-phase. Optionally, the user can choose to limit properties to Saturated Conditions in the [Advanced Transient Parameters panel](#).

Results from the MOC Transient after the transient temperature and pressure reaches the saturation temperature and pressure should be treated with caution.

## Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy

A [Warning message](#) in AFT xStream

Full Message text is:

*Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy.*

The stagnation enthalpy entering a junction should equal the stagnation enthalpy exiting, assuming no flow splits and an adiabatic junction. This warning tells you that the values differ. Results are likely incorrect, and may not represent a converged solution.

## Junction X Compressor/Fan added negative pressure

A [Warning message](#) in AFT xStream

Full Message text is:

*Junction X Compressor/Fan added negative pressure.*

A compressor junction with an assigned flow rate had a pressure loss across the [compressor junction](#) either in the steady state or transient system. This warning is meant to alert the user that the compressor is not adding pressure as they may be expecting.

## Junction X Exceeded End-of-Curve Flow Rate

A [Warning message](#) in AFT xStream

Full Message text is:

*Junction X Exceeded End-of-Curve Flow Rate*

Centrifugal [compressor](#) curves typically only extend to a given flow. AFT xStream defines this point as the End-of-Curve. Such data has been defined for this compressor in the [Compressor Configuration](#) window, and the system is operating past the End-of-Curve flow rate. Thus the results may be invalid, because the compressor may not actually operate at that point.

## Junction X Had Reverse Flow During Transient

A [Warning message](#) in AFT xStream

Full Message text is:

*Junction X had reverse flow during the transient. The head/pressure rise was estimated from the zero flow value.*



A transient event could cause reverse flow at a [compressor junction](#) in the model. In this situation, the head/pressure rise across the compressor is estimated from the value at zero flow.

## Maximum system pressure exceeded enthalpy table's maximum pressure of X

### A [Warning message](#) in AFT xStream

Full Message text is:

*Maximum system pressure exceeded enthalpy table's maximum pressure of X. Enthalpy is extrapolated for pressures exceeding this value.*

This warning indicates that the AFT Standard fluid database is being used with the Table Enthalpy model, and that the maximum system pressure exceeded the maximum pressure for the enthalpy data. The Table enthalpy model uses interpolation based on the current pressure to calculate the enthalpy at a given temperature. If the pressure at a point in the system exceeds the maximum pressure data available for the fluid in the AFT Standard fluid database, then the enthalpy must be extrapolated, leading to inaccuracy in the results. Extrapolation far above the maximum temperature will lead to increased inaccuracy, and may cause convergence issues. Consider using an alternate fluid database such as NIST Refprop or Chempak with a higher maximum pressure.

## Maximum Transient Pipe Temperature Higher Than Estimate

### A [Warning message](#) in AFT xStream

Full Message text is:

*The maximum transient pipe temperature (X) was higher than the estimate (Y). The estimated maximum pipe temperature should be set marginally above the maximum.*

This warning indicates the maximum temperature in the pipes during the transient simulation is higher than the Estimated Maximum Pipe Temperature During Transient value specified by the user in the [Sectioning panel](#). A temperature higher than the estimate may cause the maximum sonic velocity to be higher than estimated, and in turn mean the time step for the MOC Transient simulation is too small to properly simulate high-velocity flow.

The user should provide a more accurate estimated maximum temperature and re-run the simulation. If high Mach numbers are never seen during the simulation, it may be acceptable to set the estimated maximum pipe temperature lower than the actual maximum temperature.

## Maximum Transient Pipe Temperature Significantly Lower Than Estimate

### A [Warning message](#) in AFT xStream

Full Message text is:

*The maximum transient pipe temperature (X) was significantly lower than the estimate (Y). The estimated maximum pipe temperature should be set marginally above the maximum.*

This warning indicates the maximum pipe temperature during the transient simulation is significantly lower than the Estimated Maximum Pipe Temperature During Transient value specified by the user in the [Sectioning panel](#). The estimated maximum temperature is used to help determine the time step for the simulation. In this situation, it's possible that more time steps are used than are necessary to capture the system's transient behavior. Lowering the estimate will make each time step longer, reducing run time.

The user should provide a more accurate estimated maximum pipe temperature and re-run the simulation.

## MOC Steady Solution Did Not Reach Nominally Steady Before Transient

A [Warning message](#) in AFT xStream

Full Message text is:

*The MOC Steady Solution did not reach a nominally steady state before the transient started.*

The [MOC Steady Solution](#) runs a series of Block Iterations and checks the velocity change across each block. The MOC Steady Solution is considered nominally steady when the velocity change between blocks is less than the values specified in the MOC Steady Solution panel. If the velocity change across a block is never lower than the allowable change, a nominally steady state will not be reached, and this warning will be shown.

The user should increase the allowable change, increase the maximum number of MOC Steady Block Iterations, or increase the [number of sections](#) used.

## Physical flow area is less than the sonic area at junction X...

A [Warning message](#) in AFT xStream

Full Message text is:

*Physical flow area is less than the sonic area at junction X. Sonic choking probably exists but was ignored in this solution because no CdA was specified.*

AFT xStream cannot account for [sonic choking](#) at a junction unless the [CdA](#) for Sonic Choking is specified. For several junctions, a physical area is specified by the user. For example, for an [orifice](#) the orifice area or diameter is specified. However, the real flow area is less than this physical area because of vena contracta and wall separation effects. In order to perform a proper sonic choking calculation, AFT xStream needs to know the CdA.

When this warning is displayed, AFT xStream has determined that the physical area was less than the sonic area at the specified junction, but sonic choking was ignored because no CdA was specified. Thus sonic choking may exist even though AFT xStream could not calculate it.

When this warning is displayed it is recommended you obtain the proper CdA for the junction and run the model again.

### **Pipe/Junction X property calculation exceeded maximum local iteration limit**

#### **A [Warning message](#) in AFT xStream**

Full Message text is:

*Pipe/Junction X property calculation exceeded maximum local iteration limit.*

The final network solution was not within tolerance for a local iteration. The [local iteration limit](#) set in the [AFT Arrow Steady Solution panel](#) was exceeded, but the Solver ignored this and continued on. If a network solution is obtained but this out of tolerance situation exists, AFT xStream attempts the network solution again up to five times in a row. If the out of tolerance situation remains, AFT xStream ceases further attempts and displays the convergence notification, then gives this warning in the output. The property can be any of several solver parameters and depends on the actual subroutine where the problem occurred.

Along with the warning, the Error value and Tolerance are displayed, along with a Code #. For AFT xStream to consider this local iteration to be converged, the Error value needs to be less than the displayed Tolerance. This Tolerance is not the tolerance the user can specify in the AFT Arrow Steady Solution panel, but rather an internal “hard-wired” tolerance that has been set by the software developers and arrived at through experience.

If the Error value is relatively close to the displayed Tolerance, it is generally safe to ignore this warning. If it is much larger, the remainder of the output is probably invalid.

In either case, changing the solution method in the AFT Arrow Steady Solution panel frequently solves the problem.

The Code # identifies the subroutine where the problem occurred. This information is not of much assistance to the user, but can be of help to AFT technical support staff when troubleshooting models.

### **Pipe/Junction X property calculation failed to continue towards convergence**

#### **A [Warning message](#) in AFT xStream**

Full Message text is:

*Pipe/Junction X property calculation failed to continue towards convergence.*

The final network solution was not within tolerance for a local iteration. This was determined by AFT xStream’s Auto-Detect feature. The Solver ignored this and continued on. If a network solution is obtained but this out of tolerance situation exists, AFT xStream attempts the network solution again up to

five times in a row. If the out of tolerance situation remains, AFT xStream ceases further attempts and displays the convergence notification, then gives this warning in the output. The property can be any of several solver parameters and depends on the actual subroutine where the problem occurred.

Along with the warning, the Error value and Tolerance are displayed, along with a Code #. For AFT xStream to consider this local iteration to be converged, the Error value needs to be less than the displayed Tolerance. This Tolerance is not the tolerance the user can specify in [AFT Arrow Steady Solution](#), but rather an internal “hard-wired” tolerance that has been set by the software developers and arrived at through experience.

If the Error value is relatively close to the displayed Tolerance, it is generally safe to ignore this warning. If it is much larger, the remainder of the output is probably invalid.

In either case, changing the solution method in the AFT Arrow Steady Solution panel frequently solves the problem.

The Code # identifies the subroutine where the problem occurred. This information is not of much assistance to the user, but can be of help to AFT technical support staff when troubleshooting models.

### Pipe X temperature is outside temperature limits in AFT Arrow Steady Solution Control

#### A [Warning message](#) in AFT xStream

Full Message text is:

*Pipe X temperature is equal to or greater than the maximum temperature limit (equal to or less than the minimum temperature limit) of Y set by the user in Solution Control. Results may be incorrect.*

The user has [temperature limits](#) defined in the [AFT Arrow Steady Solution Control panel](#). These limits control the search space for temperatures while solving. However, the final solution had one or more pipes at the temperature limit, meaning the limit restricted the solution and may give incorrect results. The user should either remove or change the temperature limit.

### Pipe X Temperature Outside Valid Fluid Temperature Range

#### A [Warning message](#) in AFT xStream

Full Message text is:

*Temperature at inlet/outlet of Pipe X is above the max (below the min) fluid temperature range.*

The temperature specified for the pipe is outside the range of valid temperatures. See [Fluid Panel](#).

### PRV could not control pressure at Junction X

#### A [Warning message](#) in AFT xStream

Full Message is:

*PRV could not control flow at Junction X*

This warning alerts users that the [Pressure Reducing Valve](#) at junction X was not able to control the flow, and backflow may have occurred.

### **PSV could not control pressure at Junction X**

A [Warning message](#) in AFT xStream

Full Message text is:

*PSV could not control flow at Junction X*

This warning alerts users that the [Pressure Sustaining Valve](#) at junction X was not able to control the flow, and backflow may have occurred.

### **Temperature is below the saturation line for the following. . .**

A [Warning message](#) in AFT xStream

Full Message text is:

*Temperature is below the saturation line for the following:*

The solution indicates that the temperature and pressure conditions at the specified pipe are such that one or more gas components has reached saturation conditions. This warning only occurs with Chempak fluids, NIST REFPROP fluids, or ASME Steam selected on the [Fluid panel](#). When temperature falls below saturation, xStream's solution accuracy will be decreased, as xStream always assumes the fluid is superheated and will attempt to extrapolate gas properties below the saturation line.

### **Transient choking occurred at Junction X. The set point could not be met.**

A [Warning message](#) in AFT xStream

Full Message text is:

*Transient choking exists at junction X. The specified flow or pressure could not be met.*

This warning tells the user that choked conditions exist at a junction with a specified pressure or flow rate, specifically that those choked conditions existed during the transient simulation. In this situation, xStream will limit the flow at the pipe to the choked flow rate regardless of the user-defined conditions.

## Transient temperature was below fluid minimum temperature

### A [Warning message](#) in AFT xStream

Full Message text is:

*Transient temperature was below fluid minimum temperature at X seconds. Results after this time may not be valid.*

This warning informs the user that the temperature went below the minimum temperature in the fluid database during the transient simulation. xStream will extrapolate below the minimum temperature and assume the fluid remains in the gas phase. In some cases the minimum temperature for the fluid database represents saturation conditions, in which case xStream will be extrapolating below the saturation line.

Results from the MOC Transient after the transient temperature reaches the minimum temperature should be treated with caution.

## Unique Solution Not Possible Relief Valve Cycles

### A [Warning message](#) in AFT xStream

Full Message text is:

*Relief Valve cycles, unique solution is not possible.*

This warning is generated by the AFT Arrow Steady Solution, and means the relief valve will not operate in its open or closed state during steady state. When the relief valve is closed in steady state, the pressure exceeds its set pressure, so it opens. When the relief valve is open in steady state, the pressure falls below its blowdown pressure, so it closes.

## Cautions

A Caution message in AFT xStream is of less importance than a full warning. While some Caution messages may be left unresolved without affecting model results, they should never be ignored by the user.

## CAUTIONS

---

- [All transient junction behavior was disabled](#)
- [Assigned Flow at Junction X has a constant flow rate during the transient...](#)
- [Automatic Sectioning is enabled, but flow increases substantially during the transient](#)
- [Check Valve closed at Junction X](#)
- [Flow is negative through Junction X. Loss factors may be in error](#)
- [Junction X exceeded maximum event messages](#)
- [Junction X is downstream of a choking point, stagnation/static properties cannot be calculated](#)

- [Maximum transient pipe Mach number lower than estimate](#)
- [Maximum transient pipe Mach number at least 0.2 higher than steady...](#)
- [Pipe X length is shorter than the elevation change \(between Junction Y and Z\)](#)
- [Transient data did not extend to Stop Time at Junction X](#)
- [Transient temperature was within 10 deg. F of minimum temperature](#)
- [Transient temperature was within 10 deg. F of saturation temperature](#)

### All transient junction behavior was disabled

#### A [Caution message](#) in AFT xStream

Full Message text is:

*All transient junction behavior was disabled. This is a troubleshooting mode.*

This is a cautionary notice that the *Disable transient junction behavior for the entire simulation* setting is enabled on the [Transient Solver Initialization Panel](#) under the Advanced Settings, meaning that the model was run with no user-defined transients applied. This is different than using the [AFT Arrow Steady solution method](#), and is usually only useful for troubleshooting purposes, and identifying artificial transients.

### Assigned Flow at Junction X has a constant flow rate during the transient...

#### A [Caution message](#) in AFT xStream.

Full message text is:

*[Assigned Flow](#) at Junction X has a constant flow rate during the transient. During a transient the flow usually changes. This may not be an appropriate use of the junction.*

An Assigned Flow junction fixes the flow at a boundary. When a transient occurs in the system, it is common to see all flows affected by that transient. This caution is a reminder that an assigned flow may not be the appropriate junction to model a transient boundary condition. In general it is better to use [Assigned Pressure](#) or [Tank](#) junctions at boundaries.

### Automatic Sectioning is enabled, but flow increases substantially during the transient

#### A [Caution message](#) in AFT xStream

Full Message text is:

*Automatic Sectioning is enabled, but flow increases substantially during the transient.*

This is a cautionary notice that the user enabled [Automatic Sectioning](#) to allow xStream to section the system to [minimize the Steady-State Conservation Error](#). However, Automatic Sectioning is performed based on the steady-state flow, meaning conservation error will increase if the system flow increases. When this caution is presented, the user should not assume that conservation error was minimized throughout the simulation.

### Check Valve closed at Junction X

A [Caution message](#) in AFT xStream

Full Message is:

*Check Valve closed at Junction X.*

This is a cautionary notice that a [check valve](#) closed during the AFT Arrow Steady Solution.

### Flow through Junction X is negative. Loss factors may be in error.

A [Caution message](#) in AFT xStream

Full Message text is:

*Flow through Junction X is negative. Loss factors may be in error.*

This caution alerts users to flow through junctions in the opposite direction from the connecting pipes reference directions. Any [loss information](#) specified for junctions may be in error because it was defined assuming flow in the other direction.

### Junction X exceeded maximum event messages

A [Caution message](#) in AFT xStream

Full Message is:

*Junction X exceeded maximum event messages.*

Event messages are generated each time a transient event occurs at a junction. A limit is set of how many event messages can be shown for a junction to limit how much data is displayed in the [output](#). This limit was reached, and some events have happened with no event message displayed. The maximum event messages setting can be changed in the [Output Control window](#) on the Format & Action tab.

### Junction X is downstream of a choking point, stagnation/static properties cannot be calculated

A [Warning message](#) in AFT xStream



Full Message is:

*Junction X is downstream of a shock, static/stagnation properties cannot be calculated at junction.*

This warning will only be generated by pressure type junctions (e.g., [Assigned Pressure](#), [Tank](#)). When [sonic choking](#) occurs at a pressure type of junction, there is no way to calculate a velocity downstream of the choke point. This means that it is not possible to determine static conditions if you have specified stagnation at the junction, and vice versa. All other results for the model should be accurate except for the static/stagnation pressure, temperature, and enthalpy at the junction.

## Maximum transient pipe Mach number lower than estimate

A [Caution message](#) in AFT xStream

Full Message text is:

*The maximum transient pipe Mach Number X was significantly lower than the estimate Y. Lowering the estimated maximum pipe Mach number may increase accuracy. The estimate must be set above the simulation maximum pipe Mach Number, a value of at least Z is recommended*

This is a cautionary notice that the maximum transient Mach number in the pipes remained significantly below the Estimated Maximum Pipe Mach Number During Transient, which is defined in the [Sectioning panel](#). Decreasing the estimated maximum Mach number may increase the accuracy and improve the run time. Note that if the estimated maximum Mach number is less than the maximum Mach number encountered during the simulation, the solution will become unstable, so the estimated maximum Mach number should not be defined to be less than the recommended value given in the caution.

## Maximum transient pipe Mach number at least 0.2 higher than steady...

A [Caution message](#) in AFT xStream

Full Message text is:

*The maximum transient pipe Mach number is at least X higher than the steady pipe mach number at the same location. If sectioning was determined from MOC steady results, the accuracy of the transient simulation may not be sufficient for higher flow periods.*

This is a cautionary notice that there was a large change in Mach number between the steady state and transient results. The sectioning for the model is chosen based on the steady state results, and will therefore produce the highest accuracy results for cases where the velocity is similar to the steady state velocity. The inaccuracy due to sectioning is discussed further in the [Transient Result Accuracy](#) topic.

It is recommended to run the model again with more sections to determine what effect the Mach number difference has on accuracy.

## Pipe X length is shorter than the elevation change (between Junction Y and Z)

### A [Caution message](#) in AFT xStream

Full Message text is:

*Pipe X length shorter than the elevation change (between Junction Y and Junction Z).*

This caution alerts users to a possible inconsistency in their input. When this caution is shown, a pipe is defined with a length that is shorter than the elevation change between its connecting junctions. In this situation, pressure loss along the pipe due to friction is calculated using the pipe length the user has defined while static head loss is determined using the elevation change the user has defined.

## Transient data did not extend to Stop Time at Junction X

### A [Caution message](#) in AFT xStream

Full Message text is:

*Transient data did not extend to Stop Time at Junction X. Final data point was used.*

The transient data at Junction X did not have time data that covered the entire time frame of the simulation. Any transient data after the final time data point was assumed to be at the final data value. You may want to enter additional time data points at Junction X.

This extrapolation behavior can be changed in the [User Options](#) window on the Tools menu.

## Transient temperature within 10 deg. F of minimum temperature

### A [Caution message](#) in AFT xStream

Full Message text is:

*Transient temperature was within 10 deg. F of minimum temperature at X seconds.*

This is a cautionary notice that the temperature at some point in the system approached the minimum temperature for the fluid in the database during the transient simulation. This means that some properties may be extrapolated beyond the range of the available data. In some cases this may also mean that the saturation conditions were reached. The user should review the results to verify that saturation conditions were never reached.

## Transient Temperature within 10 deg. F of Saturation Temperature

### A [Caution message](#) in AFT xStream

Full Message text is:

*Transient temperature was within 10 deg. F of saturation temperature at X seconds.*

This is a cautionary notice that the temperature at some point in the system approached saturation conditions during the transient simulation. It does not necessarily mean condensation occurred or the results cannot be trusted. However, the user should review the results to verify that saturation conditions were never reached.

## Troubleshooting

This section discusses methods of troubleshooting models that are having difficulty converging. Topics include discussions of common issues, ideas to simplify overly complex models, and methods for adapting the solution methodology through either tolerance or flow relaxation.

### Contents

---

<a href="#">Reducing Run Time</a> .....	513
<a href="#">Transient Solver Errors</a> .....	518
<a href="#">Diagnostics Window</a> .....	520
<a href="#">Use the Model Data Window</a> .....	521
<a href="#">Transfer Results to Initial Guesses</a> .....	523
<a href="#">Poor Compressor Curve Fits</a> .....	524
<a href="#">Troubleshooting Resistance Curves</a> .....	524
<a href="#">Use the Output Window Sort Feature</a> .....	525
<a href="#">Lower the Flow Rate Relaxation</a> .....	526
<a href="#">Try Absolute Tolerance</a> .....	526
<a href="#">Make Initial Flow Rate Guesses for Pipes</a> .....	526
<a href="#">Change Boundary Conditions</a> .....	527
<a href="#">Turn Off Parts of the Model</a> .....	527
<a href="#">Break a Large Model Into Submodels</a> .....	527
<a href="#">Use an AFT Standard Fluid</a> .....	527
<a href="#">Run the MOC Steady Solution</a> .....	528
<a href="#">Simplify Event Transients</a> .....	528

[Limit Properties to Saturation Conditions](#) ..... 528

[Enable Transient Debug Files](#) ..... 529

---

## Related Blogs

---

[Having model convergence problems? Start here!](#)

### Reducing Run Time

The below topics discuss several common model configurations that can cause xStream model run times to be longer than expected. Select a topic below for information on why each configuration increases the run time, and suggestions on how each configuration can be avoided.

If one of the below configurations is detected by xStream and the run time is relatively long, xStream will present a message to the user showing which of the below configurations may be applicable.

Note that the configurations listed here as causing long run times may not be inaccurate for the system being modeled. However, they are all likely inefficient modeling choices for transient analysis - especially for preliminary iterations.

The configurations discussed below are not meant to capture every possible model configuration which can cause long model run times in xStream. Many other factors can also cause long run times. Following the suggestions listed below may help reduce run time in any model, though it may not be practical to implement the listed suggestions for all models.

---

## Related Topics

Minimizing Run Time.....	28
Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51

---

## Related Blogs

---

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

[You Should Combine Your Pipes](#)

### The model has both very long and very short pipes

A model with both long and short pipes requires a larger number of pipe sections in order to perform calculations. The more pipe sections in a model, the more computations the solver needs to perform, which results in a longer run time.

## How are short and long pipe lengths contributing to long run times?

---

To run a transient simulation, AFT xStream sections pipes to apply the [Method of Characteristics](#). In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - [by default, 10](#). AFT xStream uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

When a model has both very long and very short pipes, the short pipes determine the controlling pipe length, and the long pipes must be divided into many sections to properly apply the Method of Characteristics. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times.

Consider a model that consists of two pipes: a very short pipe, such as 1 meter, and a very long pipe, such as 10,000 meters. The 1 meter pipe will be the controlling pipe with 10 sections, and the 10,000 meter pipe must be broken into 100,000 sections. The model thus requires 100,010 sections in order to perform the calculations, which results in a long run time. Increasing the short pipe's length artificially from 1 to 10 meters decreases the number of sections required by a factor of 10, reducing run time by a factor of 100. While the length increase may also increase the error in the short pipe, experience shows that the error is usually negligible in a liquid transient context.

## What can the user do?

---

Several different approaches are available to reduce run time. These approaches can work in parallel, and it is up to the engineer to make appropriate judgments as to which is best for the model.

**Consider modeling [fittings and losses](#) in pipes and combining multiple short pipes.** In the Pipe Properties window, use the Fittings & Losses tab to add in fittings to the pipe instead of using junctions on the workspace. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings. For example, a user can add valves and bends to a pipe, and the solver will still apply the losses directly to the pipe during the calculations. By removing these junctions and combining pipes in the workspace, many short pipes between these fittings are eliminated. Reducing and combining short pipes reduces the number of sections in the model and the overall run time.

**Consider adding artificial length to short pipes.** Adding length to short pipes increases the length of the controlling pipe, which results in fewer sections for the other pipes in the model. Reducing the overall number of sections in the model improves run time. Using pipe lengths that are common multiples, such as 5 feet, 10 feet, 15 feet, etc. provides improved sectioning since short pipes can be perfectly divided into longer pipes.

## Related Topics

---

Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51

## Related Blogs

---

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

### The model has many pipes that can be combined

Users often model losses in AFT xStream using junctions with no transient behavior, such as a bend, static valve, or orifice, between two pipes of identical diameter. While this may help show where these losses are occurring in the overall pipe system, it can result in extra pipes which can have a detrimental effect on pipe sectioning and run time. Combining pipes and modeling the losses within the pipe itself can improve sectioning and run time while maintaining accuracy.

### How are pipes that can be combined contributing to long run times?

---

To run a transient simulation, AFT xStream sections pipes to apply the [Method of Characteristics](#). In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - [by default, 10](#). AFT xStream uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

Each pipe in a model must be sectioned individually, meaning section count must increase with pipe count. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times. Reducing the number of pipes in a model is often an effective way to reduce run time.

Consider a model that includes a pipeline that is 100 meters long, and has 9 bends in it. If those 9 bends are modeled as unique junctions, the pipeline must be modeled as 10 individual pipes. Each pipe has 10 sections by default, meaning the pipeline will have 100 sections. If the pipeline is modeled as a single pipe, and the 9 bends are modeled as additional pipe friction using the [Fittings & Losses](#) tab in the Pipe Properties window, the number of sections in the pipeline could be reduced by a factor of 10, in turn reducing the run time by a factor of 100.

### What can the user do?

---

The [Fittings & Losses](#) tab in the Pipe Properties window can be used to add losses directly to pipes to replace individual junctions on the workspace. This process can be done manually, or automatically using the Join Selected Pipes/Jcts tool in AFT xStream.

Select the pipes to be combined and the junctions between those pipes on the Workspace, and choose the [Join Selected Pipes/Jcts](#) from the Edit menu. This option will combine the selected pipes into one pipe, add the losses from the junctions into the Fittings & Losses tab.

## Related Topics

---

Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51

### Related Blogs

---

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

[You Should Combine Your Pipes](#)

### The model has an unusually short pipe

A model with a very short pipe requires a larger number of pipe sections in order to perform calculations. The more pipe sections in a model, the more computations there are that need to be performed, which results in a longer run time.

### How are pipe lengths contributing to long run times?

---

To run a transient simulation, AFT xStream sections pipes to apply the [Method of Characteristics](#). In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - [by default, 10](#). AFT xStream uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

When the controlling pipe is unusually short, the other pipes in the model must be divided into many sections to apply the Method of Characteristics. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times.

Consider a model that has one 0.05 ft long pipe and 100 pipes that are each 10 ft long. Each 10 ft pipe would require 200 sections, requiring 20,000 sections overall. Modeling the very short pipe as 0.1 ft would reduce the number of sections by a factor of 2, which would potentially decrease the run time by a factor of 4. Alternatively if the very short pipe is not considered in the sectioning by combining the pipes, then the controlling pipe length would become 10 ft. With one section in each pipe this would decrease the number of sections from 20,000 to 100, decreasing the run time by a factor of 40,000.

### What can the user do?

---

There are several different approaches to reduce run time. These approaches can work in parallel, and it is up to the engineer to make appropriate judgments as to which approach is best for the model.

**Consider modeling [fittings and losses](#) in pipes and combining multiple short pipes.** In the Pipe Properties window, use the Fittings & Losses tab to add in fittings to the pipe instead of using junctions on the workspace. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings. For example, a user can add valves and bends to a pipe, and the solver will still apply the losses directly to the pipe during the calculations. By removing these junctions and combining pipes in the workspace, many short pipes between these fittings are eliminated. Reducing and combining short pipes reduces the number of sections in the model and the overall run time.

**Consider adding artificial length to short pipes.** Adding length to short pipes increases the length of the controlling pipe, which results in fewer sections for the other pipes in the model. Reducing the overall number of sections in the model improves run time. Using pipe lengths that are common multiples, such as 5 feet, 10 feet, 15 feet, etc. provides improved sectioning since short pipes can be perfectly divided into longer pipes.



## Related Topics

---

Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51

## Related Blogs

---

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

### The simulation has a large number of time steps

A model with a large number of time steps will require a large number of calculations. The more calculations that need to be performed, the longer the run time.

### How are the time steps contributing to long run times?

---

xStream sections pipe to apply the [Method of Characteristics](#) solution method. The Method of Characteristics requires a uniform time step to be used for all pipes in the model, which is determined during the sectioning process using the section length and the sonic velocity in the gas. On average the sonic velocity is typically on the magnitude of  $10^3$ , producing a time step on the magnitude of  $10^{-3}$ .

A time step of 0.001 seconds means that a transient simulation of 1 minute will have 60,000 total time steps. Similarly, a transient simulation of 1 hour will have 3.6 million time steps. Run time is roughly proportional to the number of time steps, and models with large numbers of time steps will generally have long run times. In addition, models with higher wave velocity may require shorter time steps, increasing the number of time steps required.

### What can the user do?

---

There are several different approaches to reduce run time when the number of time steps is larger than expected.

**Consider reducing the simulation duration.** The simulation duration is defined on the [Simulation Duration panel](#). For a simple system, the full effect of the transient response occurs over twice the [communication time](#), where the communication time is the time it takes a pressure wave to travel the length of the system and back (length of system / wave velocity). Determining the communication time can become more complex for systems with branching paths or major reflection points in the path. However, limiting the simulation duration to 2 times the communication time may be a good starting point. If multiple transient events occur the simulation duration should be extended to end after the transient response has completed for the last transient event.

In addition, for a piping network attached to a very long pipe it may not be necessary to include the length of the very long pipe in the communication time calculation. Use your engineering judgement on whether the reflections from the pipeline will be important to consider.

**Consider lengthening the controlling pipe.** The controlling pipe length is used to determine the time step for the simulation, setting the time step equal to the section length divided by twice the sonic velocity. Artificially lengthening the controlling pipe means the time step for the simulation can be longer, and fewer time steps are required for the simulation.

**Consider decreasing the Minimum Number of Sections per Pipe.** The length of the time step is determined by the length of the computational stations. If the minimum number of stations per pipe is large, then shorter sections and shorter time steps will be required. Note that decreasing the number of sections will decrease the accuracy of the solution. A sensitivity analysis is recommended to determine the effect of decreasing the number of sections on accuracy, see the [Transient Sensitivity Analysis Tutorial](#).

---

## Related Topics

Transient Result Accuracy.....	49
Transient Sensitivity Analysis Tutorial.....	51

---

## Related Blogs

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

### Transient Solver Errors

A transient solver error will be shown during the transient solution if xStream encounters a calculation error which would invalidate the calculation results for the rest of the transient. Transient solver errors halt the transient solution and must be resolved for the full simulation to be run.

The below topics offer descriptions of the more common solver errors and tips for common resolutions. Reach out to [AFT Support](#) for further assistance.

---

## TRANSIENT SOLVER ERRORS

### General error occurred during the transient calculations

A [Transient Solver Error](#) in AFT xStream

Full Message text is:

*Error occurred while running Transient Solver. General error occurred during the transient calculations.*

This solver error can be caused by any number of discrepancies. It is recommended to generate a [Transient Debug File](#) to get more information on the solution leading up to the iteration error. It may also help to increase the minimum number of sections defined on the [Sectioning panel](#).

Contact AFT Support for additional help if needed.

## Junction iteration problem

### A [Transient Solver Error](#) in AFT xStream

Full Message text is:

*Error occurred while running Transient Solver. [Junction] iteration problem.*

An iteration error for junctions such as the branch junction can occur if the flow or pressure iterations at that point exceed the maximum iterations. It is recommended to generate a [Transient Debug File](#) to get more information on the solution leading up to the iteration error. It may also help to increase the minimum number of sections defined on the [Sectioning panel](#).

Contact AFT Support for additional help if needed.

## Possible supersonic flow detected at the connection of Junction X and Pipe Y

### A [Transient Solver Error](#) in AFT xStream

Full Message text is:

*Possible supersonic flow detected at the connection of Junction X and Pipe Y. Modeling of supersonic conditions is not supported and the solver was halted.*

xStream assumes that the flow is one dimensional and does not become supersonic during the transient. In reality there are situations that could cause supersonic flow to develop behind a shockwave. In addition there could be multidimensional shocks in a real system which are not captured in a one-dimensional solution. xStream will halt the transient if the calculated Mach number exceeds one to prevent inaccurate results from being displayed to the user.

If this error message is shown for a system that should not experience supersonic flow it is recommended to verify the model inputs are correct, consider adjusting the junction transient events, and potentially increase the [minimum number of sections per pipe](#). Supersonic flow is more likely to occur when there is a high pressure differential across a junction during the steady state, when a transient event occurs rapidly, and/or when the section length is relatively large in a pipe.

The configuration which most commonly causes this transient solver error is a valve with a large pressure differential across it that opens rapidly in the transient.

## User Specified AFT Arrow to Resolved allowable mismatch was exceeded

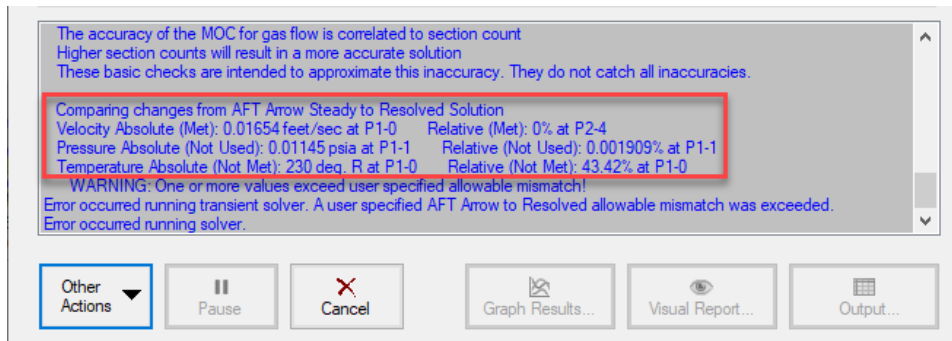
### A [Transient Solver Error](#) in AFT xStream

Full Message text is:

*Error occurred while running Transient Solver. A user specified AFT Arrow to Resolved allowable mismatch was exceeded.*

This solver error indicates that one of the allowable mismatch values defined in the Transient Solver Initialization panel was exceeded. For an explanation of how the allowable mismatch values are applied and the transient solver initialization process please reference the discussion in the [Transient Solver Initialization panel](#) topic.

xStream prints a summary of the comparison between the AFT Arrow Steady and Resolved solutions in the Solution Progress window, which can be used to determine what parameters were violated. For example, see Figure 1 below which shows a model that met the velocity criteria, but did not meet the applied temperature criteria.



**Figure 1: The bottom of the Solution Progress window provides additional information on the results of the artificial transient resolution period**

To resolve a mismatch between the AFT Arrow Steady and Resolved solutions it is recommended to re-evaluate the initial conditions selected for the model to remove inconsistent inputs.

For instance, in the model used for Figure 1 there was an inlet tank defined at 300 deg. F. Pipe P1 connected to the inlet tank was defined to use convective heat transfer with the atmospheric temperature set at 70 deg. F. There was no flow in the pipe initially, so the pipe was fixed at the atmospheric temperature (70 deg. F) for the AFT Arrow Steady solution. During the artificial transient resolution process the pipe temperature was allowed to vary and increased to about 300 deg. F due to the tank temperature, creating a mismatch between the AFT Arrow Steady and Resolved solution temperatures in the pipe.

The error message in the example model was resolved by using the [Temperature Override for Zero Flow](#) option in the Pipe Properties window to set the pipe's temperature equal to the tank temperature during the AFT Arrow Steady solution.

## Diagnosics Window

The Model Diagnostics Window is intended to help users check on a number of settings that may have been changed during the model-building process and that may affect the model results.

The Model Diagnostics window can be accessed from the Tools menu.

When the Model Diagnostics Window is opened all input will be checked, and the results will be displayed in a tabular format. Items of note will be highlighted based on the impact each identified item may have on the results. It should be noted that **not all items identified by this window will have a negative impact on results**. Rather, the identified items are settings that have frequently been incorrectly applied by other users. This is not an exhaustive list of troubleshooting items, though the most common issues are all included.

Data in the Model Diagnostics is organized by the Data Source, which is the window where that setting/input is defined. Any item that may negatively impact the model will be highlighted based on its potential to affect the model, as shown in the Potential Impact column. For pipe and junction diagnostic items, the pipe(s)/junction(s) that were identified will be listed below the diagnostic item. For an example, see Figure 1 below. The Diagnostic window indicates that multiple pipes have unusual pipe dimensions, and should be reviewed.

Data Source	Diagnostic Description	Result	Potential Impact	Diagnostic Details
> Analysis Setup			None	
> AFT Arrow Steady Solution			None	
> Environmental Properties			None	
> Sectioning			None	
User Options/General			None	
> Pipes			High	
Optional	Unusual Design Factors?	False	None	
Pipe Model	Friction model not Absolute?	False	None	
> Pipe Model	Unusual pipe dimensions?	True	High	The following pipes have an unusual geometry. (Length < 10 ft or Diameter < 1" or Diameter > 50% Length)
P13 (Pipe)				
P14 (Pipe)				
Pipe Model	User specified size?	False	None	
Steady Output	Negative absolute pressures exist?	N/A	Unknown	No output to test.
> Junctions			None	

**Figure 1: Example of the Model Diagnostics window. For this example, potential issues have been flagged in the pipe dimensions.**

## Related Blogs

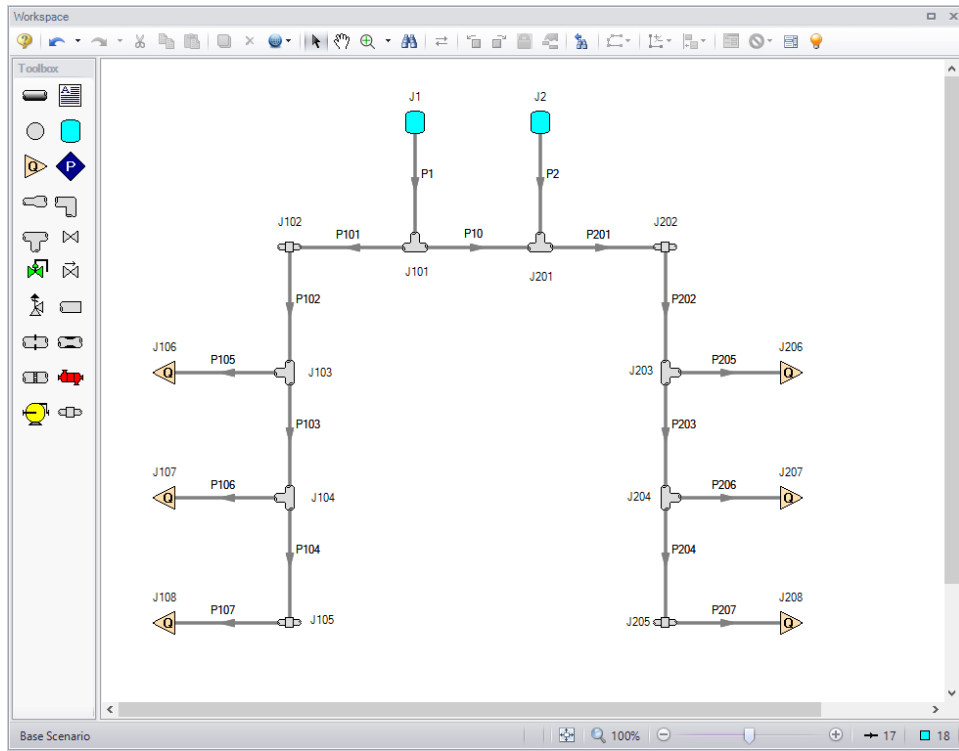
[AFT Responds to 2020 Customer Survey Feedback](#)

### Use the Model Data Window

The [Model Data window](#) is a powerful tool for reviewing and documenting model inputs, along with finding input errors in a model.

Look at the model in Figure 1. If an input error in one of the pipes was suspected, the task of looking for the error in each individual pipe input window would be tedious.

Now look at the Model Data window for the pipes in Figure 2. Here it is much easier to scan down columns of data looking for input errors. In this case, one of the pipe lengths was accidentally entered in miles rather than feet.



**Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown).**

Pipe	Name	Pipe Defined	Length	Length Units	Hydraulic Diameter	Hydraulic Diam. Units	Friction Data Set	Roughness	Roughness Units
1	Pipe	Yes	100 feet		11.938 inches		Standard	0.0018 inches	
2	Pipe	Yes	150 feet		11.938 inches		Standard	0.0018 inches	
10	Pipe	Yes	75 feet		11.938 inches		Standard	0.0018 inches	
101	Pipe	Yes	400 feet		7.981001 inches		Standard	0.0018 inches	
102	Pipe	Yes	750 miles		7.981 inches		Standard	0.0018 inches	
103	Pipe	Yes	750 feet		7.981001 inches		Standard	0.0018 inches	
104	Pipe	Yes	250 feet		7.981001 inches		Standard	0.0018 inches	
105	Pipe	Yes	100 feet		6.065001 inches		Standard	0.0018 inches	
106	Pipe	Yes	50 feet		6.065001 inches		Standard	0.0018 inches	
107	Pipe	Yes	75 feet		6.065001 inches		Standard	0.0018 inches	
201	Pipe	Yes	250 feet		7.981001 inches		Standard	0.0018 inches	
202	Pipe	Yes	500 feet		7.981001 inches		Standard	0.0018 inches	
203	Pipe	Yes	500 feet		7.981001 inches		Standard	0.0018 inches	
204	Pipe	Yes	500 feet		7.981001 inches		Standard	0.0018 inches	
205	Pipe	Yes	50 feet		6.065001 inches		Standard	0.0018 inches	
206	Pipe	Yes	75 feet		6.065001 inches		Standard	0.0018 inches	
207	Pipe	Yes	50 feet		6.065001 inches		Standard	0.0018 inches	

**Figure 2: Model Data window display for the model in Figure 1. From this perspective it is much easier to see that one of the pipes (#2) has length data entered in miles rather than feet. This is not automatically an input error, as AFT xStream accepts pipe input with mixed engineering units. However, more frequently than not the user intends to use consistent units, so this probably is an error.**

## Related Blogs

[Having model convergence problems? Start here!](#)

### Transfer Results to Initial Guesses

AFT xStream's iterative steady-state solver takes initial guesses (also called initial conditions) and applies the principles of mass and momentum balance to improve the guesses. This is called iteration. It continues to refine the solutions until agreement with the governing equations is obtained.

After a converged steady-state answer is obtained, you can choose to take these results and use them as the initial guesses for a subsequent run. Using those results as initial guesses can lead to faster convergence in subsequent steady-state runs, as the solver can start at or very close to the actual solution.

**Note:** This will not increase the speed of the transient solution progress.

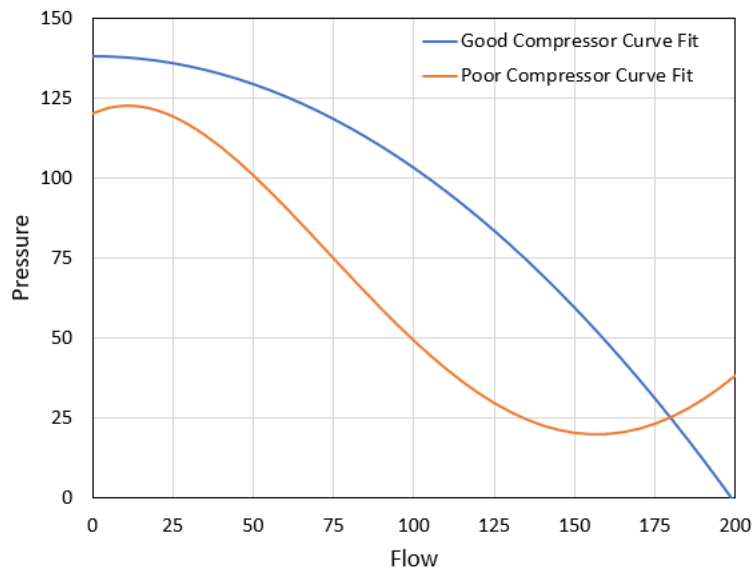
This feature is called Transfer Results to Initial Guess, and it is on the Output window, [Edit menu](#) and [tool-bar](#).

## Poor Compressor Curve Fits

The [Compressor/Fan Properties window](#) allows curve fits of compressor/fan data up to fourth order. Depending on the data points entered, compressor/fan curves fits similar to the orange or blue curves in Figure 1 below are possible.

AFT xStream can only work with the data the user inputs, and will use flow rates on the orange curve past the inflection point around 80 as acceptable operating points, which can lead to unusual results or non-convergence. Users should always ensure that their compressor/fan curves cross the flow rate axis as shown in the blue curve in Figure 1.

AFT xStream will attempt to warn the user when a model containing a poor compressor/fan curve fit is run. The user can choose to run the model with this curve fit anyways, but they should take care that the compressor/fan is operating in a region of the curve fit that matches physical expectations, such as the area to the left of the inflection point on the orange curve.

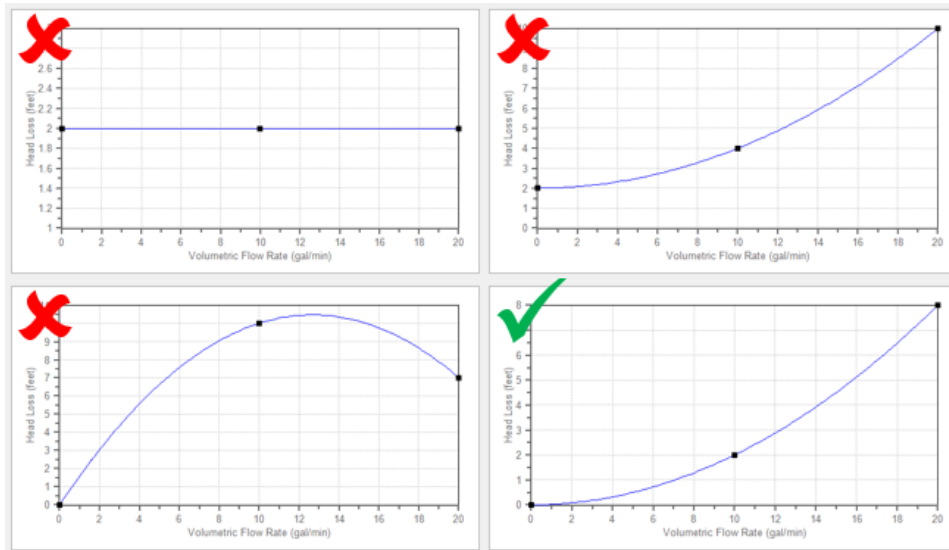


**Figure 1: A good compressor/fan performance curve fit in blue and a poor compressor/fan performance curve fit in orange. All compressor/fan curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.**

## Troubleshooting Resistance Curves

Common issues with resistance curves include curves where losses do not increase with flow, and curves with a non-zero loss at zero flow. Figure 1 below shows examples of curves with potential issues compared to a correctly defined curve.





**Figure 1: Example resistance curves showing correct and incorrect curve types. The incorrect curves either have non-zero loss at zero flow, or have losses which do not increase with flow. The correct curve has zero loss with zero flow and has losses which increase with flow.**

These common issues create two potential problems in a model.

First, if the system sees a constant pressure drop even at low flowrates, AFT xStream's iterative solver can encounter difficulty converging on a solution since the system flow and pressure drop are no longer dependent on each other.

Second, real equipment in a system will typically cause a higher pressure drop at larger flowrates, and will not provide any loss to the system at zero flow. Users should capture a single known pressure loss data point by defining a curve which passes through two points: the known pressure drop at the tested flowrate and zero pressure drop at zero flow. The topic on [Resistance Curves](#) gives more information on defining this type of curve.

## Related Topics

Resistance Curve Loss Model.....	208
General Component.....	251

## Use the Output Window Sort Feature

If a model runs but the results look incorrect, use the **Sort** feature on the [toolbar](#) in the [Output Window](#) to look for extremes in velocity, pressure drop or other parameters. An input error (like incorrect diameter) may be easier to see by looking at its effect on the results.

## Lower the Flow Rate Relaxation

The [AFT Arrow Steady Solution Panel](#) allows users to change multiple parameters, including the [numerical relaxation](#) parameter for both flow rate and pressure.

By default, an automatic relaxation is selected, and the relaxation will be enabled when AFT xStream detects the AFT Arrow Solution is getting stuck.

The user can also set this relaxation to a fixed value which is then applied for every iteration. Values of 0.5 or 1.0 are recommended for pressure relaxation, while values of 0.05 or 0.1 are recommended for flow rate relaxation.

### Related Blogs

---

[Having model convergence problems? Start here!](#)

## Try Absolute Tolerance

Most models converge reliably with relative tolerances, which is the default in AFT xStream's [AFT Arrow Steady Solution panel](#). However, some models converge better with [absolute tolerance](#). This situation usually means there is at least one element of the model that is close to the real answer in absolute terms but does not lock in on a relative (percentage) basis, likely due to a small-magnitude value for pressure or flow.

If a model will not converge with relative tolerance, try changing to absolute tolerance. Even better, try the option to converge on either absolute or relative, which will offer the most flexibility.

### Related Blogs

---

[Having model convergence problems? Start here!](#)

## Make Initial Flow Rate Guesses for Pipes

As an iterative solver, AFT xStream needs initial guesses to start the AFT Arrow Steady Solution process. xStream will make default initial guesses automatically. On occasion the AFT xStream guesses are far away from reality. For some models providing an initial guess for all the pipe flow rates in the system can help the model converge for the steady state.

User-defined initial guesses can be as simple as selecting all the pipes and initializing all of them to 1 lbm/sec, even though some operate at 0.1 lbm/sec and others at 10 lbm/sec. Note that initial guesses that are too far from the final solution can also interfere further with convergence.

### Related Topics

---

Optional Input for Pipes.....188

## Change Boundary Conditions

Changing the boundary pressures or flow rates can provide insight into what is giving the model trouble. For example, obtaining convergence only within a certain range of flowrates could indicate an issue with curve fit data being unrealistic outside of that range of flowrates.

## Turn Off Parts of the Model

Using the [Special Conditions](#) feature, users can turn off elements in your model and in effect shutdown entire sections. If the model runs when certain sections are closed, this could indicate a problem with one of the components in the closed section.

## Break a Large Model Into Submodels

AFT xStream provides convenient features for deleting parts of your model and merging parts back in. A section of your model can be quickly deleted by selecting it on the [Workspace](#) and choosing Delete from the [Edit menu](#). If the section is instead Cut, it can be Pasted into a new model of its own for testing or holding. At a later time the removed sections can be added back in to the current model by using the [Merge](#) feature on the File menu.

The advantage of breaking your model into sub-models is that you can investigate the convergence characteristics of each part of your model independently. If you can get each sub-model to converge, you should be able to get the original model to converge also. If one of your sub-models will not converge, it is doubtful your original model will converge either.

## Use an AFT Standard Fluid

AFT xStream offers multiple fluid databases for users to choose from in the [Fluids panel](#) of the [Analysis Setup window](#). Each database has benefits and drawbacks.

If the model is defined using the ASME Steam, Chempak, or NIST REFPROP databases and is experiencing convergence problems, the user should simplify the model by choosing a fluid from the AFT Standard fluid database. The fluid models used for this database are simplified from those in the other databases, and often allow AFT xStream to find a solution more easily.

Once the user has a model that runs well using an AFT Standard fluid, they can return to the more complex fluid databases.

It should also be noted that users are strongly encouraged to only use AFT Standard fluids in their AFT xStream model. The increased accuracy of other models is almost always outweighed by the increased run time associated with the more complex fluids.

## Related Topics

---

Fluid Panel.....	75
------------------	----

---

## Run the MOC Steady Solution

AFT xStream has three different Simulation Modes: AFT Arrow Steady, MOC Steady, and MOC Transient. By default, only the AFT Arrow Steady and MOC Transient solutions are available. The user can access the MOC Steady Simulation Mode by enabling Advanced Options in the Analysis Setup window.

The MOC Steady Solution is what xStream uses to move between the AFT Arrow Steady solution and the start of the MOC Transient Solution. Stopping the simulation after the MOC Steady Solution is obtained can help the user understand how the AFT Arrow Steady Solution is adjusted before the transient simulation starts. Occasionally, issues can arise during this process which need to be addressed prior to a successful MOC Transient simulation.

### Related Topics

---

AFT Arrow Steady Solution Group.....	104
Transient Solver Initialization Panel.....	98

## Simplify Event Transients

On occasion, the transient events defined for an AFT xStream model can lead to issues in the transient simulation. The user can use [Special Conditions](#) to ignore individual transients, re-run the simulation, and check to see if the issue persists. This troubleshooting tactic can be especially useful when dealing with check valves or control valves in the model.

## Limit Properties to Saturation Conditions

AFT xStream is a single-phase tool, operating under the assumption that the fluid is always in the gas/vapor phase. When saturation is reached, xStream will attempt to extrapolate gas properties past the saturation conditions. In some situations, this approach is successful, and the transient simulation can proceed. However, in other situations, the MOC Transient solution will fail as it attempts to extrapolate gas properties. In this situation, it can be helpful to limit property calculations to saturation conditions and not allow xStream to extrapolate properties.

The user can limit properties to saturation conditions in the [Advanced Transient Parameters panel](#) of the Analysis Setup window after checking the Show Advanced Options box in the lower left corner of the window.

This setting can also be useful outside of troubleshooting situations. When limited amounts of transient condensation is seen, it is unlikely that a full phase change will occur, meaning the temperature and pressure will be constant at the saturation conditions. Thus, in some situations, it may be more accurate to limit properties to saturation conditions if the system only briefly reaches saturation during a transient simulation.

### Related Topics

---

Advanced Transient Parameters Panel.....	139
Fluid Panel.....	75

---

Engineering Assumptions..... 22

## Enable Transient Debug Files

AFT xStream has the ability to write out Debug files containing useful information about events that happen during a transient simulation. xStream will write out information about the MOC Steady Solution process and information about any errors it encounters during a transient simulation.

The user can enable transient debug files in the [Advanced Transient Parameters panel](#) of the Analysis Setup window after checking the Show Advanced Options box in the lower left corner of the window.

Once the user has checked the Create Transient Debug File option, xStream will save a Transient Debug file each time a scenario is run in an xStream model. The user can view the information written to investigate the issue themselves, or work with AFT's support team to resolve the issue. In the event that a user requests support from AFT but cannot send a model file, the AFT Support team will likely ask the user to generate this file.

## Technical Support

This section focuses on the technical support options for AFT Products. Topics include descriptions of the various support options, contact information for the AFT Support team, and links to external sites to assist with installation issues related to AFT xStream.

### Contents

---

<a href="#">Technical Support Options</a> .....	530
<a href="#">Contacting AFT</a> .....	531
<a href="#">Information on Software Installation</a> .....	531
<a href="#">Chempak Database Installation</a> .....	531
<a href="#">Updated Fluids Library</a> .....	533
<a href="#">Special File Control Features</a> .....	534
<a href="#">Converting Models from Earlier Versions</a> .....	534

### Related Blogs

---

[Having model convergence problems? Start here!](#)

## Technical Support Options

### Support, Upgrade and Maintenance (SUM)

---

This primary level of support is for assistance with specific features within the software, or resolving issues with a model such as difficult to converge models. We are limited in our ability to interpret model results, and cannot recommend specific changes in a system.

One full year of SUM is provided with the purchase of a full license. Leases have SUM coverage during the period of the lease. Further support after the first year requires SUM renewal.

For technical support questions falling in this category, [contact AFT](#).

## Flow Expert Package (FEP)

---

The FEP is a higher level of consulting assistance for a particular project. This service will assign a qualified AFT engineer to assist in not only modeling your system, but also with providing operational suggestions and interpretation of results.

The intention of this service is to provide assistance to active users of AFT software and address questions on the appropriate use of the software for specific system requirements.

Please visit our website at: <https://www.aft.com/products/aft-flow-expert-package> for more information on our FEP services.

## Flow Modeling Consulting Services

---

AFT's sister company, Purple Mountain Technology Group (PMTG), provides comprehensive consulting services utilizing experienced consulting engineers and AFT tools. PMTG offers a solution to a wide variety of system analysis by building and investigating the models and providing direction on system changes and appropriate operation. Visit PMTG on the web: <https://www.pmtchgroup.com/>, or call at (719) 687-9145 for more information.

## Contacting AFT

- **Telephone** - (719) 686-1000
- **Fax** - (719) 686-1001
- **Online** - <https://www.aft.com/> download the latest maintenance releases, find out what is happening at AFT, and get all the latest information.
- **E-mail Support** - [support@aft.com](mailto:support@aft.com)
- **Mail** - 2955 Professional Place (Suite 301), Colorado Springs, CO 80904, USA

## Information on Software Installation

For information on Local and Server installations of AFT software, please visit our website at <https://www.aft.com/support/installation>.

For answers to frequently asked licensing questions, please visit our [eLicense](#) or [USB Key](#) FAQ pages.

## Chempak Database Installation

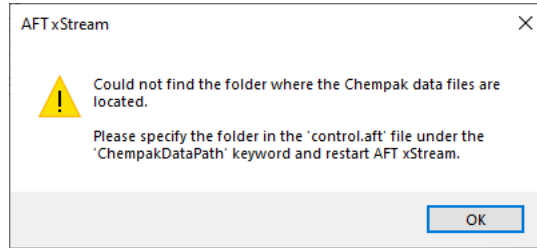
AFT xStream does not include the [Chempak Database](#) by default. The Chempak Database is a separate product that must be downloaded and installed as an addition to AFT xStream.

---

## Database File Location

---

The Chempak Database is installed in **C:\AFT Products\Chempak Data\** by default. If both AFT xStream and Chempak Database are installed to their default locations, no further configuration is necessary. However, if AFT xStream or the Chempak Database are not installed in a default location, AFT xStream will need to know where to look for the Chempak Database. This is specified in the **control.aft** file, located in the same folder as xStream.exe. If the files cannot be found the following message is displayed:



*Figure 1: Error message when Chempak data cannot be found in default location*

**Control.aft** can be edited with a text editor, and by default looks like:

```
control.aft - Notepad
File Edit Format View Help
[SEARCH PATHS]
ChempakDataPath =
CrypKeyPath =

[CONTROL FLAGS]
ProtectionType = 0
DebugTokens = False
```

*Figure 2: Default control.aft file*

If the Chempak Database was installed to a location other than the default, the line "ChempakDataPath = " will need to be edited to include the directory it was installed to. For example: "ChempakDataPath = C:\My AFT Files\Chempak Data"

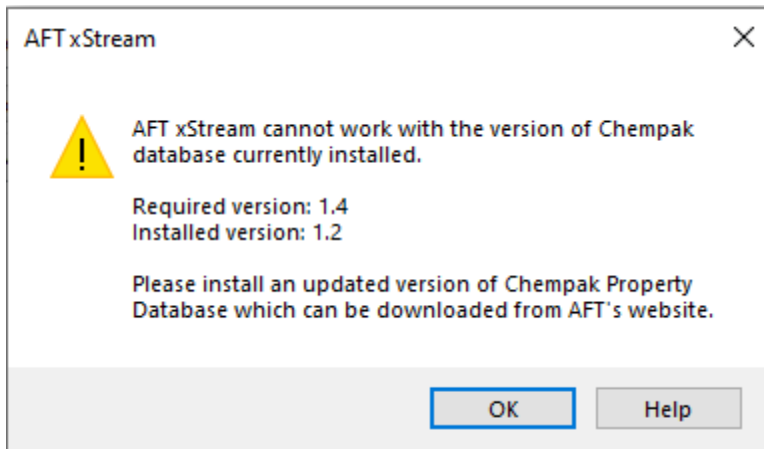
---

## Upgrading Chempak Database Version

---

If you had Chempak installed with other AFT products, you may receive a message that the library is outdated.



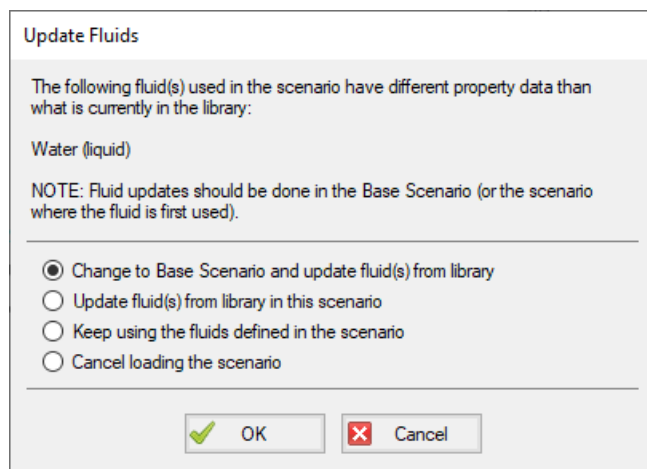


**Figure 3: Outdated Chempak data**

To fix this issue, download the newest version of the Chempak Database from the [AFT website](#). Be sure to install to the same location as the existing Chempak Database, and double check the control.aft file if necessary.

## Updated Fluids Library

If the data in a shared fluid library is changed outside of the model you are opening, the following informational message will appear when opening the file.



**Figure 1: Update Fluids dialog window which appears if the fluid properties for the model have changed since it was last opened**

This message is shown because the model is defined with a custom fluid in the fluid library and the custom fluid was modified outside of this model. If the fluid was changed in error, then choose the option the "Keep using the fluids defined in the scenario" or cancel loading the model and change the fluid. If the changes were intended then update the fluid data used in the model using the "Change to Base Scenario and update..." option.

## Special File Control Features

### What Is the Control.aft File?

---

Control.aft is read as AFT xStream begins to load and instructs AFT xStream where certain key files and copy protection tokens are located. Typically there is little need to change the settings, however this file can be used to override the internal defaults and customize your working environment.

Control.aft is typically located in the application folder. However, on networks, it may be necessary to locate it in the Windows\System folder of each client. If AFT xStream finds the file in the Windows\System folder it will use it and ignore any file in the application folder.

Control.aft can be edited using any text-based editor. Notepad, supplied as part of the Windows installation, is a good choice. Be careful to avoid changing any of the headings or key words. If these are changed AFT xStream will not find the important data.

### Specifying Where Chempak is Located

---

Before AFT xStream can use the [Chempak Fluid Database](#) it needs to know where both the data is located. These paths are specified in the control.aft file.

When Chempak is installed the setup procedure automatically updates any control.aft file specified with the path to the data.

You can manually edit the file by opening it in Notepad (or any text editor) and adding the following line to the [SEARCH PATHS] section (replace C:\AFT Products\Chempak Data with the path information based on your system):

```
ChempakDataPath = C:\AFT Products\Chempak Data
```

This will instruct AFT xStream to look in the C:\AFT Products\Chempak Data directory for the data.

Also see [Chempak Data Location](#).

## Converting Models from Earlier Versions

Models from earlier version of AFT xStream can be used with this version. In most cases, the model file can be opened in the newer version and automatically converted to the updated format. It is considered good practice to do a complete check on the model to ensure the conversion process was complete and accurate. In some cases, especially with older models, the automatic conversion is not successful. If this occurs, please contact our support team with a copy of the model file.

Other files such as libraries ([previously databases](#)) or preference files can also be converted.

## Examples Overview

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

### Tank Blowdown Example (English Units)

#### Summary

---

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

#### Topics covered

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

#### Required knowledge

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

#### Model file

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

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

#### Step 1. Start AFT xStream

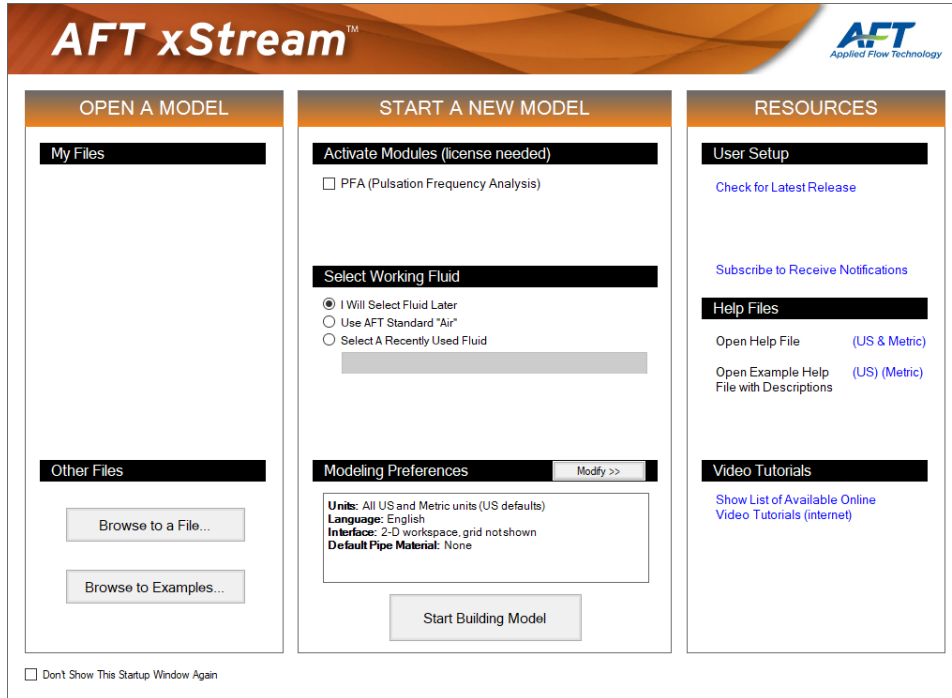
---

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

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

## Tank Blowdown Example

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



**Figure 1: AFT xStream Startup window**

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

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

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

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

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

The Workspace window is the primary interface for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox is the bundle of tools on the far left. The Quick Access Panel is displayed on the far right. It is possible to collapse the Quick Access Panel by clicking on the thumbtack pin in the upper right of the Quick Access Panel in order to allow for greater Workspace area. The Workspace takes up the rest of the window.

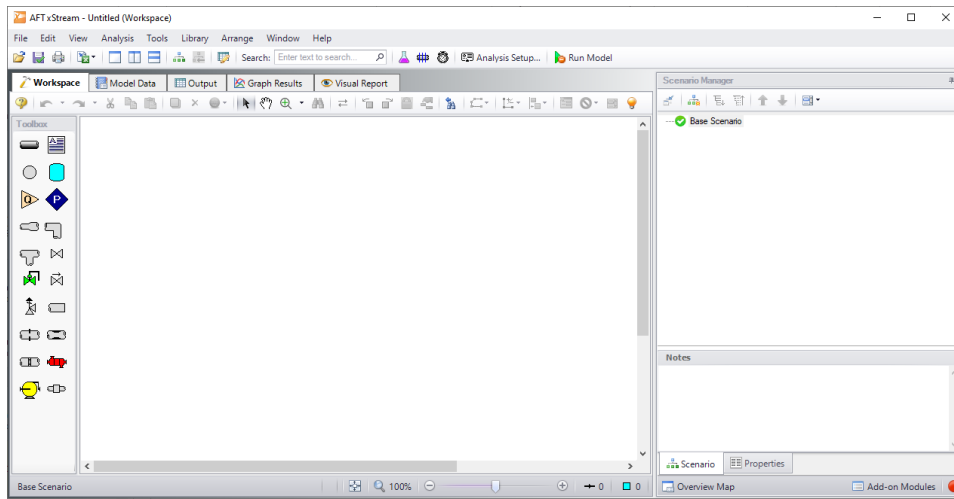
## Tank Blowdown Example

---

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

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

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



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

## Step 2. Lay out the model

---

To lay out the tank blowdown model, you will place a Tank junction, a Valve junction, and an Assigned Pressure junction on the Workspace. Then you will connect the junctions with pipes.

### I. Place the Tank junction

➤ **To start, click and drag a Tank junction from the Toolbox and drop it on the Workspace.** Figure 3 shows the Workspace with one Tank.

#### **Objects and ID numbers**

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

All pipe and junction objects on the Workspace have an associated ID number. For junctions, this number is, by default, placed directly above the junction and prefixed with the letter "J". Pipe ID numbers are prefixed with the letter "P". You can optionally choose to display either or both the ID number and the

name of a pipe or junction. You also can drag the ID number/name text to a different location to improve visibility.

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

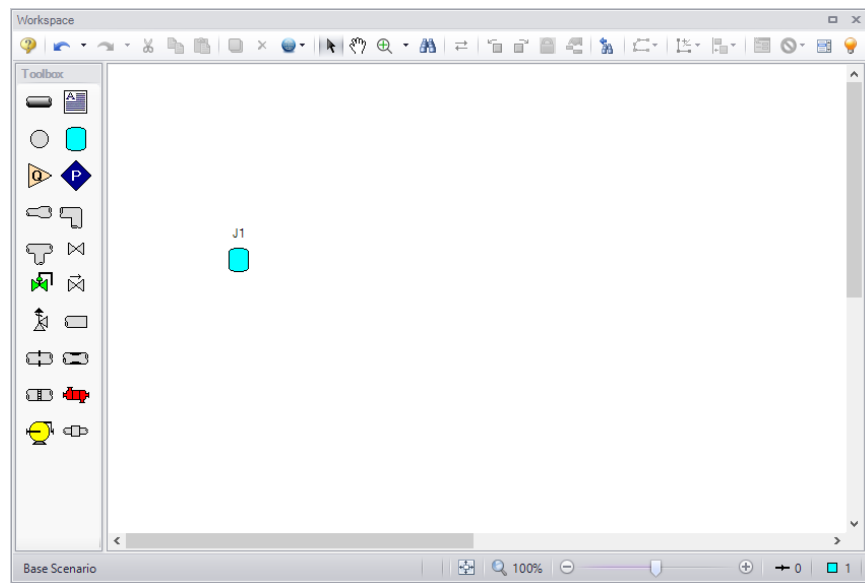
### ***Editing on the Workspace***

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

---

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

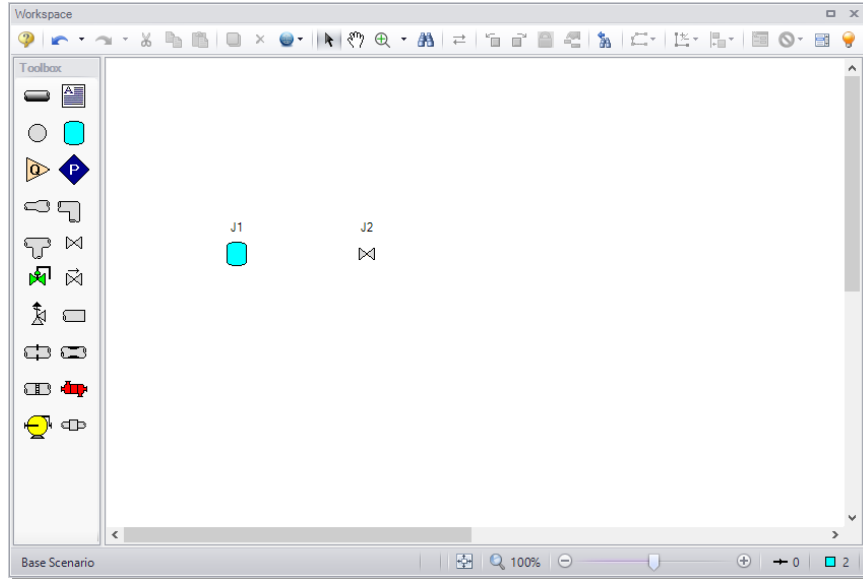
---



***Figure 3: Tank junction placed***

## **II. Place the Valve junction**

➤ **To add a Valve junction**, click and drag a valve from the Toolbox and place it on the Workspace as shown in Figure 4. The Valve will be assigned the default number "J2".



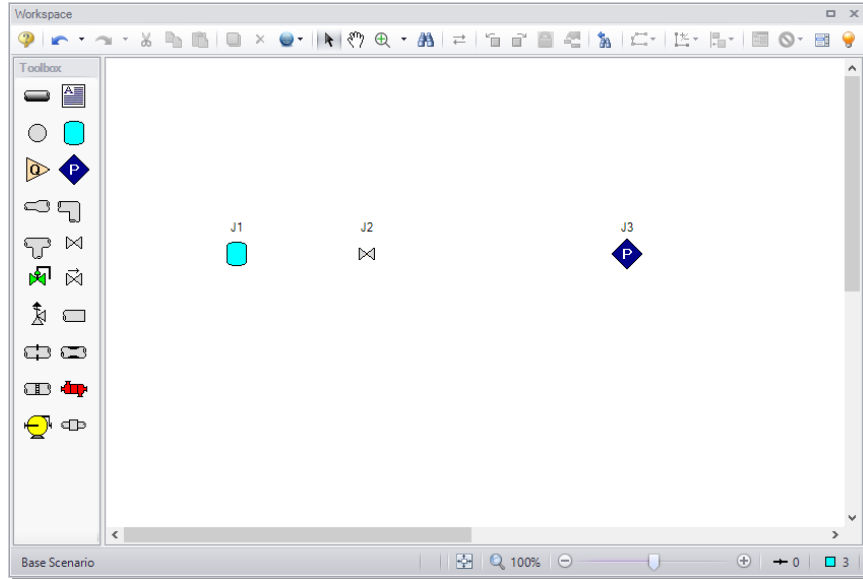
**Figure 4: Tank Blowdown model with Valve placed**

### III. Place an Assigned Pressure junction

➤ **To add an Assigned Pressure junction**, click and drag an Assigned Pressure junction from the Toolbox and place it on the Workspace as shown in Figure 5. The Assigned Pressure junction will be assigned the default number "J3".

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





**Figure 5: All junctions placed**

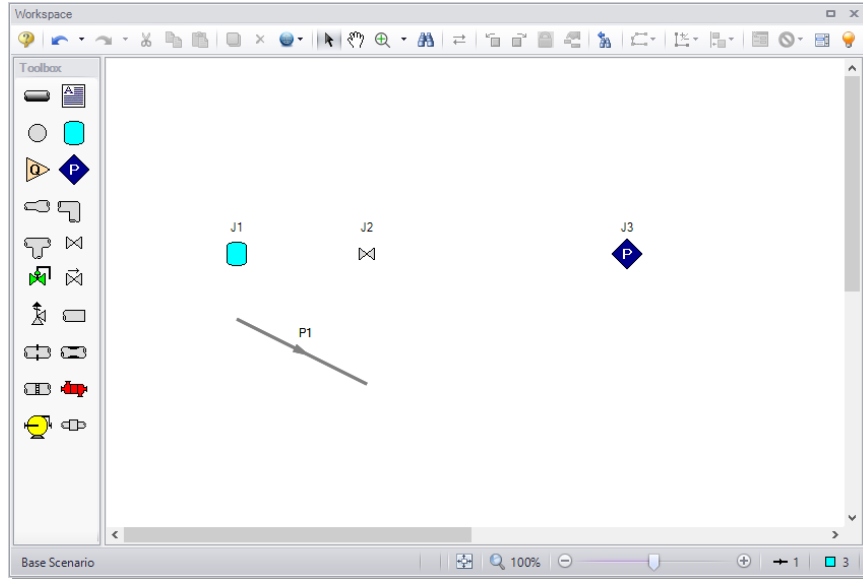
#### **IV. Draw a pipe between J1 and J2**

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

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

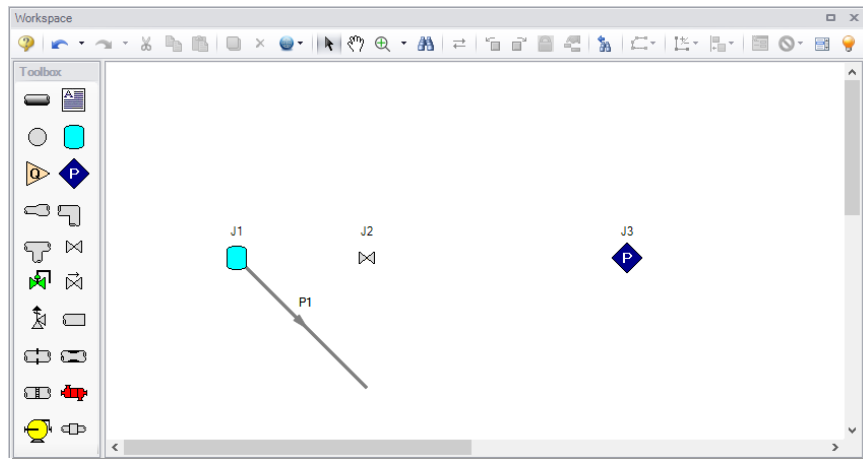
The pipe object on the Workspace has an ID number (P1) that is shown near the center of the pipe.

## Tank Blowdown Example



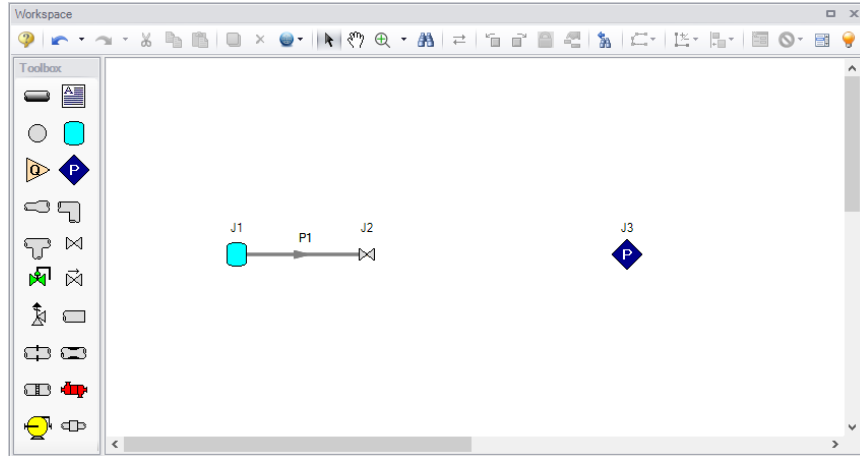
**Figure 6: First pipe drawn**

➤ **To place the pipe between J1 and J2**, use the mouse to grab the pipe in the center and drag it so that the left endpoint falls within the J1 Tank icon, then drop it there (see Figure 7). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J2 Valve icon (see Figure 8).



**Figure 7: First pipe partially connected**

## Tank Blowdown Example



**Figure 8: First pipe fully connected**

### **Reference positive flow direction**

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

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

### **V. Add the remaining pipe**

A faster way to add a pipe is to draw it directly between the desired junctions.

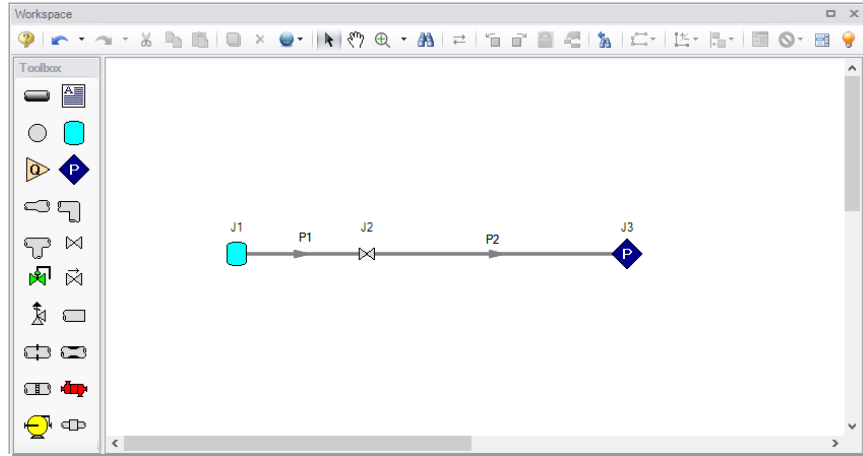
➤ **Activate the Pipe Drawing Tool again.** Position the cursor on the J2 Valve, then press and hold the left mouse button. Stretch the pipe across to the J3 Assigned Pressure junction, then release the mouse button. All objects in the model should now be graphically connected as they are in Figure 9.

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

---

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

---



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

### A. Define the model components

#### **Object status**


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

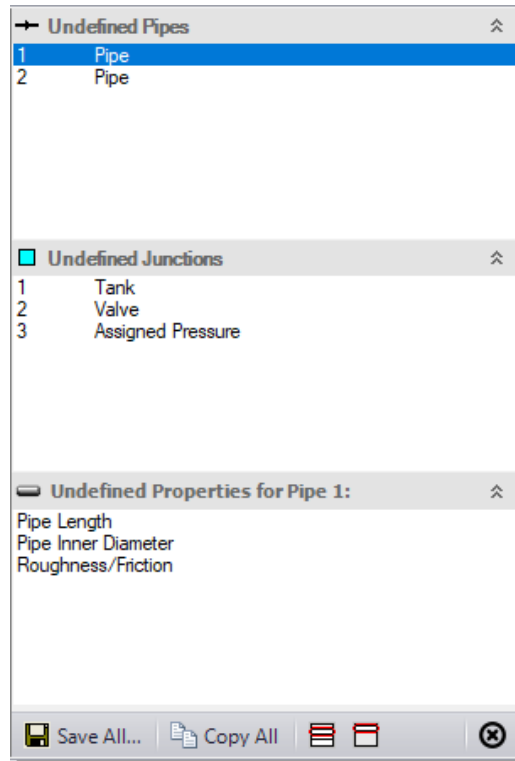
When "Show Object Status" is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. (These colors are configurable through User Options from the Tools menu.)

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

#### **Showing undefined objects**

Another useful feature is the List Undefined Objects window (Figure 10). This can be opened from the View menu by clicking on "List Undefined Objects", or by clicking the List Undefined Objects icon on the

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



**Figure 10:** The List Undefined Objects window lets you see the undefined properties for each undefined object

### I. Enter data for Tank J1

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

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

---

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

---

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

---

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

---

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

## Tank Blowdown Example

---

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

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

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

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

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

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

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

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

## Tank Blowdown Example

Tank Properties

Number: 1

Name: Supply Tank

Library Jct:

Copy Data From Jct...:

Elevation: 0 feet

OK

Cancel

Jump...

Help

Tank Model Optional Design Alerts Notes Status

Tank Model

Infinite

Finite

Initial Conditions

Pressure: 600 psia

Temperature: 300 deg. F

Enthalpy: 300 deg. F

Finite Tank Parameters

Fixed Volume: 300 feet3

Initial Conditions

Known

Calculate From Steady State

Thermodynamic Process

Adiabatic

Isothermal

Known Heat and Volume vs. Time

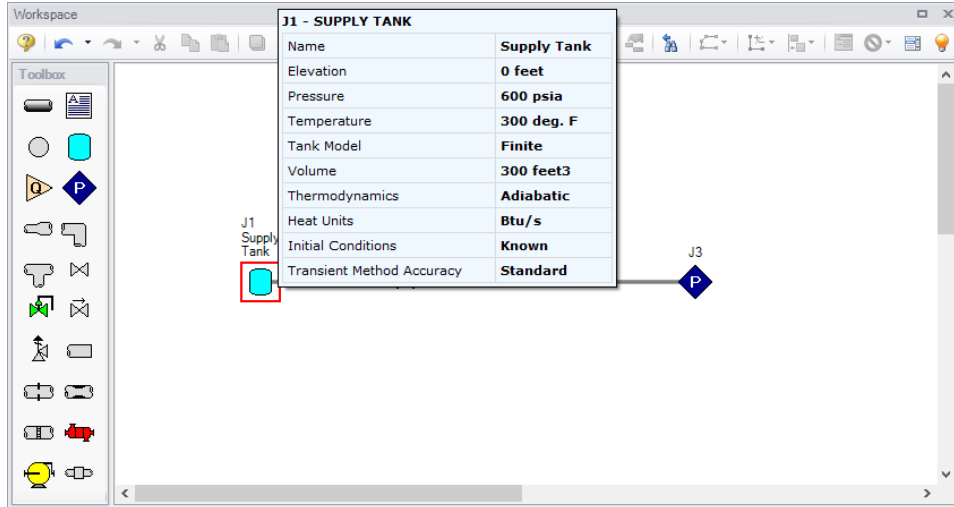
**Figure 11: Properties window for Tank J1**

### ***The Inspection feature***

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspection feature. Position the cursor on the Tank J1 junction and hold down the right mouse button. An information box appears, as shown in Figure 12.

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

# Tank Blowdown Example



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

## II. Define Valve J2

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

<i>Time (seconds)</i>	$C_v$	$X_t$
0	0	0.7
0.1	60	0.7
240	60	0.7

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

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

The J2 Valve is the element which causes the transient in this model. The purpose of the model will be to understand how the conditions within the supply tank and discharge pipe change after the valve opens.



## Tank Blowdown Example

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

Valve Properties

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

OK  
Cancel  
Jump...  
Help

Loss Model Transient Optional Design Alerts Notes Status

Subsonic Loss Model  
 Cv (ANSI/ISA) ?  
 Kv

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

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

Open Percentage Data

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

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

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

Exit Valve (optional)  
Exit Pressure:   
Exit Temperature:

Figure 13: Input data for Valve junction J2

# Tank Blowdown Example

Valve Properties

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

Loss Model  Transient  Optional Design Alerts Notes Status

Transient Special Condition  
 None  
 Ignore Transient Data

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

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

Data Point	Time (seconds)	Cv	Xt
1	0	0	0.7
2	0.1	60	0.7
3	240	60	0.7
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			

Xt Source  
 User  
 Open % Table  
Fill Xt from table

Edit Table Show Graph...  
 Repeat Transient  
Transient

OK Cancel Jump... Help

Figure 14: Transient input data for Valve junction J2

### III. Define Assigned Pressure J3

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

## Tank Blowdown Example

Assigned Pressure Properties

Number:

Name:

Library Jct:

Copy Data From Jct...:

Elevation:

OK Cancel Jump... Help

Pressure Model Loss Coefficient Transient Optional Design Alerts Notes Status

Boundary Specification

Stagnation  Static

Stagnation Pressure:

Stagnation  Temperature  Enthalpy

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

### IV. Define Pipe P1

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

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

## Tank Blowdown Example

The screenshot shows the 'Pipe Properties' dialog box for 'Pipe P1'. The 'Pipe Model' tab is selected. The 'Number' field is set to 1, and the 'Name' is 'Pipe'. The 'Upstream Junction' is 1 and the 'Downstream Junction' is 2. The 'Pipe Material' is 'Steel - ANSI', 'Size' is '2 inch', 'Type' is 'STD (schedule 40)', and 'Inner Diameter' is '2.067 inches'. The 'Length' is '20 feet'. The 'Friction Model' section has 'Standard' selected under 'Data Set' and 'Absolute Roughness' set to '0.0018 inches'. The 'Load Default' button is visible next to the roughness field.

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

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

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

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

Click OK to finish defining P1.

## V. Define Pipe P2

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

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

## B. Review model data

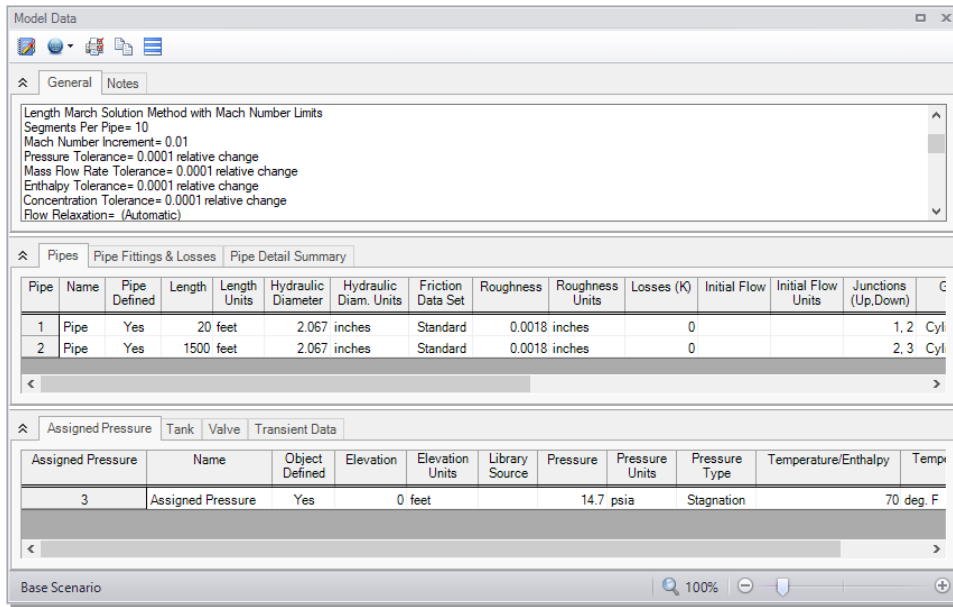
➤ **Save the model.** It is also a good idea to review the input using the Model Data window.

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

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

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

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



**Figure 17: The Model Data window displays all input in text form**

## Tank Blowdown Example

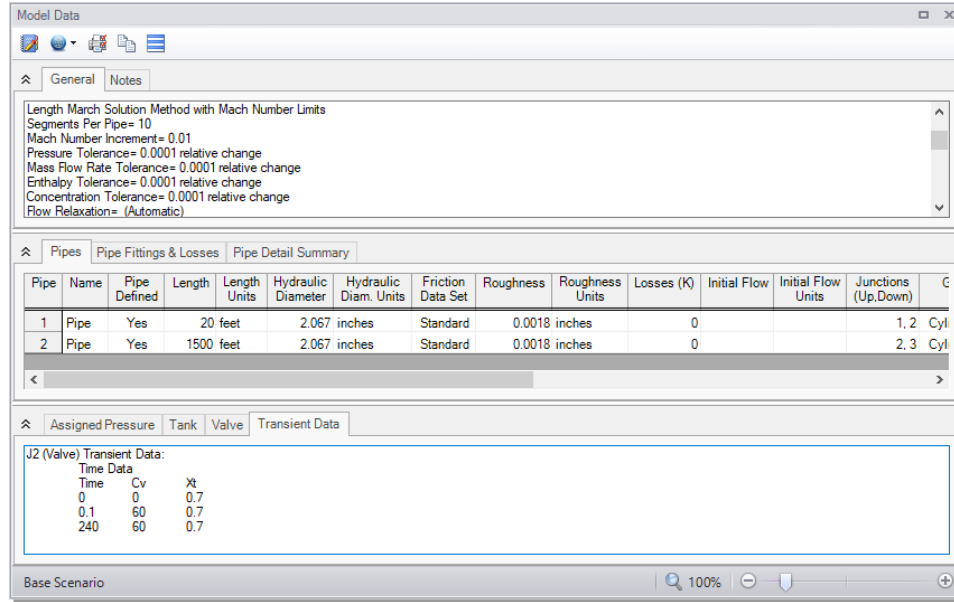


Figure 18: The Transient data tab in the Model Data window displays transient input data

### Step 3. Specify Analysis Setup

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

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

# Tank Blowdown Example

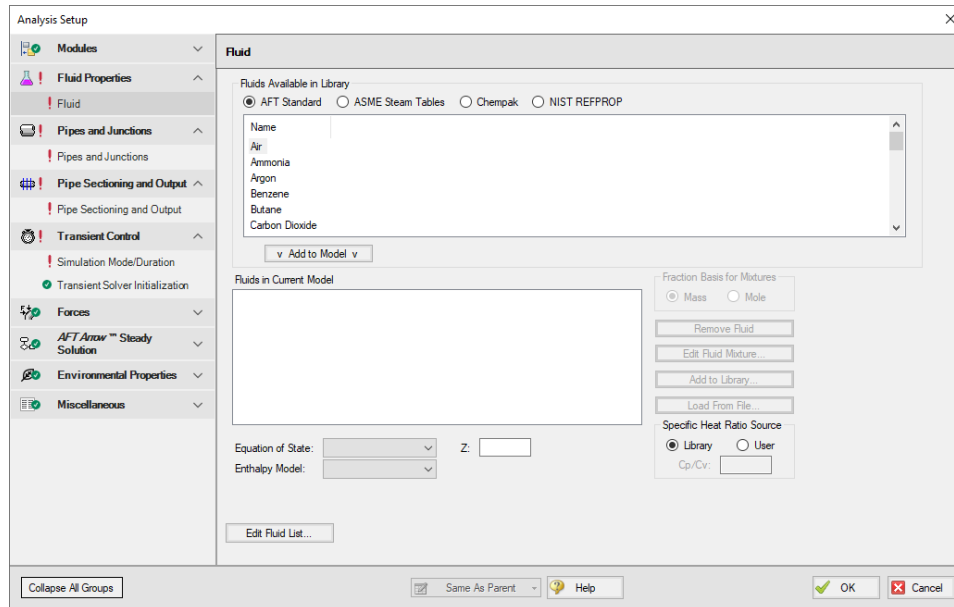


Figure 19: The Analysis Setup window tracks the defined and undefined model input

## A. Specify the Modules group

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

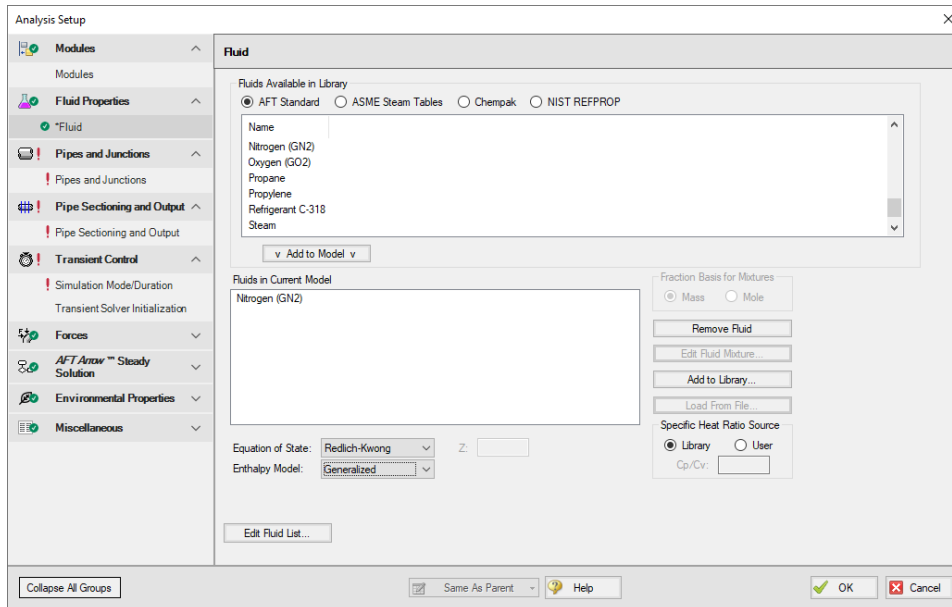
## B. Specify the Fluid Properties group

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

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

You can select a fluid from the standard AFT xStream fluid library (AFT Standard), or select ASME Steam Tables. Additionally, you can select multiple fluids and/or create mixtures using the included NIST REFPROP library or optional add-on Chempak library, or create custom fluids. Custom fluids are created by opening the Library Manager from the Library menu or by clicking the Edit Fluid List button in the Fluid panel and clicking "Add New Fluid...".

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



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

## C. Inspect the Pipes and Junctions group

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

### *Running models in steady-state*

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

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

## D. Specify the Pipe Sectioning and Output group

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

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



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

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

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

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

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

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

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

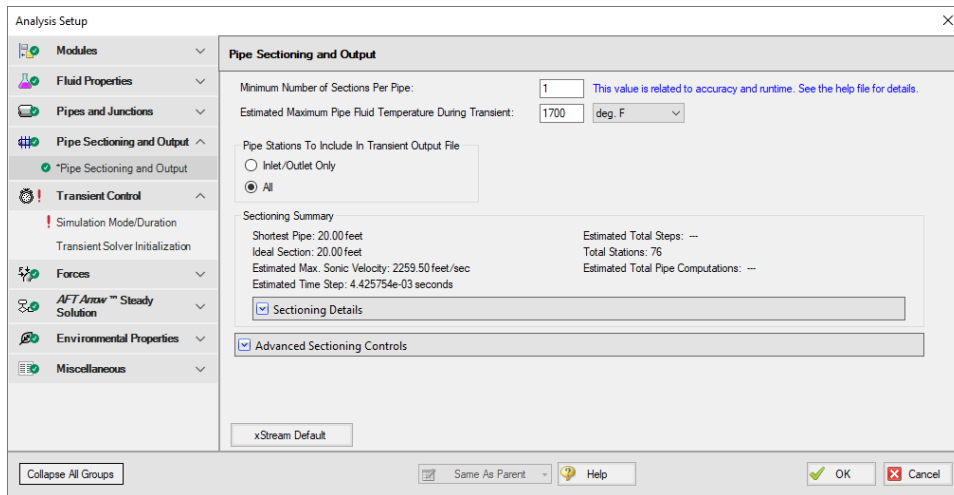


Figure 21: The Sectioning panel in the Analysis Setup window is used to calculate the time step of the model

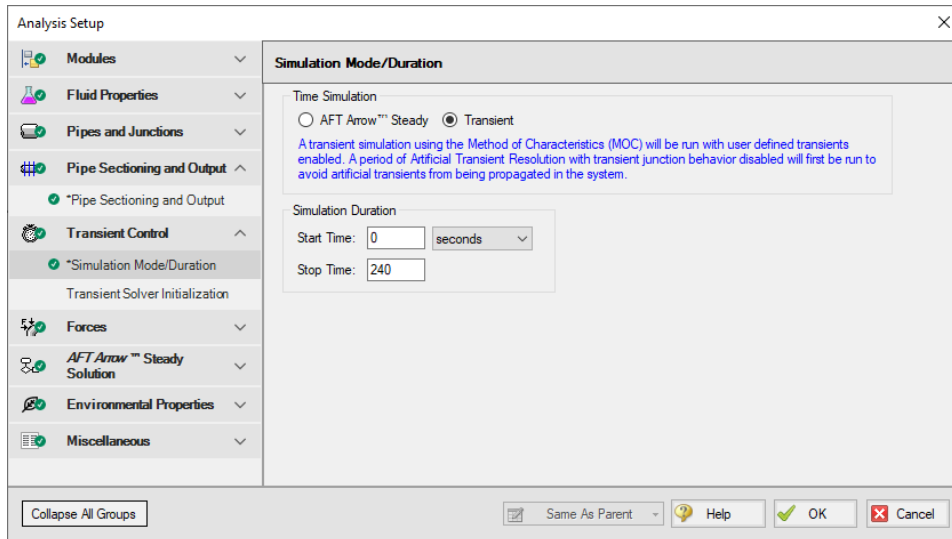
## E. Specify the Transient Control group

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

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

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

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



**Figure 22: The Simulation Mode/Duration panel is used to specify the simulation type and the time span for the transient**

➤ **Save the model.**

## Step 4. Run the solver

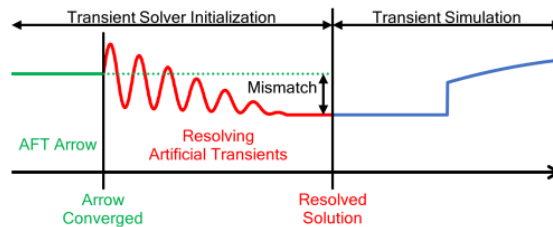
➤ **Choose "Run Model..." from the Analysis menu** or click the "Run Model" button on the Main Toolbar. During execution, the Solution Progress window displays the state of the simulation (Figure 24). You can use this window to pause or cancel the solver's activity.

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

### The two solvers

AFT xStream has two solvers. The first is called the *AFT Arrow*<sup>TM</sup> 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*<sup>TM</sup> Steady solver and Transient solver use different sectioning methods, which causes small discrepancies between the converged *AFT Arrow*<sup>TM</sup> 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*<sup>TM</sup> Steady solution. If the mismatch between the beginning and end of the artificial transient resolution process is sufficiently small AFT xStream proceeds with the transient simulation. See the chart in Figure 23 as a visual representation of the process described above.



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

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

## Tank Blowdown Example

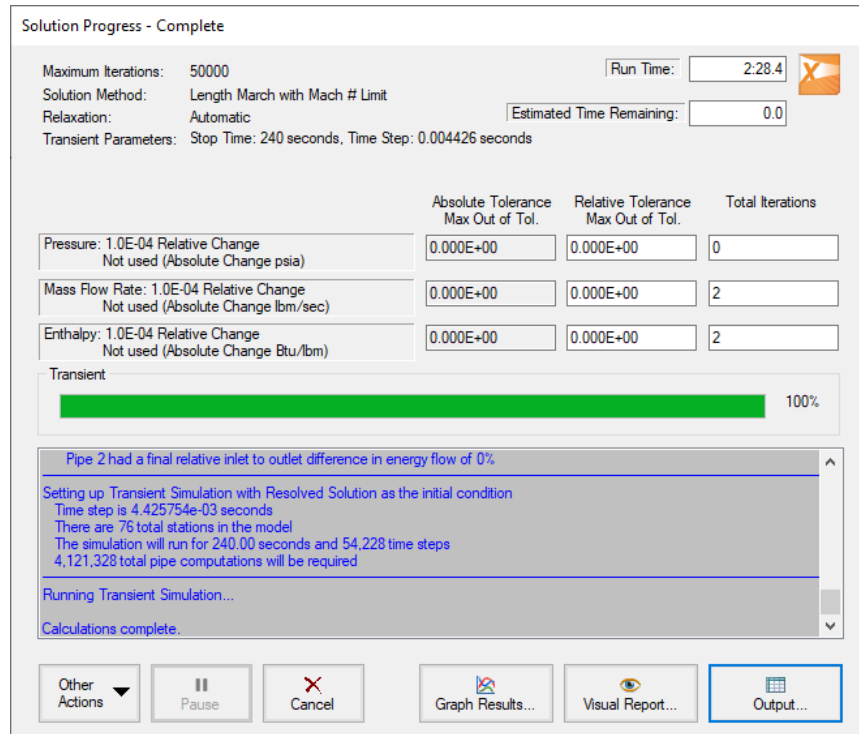


Figure 24: The Solution Progress window displays the state of the simulation

### The transient output file

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

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

## Step 5. Review the Output window

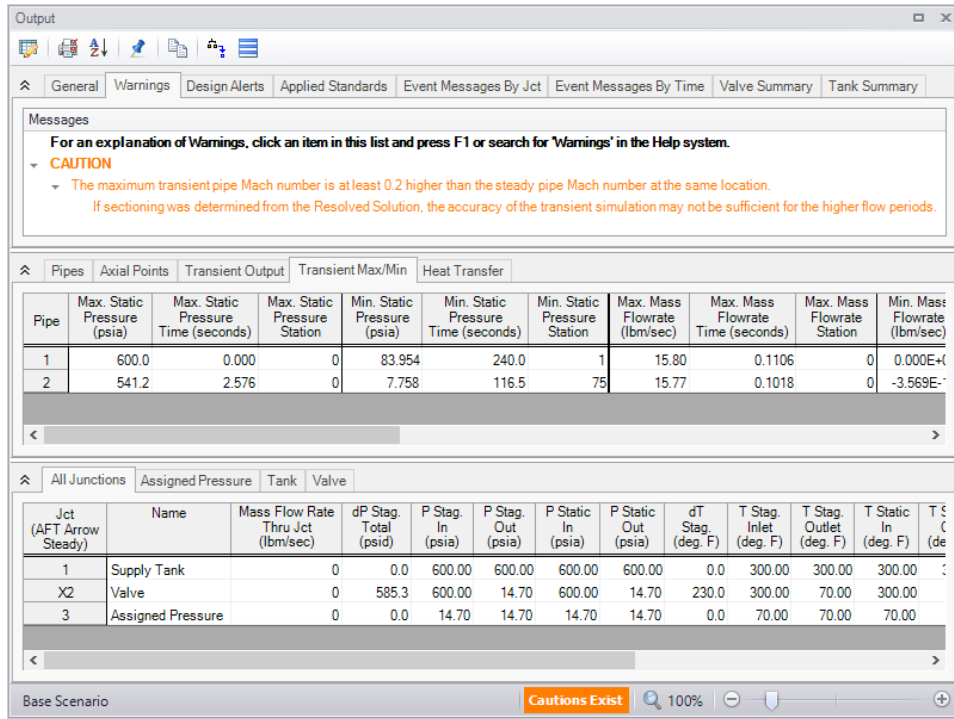
Click Output after the solver is finished. The Output window (Figure 25) is similar in structure to the Model Data window. Three areas are shown, and you can minimize or enlarge each section by clicking the arrow next to the General, Pipes, and All Junctions tabs. The items displayed in the tabs can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

The General section will open by default to the Warnings tab if any Warnings or Cautions were encountered during the simulation. This model should generate a Caution that the Maximum Transient Pipe Mach Number is at least 0.2 higher than the steady pipe Mach number, which occurs since there is no flow in the system at steady state. When this Caution is present, the effect of pipe sectioning on results can be significant. The impact of pipe sectioning will be addressed in greater detail later in Step 10.

## Tank Blowdown Example

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

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



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

## Tank Blowdown Example

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

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

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

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

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

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

### A. Modify the output format

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

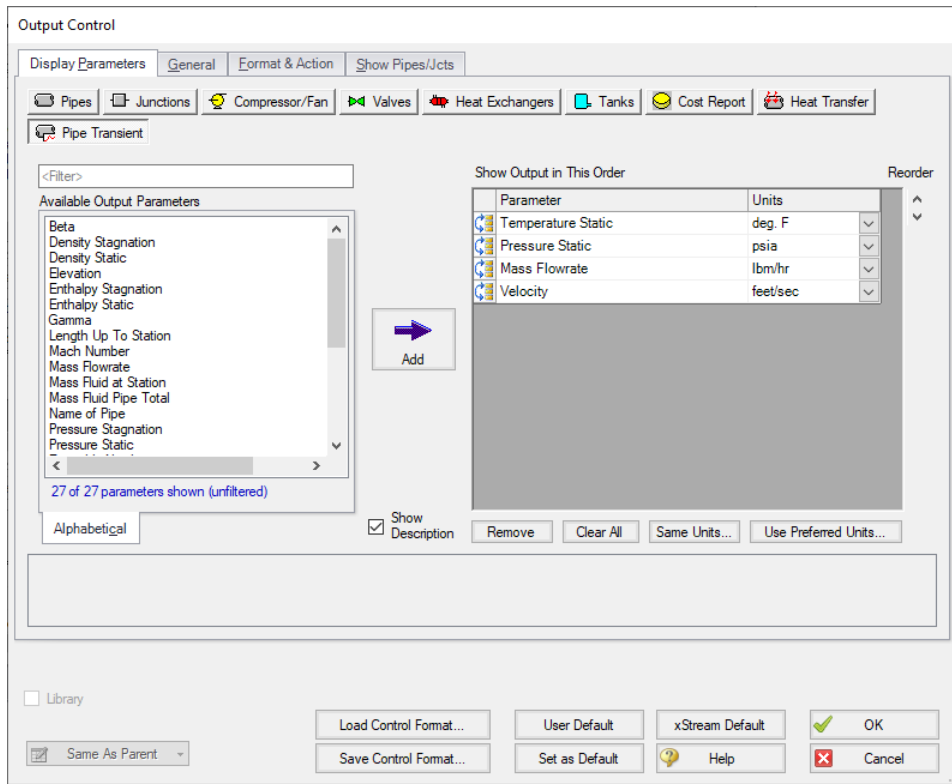
➤ **Select Output Control from the Tools menu or the main Toolbar.** Open the Pipe Transient results section. On the right side of the Pipe Transient results section is the list of currently selected output parameters (see Figure 29). Click and change the units by clicking the arrow beside the units,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

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

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

## Tank Blowdown Example

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




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

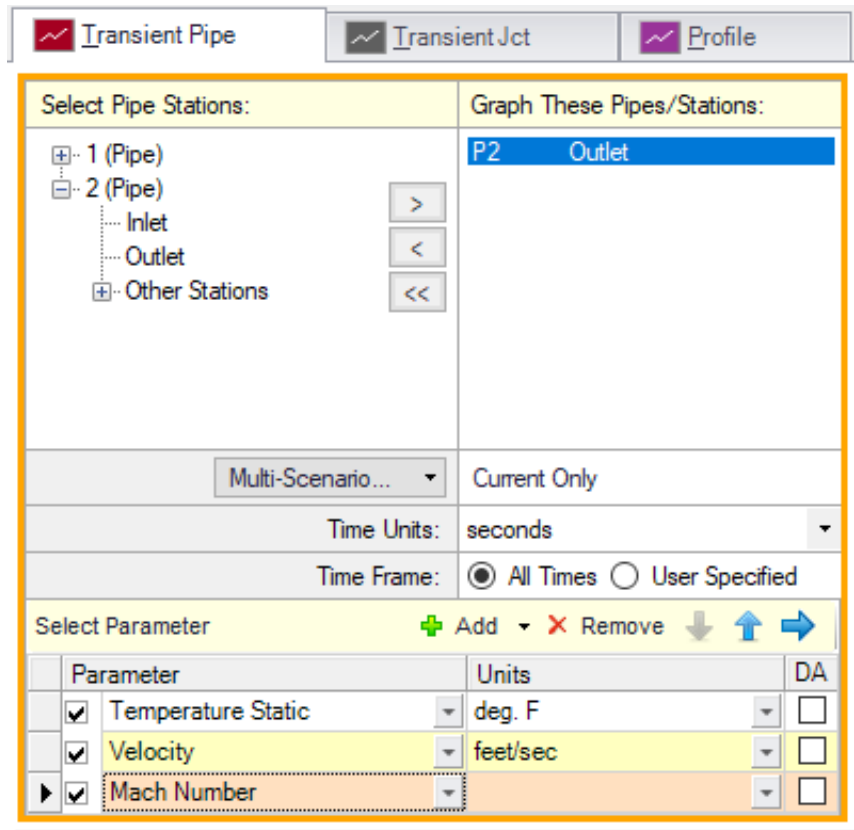
### B. Graph the results

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

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

## Tank Blowdown Example

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



**Figure 30:** The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area

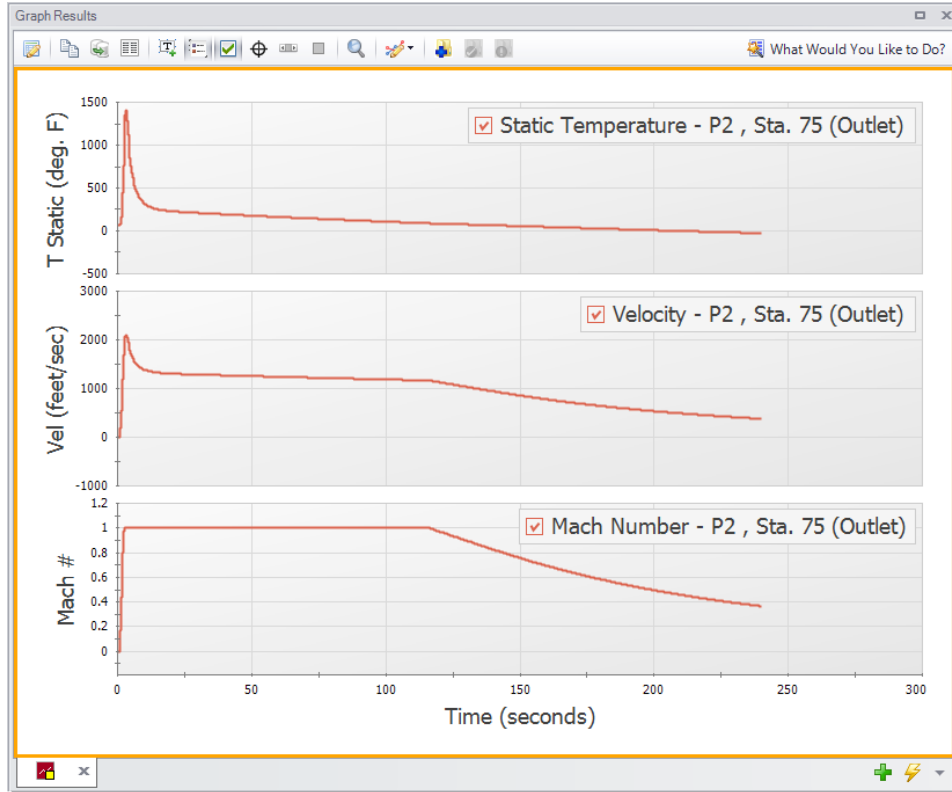
➤ **Click the Generate button.** The graph shows the temperature, velocity, and Mach number at the exit for Pipe 2 (Figure 31).

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

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



## Tank Blowdown Example



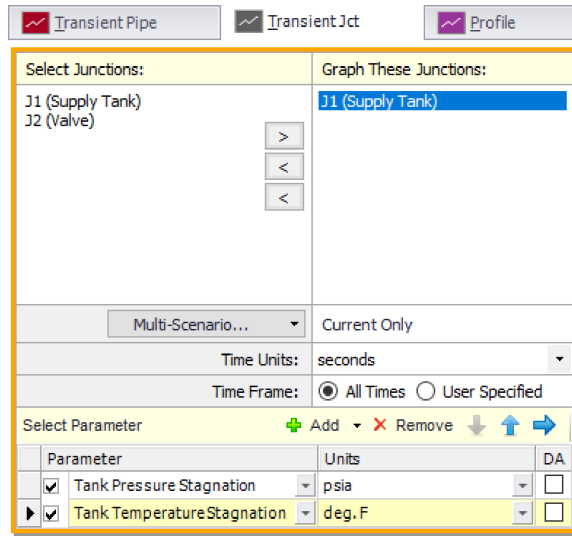
**Figure 31: The Graph Results window offers full-featured plot generation. The static temperature, velocity, and Mach number of Pipe P2 outlet are shown.**

Further review of the graph results in Figure 31 shows that sonic choking occurs after several seconds as was seen on the Event Messages tab in the Output window, and continues for the next 115 seconds. Also note that the temperature, velocity and Mach number all rise sharply shortly after the valve opens, with the Mach number rapidly reaching one. After the initial rise, the temperature and velocity both decay while the Mach number remains at one. The velocity drops because sonic velocity has a positive correlation with temperature, so as the temperature decays, the velocity required for choked flow also decreases. It is good to keep in mind this correlation between temperature and sonic velocity. This relationship is important to understand when looking at systems that encounter choking.

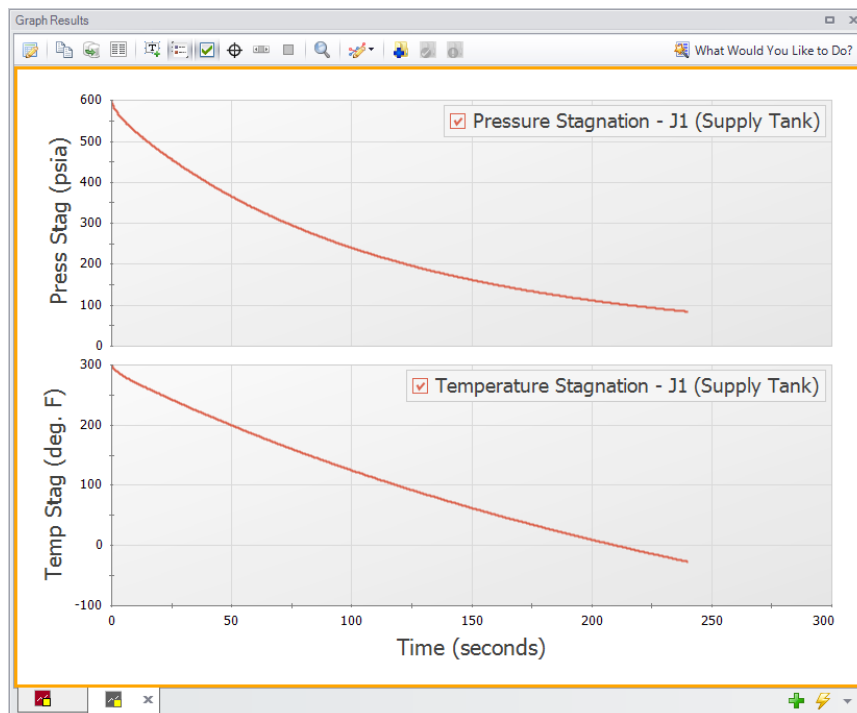
➤ **In the Quick Access Panel**, open the Transient Jct Tab. Under Select Junctions, add J1 (Supply Tank). Select the Graph Parameters as Tank Pressure Stagnation and Tank Temperature Stagnation (see Figure 32). Set the units to psia and deg. F. Click the Generate button. This will show the temperature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

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

## Tank Blowdown Example



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



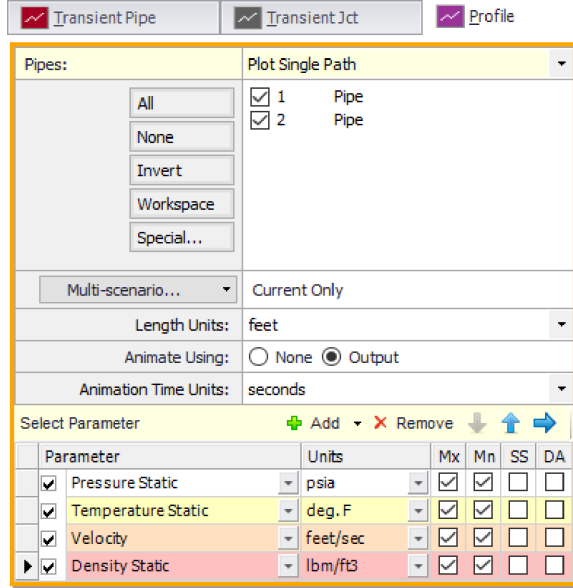
**Figure 33: The Transient Junction graph shows how variables within a junction change over time. The temperature and pressure of Tank J1 over time are plotted.**

This junction graph could be used to determine when certain conditions within the tank were met. For instance, suppose there was a need to know when the tank pressure dropped below 300 psia. The junction graph would indicate the tank pressure would reach such a point around 75 seconds after beginning of the tank blowdown. Also note that the rate of decrease in the tank pressure decays at later time points as the pressure gradient driving flow out of the tank becomes smaller as time progresses.

## Tank Blowdown Example

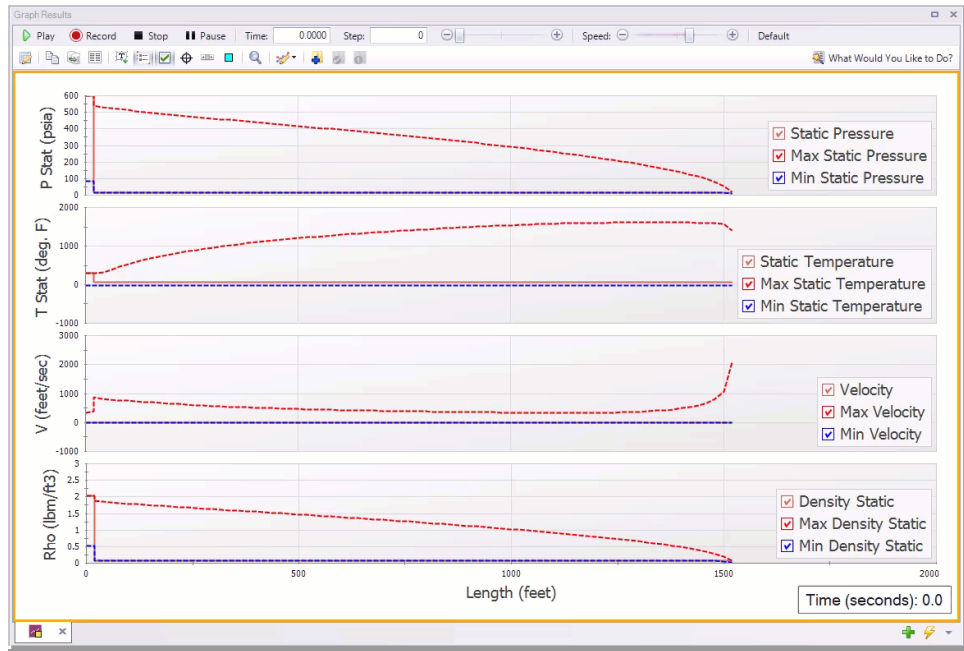
Lastly, we will animate the profile of the Blowdown Tank system to show the wave caused by opening the valve. In the Quick Access Panel, open the Profile tab. Under Plot Single Path, select Pipe 1 and Pipe 2. After ensuring your length is feet, select the option to Animate Using Output and ensure the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

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



**Figure 34: Profile Graph Parameters window**

## Tank Blowdown Example



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

Take note of the pressure wave that is created at the valve just after animation is started. This wave travels to the Assigned Pressure junction before reflecting back towards the valve. The amount of time it takes this wave to travel to the reflection point and back is called the communication time. It should be noted when creating your transient that any event that occurs over a shorter length of time than the communication time of the system is considered to happen instantaneously.

### C. View the Visual Report

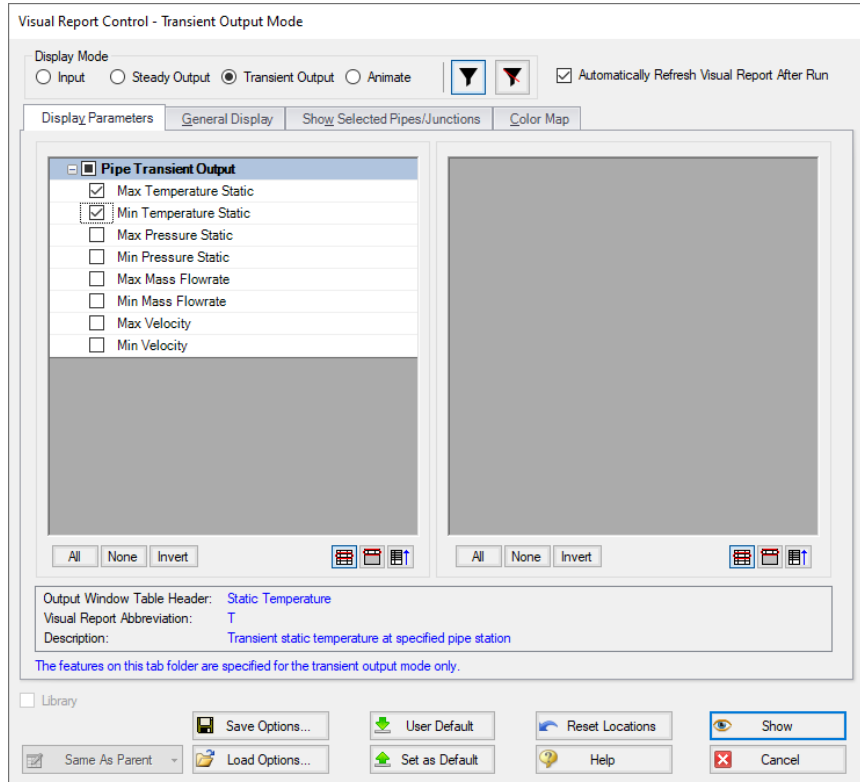
The Visual Report allows you to show text output alongside the model schematic. This is useful to quickly show pertinent information in relation to location in the model. The Visual Report can also animate the transient pipe results in a color animation overlaid on the model.

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

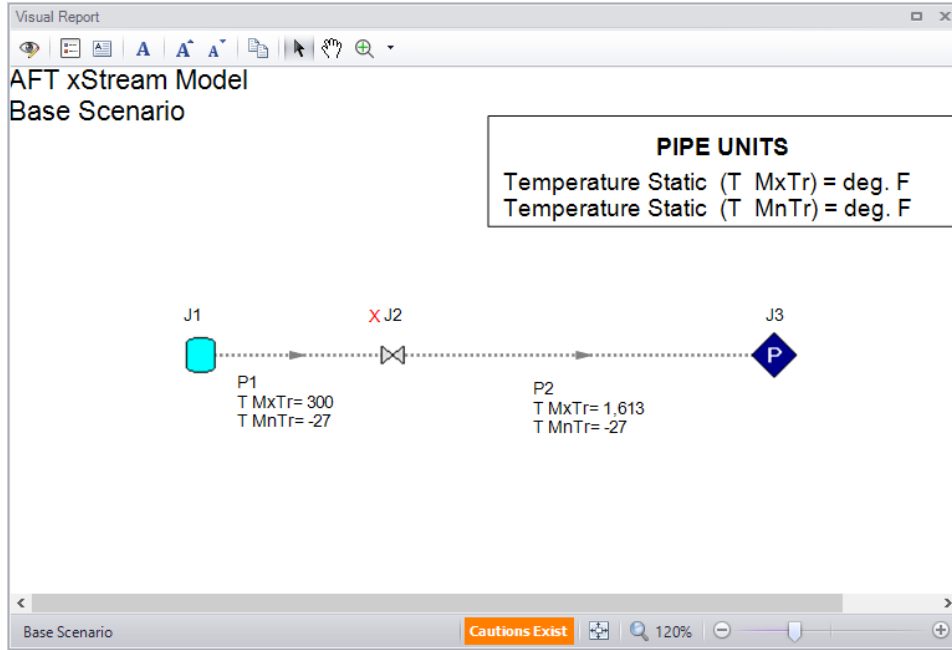
➤ **Select "Max Temperature Static" and "Min Temperature Static"** in the Pipe Transient Output area of the Visual Report Window as shown in Figure 36. Click the Show button. This will generate the Visual Report window graphic seen in Figure 37.

## Tank Blowdown Example

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



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



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

## D. Examine the Effect of Heat Transfer

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

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

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

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

After updating each of the pipes open the Analysis Setup window and go to the Pipe Sectioning and Output panel. Change the Estimated Maximum Pipe Fluid Temperature During Transient to 350 deg. F and click OK to accept the changes.

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

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

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

## Tank Blowdown Example

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

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

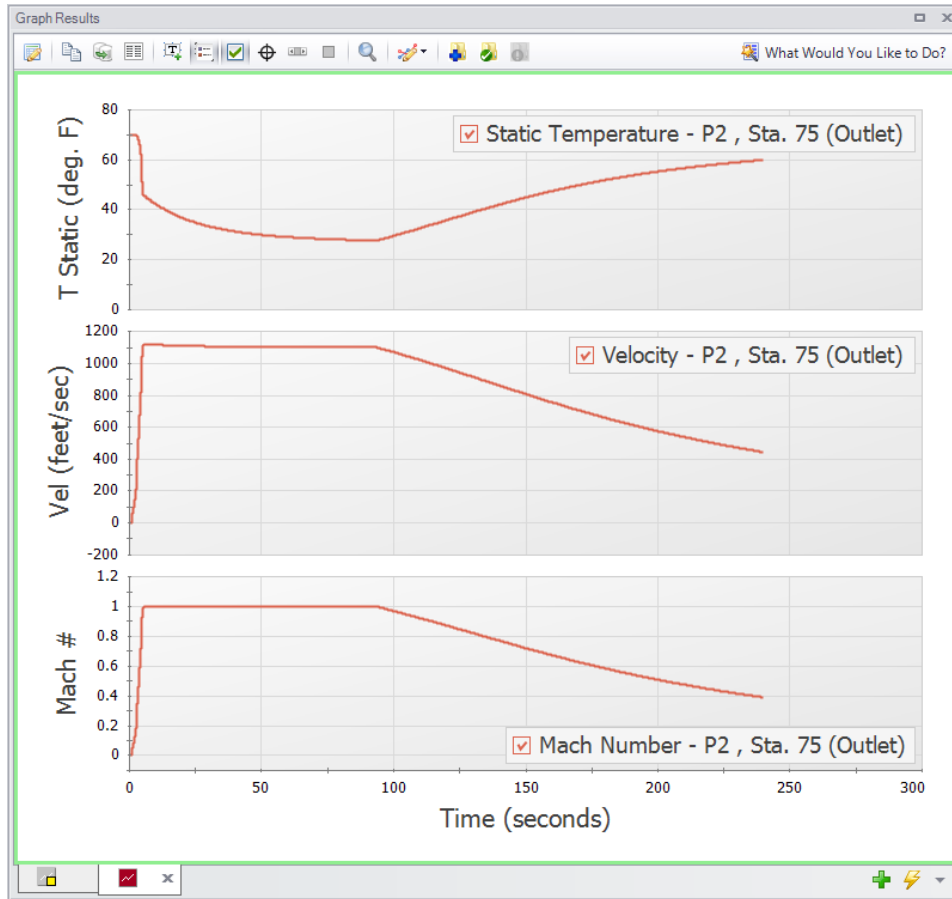


Figure 39: Temperature, velocity, and Mach number for the discharge of Pipe P2 with convective heat transfer

### E. Examine the Effect of Sectioning

In AFT xStream, sectioning plays an important role in model accuracy. Due to how the MOC operates, accuracy increases with the number of sections you have in the model. To illustrate this, the model was re-run with a Minimum Number of Sections Per Pipe of 6. Comparing Figure 33 to Figure 42, values such as the tank temperature and pressure for Tank J1 remain similar with the increase in the number of sections. However, values such as the maximum temperature reached in Pipe 2 (Figure 25 and Figure 40), changed substantially. This value increased from 1613 deg. F to 2097 deg. F. This change in the maximum temperature indicates that the difference in sectioning has significant implications if your interest is

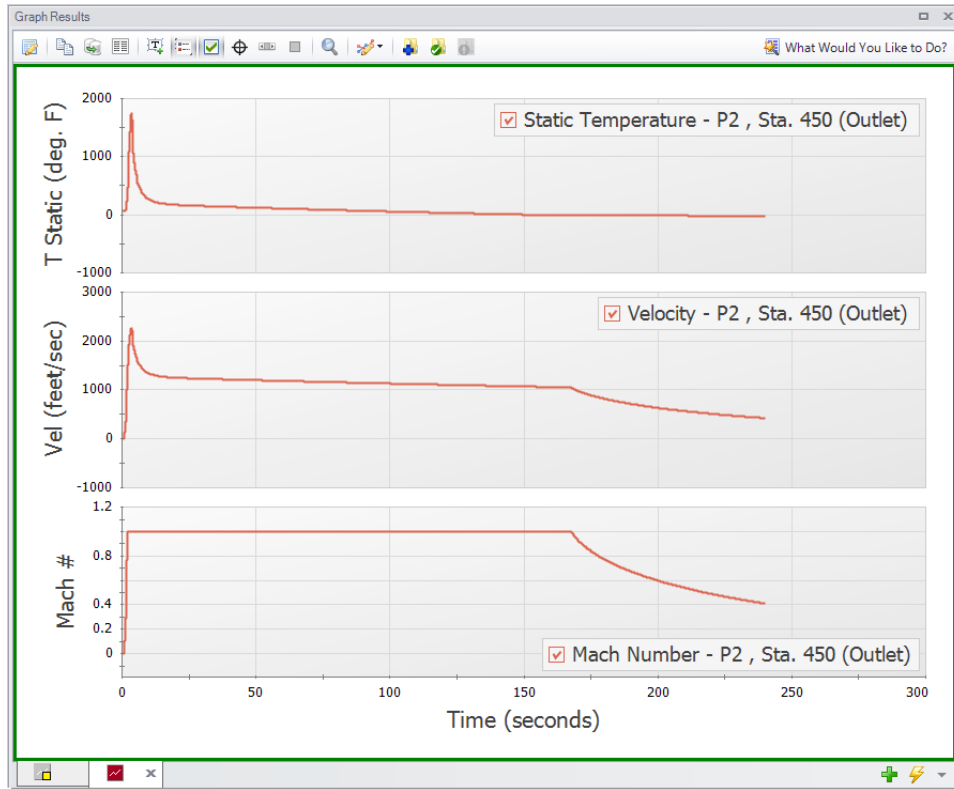
## Tank Blowdown Example

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

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

Pipe	Max. Static Temperature (deg. F)	Max. Static Temperature Time (seconds)	Max. Static Temperature Station	Min. Static Temperature (deg. F)	Min. Static Temperature Time (seconds)	Min. Static Temperature Station	Max. Static Pressure (psia)	Max. Static Pressure Time (seconds)	Max. Static Pressure Station
1	300.0	5.163E-03	0	-23.69	240.0	0	600.0	0.000	0
2	2,098.2	3.049	365	-33.33	240.0	450	540.4	2.625	0

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



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



# Tank Blowdown Example

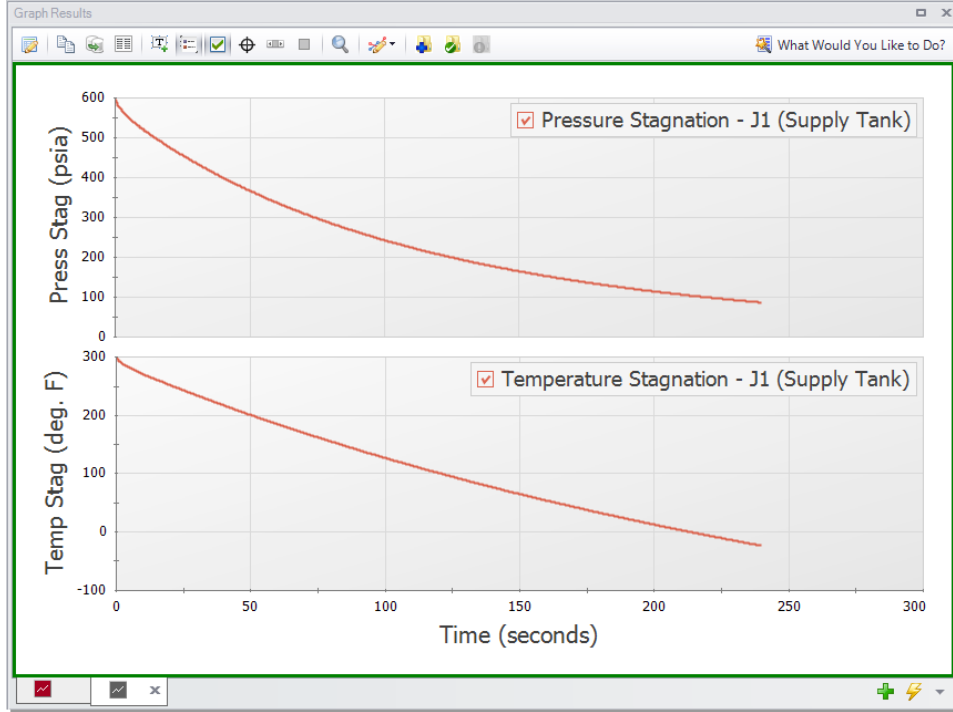


Figure 42: Temperature and pressure stagnation for Tank J1 with 6 sections minimum per pipe

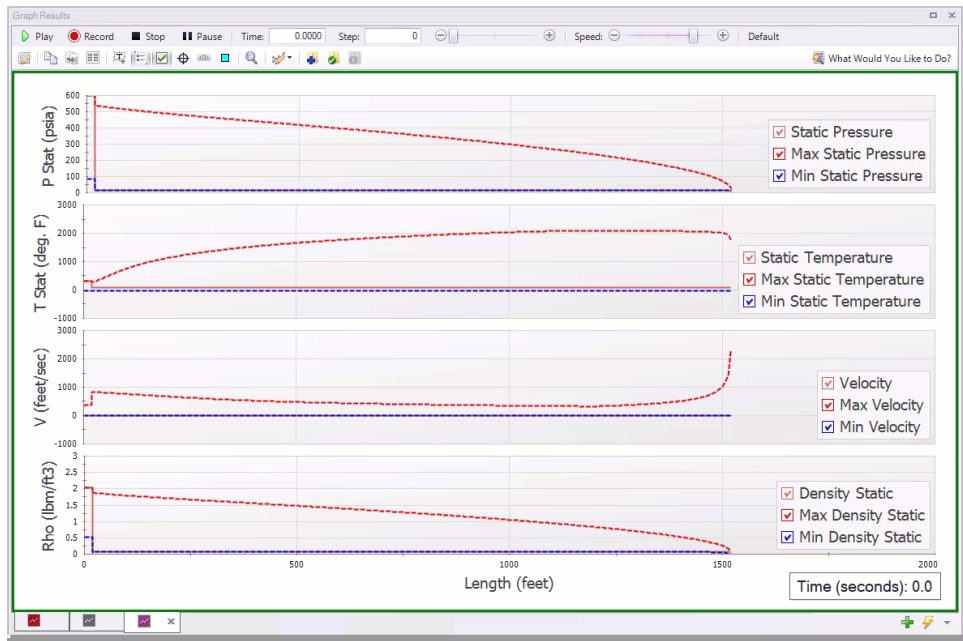


Figure 43: Profile Graph for Tank Blowdown system with 6 Sections Minimum Per Pipe where the blue line represents the minimum value, the bright red line represents the maximum value and the dark red line represents the value at the current time step

## **Conclusion**

---

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

# Gas Turbine Fuel System Example

## Summary

---

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

## Topics covered

This example will cover the following topics:

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

## Required knowledge

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

## Model file

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

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

## Step 1. Start AFT xStream

---

Open AFT xStream. Click "Start Building Model".

## Step 2. Build the model

### A. Place the pipes and junctions

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

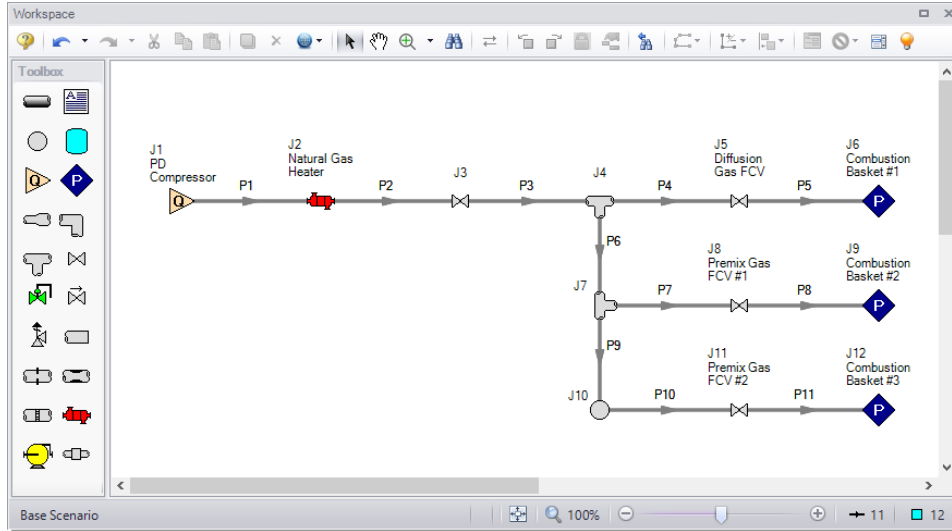


Figure 1: Layout of gas turbine fuel system

### B. Enter the pipe and junction data

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

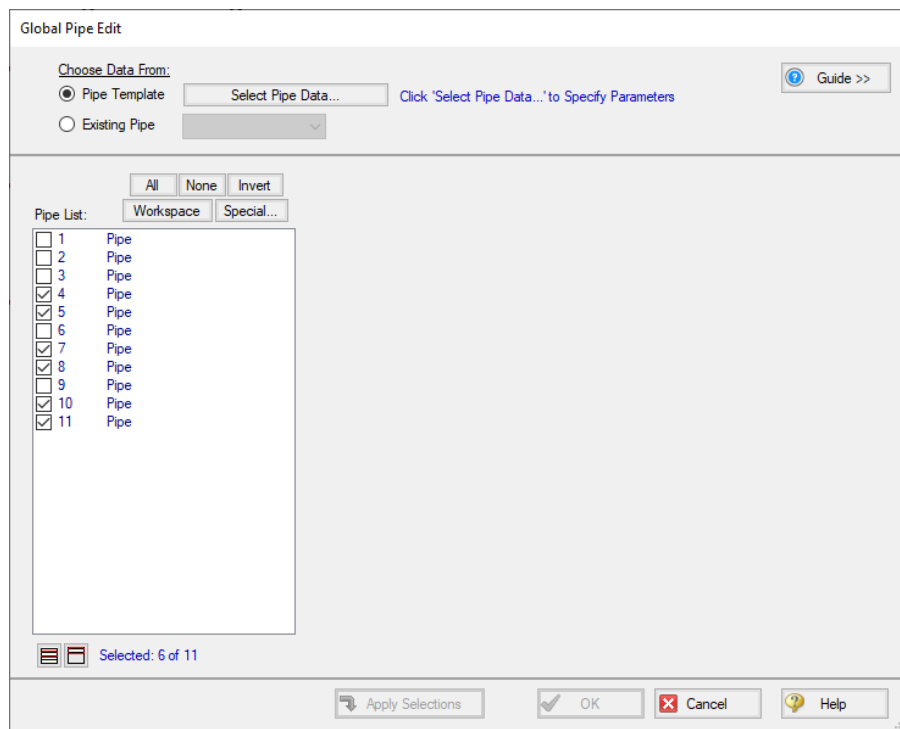
All Pipes are Stainless Steel - ANSI, Schedule 10S, and have standard roughness. The rest of the pipe information is as follows:

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

## Gas Turbine Fuel System Example

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

Because several of the pipes are the same, you can use the Global Pipe Editing tool to speed up the process of entering the data. The Global Pipe Editing tool allows you to apply changes to multiple pipes at the same time. Select Global Pipe Edit either from the Edit Menu, or by selecting the "Global Edit" drop-down on the toolbar and selecting the "Global Pipe Edit" option. This will open the Global Pipe Edit window. Select pipes P4, P5, P7, P8, P10, and P11 from the list of pipes as shown in Figure 2.

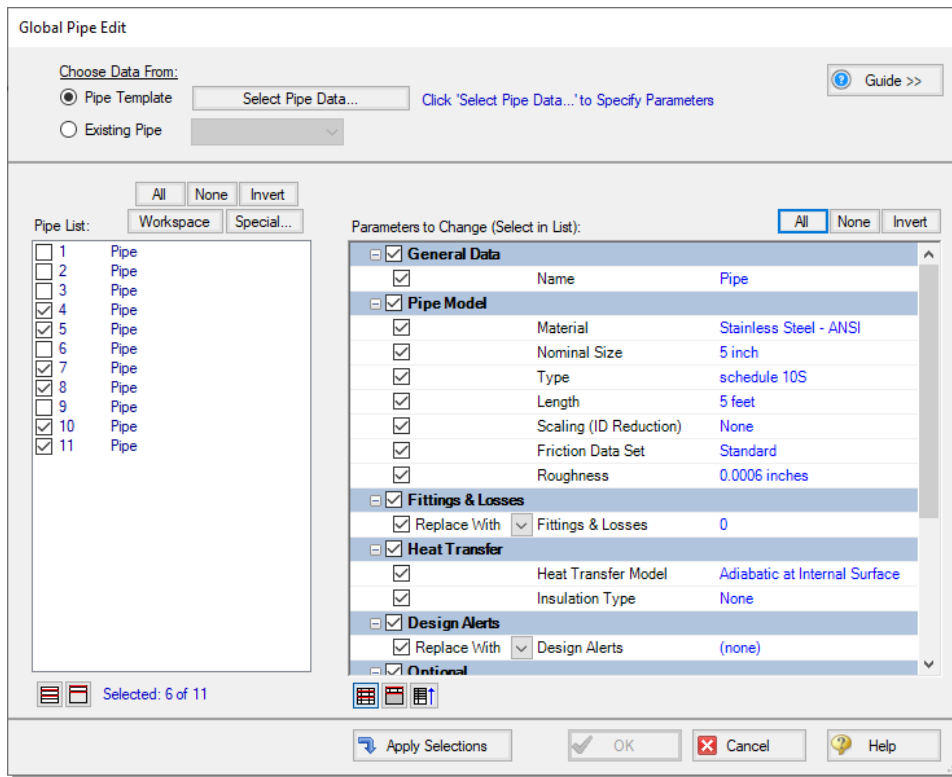


**Figure 2: Global Pipe Edit window**

After selecting the pipes, click the "Select Pipe Data" button. This will open a template Pipe Properties window for entering the data to be applied to all of the selected pipes. Fill out the data for the selected pipes. When you have entered all of the data for the pipes, close the Pipe Properties window by selecting OK. The Global Pipe Edit window will now display a list of all of the parameters that may be applied to the selected pipes. The parameters are categorized as they are displayed on the tabs in the Pipe Properties window. For this example, you want all of the parameters for all of the selected pipes to be updated to the

## Gas Turbine Fuel System Example

specified values. This can be accomplished by selecting the "All" button above the list of parameters. The Global Pipe Edit window should now appear as shown in Figure 3.



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

Apply the changes to the selected pipes by clicking the "Apply Selections" button. AFT xStream will notify you that the changes have been completed. Select OK. At this point, all of the changes have been applied but not saved. If you wish to cancel all of the changes you just implemented, you may do so by clicking Cancel on the Global Pipe Edit window. Save the applied changes by clicking OK.

---

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

---

1. Assigned Flow J1
  - a. Name = PD Compressor
  - b. Elevation = 0 feet
  - c. Mass Flow Rate = 90,000 lbm/hr

---

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

---

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

## Gas Turbine Fuel System Example

---

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

---

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

---


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

<b>Time (seconds)</b>	<b><math>C_v</math></b>	<b><math>X_t</math></b>
0	195	0.7
0.01	0	0.7
2	0	0.7

- 7. Assigned Pressures J6, J9, J12
  - a. J6 Name = Combustion Basket #1
  - b. J9 Name = Combustion Basket #2
  - c. J12 Name = Combustion Basket #3
  - d. Elevation = 0 feet
  - e. Stagnation Pressure = 15.5 atm
  - f. Temperature = 140 deg F.

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

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

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

## Step 3. Specify Analysis Setup

### A. Specify the Fluid panel

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

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

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

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

**Note:** If following along in the completed model file, the Estimated Maximum Pipe Temperature During Transient will have a different value. This will be discussed in Steps 8 and 9.

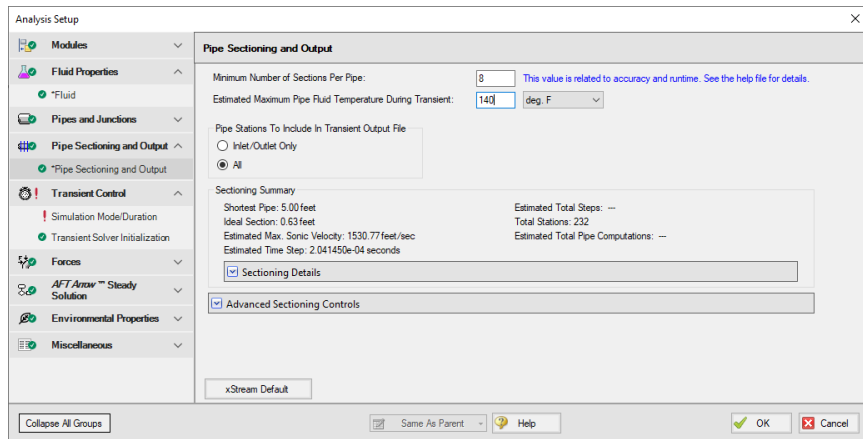


Figure 4: Sectioning panel for Turbine Fuel System



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

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

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

---


### A. Create scenarios

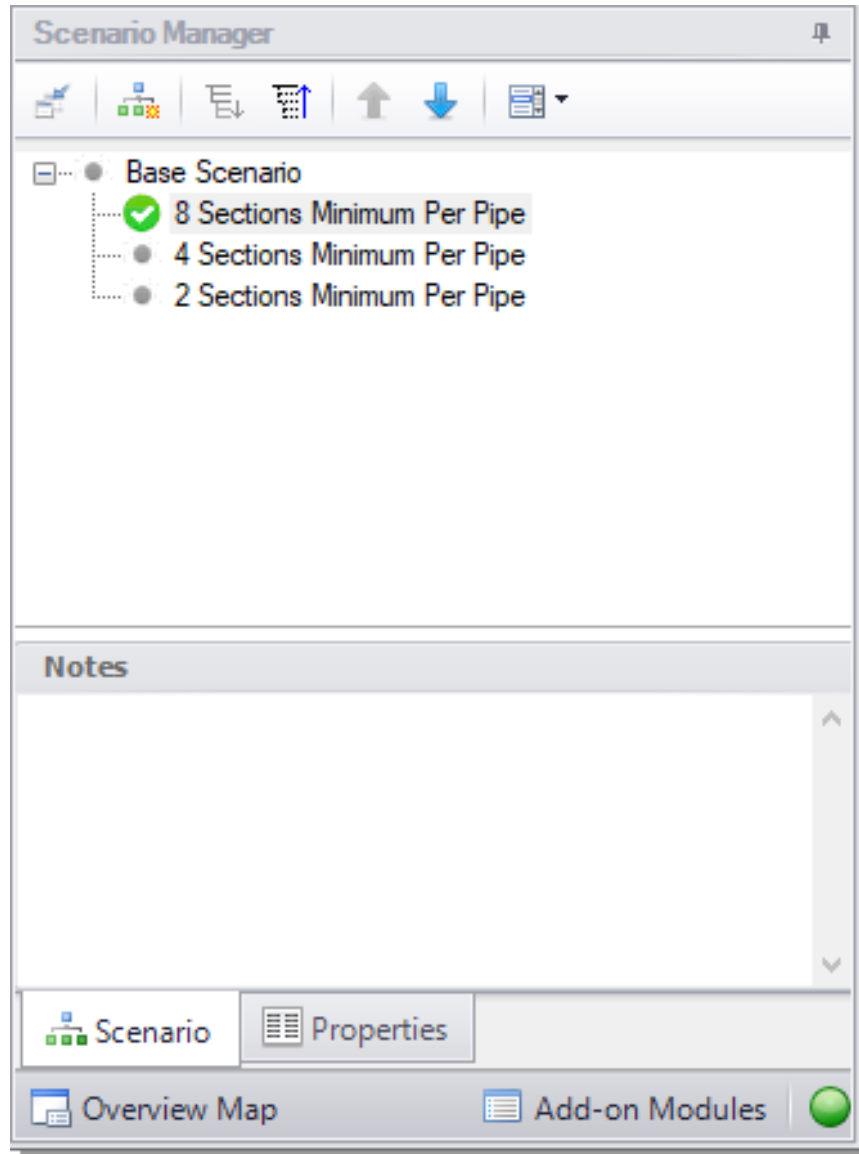
In this model, the following three types of Sectioning cases for the system are to be evaluated:

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

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

Open the Scenario Manager on the Quick Access Panel. Create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on the

Scenario Manager on the Quick Access Panel and then selecting the Create Child icon . Enter the name "8 Sections Minimum Per Pipe" in the Create Child Scenario window, and select the OK button. The new "8 Sections Minimum Per Pipe" scenario should now appear in the Scenario Manager on the Quick Access Panel below the Base Scenario. Select the Base Scenario. Create another child and call it "4 Sections Minimum Per Pipe". Finally, create one more child and call it "2 Sections Minimum Per Pipe". The Child Scenarios should now be displayed as shown in Figure 5.



*Figure 5: The Scenario Manager on the Quick Access Panel displays the scenario tree which allows you to create scenarios and keep them organized within the same model file*

## B. Set up scenarios

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

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

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

### Step 5. Run the first scenario

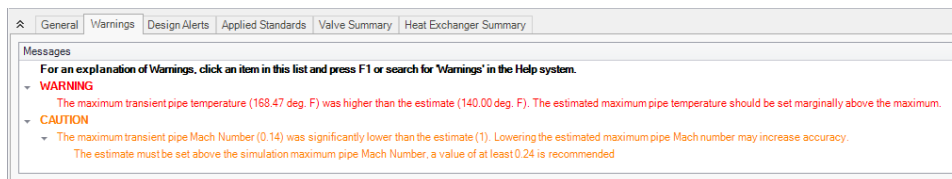
➤ **Double-click the "2 Sections Minimum Per Pipe" scenario** in the Scenario Manager. Select "Run Model" on the toolbar. AFT xStream will prompt you to save, then will open the Solution Progress window. This model is estimated to run in approximately one minute, but the run time is dependent on the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

### Step 6. Review the results

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

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

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



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

## Step 7. Adjust the Pipe Sectioning panel

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

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

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

### A. Review Output window

➤ **Double-click the "8 Sections Minimum Per Pipe" scenario.** Select "Run Model" from the Analysis menu, then proceed to the Output window. There should be a caution indicating that the maximum transient pipe mach number was significantly lower than the estimate. Disregard this error as discussed previously. Select the Transient Max/Min tab (Figure 7). Here we can see that the maximum temperature in the inlet lines (P5, P8, & P11) to the combustion baskets is 163.6 deg. F and occurs in the inlet line to combustion basket 1.

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

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

### B. Graph the results

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


Create a graph that shows the temperature over time at the entrance to Combustion Basket #1 using the following steps:

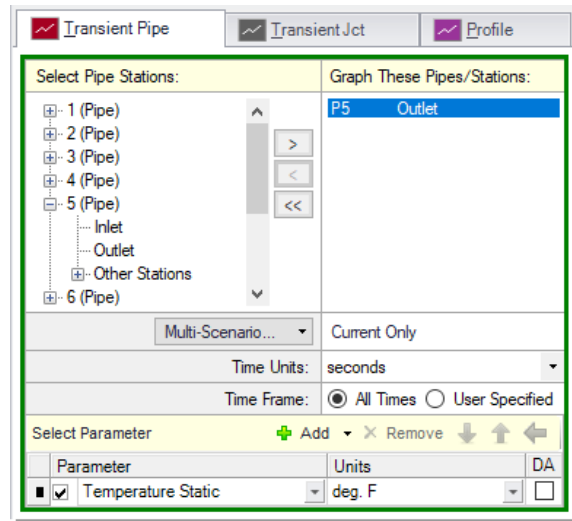
1. From the Graph Control tab on the Quick Access Panel, select the Transient Pipe tab as shown in Figure 8

2. Select Pipe P5 and add the outlet station
3. Verify that the time units are set to seconds
4. In the Parameters definition area, select "Temperature Static" and specify "deg. F" in the drop-down box under "Units"
5. Click the Generate button

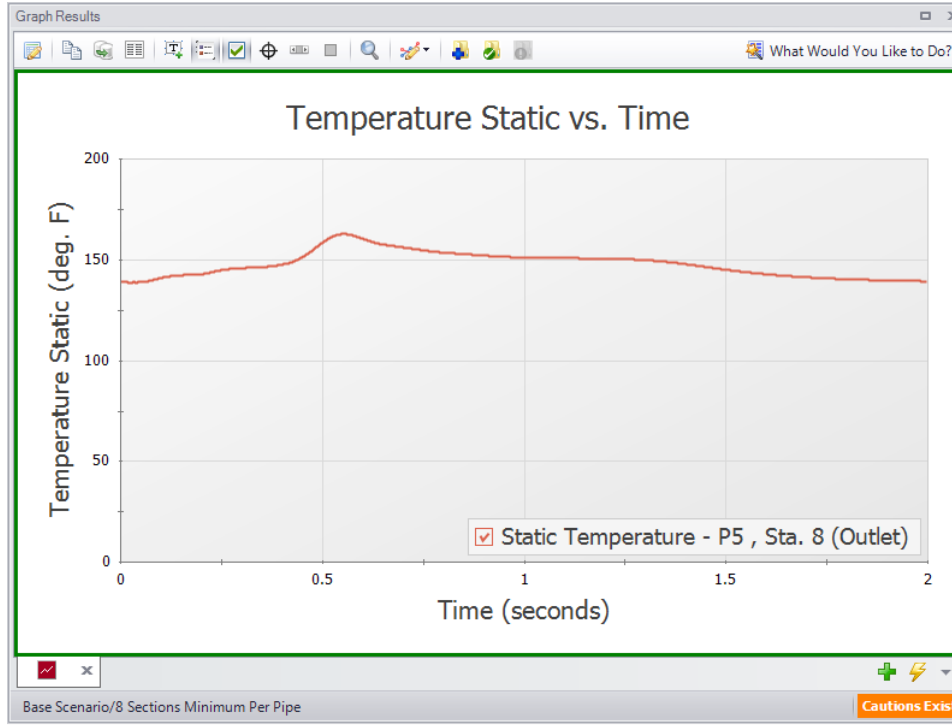
The results (see Figure 9) show a maximum temperature of 162.9 deg. F at the entrance to combustion basket #1. Note this temperature is lower than the temperature shown on the Transient Max/min tab because this graph is showing the outlet of pipe P5, while the maximum temperature in pipe P5 occurs at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

To easily recreate this graph in the other scenarios or in the future, it can be added into the Graph List

Manager on the Quick Access Panel. After generating the graph, click the "Add Graph to List" icon  on the Graph Results Toolbar (or simply right-click the graph itself and choose "Add Graph to List"). Give the graph a name, such as "Combustion Basket #1 Temperature vs Time", then click OK. The graph will be added to the currently selected folder in the Graph List Manager. Note that graphs added to the Graph List Manager are not permanently saved until the model is saved again. Also be aware that only the graph templates are saved as opposed to the graph data.



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



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

### C. Animate the results

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

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

*To view the animated profile*

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

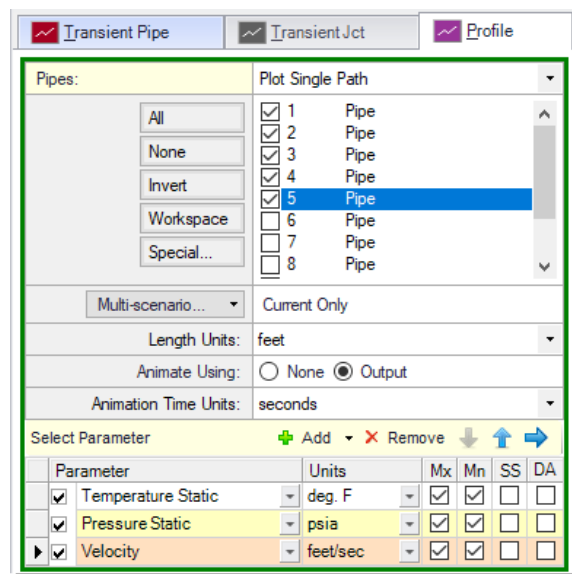
You should generate a graph that looks like the one shown in Figure 10. Playing the animation reveals that the highest temperature in the system is observed at the J2 heat exchanger. It should also be noted

## Gas Turbine Fuel System Example

that the rise to this maximum temperature occurs when the pressure wave from the valve closing passes through this junction. This heightened temperature is not immediately seen by the combustion baskets downstream, however, as the gas velocity throughout the system drops immediately following the valve closure. Instead, the heightened temperature moves slowly downstream until the gas wave is reflected at the positive displacement compressor and passes through the heat exchanger going the opposite direction. This causes the local gas velocity to rise and causes the temperature wave to move downstream towards the combustion baskets more quickly. This wave eventually reaches combustion basket #1 at  $T = 0.5$  seconds, at which point the inlet to the combustion basket reaches its maximum temperature.

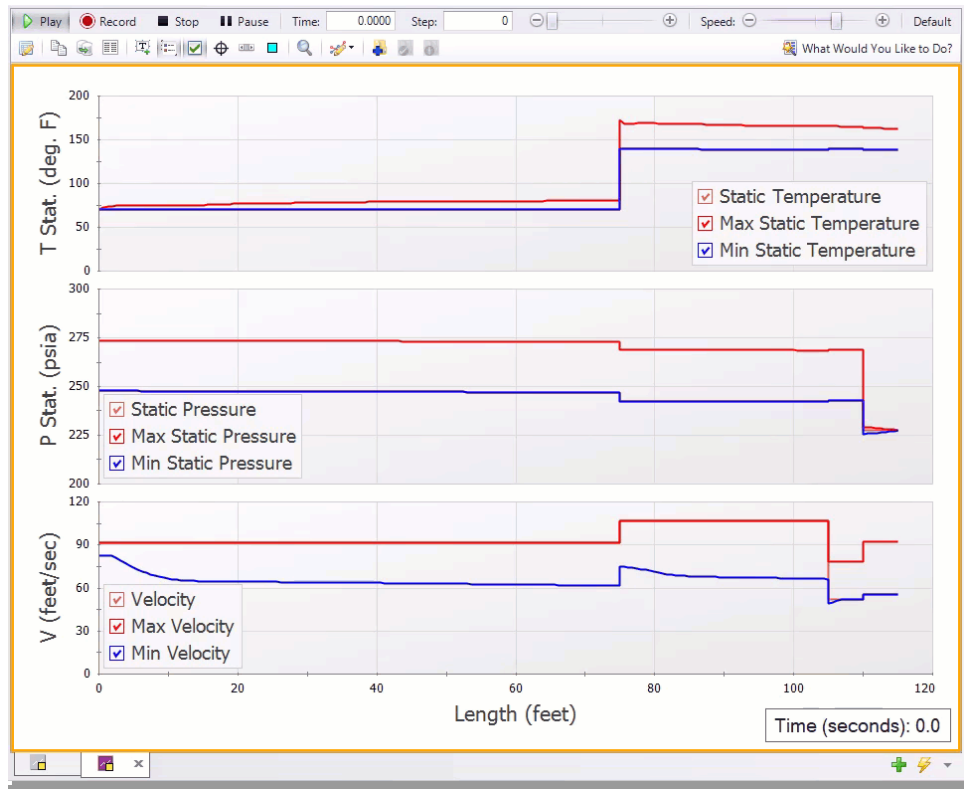
Click the "Add Graph to List" icon and give it a name such as "Combustion Basket #1 Profile". Click OK to add the Graph to the Graph List.

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



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

## Gas Turbine Fuel System Example



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

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

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

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

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

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



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

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

## **Conclusion**

---

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

Additionally, this example identifies the Estimated Maximum Temperature During Transient as a critical variable in the tabulation of the transient results. This value sets the boundary for the characteristic grid the Method of Characteristics uses. If sonic choking occurs at a temperature higher than the estimated maximum, the model may not converge. As such, this value should be conservatively high on early runs of a model and then refined once the temperature that the transient approaches is known.

# High Pressure Steam - Fluid Property Models Example

## Summary

---

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

## Topics covered

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

## Required knowledge

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

## Model file

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

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

## Step 1. Start AFT xStream

---

Open AFT xStream. Click "Start Building Model".

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

## Step 2. Build the model

---

### A. Place the pipes and junctions

The next step is to define the pipes and junctions.

#### *Isometric Drawing Mode*

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

1. From the Arrange menu, select the option to Show Grid
2. Open the Arrange menu again, go to Pipe Drawing Mode, and select Isometric
3. Place the junctions as shown in Figure 1
4. Click the Pipe Drawing Tool and draw the pipes in the model as shown. When drawing segmented pipes such as P5, a red dashed preview line will show how the pipe will be drawn on the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse.
5. Use the Rotate Icons buttons on the Workspace Toolbar, or right-click on the junction and select the Auto-rotate option to automatically align the junction with the connected pipes. Alternatively, Customize Icon can be selected from the right-click menu to manually choose an icon/orientation.
6. The grid can be shown or turned off in the Arrange menu, as shown in Figure 2

---

**Note:** You can hold the "Alt" key while adjusting a pipe by the endpoint to add an additional segment. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

---

## High Pressure Steam - Fluid Property Models Example

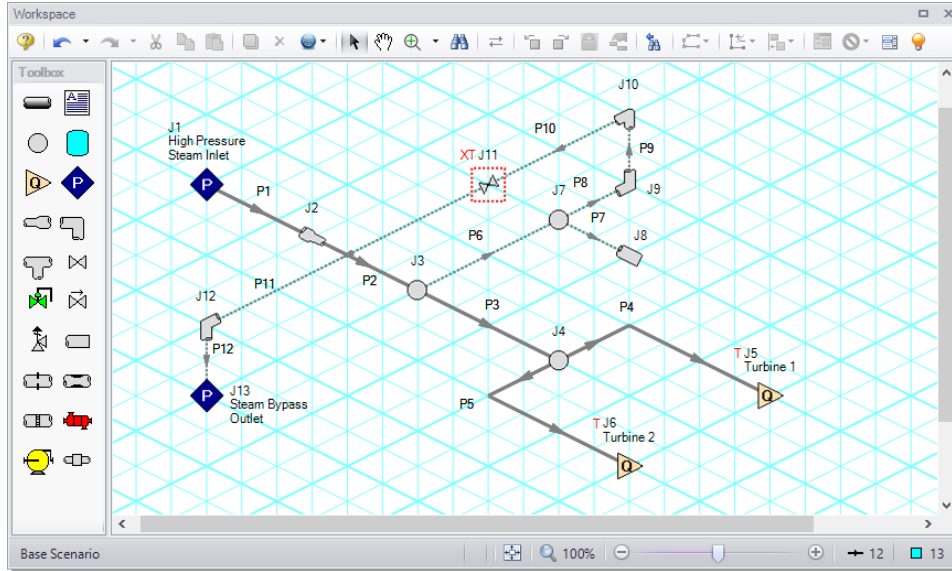


Figure 1: The isometric layout with the grid shown

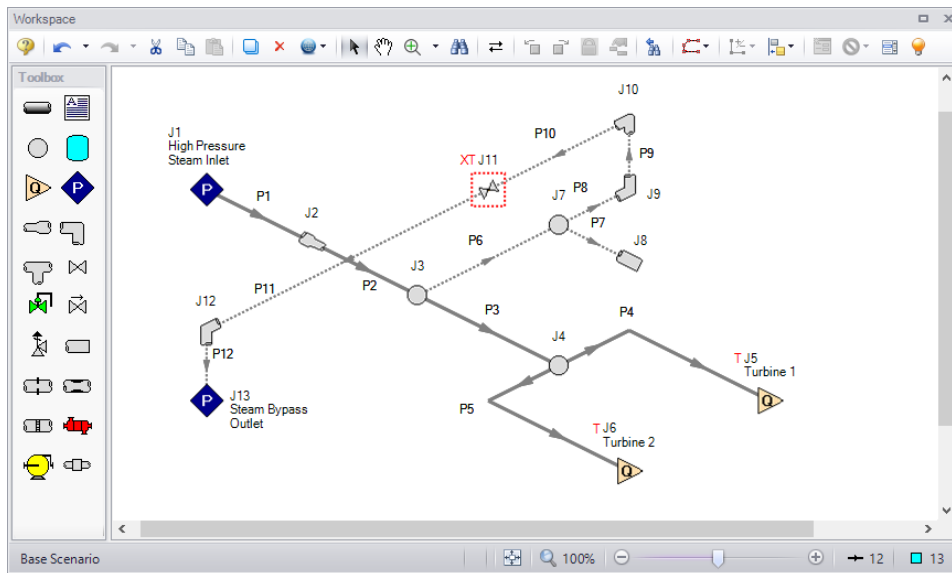


Figure 2: The isometric layout with the grid turned off

### B. Enter the pipe and junction data

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

All pipe materials are User Specified in this model (see Figure 3). Use the Absolute Roughness of Steel (0.0018 in) as the Friction Model, along with the following data:

## High Pressure Steam - Fluid Property Models Example

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

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

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

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

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

## High Pressure Steam - Fluid Property Models Example

---

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

---

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

---

- e. Temperature = 1060 deg. F
5. Dead End J8
  - a. Elevation = 0 feet
6. Bend J9
  - a. Elevation = 0 feet
  - b. Type = Standard Elbow (knee, threaded)
7. Bend J10, J12
  - a. Elevation = 5 feet
  - b. Type = Standard Elbow (knee, threaded)
8. Valve J11
  - a.  $C_v = 3000$
  - b.  $X_t = 0.7$
  - c. Elevation = 5 feet
  - d. In the optional tab, set the special condition to closed
  - e. Click on the Transient Tab and enter the following data

<b>Time (seconds)</b>	<b><math>C_v</math></b>	<b><math>X_t</math></b>
0	0	0.7
2	3000	0.7
5	3000	0.7

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

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

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

## Step 3. Specify Analysis Setup

### A. Specify Fluid panel

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

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

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

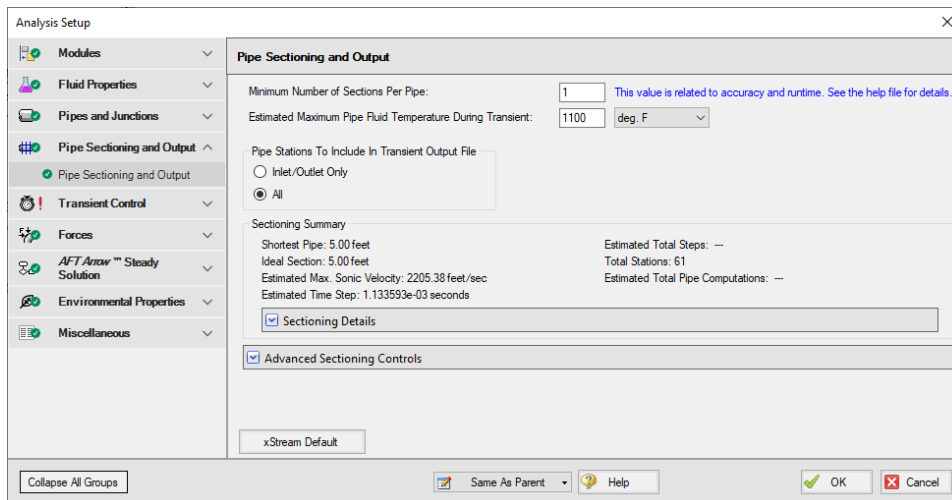


Figure 4: Sectioning panel for High Pressure Steam System

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

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

### Step 4. Run the model

---

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

### Step 5. Graph the results

---

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

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

1. From the Graph Control tab on the Quick Access Panel, select the Transient Pipe tab
2. Select the pipe P5 outlet station
3. Ensure that time units are set to seconds
4. In the Parameters definition area, select "Pressure Static" and specify "psia" in the dropdown box under Units
5. Click Generate

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

1. Right-click the y-axis
2. Uncheck the Auto Scale box
3. Set the Minimum to 2200 psia, the Maximum to 2600 psia, and Major Val to 100 (see Figure 7)

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



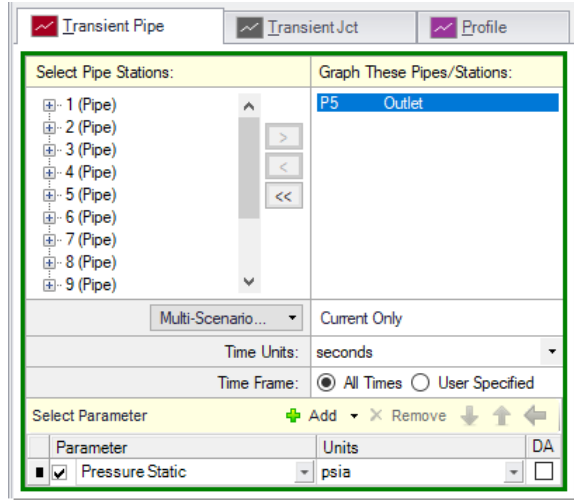


Figure 5: Graph Parameters for Inlet Pressure to Turbines

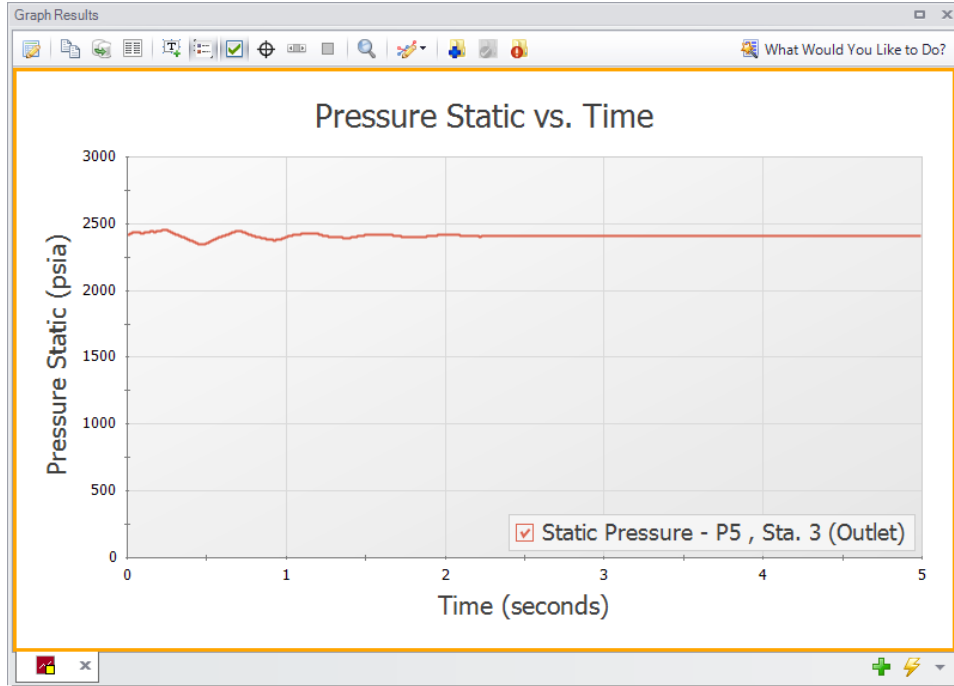


Figure 6: Static pressure at steam turbine 2 entrance

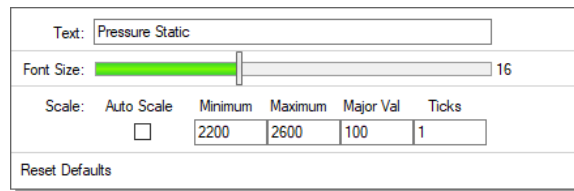
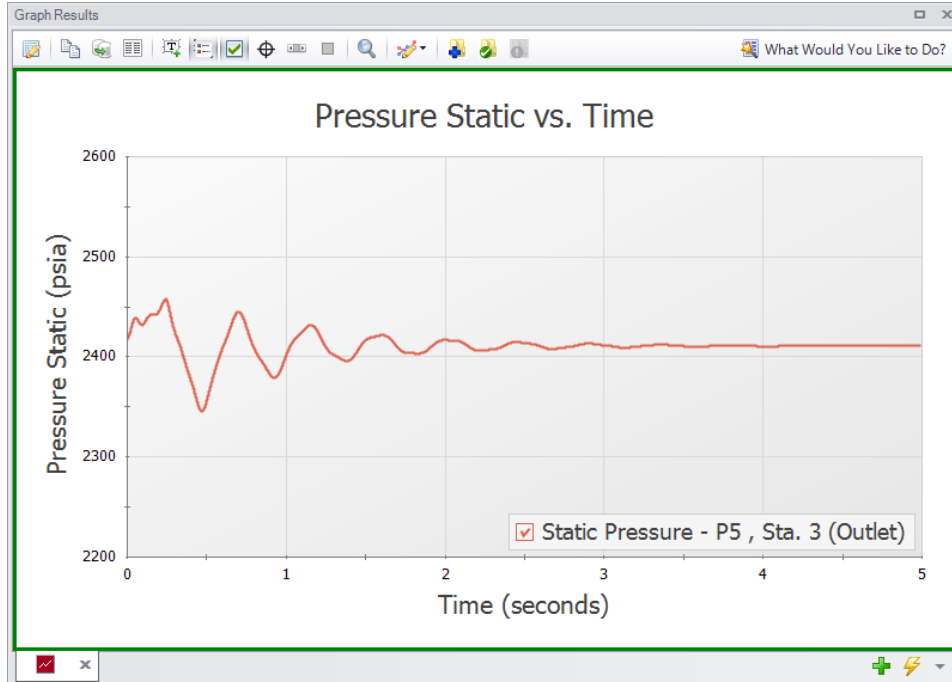


Figure 7: Axis Modification Window opened by right-clicking the y-axis



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

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

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

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

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

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

## High Pressure Steam - Fluid Property Models Example

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

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

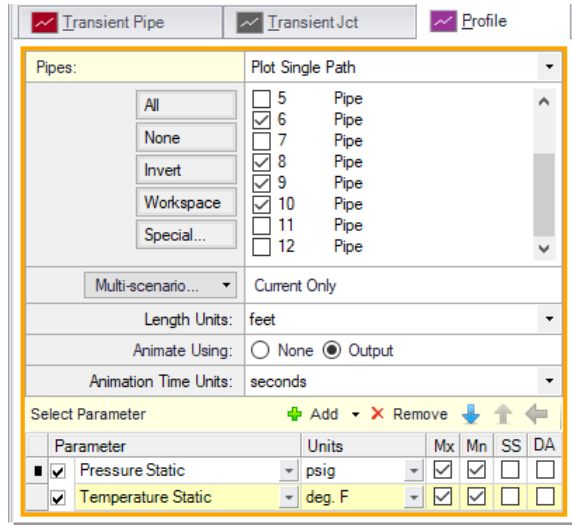


Figure 9: Graph Parameters for bypass line profile

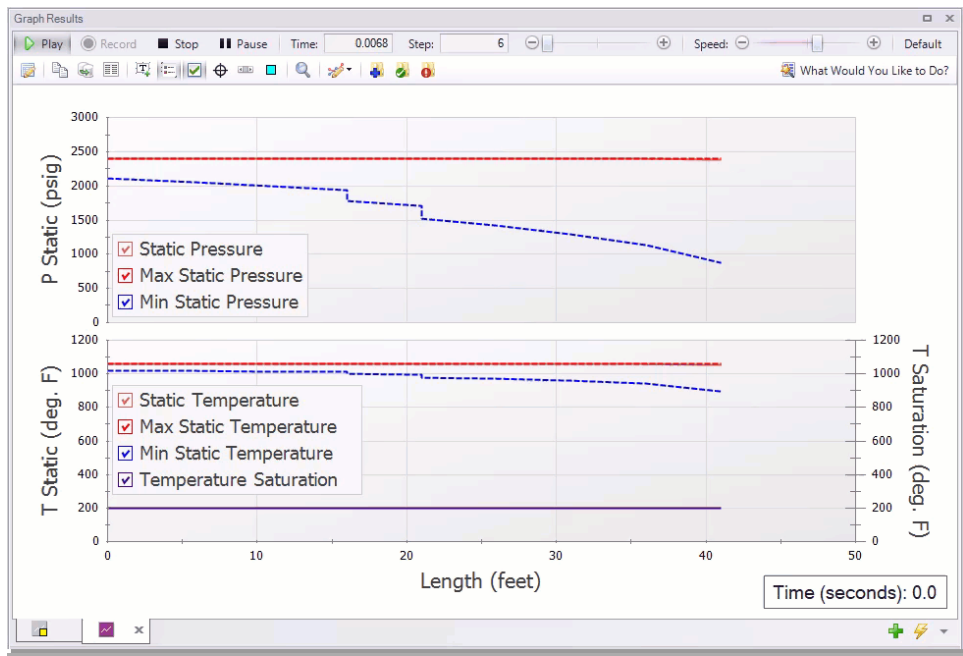


Figure 10: Bypass line animation profile

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

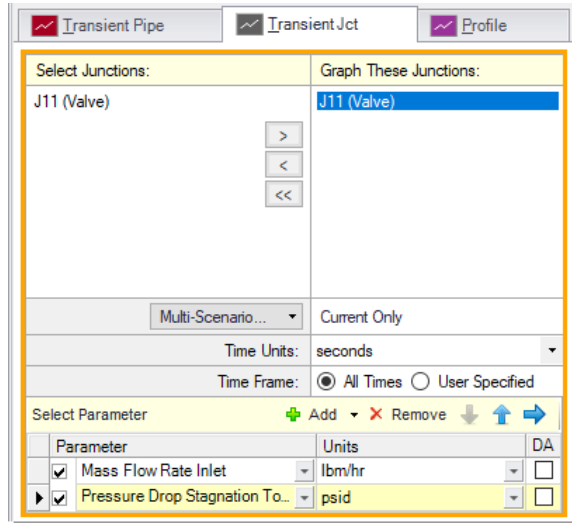
### C. Graph the mass flow rate and pressure drop through the bypass valve

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

1. Click the "New Tab" button to add a new graph tab
2. From the Graph Control tab on the Quick Access Panel, select the Transient Jct tab
3. In the Select Junctions section, select J11 and click the ">" arrow
4. Add a second parameter in the Select Parameters section. Specify the parameters as "Mass Flow Rate Inlet" and "Pressure Drop Stagnation Total".
5. Set the units for Mass Flow Rate Inlet to "lbm/hr" and Pressure Drop Stagnation Total to "psid"
6. Click Generate

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

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



**Figure 11: Bypass valve mass flow and pressure drop Parameters**

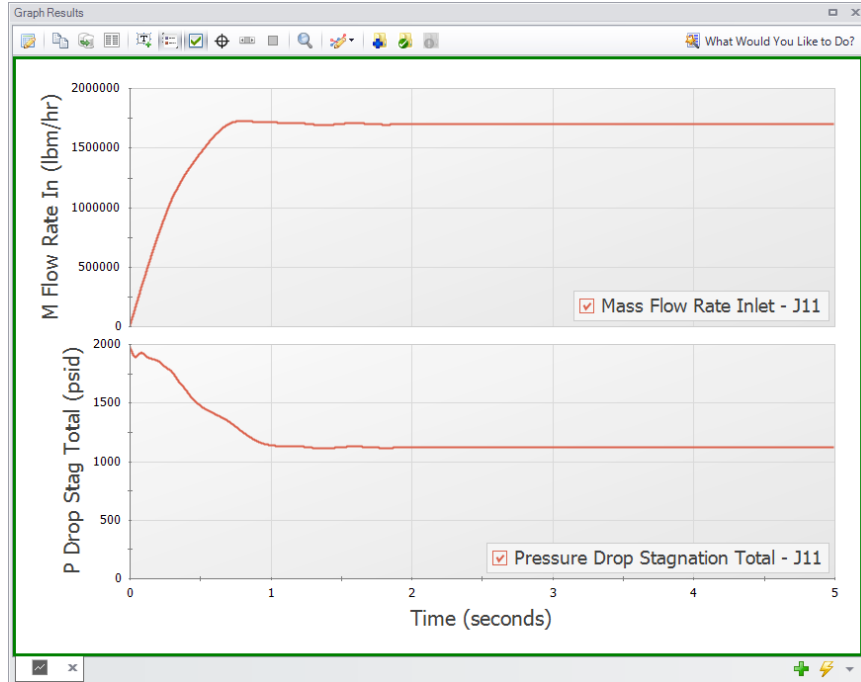


Figure 12: Bypass valve mass flow rate and pressure drop transient

## Step 6. Results of Other Libraries and Discussion

### A. Review the results of other libraries

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

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

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

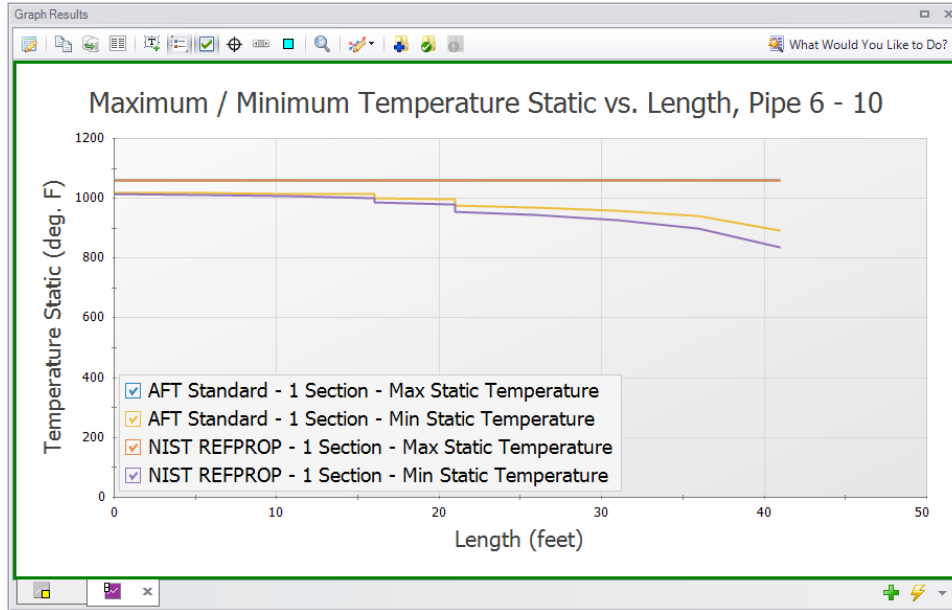


Figure 13: Bypass line profile obtained using NIST REFPROP

### B. Advice for preliminary runs

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

#### **Step 1: Run the steady-state model**

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

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

You should then run your simplified model in the Transient Solver using an AFT Standard fluid with minimal sectioning. This is not always possible, such as when mixtures are present, but this step should always be attempted if allowable. This will allow for larger issues to be caught before committing to extended run times. Only after you are satisfied with the model should you attempt increasing the number of sections.

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

A number of factors must be taken into consideration while selecting the fluid property model for an AFT xStream model. If there are no mixtures, and the compound is present in the AFT Standard library, it is

recommended that the AFT Standard library be the first library used. AFT Standard has the shortest run times of the four libraries and is an excellent choice for early runs focused on troubleshooting the model. However, if there is a need to model mixtures, and simplifying the system is not possible, initial runs should be conducted with NIST REFPROP or Chempak.

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

Lastly, ASME Steam provides a well-established, trustworthy source of steam data. ASME Steam has the longest run times of the four, and it is recommended that a high processing power computer paired with ample time be used to run your analysis if this library is selected. If either of those elements is unavailable, it is recommended that one of the three other libraries be used, with the added note that NIST REFPROP is the most similar in accuracy to ASME steam. Table 1 summarizes this example's run time with the four different fluid property models. The run times will be affected by the specifications of your computer as well as the memory available.

Table 1 shows data for runs conducted on the computers at AFT as of publication. Additionally, Table 2 provides a library compatibility chart to assist with fluid property model selection.

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

**Table 1: Comparison of High Pressure Steam System run times**

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



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

Fluid Library	Model Requirements		
	Mixtures	Complex Systems*	User Specified Fluids
AFT Standard	▪	✓	✓
NIST REFPROP	✓	✓	▪
Chempak	✓	✓	▪
ASME Steam	▪	▪	▪

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

## Conclusions

---

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

## High Pressure Steam - Forces Example

### Summary

---

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

### Topics covered

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

### Required knowledge

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

### Model file

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

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

### Step 1. Start AFT xStream

---

Open AFT xStream. Click the Start Building Model button.

### Step 2. Build the model

---

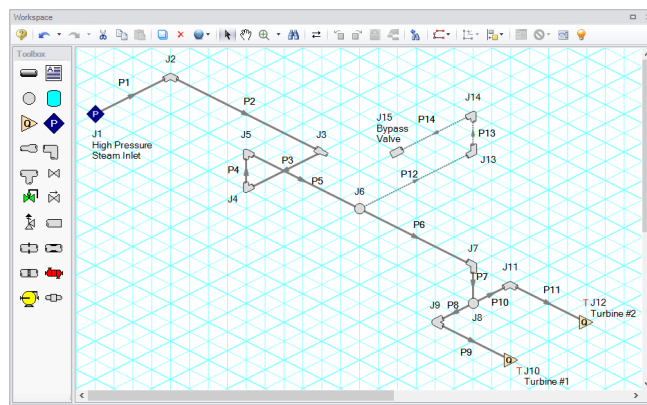
#### A. Place the pipes and junctions

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

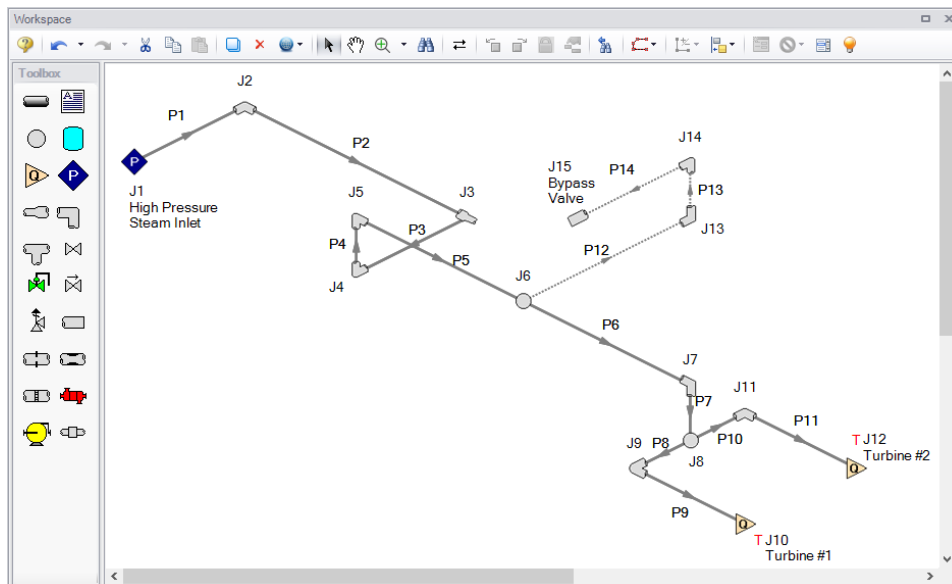
## High Pressure Steam - Forces Example

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

**Note:** You can hold the "Alt" key while adjusting a pipe by the endpoint to add an additional segment . This can be used with the arrow key or mouse scroll wheel to change between different preview line options.



**Figure 1: Model layout with grid**



**Figure 2: Model layout without Grid**

## B. Enter the pipe data

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

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

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

## C. Enter the junction data

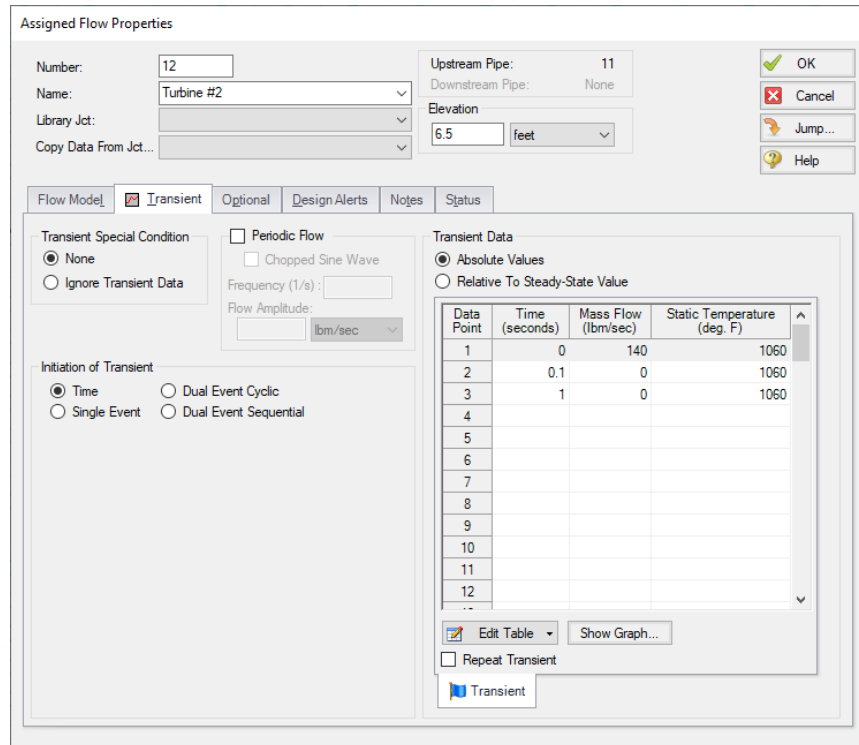
1. Assigned Pressure J1
  - a. Name = High Pressure Steam Inlet
  - b. Elevation = 0 feet
  - c. Static Pressure = 2420 psia
  - d. Static Temperature = 1060 deg.F
2. Bend J2, J4
  - a. Elevation = 0 feet
  - b. Type = Standard Elbow (knee, threaded)
3. Area Change J3

## High Pressure Steam - Forces Example

---

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

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



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

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

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

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

## Step 3. Specify Fluid

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

## Step 4. Specify the Pipe Sectioning and Output panel

➤ **Open the Pipe Sectioning and Output panel in the Analysis Setup window.** Define the Minimum Number of Sections Per Pipe as 1 section and the Estimated Maximum Pipe Temperature During Transient as 1100 degrees F.

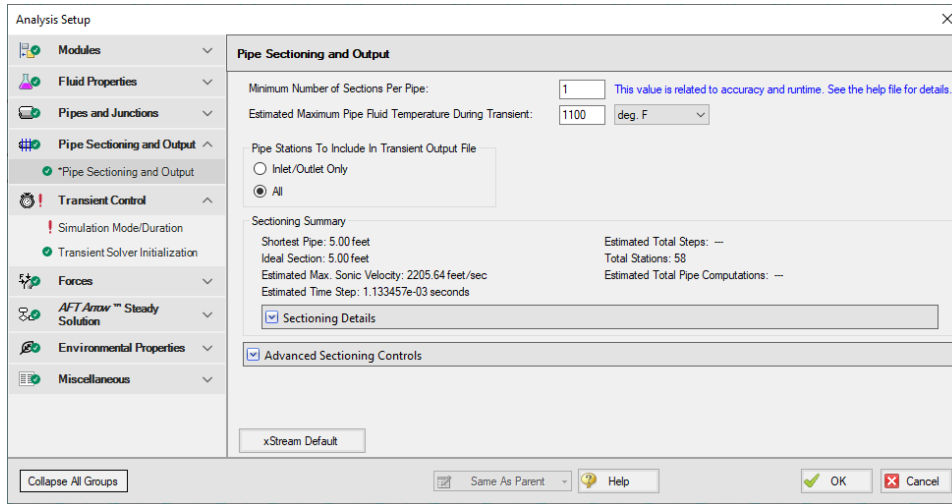


Figure 4: Sectioning panel for high pressure steam system in the Analysis Setup window

## Step 5. Specify the Simulation Mode/Duration panel

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

## Step 6. Specify Force Sets

We will first add one Force Set from the Workspace.

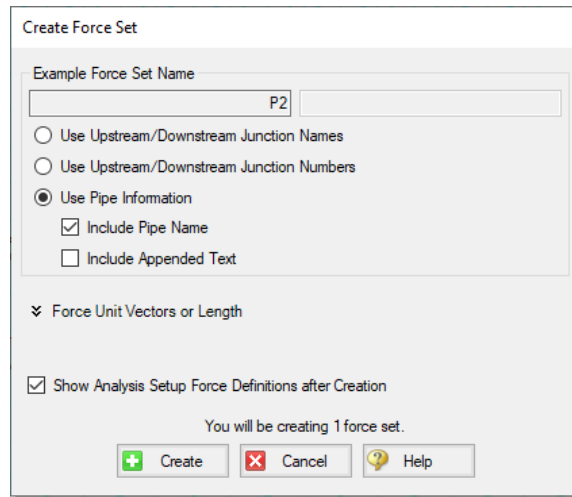
➤ **On the Workspace, right-click on pipe P2** and from the right-click menu select "Create a Force Set with Selected Pipe." Select the Use Pipe Information radio button and uncheck the "Include Appended Text" option (Figure 5). Click Create. This will open the Force Definitions panel of the Analysis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

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

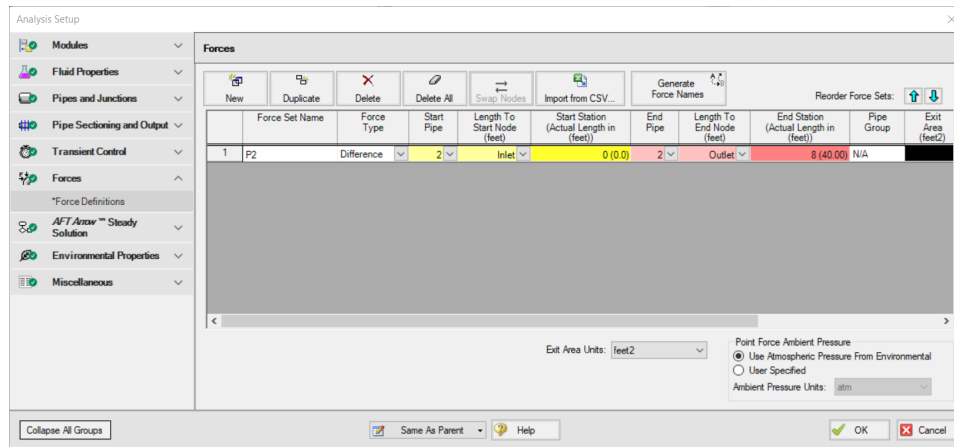
## High Pressure Steam - Forces Example

If a more robust view of the 3D orientation of the pipes is required, directional information can be entered using the optional Force Unit Vector columns. The Force Unit Vectors do not impact AFT xStream's calculations, but may be useful to export the output to piping stress analysis software as discussed in Step 12

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



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



**Figure 6: High Pressure Steam System Force input in the Analysis Setup window**

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

➤ **Define a force set by clicking "New"**, selecting the Force Type (in this example, we will be using "Difference"), entering a name, and defining the starting and ending locations of the force set. Set up the force sets as shown in Figure 7. Click OK to exit the Analysis Setup window.



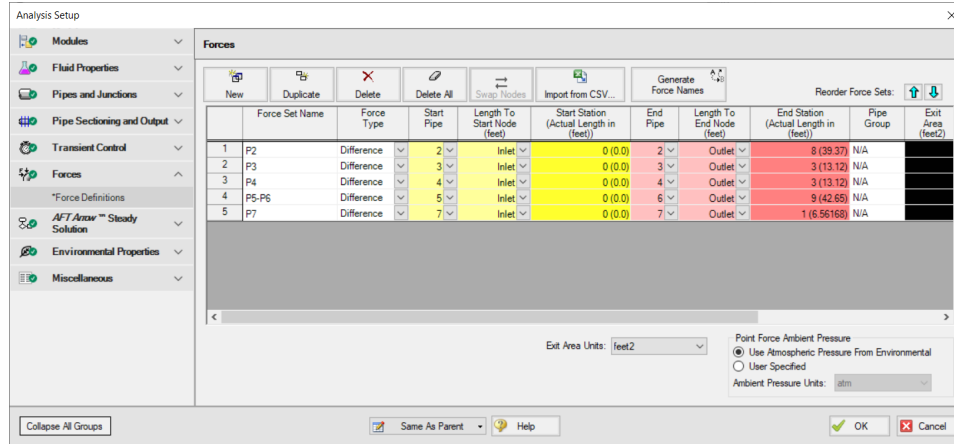


Figure 7: Completed Forces panel in the Analysis Setup window

## Step 7. Add child scenarios using Scenario Manager


### A. Create scenarios

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

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

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

➤ **Go to the Scenario Manager on the Quick Access Panel.** Create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child, or by first selecting the Base Scenario on

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

➤ **Right-click the Base Scenario in the Quick Access Panel.** Create another child and call it "0.25 Second Valve Closure". Finally, create one more child and call it "0.5 Second Valve Closure". The child scenarios should now be displayed as shown in Figure 8.

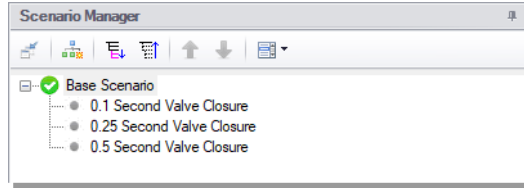


Figure 8: Completed Scenario Manager for High Pressure Steam - Forces

## B. Define the child scenarios

Since the "0.1 Second Valve Closure" scenario uses the same transient definitions as the base scenario, we do not need to modify it. We will set up the remaining scenarios using the following steps:

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

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

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

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

6. Enter the same Transient Data for Assigned Flow J12

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

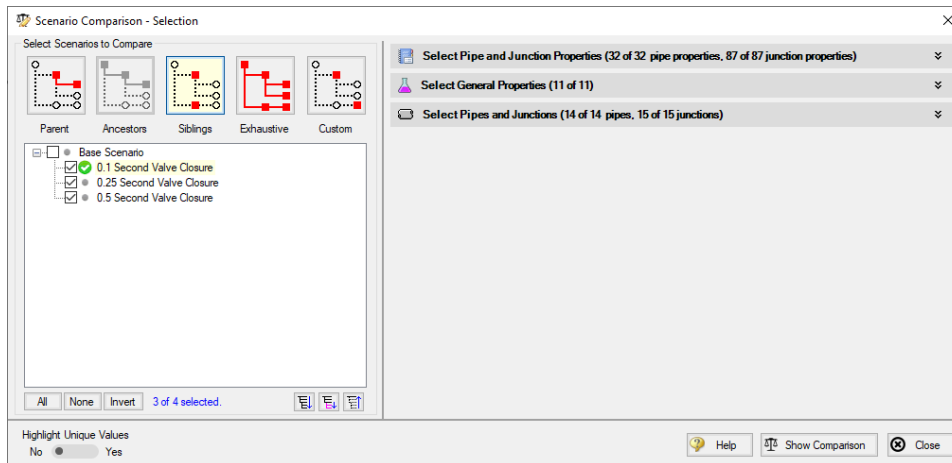
The three scenarios created should be identical except for the different transient data. To confirm the scenarios are otherwise identical, we will use the Scenario Comparison Tool.

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

## High Pressure Steam - Forces Example

3. In the Scenario Comparison - Selection window (Figure 9), select "Siblings"
4. Click "Show Comparison"
5. In the Scenario Comparison Grid, set the "Item" and "Parameter" sliders to show "Differences". Also set the "Highlight Unique Values" slider to "Yes".

The Scenario Comparison Tool will run a comparison of all Pipe and Junction Properties as well as the General Properties used in each scenario and highlight any differences in the Scenario Comparison Grid. In this model, the only highlighted differences should be for the "First Transient Data" parameter for junctions J10 and J12 (Figure 10). Note that each column in the Scenario Comparison Grid is highlighted in a different color. This indicates that each scenario possesses a unique value for the given parameters. Click close to exit the Scenario Comparison Tool.



**Figure 9: Scenario Comparison Tool - Selection window**

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

**Figure 10: Scenario Comparison Grid of the 3 scenarios**

## Step 9. Run the first scenario

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

## Step 10. View the results

### A. Open the Forces tab

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

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

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

### B. Graph the force results

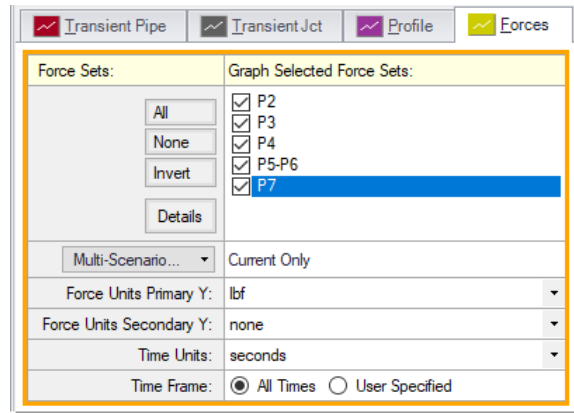
➤ **Open the Graph Results window.** We will create a graph that shows the transient forces over time for the force sets. To do so, perform the following steps:

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

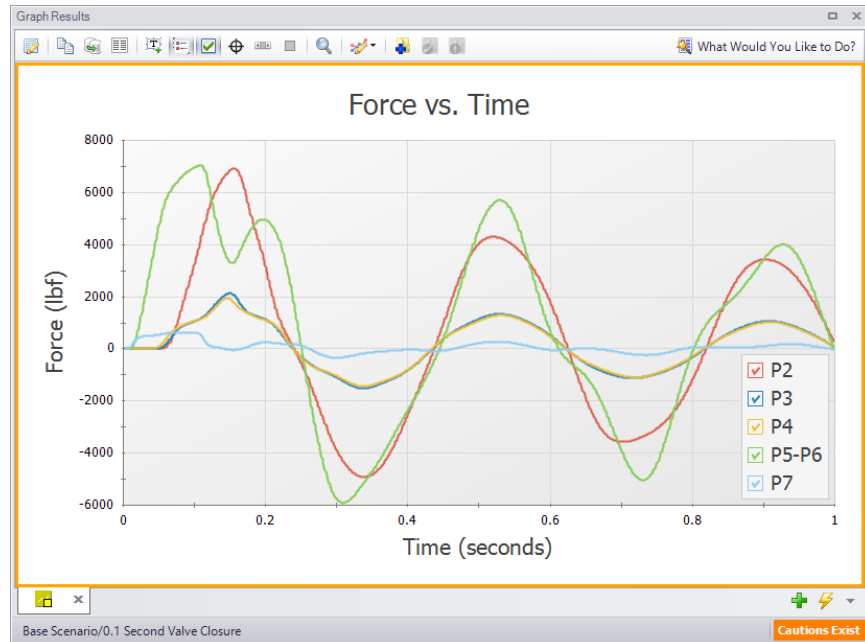
This should generate the graph shown in Figure 13.

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

## High Pressure Steam - Forces Example



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



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

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

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

There are two important points to be observed here:

## High Pressure Steam - Forces Example

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

### Step 11. Run the other child scenarios

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

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

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

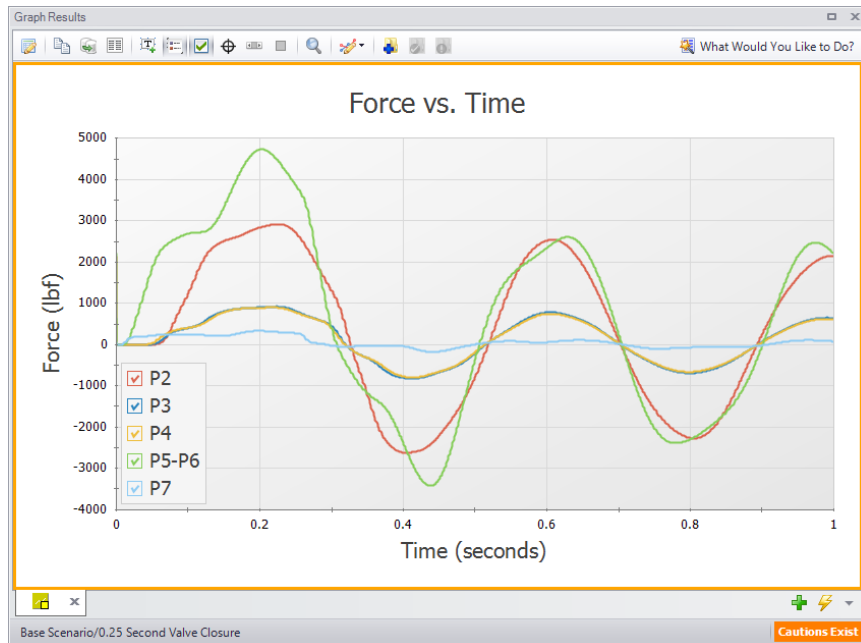
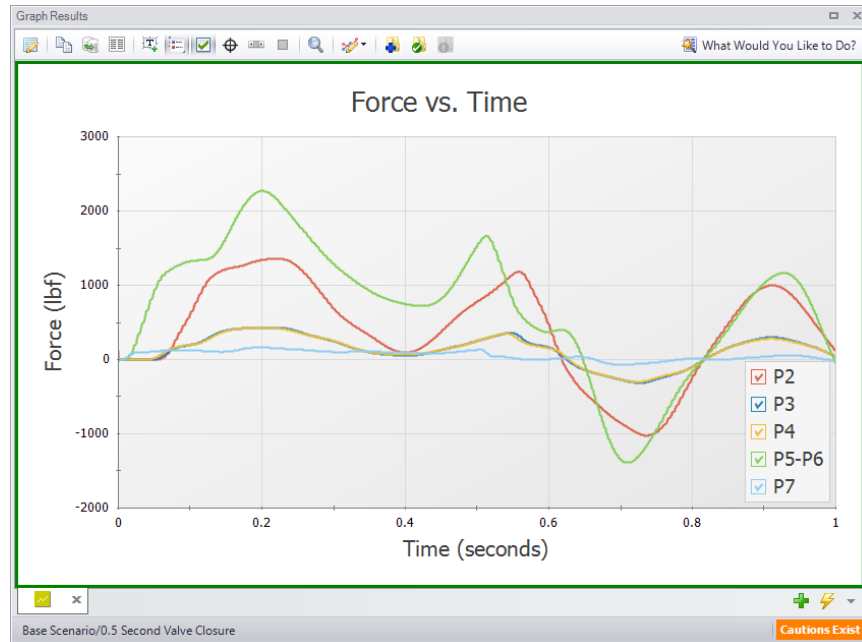


Figure 15: Graph for forces of the "0.25 Second Valve Closure" scenario

## High Pressure Steam - Forces Example

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

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



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

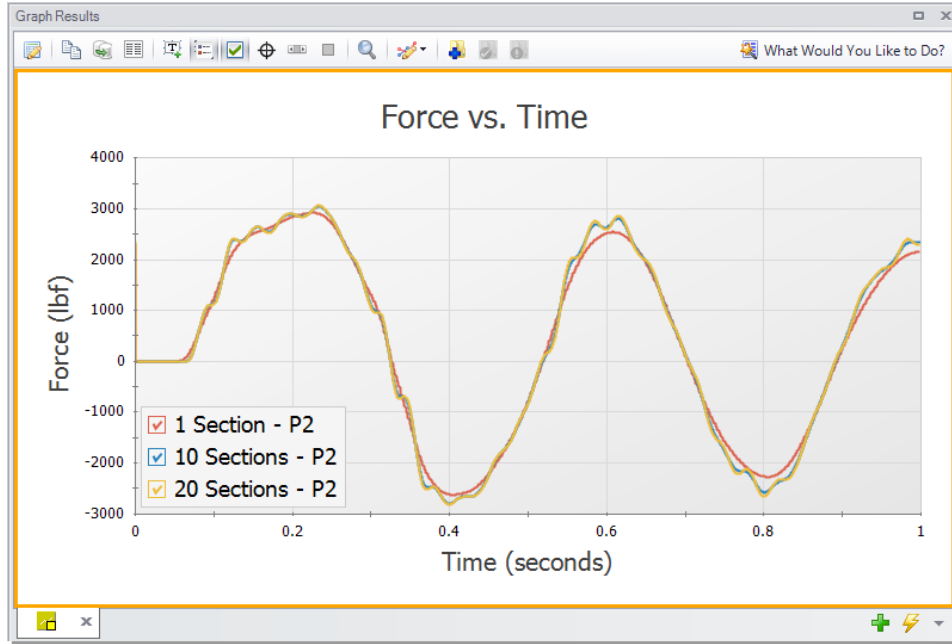
These figures show that the maximum force magnitude increases as the valve closes faster. Typically, the more rapidly a change occurs, the more severe the resulting forces are. Therefore, a utility of AFT xStream would be to evaluate the correlation between transient event duration (e.g., valve closing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

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

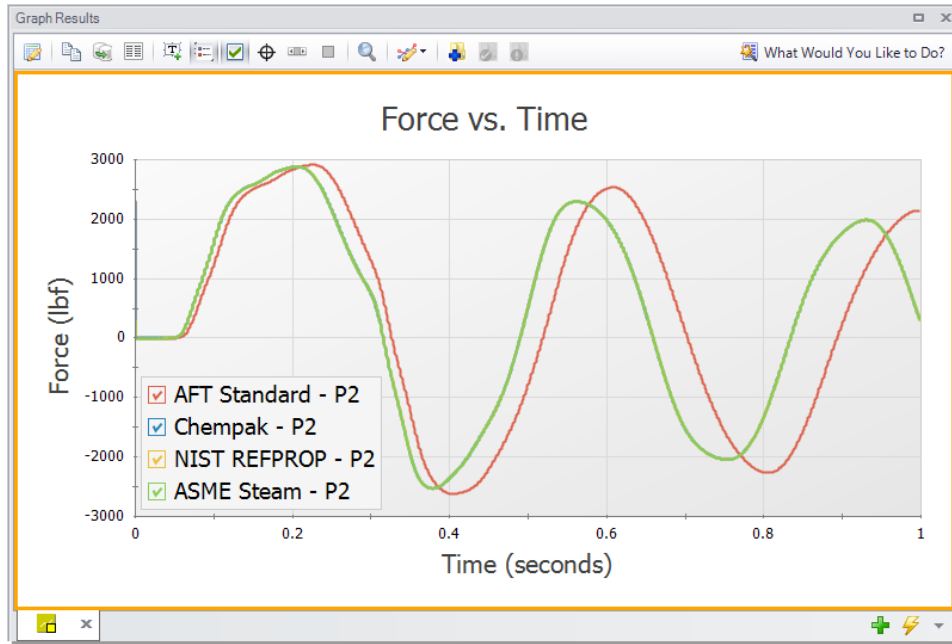
Pipe sectioning, the selection of fluid property model, and specification of Maximum Pipe Mach Number During Transient can have a significant effect on the accuracy of the forces calculated by AFT xStream. This effect was not significant in the system built within this example. However, the results for altering each of these three parameters are shown in Figures 18-20 so that you may see the impact of changing these three parameters. Note that in Figure 19, ASME Steam, Chempak, and NIST REFPROP are superimposed on each other, indicating no meaningful difference for this case.

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

## High Pressure Steam - Forces Example



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



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



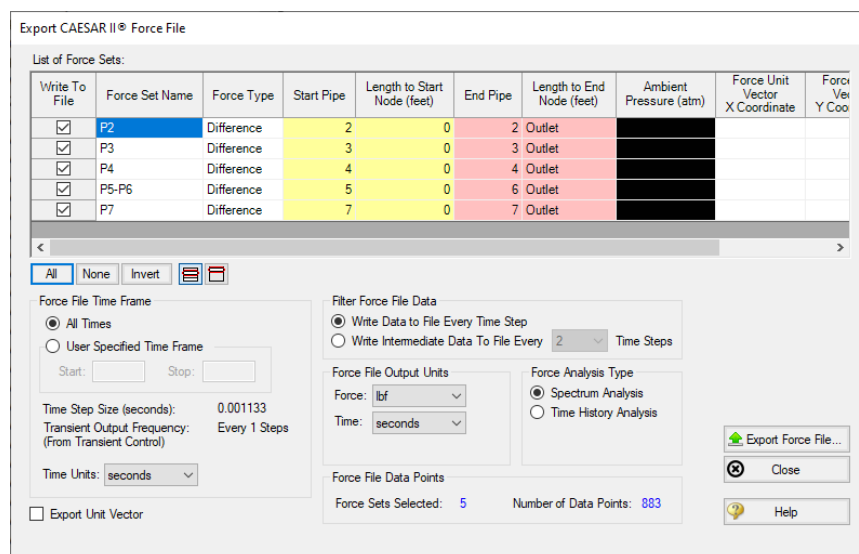
## High Pressure Steam - Forces Example



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

### Step 12. Export CAESAR II Force file

➤ Open the Output window, and click on the File menu. Select "Export Force File..." and choose "CAESAR II® Force File." This will open the export file window shown in Figure 21.



**Figure 21: Export CAESAR II® Force File window**

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

### **Conclusion**

---

We created a force model to calculate the forces experienced as a result of two simultaneous closures in a steam header. We also identified how the length of the transient can have a significant impact on the forces a system experiences. Finally, we discussed how to export these forces into a force file to be used in pipe stress analysis software.

## Frequency Analysis - PFA Example

### Summary

---

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

### Topics covered

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

### Required knowledge

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

### Model file

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

- ***US - Frequency Analysis PFA.xtr***

### Step 1. Start AFT xStream

---

Open AFT xStream. Click the Start Building Model button.

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

## Step 2. Build the model

### A. Place the pipes and junctions

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

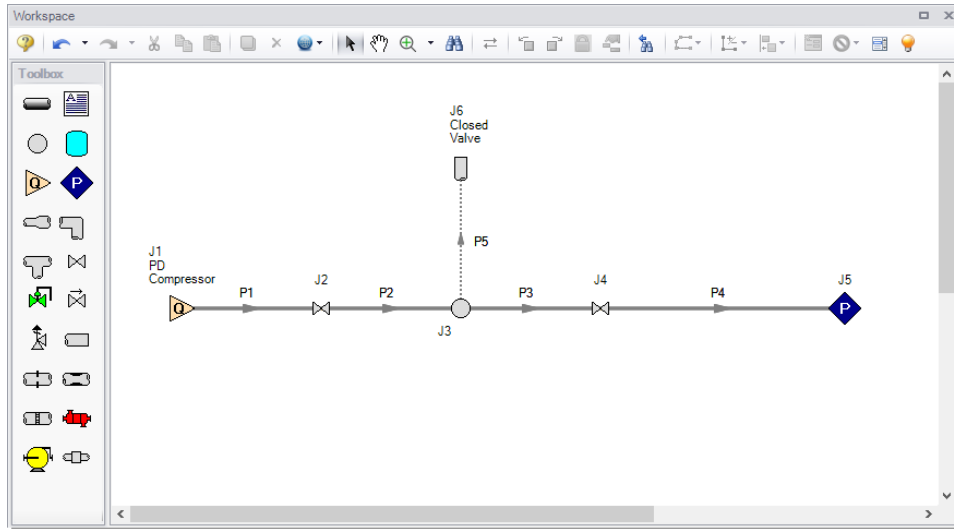


Figure 1: Layout of frequency analysis model

### B. Enter the pipe and junction data

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

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

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

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

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

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

## **Step 3. Specify Analysis Setup**

---

### **A. Specify the Fluid panel**

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

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

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

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

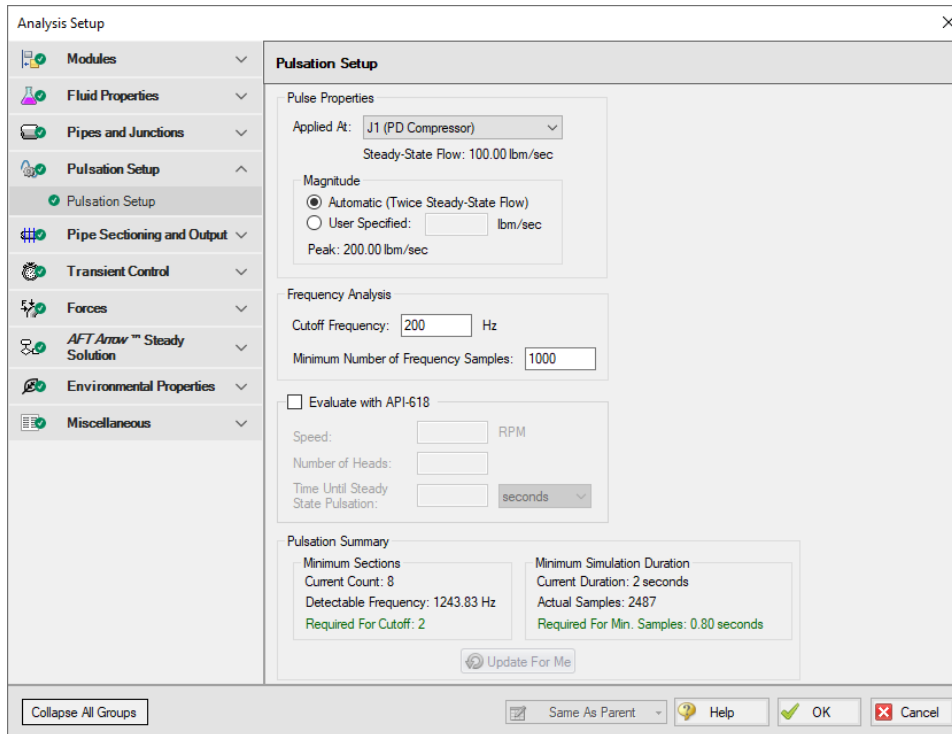
➤ **Open the Simulation Mode/Duration panel** in the Transient Control group. Enter the Stop Time as 2 seconds.

## **D. Specify the Pulse Setup panel**

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

Define the Pulsation inputs as follows (Figure 2):

1. Check that junction J1 is selected for the pulse to be applied at. Note that the pulse may only be applied at Assigned Flow or Compressor junctions.
2. For Magnitude, ensure that Automatic (Twice Steady-State Flow) is selected.
3. Under Frequency Analysis, define the Cutoff Frequency as 200 Hz and verify that the Minimum Number of Frequency Samples is set to the default value of 1000.
4. Leave the Evaluate with API-618 option unchecked. This option will be used in a later step once the acoustic frequencies have been determined.



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

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

Checking the Pulsation Summary for this example, the Minimum Sections Required for Cutoff are 2, and the Minimum Simulation Duration Required for Min. Samples is 0.80 seconds. Both of these requirements are satisfied by the current model settings. The "Update For Me" button can be used to automatically change the minimum number of sections and/or the minimum simulation duration if they are insufficient. Click OK to exit the Analysis Setup window.

## Step 4. Run the model

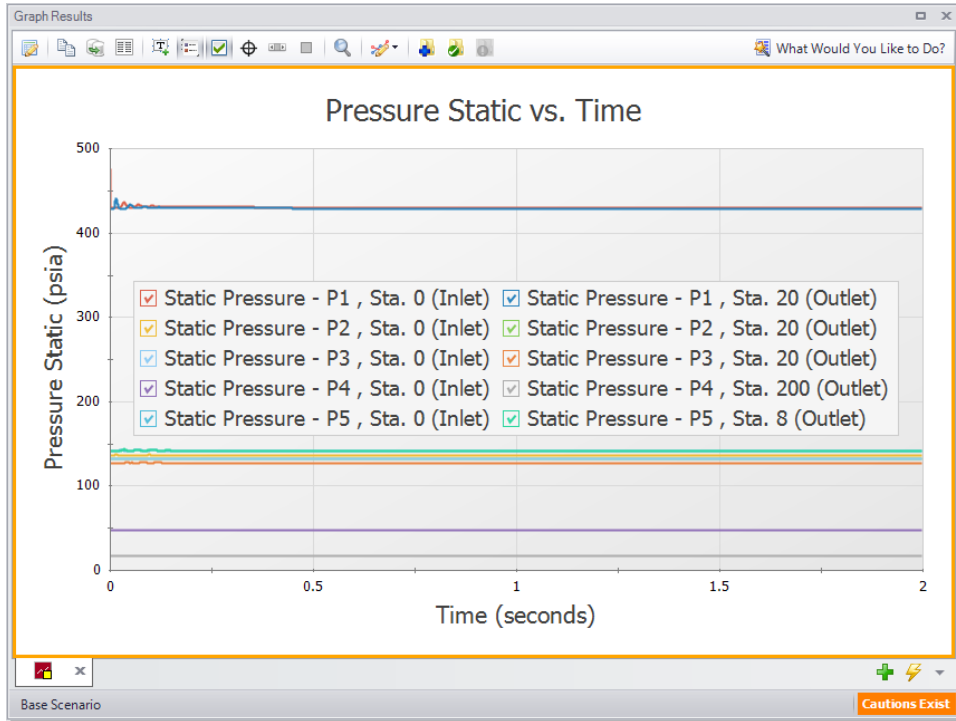
➤ **Select "Run Model"** from the Main Toolbar. This will open the Solution Progress window. This model has an estimated run time of 5 minutes, but the run time is dependent on the speed of your PC.

## Step 5. Review the frequency results

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

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

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

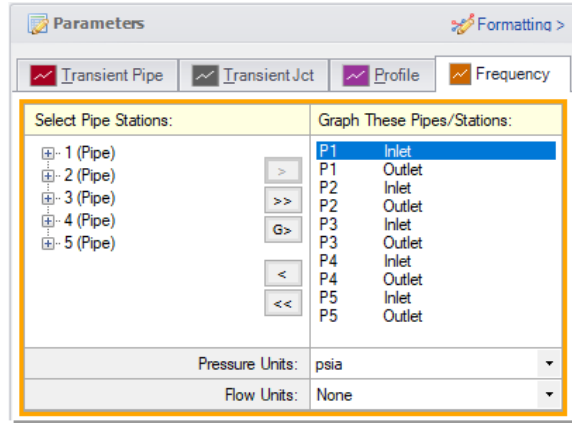


**Figure 3: Pressure Static in each of the pipes for the frequency analysis**

Next, graph the acoustic frequency response as follows:

1. Click the "New Tab" button
2. Select the Frequency tab in the Graph Parameter section of the Quick Access Panel (Figure 4)
3. Add the inlet and outlet station for each of the pipes
4. Verify that the pressure units are set to psia
5. Click Generate



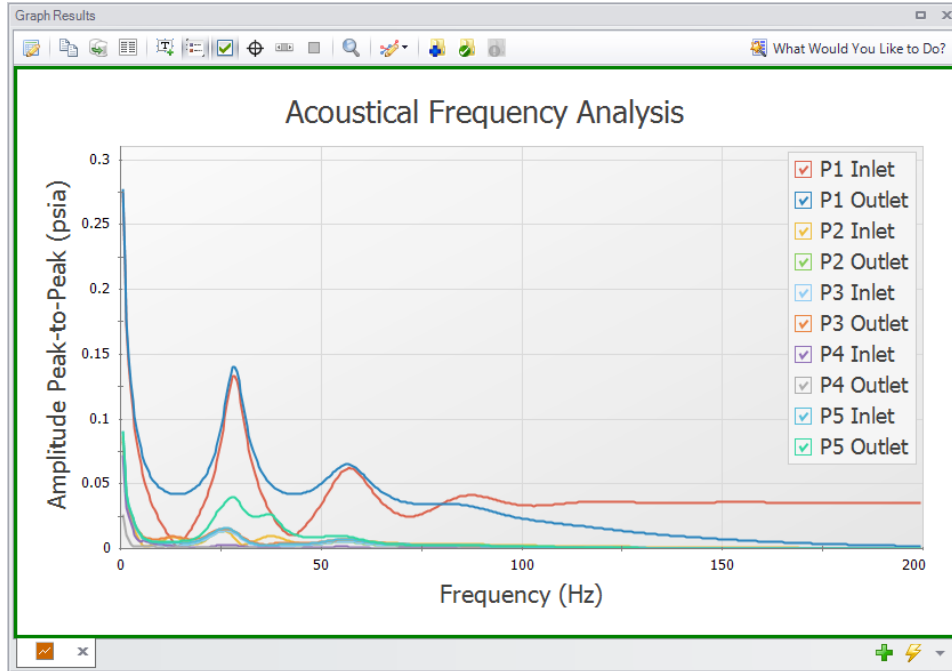


**Figure 4: Graph Parameters section for the frequency analysis**

**Identify excitation frequencies to study**

From the graph in Figure 5, there are several frequencies that produce a large pressure response. Points on the plot with a large response magnitude are easily identified by finding local maxima on the graph. These maxima are the acoustic natural frequencies that are potentially the most damaging when excited by a positive displacement compressor. These frequencies can also be caused by equipment that contributes large pressure drops, such as control valves.

Two natural frequencies are identified for the system based on the peaks in the graph at roughly 28 Hz and 57 Hz. Note that the number of sections used for the analysis may affect the location of the peaks, so it is prudent to run the analysis with more sections to reduce uncertainty in the results. It is also recommended to analyze frequencies near identified peaks (for example, analyzing 55 and 59 Hz for the peak at 57 Hz) to account for potential shifting of the frequency peaks that can result from the assumptions made for the frequency analysis in AFT xStream.



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

## Step 6. Determine the pressure response for API-618

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

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

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

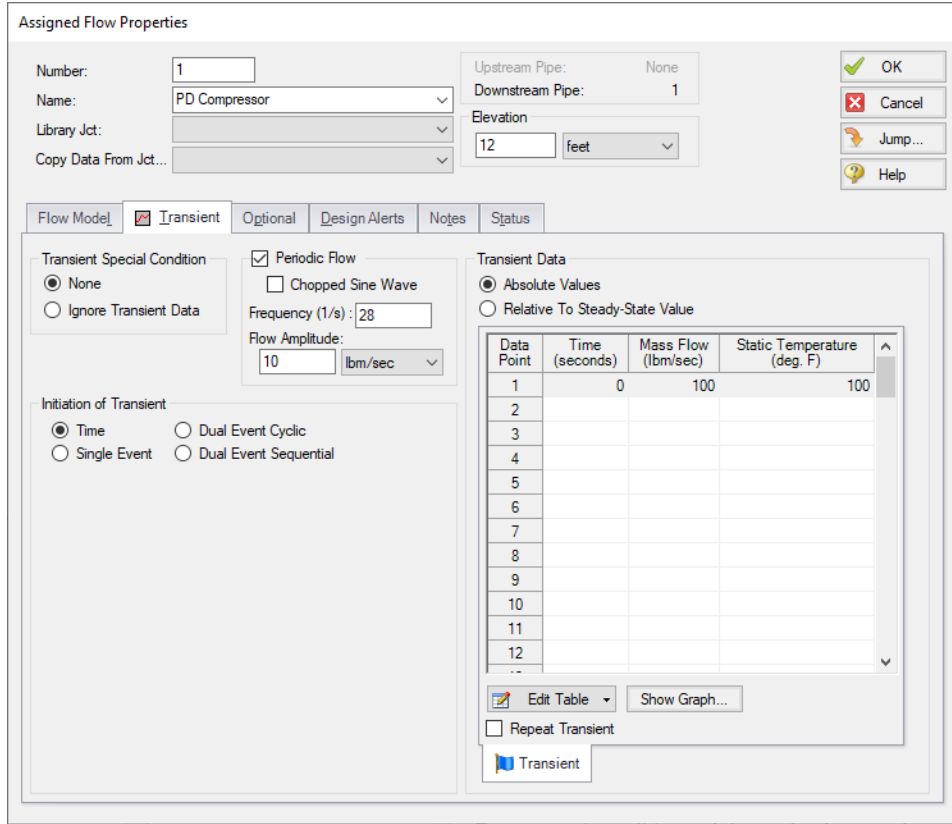


Figure 6: Assigned Flow Properties window with Periodic Flow transient applied

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

In the Output window there is now a warning stating that the maximum pressure limit for API-618 was exceeded as is shown in Figure 7. When the API-618 evaluation is performed the Pulsation Summary tab is shown in the Output window with additional information on where pressure limit violations occurred, and what inputs were used to calculate the API-618 pressure limits for the scenario (Figure 8).

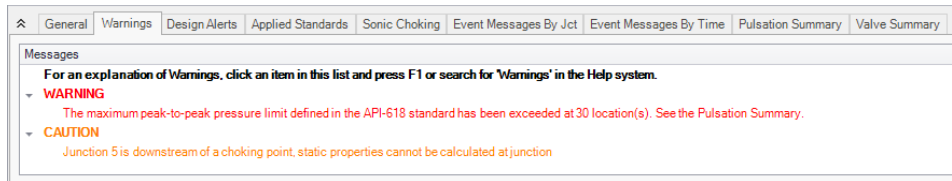
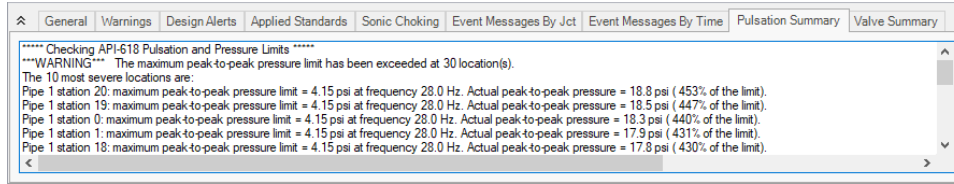


Figure 7: Warning showing that the API-618 pressure limits have been exceeded at multiple locations



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

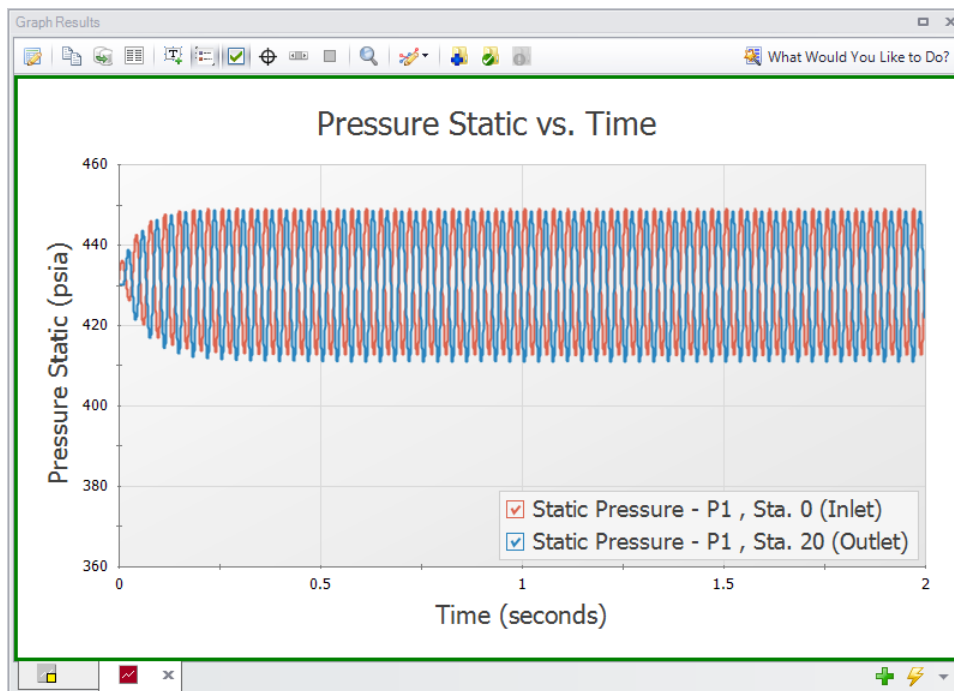
In the Graph Results window, generate a graph of the static pressure following the same process used to create Figure 3. However, only include the pipe P1 inlet and outlet station. The resulting graph is shown in Figure 9. These pressure results can be used to check for compliance with standards such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

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

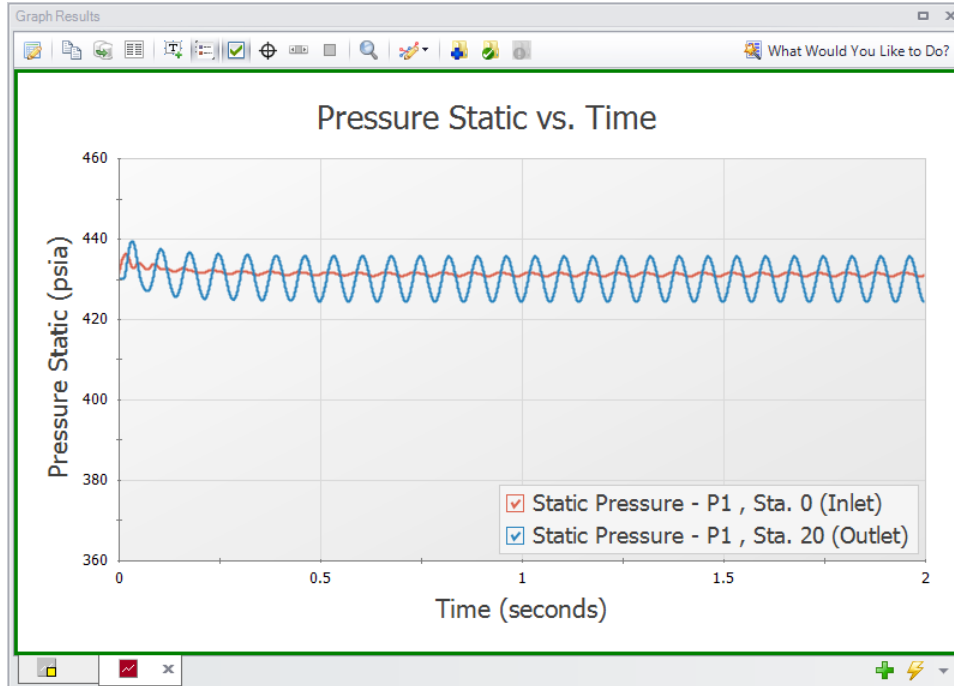
Note for this example the value of 0.25 seconds was given in advance to expedite the process, but normally the user would need to run the model once with the "Time Until Steady..." set to 0, then determine the time input and re-run the model with the correct value.

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

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



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



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

## Conclusion

The PFA add-on module for AFT xStream can be used to apply a pressure pulse to the system in order to identify acoustic frequencies which may result in damage. These frequencies can be used in further pulsation analysis to determine if pulsation mitigation is required, and what mitigation would be effective.

## Index

---

### A

Activate Modules 273

Add Component to Library 273

AFT Arrow Steady Solver 559, 603

AFT Standard library See Fluid libraries, AFT Standard library

Align 275

Analysis Setup 272

Analysis Setup window 554, 580, 582

Animation

    Graph 599

    Speed adjustment 599

Annotation Tool See Toolbox, Annotation tool

API 618 623, 632

Area Change

    Specifications window 265

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

Assigned Flow

    Specifications Window 265

Assigned Pressure

    Specifications Window 265

### B

Bend

    Specifications Window 265

Bounded by known flow rates 61

Branch

Specifications Window 265

**C**

Chempak library See Fluid libraries, Chempak library

Clear Initial Guesses 271

Compressor/Fan

Specifications Window 265

Compressor/Fans

Specifications Window 265

Condensation 599, 603

Control Valve

Specifications Window 265

Conventions 59

Converting models and libraries 534

Copy Data From Jct... 546

Custom Libraries 391, 394-395, 403

Cutoff Frequency 92, 386

**D**

Dead End

Specifications Window 265

Defining force sets See Forces, Force sets, Definition

Defining objects

List Undefined Objects window 544, 580, 595, 610, 625

Design Factors 189

Disable Pipes Snapping to Isometric Grid 275

Display on Workspace 189

Duplicate Special 270

## E

Edit menu 539

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

Export force files See Forces, Exporting force files

Export Model Data 268

## F

Fittings & Losses 274

Flip Vertex 275

Fluid libraries

AFT Standard library 555, 580, 595, 603, 625

ASME Steam Tables library 555, 604

Chempak library 555, 604

Effect of 602, 619

NIST REFPROP library 555, 604

Selection 603, 605

Fluid Library 274

Fluid panel 555, 580, 595, 625

Fluid Property Viewers 273

Forces

Exporting force files 621

Force sets 611

Definition 611-612

Direction 611



Force Unit Vectors 612

Methodology 617

Forces panel 622

## G

General Component

Specifications Window 265

Graph Animations See Animation, Graph

Graph Control tab

Forces 616

Profile tab 586, 599

Transient Jet tab 601

Transient Pipe tab 584, 596

Graph List Manager 585, 588, 616

Graph Results window 563

Modifying axes of graphs 596

## H

Heat Exchanger

Specifications Window 265

Highlight feature 546

## I

ID numbers 538, 544

Import Graph Data 268

Initial Flow Guess 188

Inspection feature 547

Insulation 173

Insulation Library 274

Integration Query 272

Isometric See Pipe Drawing Mode, Isometric

## J

Jump button 552

Junction library 273, 546

Junctions 265, 538

    Tank 538, 545

## K

Keep Network eLicense Checked Out 273

## L

Libraries 391, 403

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

Load Background Picture 269

Lock Object 275

## M

Make Selected Pipes and Jcts Orthogonal 275

Merge 268

Method of Characteristics 556, 583

Minimum Number of Frequency Samples 93, 386

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

Mixtures 555, 603-604

Model building process 603

Model Data window 553

**N**

Network Libraries 392

NIST REFPROP library See Fluid libraries, NIST REFPROP library

**O**

Objects 538

Open Pipe/Jct Window 545

Orifice

    Specifications Window 265

Output Control window 560

Output file See Transient output file

Output window 559-560, 583

    Forces tab 616, 618

    Transient Max/Min tab 584

**P**

Parallel Pipes 189

PFA Module

    Excitation frequencies 629-630

Pipe Drawing Mode 275

    2D Freeform 591

    2D Orthogonal 591

    Isometric 591, 607

        Adding segments 591

Pipe Drawing Tool See Toolbox, Pipe Drawing Tool

Pipe Line Size and Color 188

Pipe Material Library 274

Pipe Properties window 551

Pipe Sectioning panel 556, 580, 584, 595, 611, 626

    Estimated Maximum Pipe Mach Number During Transient

        Effect of 619

    Estimated Maximum Temperature During Transient 557, 580, 595, 611, 626

        Effect of 583

    Minimum Number of Sections Per Pipe 557, 580, 595, 611, 626

        Effect of 571, 619

    Sectioning Summary 557

Pipe Segments 275

Properties windows

    Notes tab 546

    Optional tab 546

    Status tab 546, 552

    Transient tab

        Specification 614, 630

Pulsation Frequency Analysis 272

Pulsation panel 626

    Cutoff Frequency 626

    Minimum Number of Frequency Samples 626

    Pulsation Summary 627

## Q

Quick Access Panel 537

    Graph List Manager 585

**R**

Reset Label Locations 275  
Reverse Direction 275  
Rotate Icon 275  
Rotate Selection 275  
Rotating Icons 591, 607  
Run Model 272, 558, 583-584, 596, 616, 627, 631

**S**

Satisfying Connectivity Requirements 161  
Scale/Flip Workspace 275  
Scenario Comparison Tool 614-615  
Scenario Manager 584, 588, 613, 630  
Screen  
    Specifications Window 265  
Select Flow Path 270  
Select Special 270  
Selecting a fluid library See Fluid libraries, Selection  
Set Pipe and Junction Locations to Parent 275  
Show Grid 275, 591, 607  
Show Object Status 162, 544, 546  
Simulation Mode/Duration panel 556, 558, 581, 596, 611, 626  
Snap to Grid 275  
Solution Progress window 558  
Sonic choking 565, 589  
Sort Table By 275

Special Conditions 197

Status Bar 554

## T

Tank

    Specifications Window 265

Tank junction See Junctions, Tank

Tee/Wye

    Specifications Window 265

Time Simulation 272

Toolbars 554

Toolbox 537-538

    Annotation tool 538

    Pipe Drawing Tool 538, 541, 543

Transfer Results to Initial Guesses 270

Transient Control group 558

Transient event specification See Properties windows, Transient tab, Specification

Transient Jct tab See Graph Control tab, Transient Jct tab

Transient output file 560

Transient Pipe tab See Graph Control tab, Transient Pipe tab

Transient Solver 559, 603

## U

Undefined objects list See Defining objects, List Undefined Objects window

Undefined Objects Window 161

Unique solution 61

Use Status Feature 161

User Options 544

**V**

Valve

    Specifications Window 265

Venturi

    Specifications Window 265

Visual Report Control 568

Visual Report window 568

**W**

Warnings in Output 492

Workspace window 537, 576