

AFT xStream[™]

Product Help Guide

AFT xStream Version 3

Gas Transient Modeling in Piping Systems

Published: March 08, 2024



Dynamic solutions for a fluid world[™]

Contents

Overview of AFT xStream	17
Engineering Assumptions	18
New in Version 3	18
Interface Improvements for xStream 2	29
New in Version 2	31
xStream Learning Resources	33
Startup Window	34
Overview of Fluid Transients	37
Wave Velocity and Communication Time	37
Pipe Sectioning - Introduction to Method of Characteristics	38
Minimizing Run Time	41
Transient Forces	43
Defining Forces	43
Force Set Types	46
Exporting Force Data	46
Force Theory	47
Interpreting Transient Results	51
Animating Graph Results	51
Animation Layers	55
Transient Result Accuracy	59
Transient Sensitivity Analysis Tutorial	62
Fundamental Modeling Concepts	69
Pipes and Junctions	69
Features for Modeling Irrecoverable Losses	70

Reference Positive Flow Direction	70
Philosophy of Computer Modeling	71
Role of Pressure Junctions and How They Work	72
Open vs. Closed Systems	75
Curve Fitting	77
Stagnation vs. Static Pressure Boundaries	79
Analysis Setup Overview	83
Modules Group	84
Modules Panel	84
Fluid Properties Group	87
Fluid Panel	87
Pipes and Junctions Group	102
Pipes and Junctions Panel	102
Pulsation Setup Group	103
Pulse Setup Panel	103
Pipe Sectioning and Output Group	111
Pipe Sectioning and Output Panel	112
Transient Control Group	114
Simulation Mode/Duration Panel	114
Transient Solver Initialization Panel	114
Forces Group	117
Force Definitions Panel	117
AFT Arrow Steady Solution Group	120
Solution Method Panel	120
Tolerance Panel	148
Iterations Panel	152

Pressure and Temperature Limits Panel	154
Environmental Properties Group	155
Environmental Properties Panel	155
Miscellaneous Group	157
Advanced Parameters	157
Transient Solver Advanced Parameters	157
Working with Pipe and Junction Objects	159
Selecting and Finding Objects	159
Selecting Objects	159
Finding Workspace Objects	162
Groups	163
Pipe Drawing Modes	163
Global Edit	171
Global Pipe Editing	171
Global Junction Editing	172
Toolbox	173
Creating Pipes - the Pipe Drawing Tool	175
Joining or Splitting Pipes	176
Creating Junctions	178
Connecting Objects	178
Moving Objects	179
Editing Objects	180
Morphing Junctions	180
Defining Objects	180
Highlight Required Input	182
Inspecting Objects	182

Notes	183
Status	183
Workspace Pan	184
Moving Pipe and Junction Labels	184
Special Conditions	185
Pipes	187
Pipe Properties Window	187
Common Pipe Input Parameters	188
Pipe Model	189
Fittings & Losses	190
Heat Transfer in Pipes Overview	192
Constant Heat Transfer	194
Convective Heat Transfer	194
Heat Tracing Flux Based On Wire Turns and Length	197
Steady State Heat Transfer In Pipes	201
Transient Heat Transfer in Pipes	207
Insulation for Pipes	210
Optional Input for Pipes	210
Add/Remove Pipe Segment	211
Reverse Direction	212
Pressure Drop in Pipes - Detailed Discussion	212
Junctions	215
General Junction Information	215
Two Window Formats	217
Common Junction Input Parameters	219
Optional Input for Junctions	220

Specifying Losses	220
Conventions for Specifying Basis Area	221
Reference Density	221
Equal Percentage Valves	223
Junction Loss Models	224
Discharge Coefficient Loss Model	224
K Factor Loss Model	225
Cv/Kv and XT Loss Model	228
Resistance Curve Loss Model	231
Defining Junction Transients	233
Junction Transient Data	233
Initiation of Transients	235
Multi-Condition Events	239
Area Change	242
Area Change Loss Model	243
Assigned Flow	244
Assigned Flow Transient Theory	245
Assigned Pressure	246
Assigned Pressure Transient Theory	247
Bend	248
Bend Loss Model	249
Branch	252
Branch Transient Theory	252
Check Valve	254
Compressor/Fan	255
Centrifugal Compressor	257

Positive Displacement Compressor	269
Compressor/Fan Efficiency Data	276
Control Valve	277
Control Valve Transient Theory	279
Dead End	280
Dead End Transient Theory	280
General Component	281
Heat Exchanger	282
Orifice	282
Orifice Loss Model	283
Relief Valve	285
Relief Valve Transient Theory	287
Screen	287
Screen Loss Model	288
Tank	289
Tank Transient Theory	291
Tee/Wye	292
Tee/Wye Transient Theory	293
Valve	293
Venturi	295
Interface and User Options	297
Menus and Toolbars	297
File Menu	298
Edit Menu	299
View Menu	301
Analysis Menu	301

Tools Menu	302
Library Menu	303
Arrange Menu	304
Window Menu	305
Help Menu	306
Toolbars	307
Printing	308
Primary Window Overview	309
Status Bar	311
Quick Access Panel	311
Model Status Light	322
Solution Progress Window	322
Workspace Window	322
Workspace Layers	324
Merging Models	333
Renumbering Workspace Objects	333
Scale/Flip Workspace	334
Magnifier Tool	334
Annotations	335
Extended Model Check	339
Keyboard Shortcuts for Workspace	340
Undefined Objects Window	342
Raw Data Changes	343
Duplicate Special	343
Model Data Window	343
Model Data Control Window	345

Output Window	347
Output Control	353
Output Parameters	357
Applied Standards	358
Graph Results Window	359
Quick Graph Formatting Window	364
Graph Types	364
Basic Graph Tutorial	367
User Options Overview	379
Unit System	380
Workspace	382
Parameters	385
Files	386
File Import and Export	389
Importing Piping Component Files	389
Importing GIS Shapefiles	393
Importing Caesar II Netral Files	395
Importing and Exporting Epanet Files	397
Importing and Exporting from Other AFT Products	402
Exporting to Excel	403
Importing Excel Data	407
Design Alerts	413
Design Alerts From Output	416
Design Alert Importing & Exporting	417
Scenario Manager	419
Creating, Organizing, and Editing Scenarios	420

Modifying Individual Scenarios	423
Scenario Inheritance	423
Running Models in Batch Mode	426
Open Model Using Last Scenario	427
Scenario Comparison Tool Overview	427
Accessing Scenario Comparison Tool	428
Modify Comparison	429
Scenario Comparison Grid	430
Add-on Modules	433
Introduction to the PFA Module	433
Defining Pulsation Settings	434
Interpreting Pulsation Results	441
PFA Module Glossary	442
Libraries	443
Library Manager	444
Library Browser	444
Junction Library	447
Fluid Library	447
Pipe Material Library	448
Insulation Library	450
Fittings & Losses Library	451
Network Library Overview	456
Benefits of Shared Libraries	457
Creating an enterprise-wide, network library system	457
Library Verification	460
Library Locations in General Preferences	462

Theory and References	463
Nomenclature	463
Review of Compressible Flow Theory	465
Transient Theory	483
Transient Flow Equations	483
Method of Characteristics	490
Compatibility Equations	495
Transient Sonic Choking	497
Compressible Flow Theory in Single Pipes	498
Extension of Single-Pipe Methods to Networks	498
Fundamental Equations	499
Continuity Equation at a Branch	500
Energy Equation at a Branch	500
Network Implementation	501
Verifying Network Solutions	502
Sonic Choking	503
Sonic Choking Detailed Description	504
Subsonic vs. Sonic Pressure Losses	509
Stagnation Properties	510
Pipe Material Library References	511
References	516
Chempak Technical Information - Detailed Discussion	518
Role of Pressure Junctions - Detailed Discussion (Long)	535
Output Messages & Warnings	547
Critical Warnings	547
Characteristic speed exceeded maximum determined from...	548

Global Energy Balance Disabled	548
Global Mass Balance Disabled	548
Junction X Had Reverse Flow - Compressor Pressure/Head Could Not Be Predicted	549
Maximum transient pipe mach number X higher than estimate Y	549
Pipe Mach number exceeded 1 during simulation	549
Required flow to supply the pressure control valve at junction X is above sonic flow	550
Setting relaxation lower than relative pressure/flow tolerance gives unreliable results	550
Specified flow through/out of junction X is above sonic flow and cannot be supplied	550
These results do not represent a converged solution	550
Warnings	551
Automatic Sectioning Did Not Achieve Steady Conservation	551
Chempak Temperature/Pressure Outside Valid Range	551
Control Valve Open Percentage Outside Valid Range	551
Energy may not be balanced at Junction X	552
FCV could not control flow at Junction X	552
Flow may not be balanced at Junction X	552
Fluid is no longer in the vapor phase because the fluid temperature/pressure is... ..	553
Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy	554
Junction X Compressor/Fan added negative pressure	554
Junction X Exceeded End-of-Curve Flow Rate	555
Junction X Had Reverse Flow During Transient	555
Maximum system pressure exceeded enthalpy table's maximum pressure of X	555

Maximum Transient Pipe Temperature Higher Than Estimate	555
Maximum Transient Pipe Temperature Significantly Lower Than Estimate	556
Physical flow area is less than the sonic area at junction X... ..	556
Pipe/Junction X property calculation exceeded maximum local iteration limit	557
Pipe/Junction X property calculation failed to continue towards convergence	557
Pipe X temperature is outside temperature limits in AFT Arrow Steady Solution	558
PRV could not control pressure at Junction X	558
PSV could not control pressure at Junction X	559
Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range	559
The maximum difference between the AFT Arrow Steady solution and the MOC Steady Solution for velocity is X%	560
The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X locations	560
The MOC Steady Solution did not reach a nominally steady state before the transient started	561
Transient choking occurred at Junction X	561
Transient temperature was above/below fluid maximum/minimum temperature	562
Transient temperature was below saturation temperature	563
Cautions	564
All transient junction behavior was disabled	564
Assigned Flow at Junction X has a constant flow rate during the transient... ..	565
Automatic Sectioning is enabled, but flow increases substantially during the transient	565

Check Valve closed at Junction X	565
Flow through Junction X is negative. Loss factors may be in error.	565
Junction X exceeded maximum event messages	566
Junction X is downstream of a choking point	566
Maximum transient pipe Mach number significantly lower than estimate	567
Maximum transient pipe Mach number at least 0.2 higher than steady...	567
Pipe X length is shorter than the elevation change (between Junction Y and Z)	567
Transient data did not extend to Stop Time at Junction X	568
Transient temperature was within 10 deg. F of fluid minimum/maximum	568
Transient temperature was within 10 deg. F of saturation	569
Information	570
Flow at the Following Junctions is Laminar	570
Pulsation peak-to-peak pressures are within allowable API 618 levels ...	570
Troubleshooting	571
Reducing Run Time	572
The model has both very long and very short pipes	573
The model has many pipes that can be combined	574
The model has an unusually short pipe	575
The simulation has a large number of time steps	576
Transient Solver Errors	577
General error occurred during the transient calculations	578
Junction iteration problem	578
Possible supersonic flow detected at the connection of Junction X and Pipe Y	578
User Specified AFT Arrow to Resolved allowable mismatch was exceeded	579

Diagnostics Window	580
Use the Model Data Window	581
Transfer Results to Initial Guesses	582
Poor Compressor Curve Fits	583
Troubleshooting Resistance Curves	583
Use the Output Window Sort Feature	584
Lower the Flow Rate Relaxation	585
Try Absolute Tolerance	585
Make Initial Flow Rate Guesses for Pipes	585
Change Boundary Conditions	586
Turn Off Parts of the Model	586
Break a Large Model Into Submodels	586
Use an AFT Standard Fluid	586
Run the MOC Steady Solution	587
Simplify Event Transients	587
Limit Properties to Saturation Conditions	587
Enable Transient Debug Files	588
Identifying and Restoring Corrupted Model Files	588
Technical Support	593
Technical Support Options	593
Contacting AFT	594
Information on Software Installation	594
Chempak Database Installation	594
Output Files	596
Updated Fluids Library	596
Special File Control Features	597

Converting Models from Earlier Versions	598
Index	599

Overview of AFT xStream

Welcome to AFT xStream™ - a visual platform for modeling compressible fluid transients in pipe networks! [See what's new in this version.](#)

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 and video tutorials 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.....	33
Primary Window Overview.....	309
Overview of Fluid Transients.....	37
Philosophy of Computer Modeling.....	71
Method of Characteristics.....	490

Engineering Assumptions

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

- [Compressible flow](#)
- One-dimensional flow
- System is fluid-full
- All gases are superheated (single phase)
- No chemical reactions
- No supersonic flow in the system
- Pipes are rigid (pipe geometry does not change during the transient)
- The Method of Characteristics (MOC) solution used 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

New in Version 3

View the links below for key updates from past versions:

[New in Version 2](#)

Workspace Layers

The [Workspace Layers](#) feature provides a powerful new tool for customizing and viewing model appearance, input data, and results - all in one place. The Workspace Layers toolset significantly enhances the model building process, providing unparalleled customization options. You can easily update pipe/junction size, color, display text, visibility, and more from a centralized

location on the Quick Access Panel. Workspace Layers surpass the capabilities of the previous Visual Report feature. With Layer Settings, you now have easy control over Color Mapping and the visibility of both Input and Output parameters. Any change you make to these settings will have an immediate impact on the model's display, providing real-time feedback. The dynamic combination of unique Layers offers an incredibly flexible framework, allowing you to display precisely what you desire on the Workspace. You can easily toggle the visibility of individual layers on or off, and the state of all layers can be saved to a Layer Preset for future use.

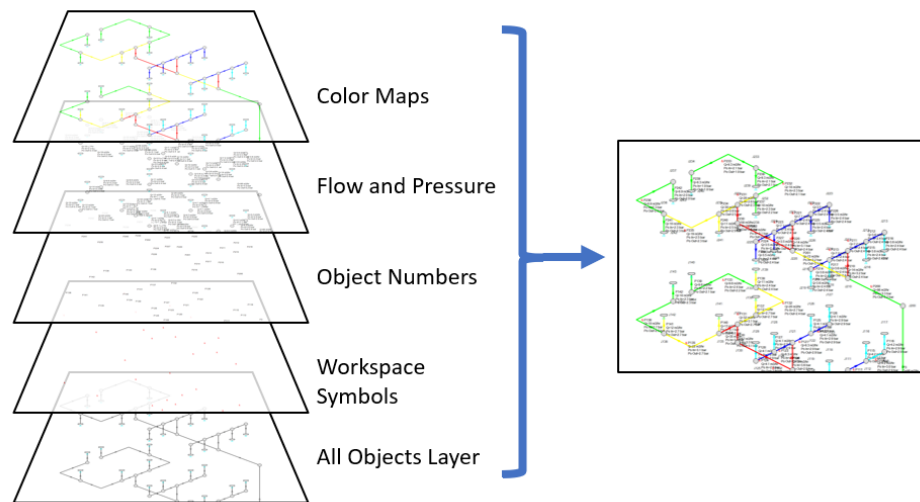


Figure 1: Workspace Layers can display input and output values superimposed over the model, show Color Maps, and control visibility of objects

Reciprocating Compressors

Model single or double-acting multi-cylinder [reciprocating compressors](#), accounting fully for all fundamental thermodynamics. Completely new in this version of AFT xStream is a fully transient reciprocating compressor model that considers the thermodynamic effects of gas being compressed or expanded in a closed chamber. Combined with the fundamental gas dynamics considered by the AFT xStream transient solver, this offers an exceptionally powerful simulation tool. The reciprocating compressor model offers the flexibility to specify single or double-acting compressors, with various arrangements including opposed, inline, or V configurations, accommodating up to six cylinders. The model incorporates a physical representation of the cylinder geometry and valve opening crank angles. In combination with the PFA module improvements in this version, this enables engineers to analyze the pulsation effects of reciprocating compressor systems with greater precision and confidence.

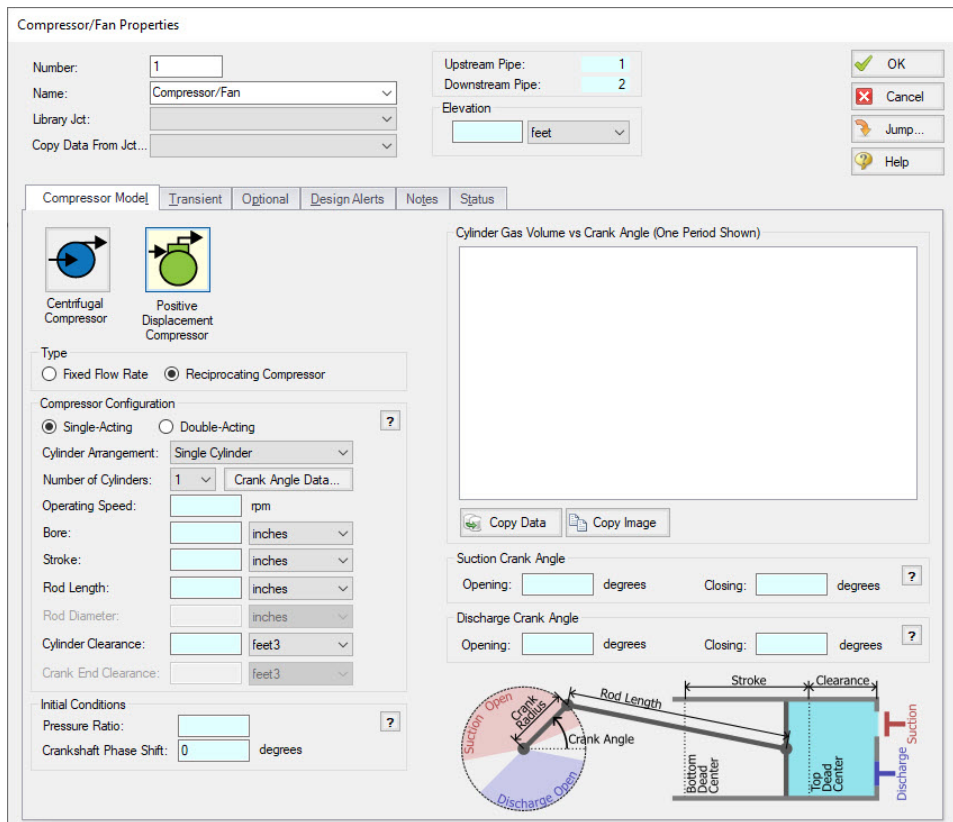


Figure 2: Reciprocating Compressors can be modeled inside the Compressor Properties menu under the Positive Displacement option

Transient Checkpoints

Extend the length of an existing transient run without the need to re-run the entire simulation, or stop and continue a run from other points in the simulation. [Transient Checkpoints](#) enable you to save the state of a transient simulation at any given point and resume it later from that exact point. This functionality proves particularly useful in scenarios where you need to pause a simulation temporarily to provide access to another user or when unexpected interruptions, such as system updates, occur during the simulation. This feature also allows for the extension of a transient run. If you have ever experienced the frustration of waiting for results only to realize that you should have run the simulation for a slightly longer duration, this feature provides a solution. Instead of rerunning the entire simulation, you can simply extend an existing run by leveraging the automatically created Transient Checkpoints.

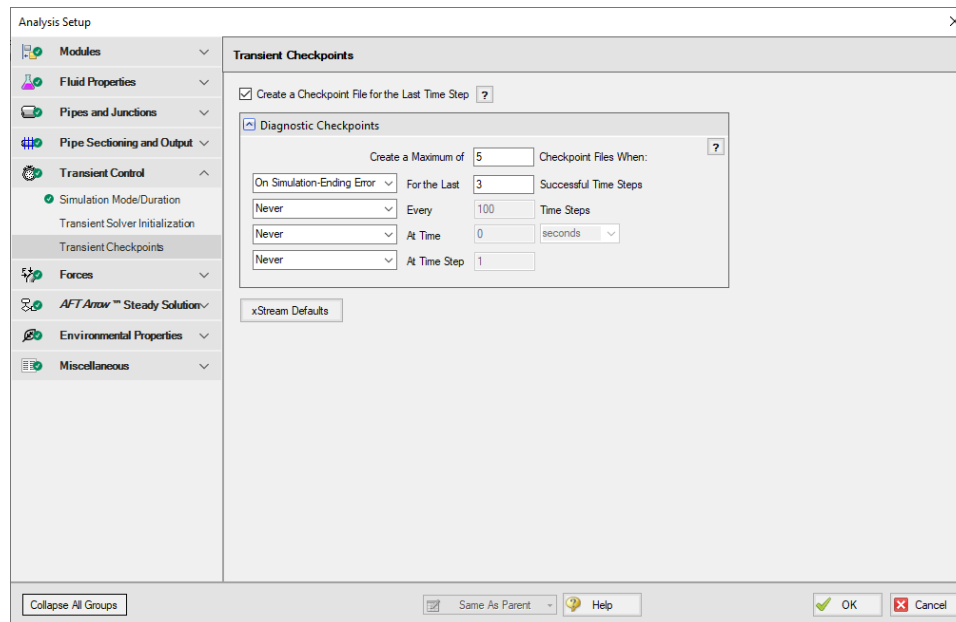


Figure 3: The Transient Checkpoint panel within Analysis Setup

Retain Partial Transient Output

If the simulation is stopped - by the user or an error - show the transient results up to that point. [Transient Output](#) is now saved and viewable regardless of what caused the simulation to stop. This means that you can access and analyze the results up to the point where the simulation was interrupted, just like regular results. This eliminates a lot of guesswork involved in interpreting system behavior when it deviates from expectations, particularly when troubleshooting simulation errors. For instance, if a simulation encounters difficulties in converging on a compressor solution due to factors like reverse flow or unexpectedly low temperatures, the availability of partial output enables you to quickly identify these situations. You can then take appropriate corrective measures or provide our Support Team with detailed information to expedite problem resolution.

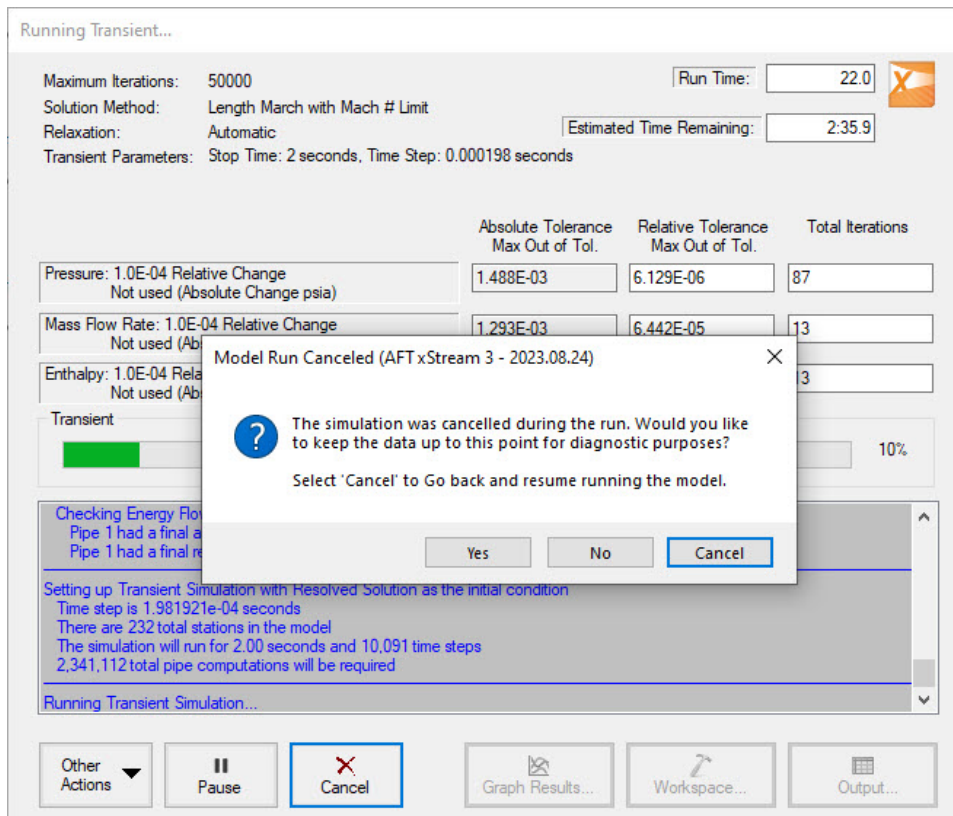


Figure 4: Transient Output can be retained on errors or cancel

Improved Valve Properties Window

The new [Valve Properties window](#) greatly enhances clarity and usability when it comes to modeling valves. The window now clearly distinguishes between different data sources such as User Specified, Characteristic, or Handbook, making it easier to understand the available options. The form has been streamlined to focus on the most essential data, providing a clear understanding of what is required to define a valve accurately. Valve Characteristics, represented by an Open Percent vs. Flow Coefficient table, have been brought front and center. While using a Characteristic is optional, it is highly recommended as it ensures that the valve behavior aligns with real-world scenarios, particularly for valve transients. The process of selecting a pre-defined Characteristic Source has been simplified, which helps obtain more accurate values in case manufacturer data is unavailable.

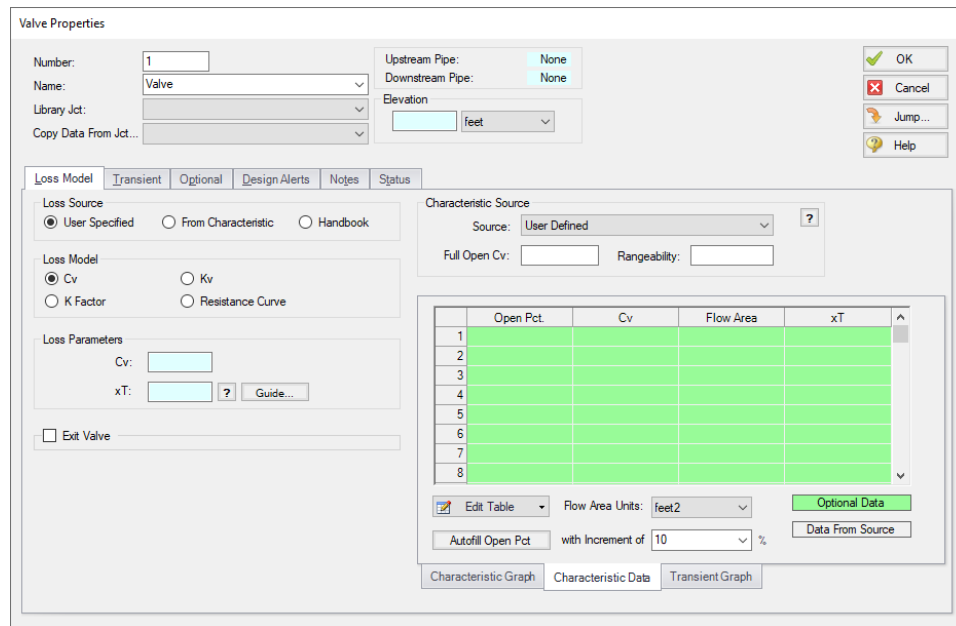


Figure 5: Redesigned Valve Properties window

Streamlined Design Alerts

The [Design Alert Manager](#) has been reorganized to improve the process of creating and applying user-defined alerts. Now, you can effortlessly create or edit a Design Alert with just a few clicks. As you modify settings, you will immediately see the criteria you are applying, ensuring a transparent experience. We have also reduced the amount of text on the form, while ensuring grammatical clarity. These improvements aim to make it easier to understand the limits being placed on the design.

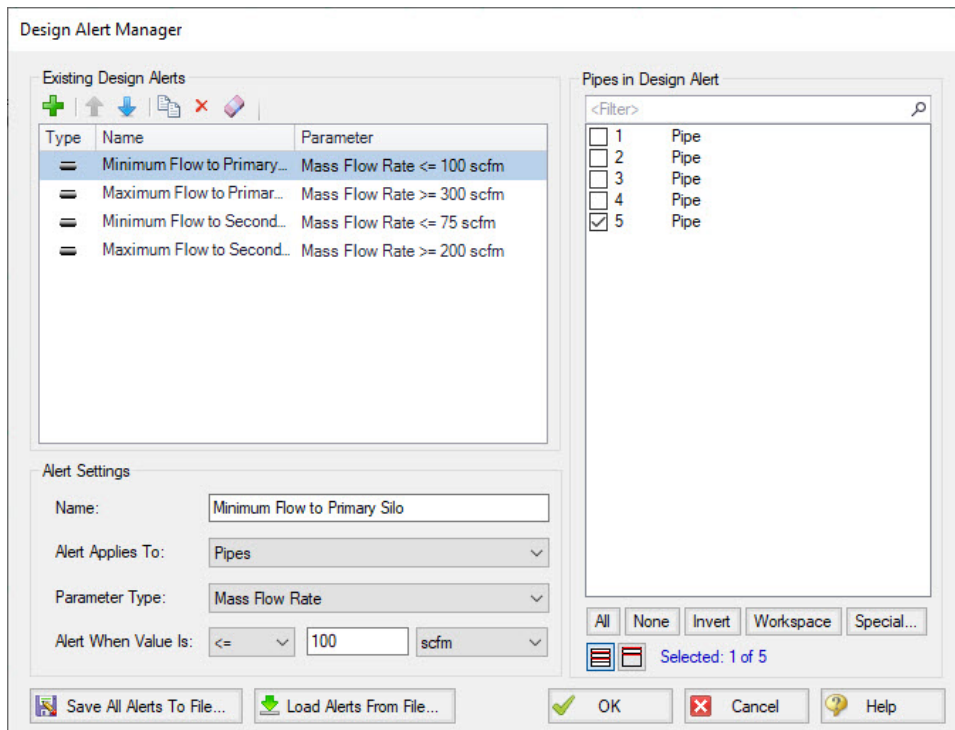


Figure 6: Steamlined Design Alerts Manager

Other Notable Features

- **Custom Junction Icons** - Use your own images for [junction icons](#). Utilize any image as a junction icon, simply by right-clicking on the Toolbox icon for the junction, selecting Add Image, and browsing to the image file. This functionality enables you to use standard P&ID icons or custom images to visually represent your equipment. By using recognizable icons, you can improve the clarity and comprehension of model results, especially for those who may not be familiar with the default AFT junction icons.

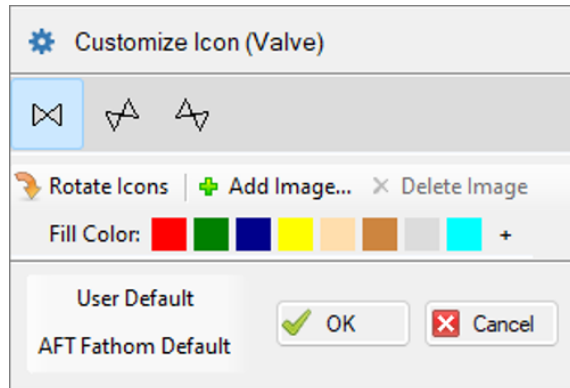


Figure 7: The Customize Icon window allows a custom image to be applied to any junction

- **Isometric Freeform Drawing Mode** - Draw on the [isometric grid](#) without forcing any particular pipe routing.

The Isometric drawing tools have been enhanced by adding a new Isometric Freeform drawing mode. This new drawing mode allows you to snap junction and pipe anchors to the Isometric grid, while having the flexibility to draw pipes that do not necessarily align with the isometric gridlines. This allows you to easily draw complex systems where not all piping follows a Cartesian direction.

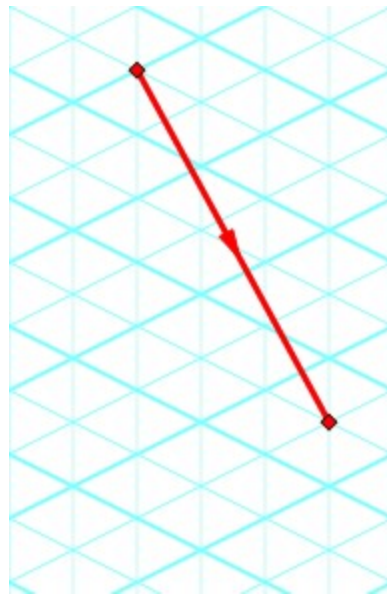


Figure 8: Isometric Freeform drawing mode does not force pipes to follow gridlines

- **New Annotations** - There are new shapes and a line tool, along with a more advanced [Annotation](#) editing panel.

New Annotation shapes and a line tool have been introduced to allow more flexible markup such as revision numbers or change requests. Editing of Annotations has been revamped with the introduction of a live-edit Quick Access Panel, which provides

immediate visual feedback of the changes you make, and simple global editing of all selected Annotations. The new edit panel also offers expanded functionality, providing you with more customization than was possible in the past.

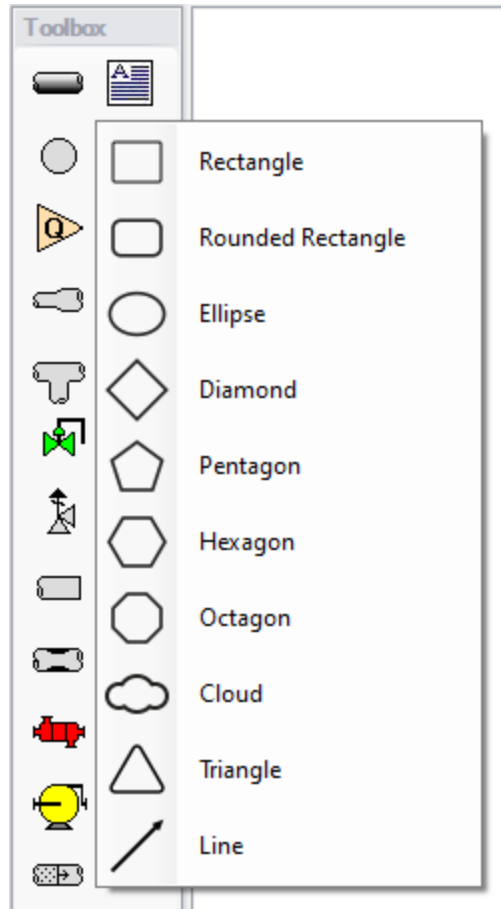


Figure 9: New Annotation options

- **Set Workspace Print Area** - Show printing page borders on the Workspace and customize the printing area.

The new [Printer Page Layout](#) tool offers a convenient and effortless way to adjust the printing area according to your preferences. With a clear and intuitive visual representation, you can easily identify where page breaks will occur. Now, you can confidently print a specific section or the entire model at your desired scale, eliminating the need for trial and error.

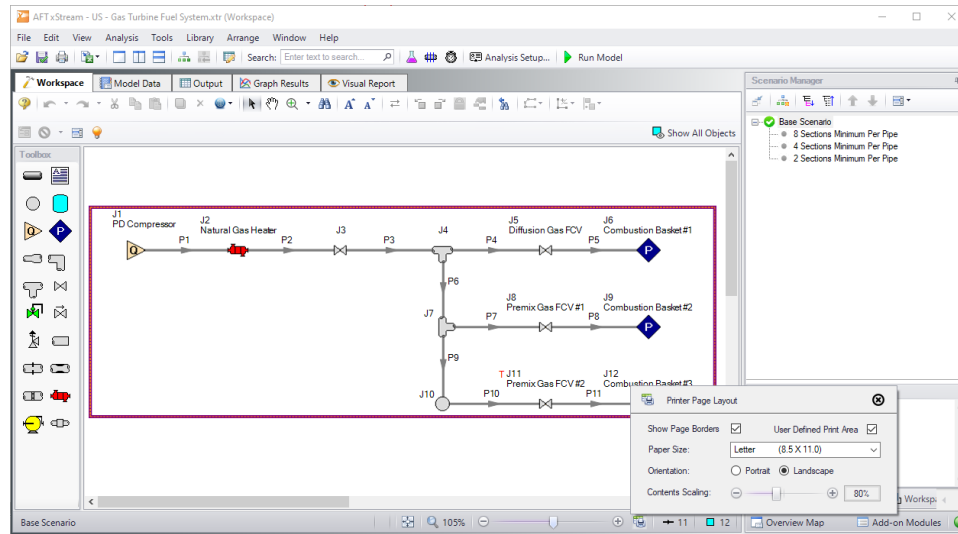


Figure 10: The Printer Page Layout feature allows the user to set the Workspace print area

- Customize Parameters Available to Graph from the Workspace - New [User Options](#)** allow custom selection of any graph parameter on the right-click Workspace context menu.

Did you know you can easily graph results from the Workspace by right-clicking on a pipe or junction? This functionality has existed for a long time, but it was previously limited to a predefined set of parameters. Now, you can specify what graph types you would like to see in these menus to simplify your analysis process.

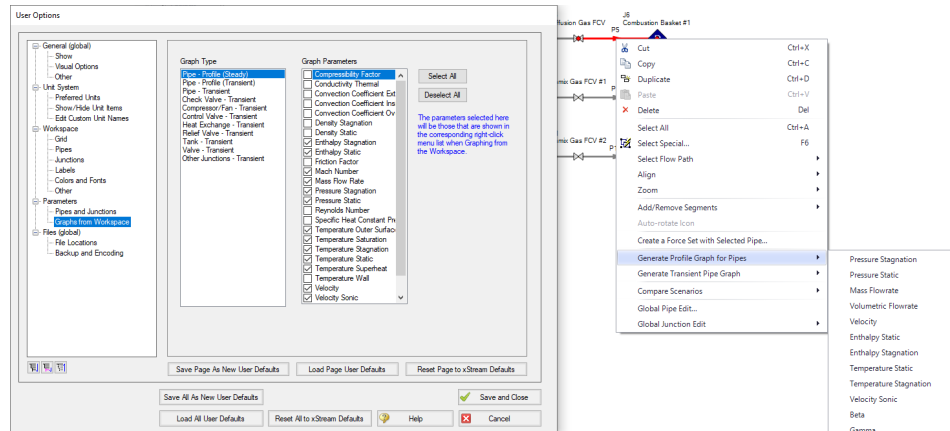


Figure 11: The right-click context menu for pipes on the Workspace can have the graphing parameter options modified

- PFA Module Improvements** - The [PFA module](#) has an improved interface, and can now find a Steady State Pulsation solution, allowing better analysis of pulsating systems.

The PFA module interface for xStream has been overhauled to enhance clarity and streamline the process, while introducing new tools essential for pulsation analysis. Chief among these is the addition of a Steady State Pulsation mode, which allows the transient simulation to run until standing waves are formed by the device causing pulsation. This new mode eliminates artificial transients that could affect the accuracy of conclusions drawn from a pulsation study. By focusing on the formation of standing waves, the modal analysis becomes more precise and reliable.

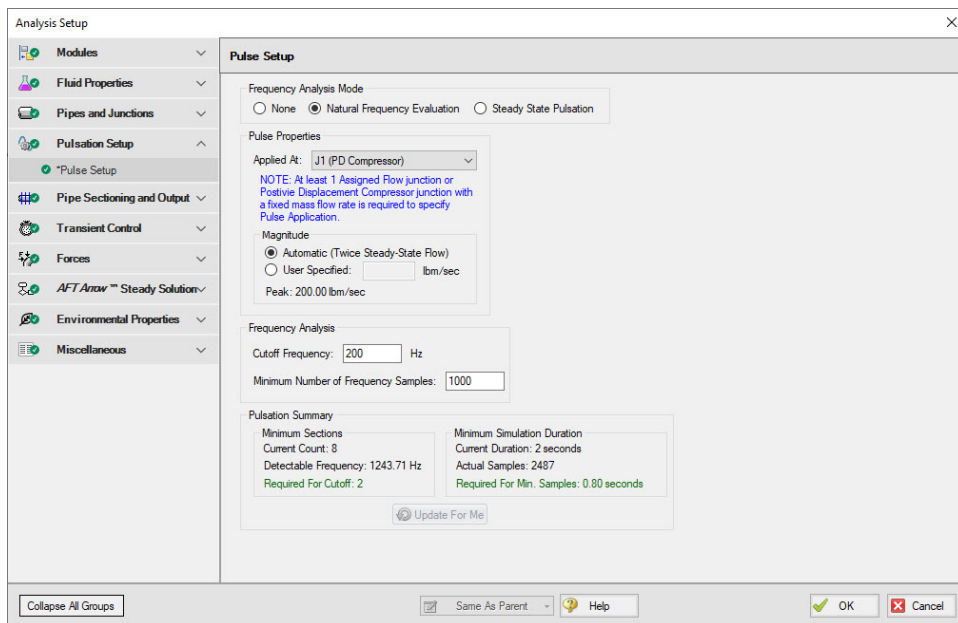


Figure 12: The Pulse Setup panel inside Analysis Setup

Miscellaneous Improvements

- **Modeling Additions and Improvements**
 - **Zero-Length Connectors** - Zero-Length Connectors now behave like standard pipes for most features, including selection and graphing tools.
 - **Fluid Libraries** - New category for user-defined fluids, making it easier to find custom entries.
 - **Simpler Force Definitions** - Difference forces spanning more than 2 pipes no longer require the definition of a group.
 - **Control Valve Fully Open Cannot Regain Control** - New special condition that keeps the control valve in a locked-open position during transients.
 - **API 618 Limitations in PFA Module** - API 618 outlines frequency-dependent pulsation limits - you can now check for violations graphically, and see locations where the limit is exceeded.
- **Importing of PCF, EPANET, GIS, and CEASAR II Models**

- **Bend r/D** - Import PCF bends with appropriate r/D values.
- **Speed Increase** - Importing is up to 10x faster in some cases.
- **Defaults and Display** - Better defaults and display options for imported models.
- **Pipe Materials** - Improved ability to link AFT pipe materials to imported pipes.
- **CEASAR II Node Names** - Display imported CAESAR II node names as junction names.
- **Behind the Scenes Improvements**
 - **Graphing Optimization** - Significant improvements to backend graphing code which provides additional stability now and more capabilities in the future.
 - **eLicense Access Improvements** - Added a "heartbeat" functionality to eLicenses which improves security and the release of license seats not in use.
 - **Output** - Significant improvements to backend output code which provides additional stability now and more capabilities in the future.
 - **Timed Recovery File** - Automatically generate a backup of your *unsaved* model that can be used to recover unsaved work in the event of unexpected application closure. This is separate from the *.bak files which are only generated *after* saving a model.
- **New and Improved Output Parameters and Messages**
 - **Compressor RPM Output Parameter** Report compressor speed in RPM, in addition to percent.
 - **Dimensionless Heat Transfer Parameters** - Grashof, Nusselt, Prandtl, and Rayleigh Numbers have been added as Output parameters.
 - **Valve xT Output Parameter** - Report and graph the xT value used for valves.
 - **Estimated xT Output Parameter** - Graph saturation temperature and temperature above saturation (superheat) along flow paths.
 - **Laminar Caution** - Alerts the user if flow at junctions is in a laminar range.
 - **Profile Graphs for Temperature Saturation and Temperature Superheat** - Graph saturation temperature and temperature above saturation (superheat) along flow paths.
 - **POR/AOR in Compressor Summary** - Show the values for Preferred and Allowed Operating Ranges in the Compressor Summary.
 - **Events Messages in New Format** - Events are now shown in a cleaner, collapsible format similar to Warnings.

Interface Improvements for xStream 2

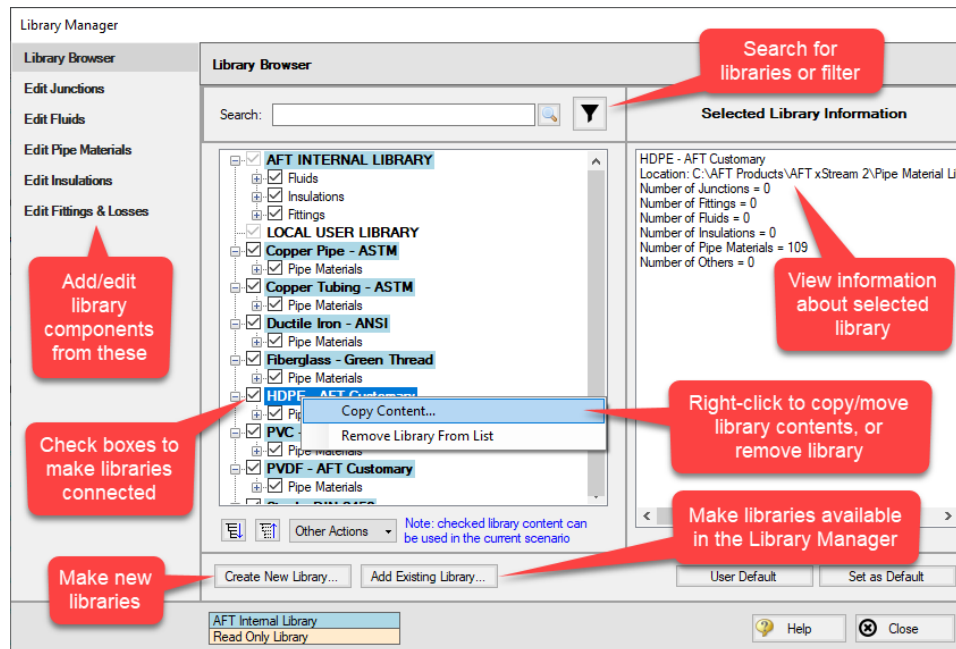
There were many improvements made to the interface in xStream 2. These changes primarily involve consolidating inputs with the new Analysis Setup window, and updating the functionality of databases, which have been renamed as libraries.

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

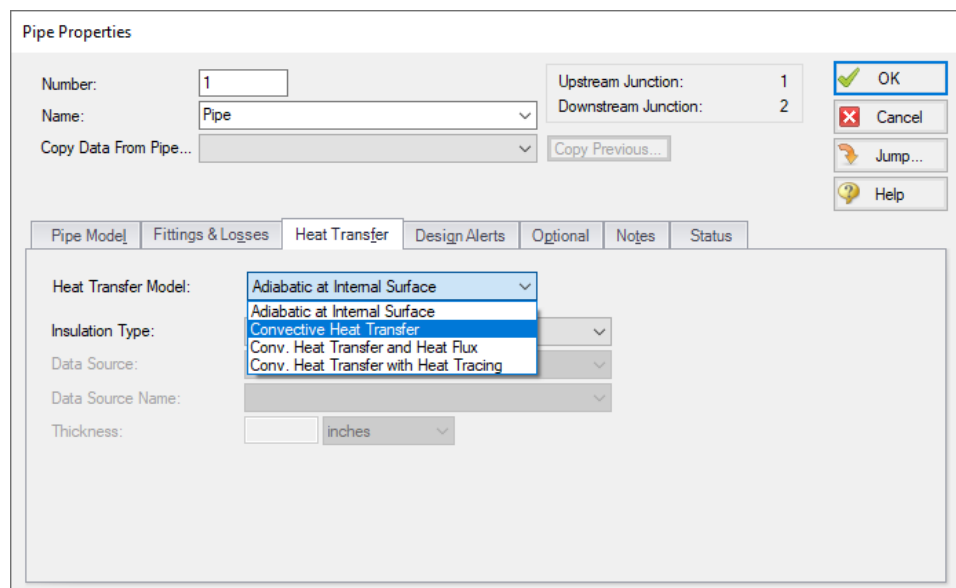
- Databases have been renamed as libraries throughout the interface to better reflect their functionality
- Connecting, creating, and transferring content between libraries can all be done from the [Library Browser](#), rather than switching between tabs in the Database Manager to do this
- Viewing the 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



New in Version 2

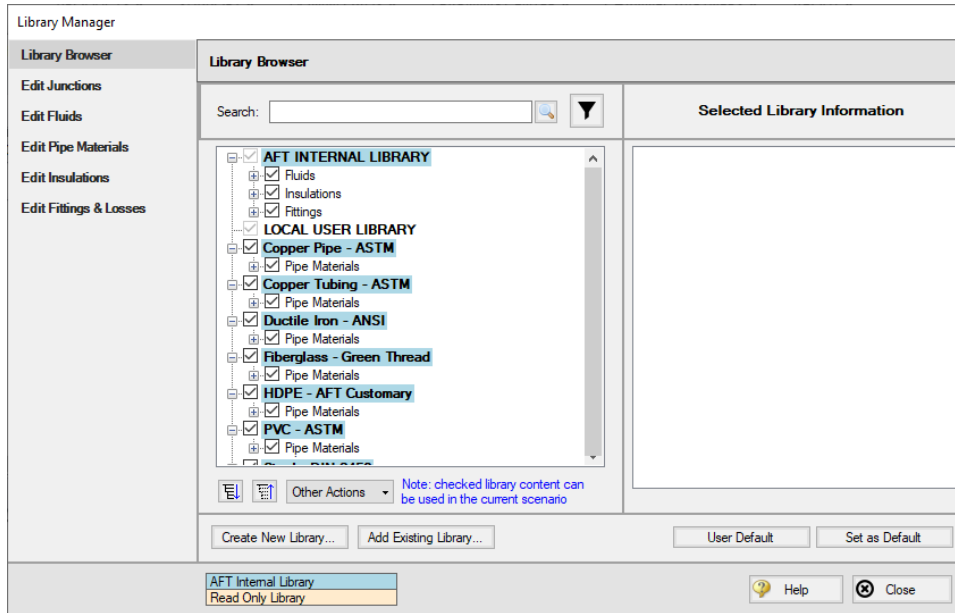
Convective Heat Transfer

[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

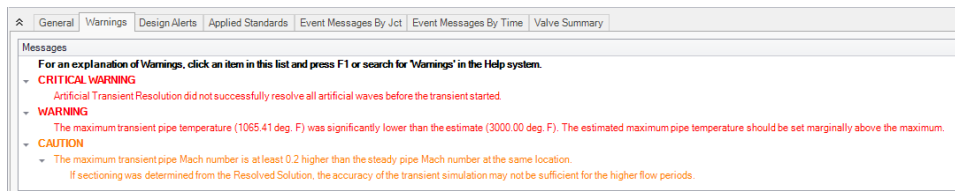
The database feature interface was completely overhauled to improve workability. The new system is called the [Library Manager](#) and significantly improves the ease and workability for users to browse and connect input information, now called libraries, into their model. For a list of differences between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

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**

Online Quick Start Video Tutorial

The Quick Start video tutorial series provides a short introduction to the interface of AFT xStream using a simple example model.

- [Online Quick Start Video Tutorial Series](#)

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

A list of examples using the add-on modules of AFT xStream to solve a variety of problems:

- Module Walk Through Examples

Related Topics

Overview of AFT xStream.....	17
Primary Window Overview.....	309
Overview of Fluid Transients.....	37
Philosophy of Computer Modeling.....	71

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 [Analysis Setup](#) on the [Fluid panel](#) before running the model.
 - **Use AFT Standard Air** - Selects AFT Standard Air to be used as the working fluid of the model.
 - **Select A Recently Used Fluid** - Allows the selection of a recently used fluid from the dropdown menu.
 - **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 for the display of the Menus, [Workspace Layers](#), [Output](#), [Graph Results](#), and the Startup Window.
 - **2D or Isometric Workspace** - Set up the grid to enable either [2D](#) or [iso-metric](#) drawing modes.
 - **Default Pipe Material** - Set the [pipe material](#) that is initially selected for new pipes.
- **Resources**
 - **Current Version** - Open the AFT website to view the latest AFT xStream release, or sign up for notifications of new releases via email.
 - **Quick Start Guides** - Provides links to various [Learning Resources](#) that are designed to help get a new user up and running with AFT xStream as quickly as possible. The resources include the Quick Start Guide PDF, Examples, and video tutorials.

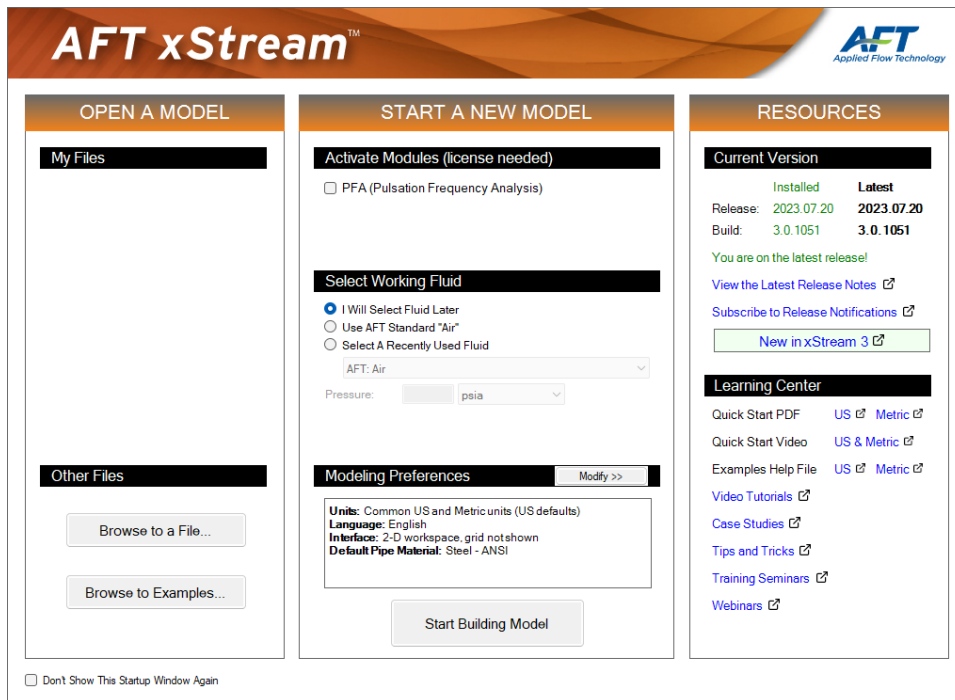


Figure 1: Startup Window in AFT xStream

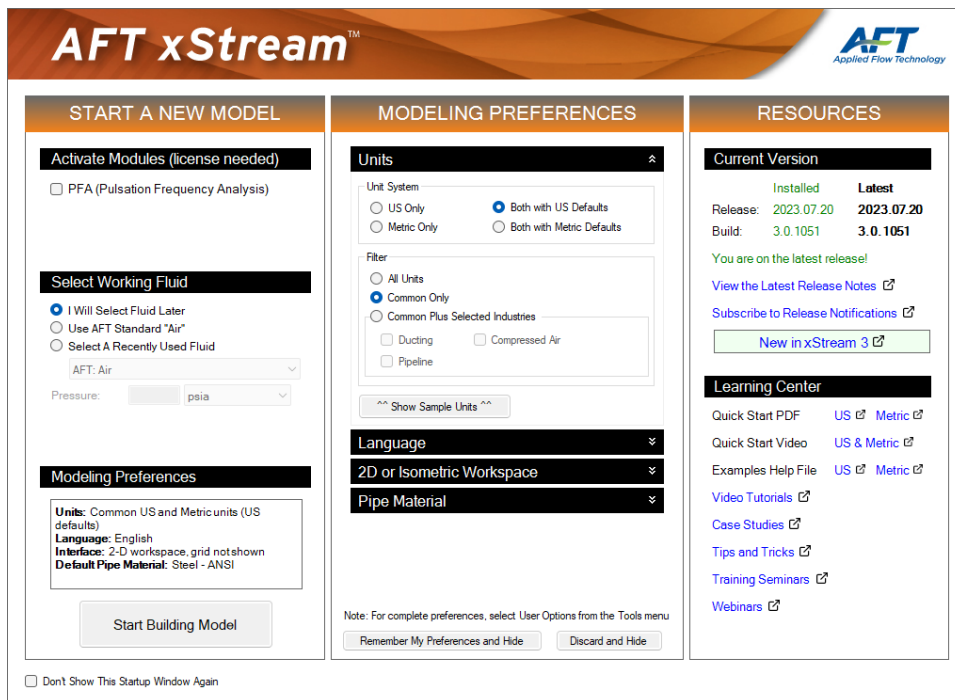


Figure 2: Startup Window showing Modeling Preferences

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*. This is required to use the Method of Characteristics (MOC) solution approach. A brief overview of the impact the Method of Characteristics has on the model is covered below. A detailed discussion on the mathematics behind the method is covered on the [Method of Characteristics](#) help topic.

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 Δt it takes to travel from one point to another is determined by the linear distance Δx between those points and the [wave velocity](#) ($a + V$):

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

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.

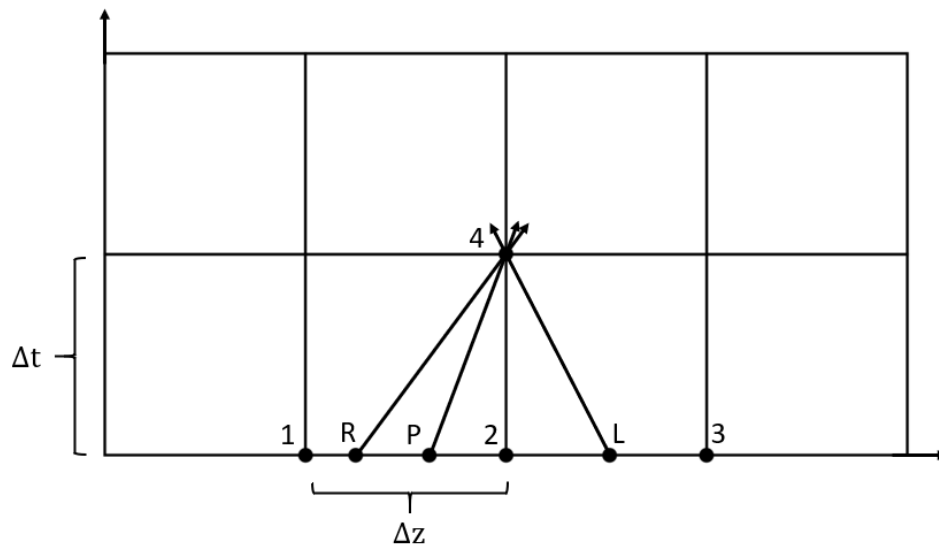


Figure 1: Characteristic grid in AFT xStream, where point 4 sees effects from points R, P, and L after some time dt , which has equal magnitude of dx/a . Points R, P, and L must be interpolated from known points 1, 2, and 3.

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, Δx , 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..... 490

Minimizing Run Time

Models built in AFT xStream can experience long run times, sometimes requiring hours or days 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 [Join Pipes/Junctions](#).
- 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..... 59
 Transient Sensitivity Analysis Tutorial..... 62
 Reducing Run Time..... 572

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

Defining Forces	43
Force Set Types	46
Exporting Force Data	46
Force Theory	47

Defining Forces

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

Note: As AFT xStream is only a pneumatic solver, only pneumatic 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. Pneumatic force files can be generated for [import into software](#) designed for force analysis.

Defining Force Sets from Analysis Setup

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

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 Pipe Forces from Turbine Shutdown Example.

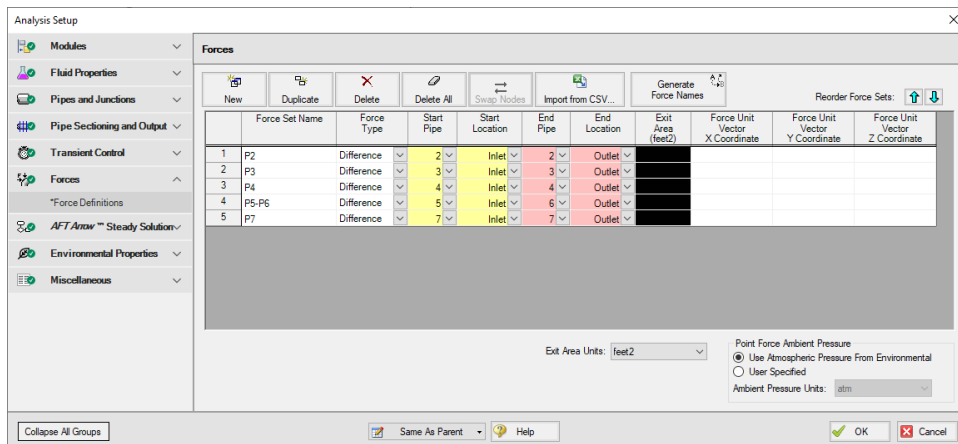


Figure 1: Force set data is defined on the Force Definitions panel in the Analysis Setup window

- **New** - Add a new force set to the table.
- **Duplicate** - Copy an existing force set.
- **Delete/Delete All** - Remove the selected force set / remove all 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 panel](#) value, but can be changed here.

Each individual force set has a number of required parameters:

- **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 node (for Difference force types). 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.
- **Exit Area** - For Exit force types, 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 Force Sets 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.

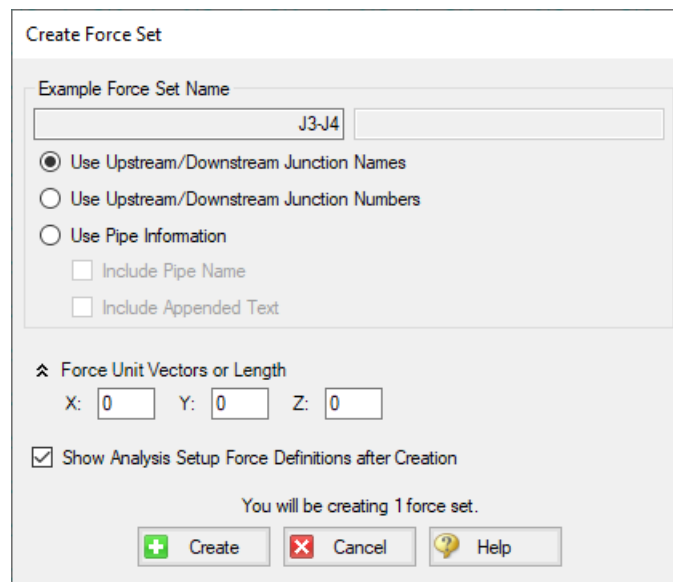


Figure 2: Create Force Set window in the Workspace

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

Related Topics

Force Definitions Panel 117

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 the ambient environment multiplied by the pipe area, which can be used for Dead End junctions.
- **Difference** - Difference forces are defined as the unbalanced force between two nodes, or [computation stations](#). These nodes should be located at 90 degree direction changes. There may be one or several junctions between the elbows so long as the flow does not branch between the nodes and all components are axially aligned. 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 first node must be a 90 degree elbow, and the second node must be a location where flow exits the system. A user defined exit area will be used at the exit node for calculating the force across the section. This is useful for modeling forces on exit valves, exit orifices, and relief valves.

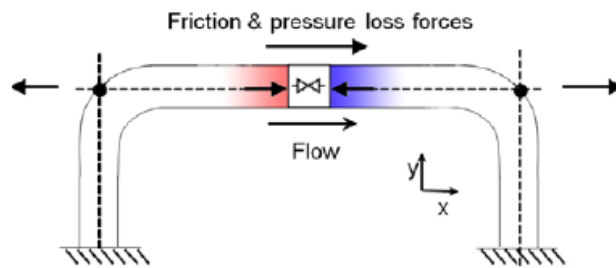


Figure 1: Schematic of the Difference force type

Related Topics

Force Definitions Panel 117

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.

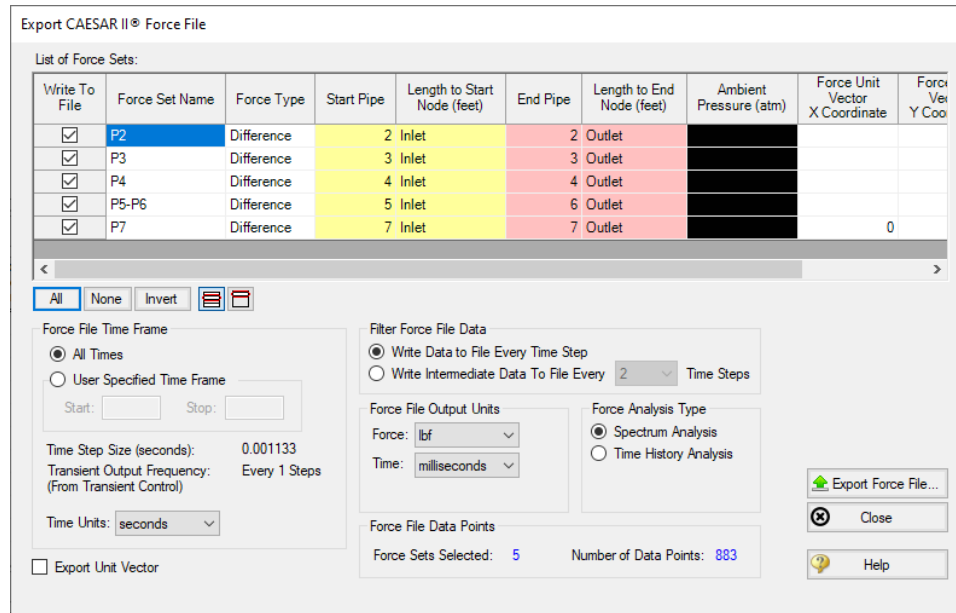


Figure 1: Exporting force results to a Caesar II Force File

Related Topics

Force Definitions Panel..... 117

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 white paper 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.

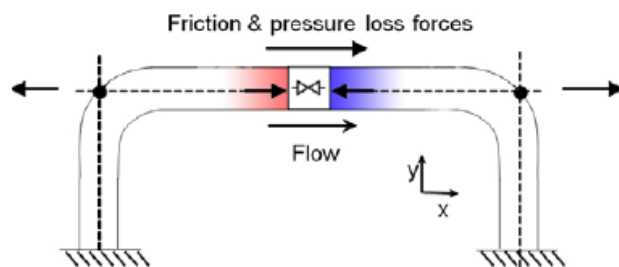


Figure 1: Simplified force diagram for fluid flow in a pipe

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.....	117
------------------------------	-----

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

Animating Graph Results	51
Animation Layers	55
Transient Result Accuracy	59
Transient Sensitivity Analysis Tutorial	62

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.

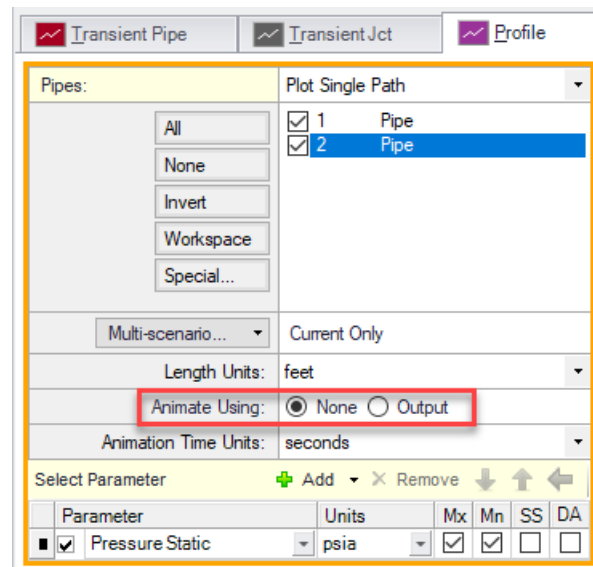


Figure 1: Options for animation under Profile type graphs in the Graph Results window

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.

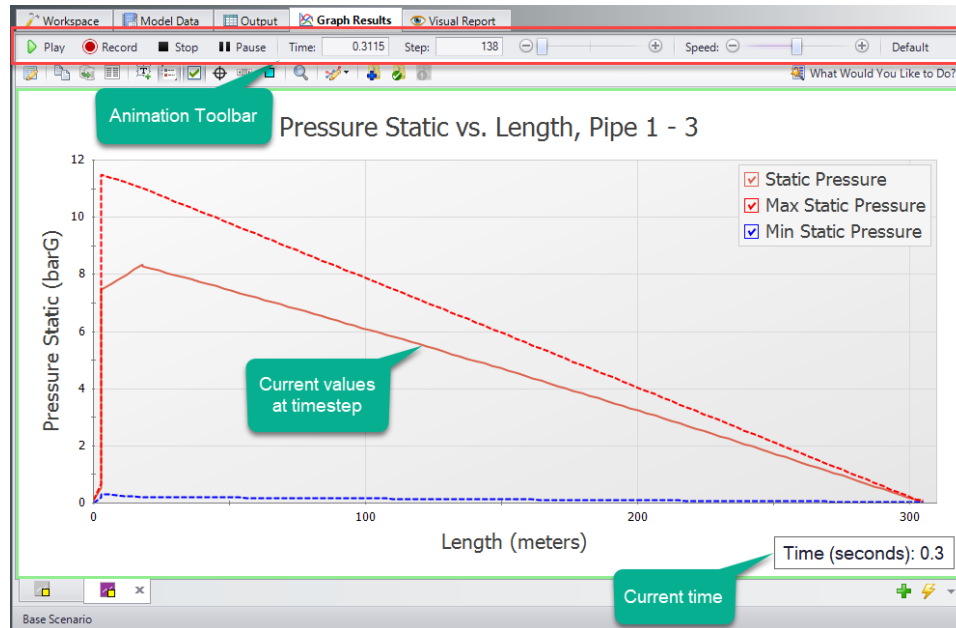


Figure 2: Components added to the Graph Results window when animating a Profile graph

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.

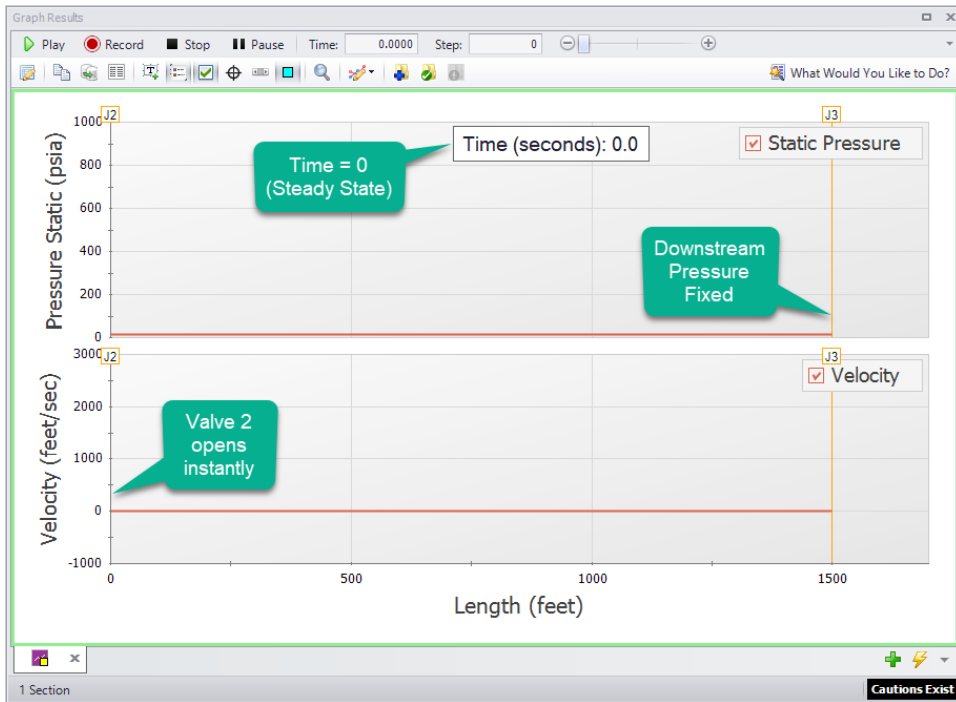


Figure 3: Animation showing steady-state (time step zero) results

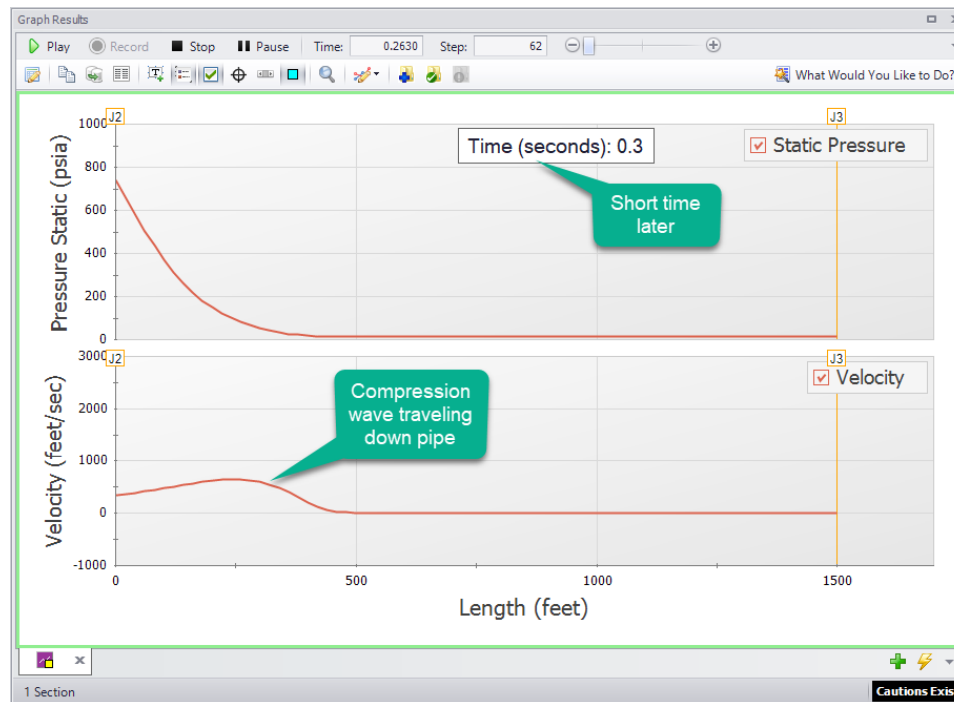


Figure 4: The movement of the wave within the pipe can be directly visualized

Animation Layers

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](#), animated color maps provide 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 Animation Layers 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 Workspace, a [Color Map/Animations layer](#) must first be created using [Workspace Layers](#). The Layer Type will then need to be set to "Animation" in the Color Map/Animation panel in the Layer Settings window.

When Animation is selected in Layer Settings, the following Animation Settings become available.

- **Data Type**
 - **Absolute Values** - This option directly uses the transient values for the pipes being animated. Impulse 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. Impulse 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.
- **Smooth colors between pipe sections** - This choice sets the color gradient on the Workspace 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 on the Workspace window to be adjusted for the animated model. The junctions can also be set to 0% to automatically hide all junctions.
- **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 animation settings to the defaults.
- **Parameter** - The parameter and units for the animation can be selected
- **Auto Scale** - If this box is checked, Impulse 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.

Note: With Auto Scale active, minimums and maximums will be adjusted as appropriate for the active scenario and are not fixed to the maximum and minimum values that were applicable at the time of creating the layer.

- **Use Inverted Scale** - This option flips the minimum and maximum values on the color scale so that lower values are warmer colors and higher values are cooler colors.
- **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.
- **Reset Label Locations** - This resets the location of pipe and junction labels to their default positions.

Animation settings are specific to each individual animation layer. While only one animation layer can be viewed at a time, many can be present in a model and can be toggled with visibility settings or via [Layer Presets](#). Animation layers are not scenario-specific, but can be linked by Layer Presets to scenarios to automatically display when loading those scenarios in the same fashion as other layer types.

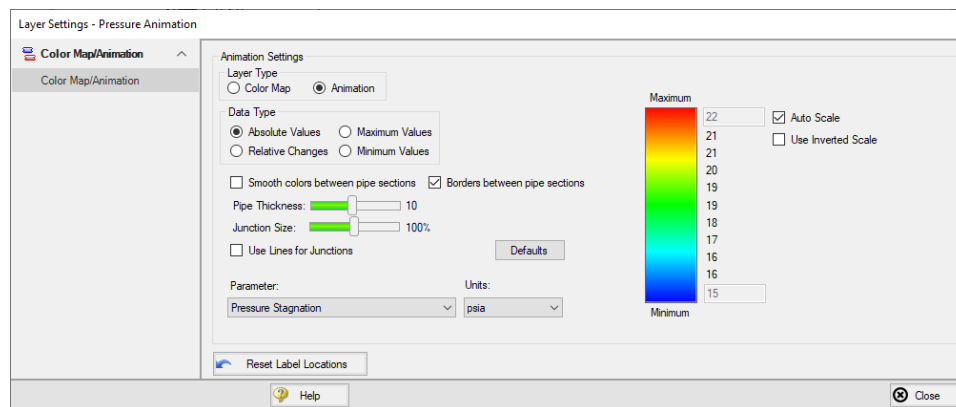


Figure 1: The Layer Settings menu for creating an animated color map

Viewing Animation

Once the Layer Settings have been configured and the animation layer is toggled to visible, several animation-specific components will appear, including the animation toolbar as shown below.

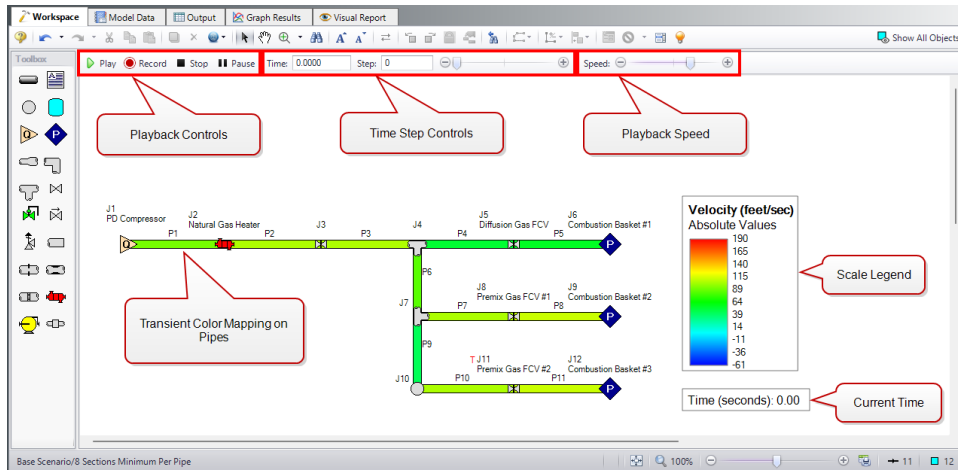


Figure 2: Components added to the Workspace window when an animation layer is visible

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.

Note: Input can be typed directly into the Time or Step text boxes to quickly navigate to a desired time step or to the one nearest to a desired point in time.

- **Time/Step slider** - Seek for a certain point in the animation.
- **Speed slider** - Change the speed of the playback.

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. Visualizing pressure changes or velocities over time allows an engineer to see how pressure waves propagate through a system, including where they originate, their timing and magnitude, and how they reflect within the system. Animation Layers communicate this information more clearly than graphs and numbers alone, providing additional insights.

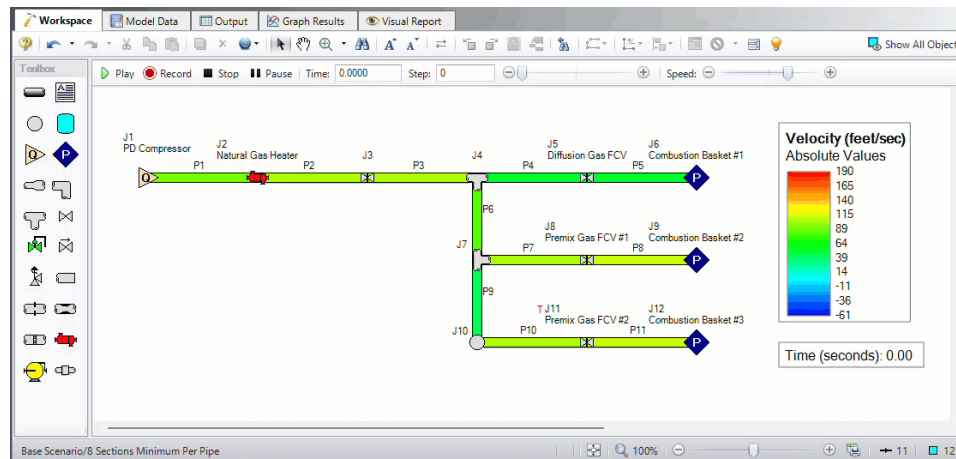


Figure 3: Transient velocities in the pipes can be directly visualized

Related Topics

Animating Graph Results.....	51
Transient Sensitivity Analysis Tutorial.....	62
Transient Result Accuracy.....	59
Workspace Layers.....	324

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 [AFT 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.....	483
Method of Characteristics.....	490
Solution Method Panel.....	120
Transient Solver Initialization Panel.....	114
Animating Graph Results.....	51
Animation Layers.....	55
Transient Sensitivity Analysis Tutorial.....	62
Minimizing Run Time.....	41
Reducing Run Time.....	572

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.

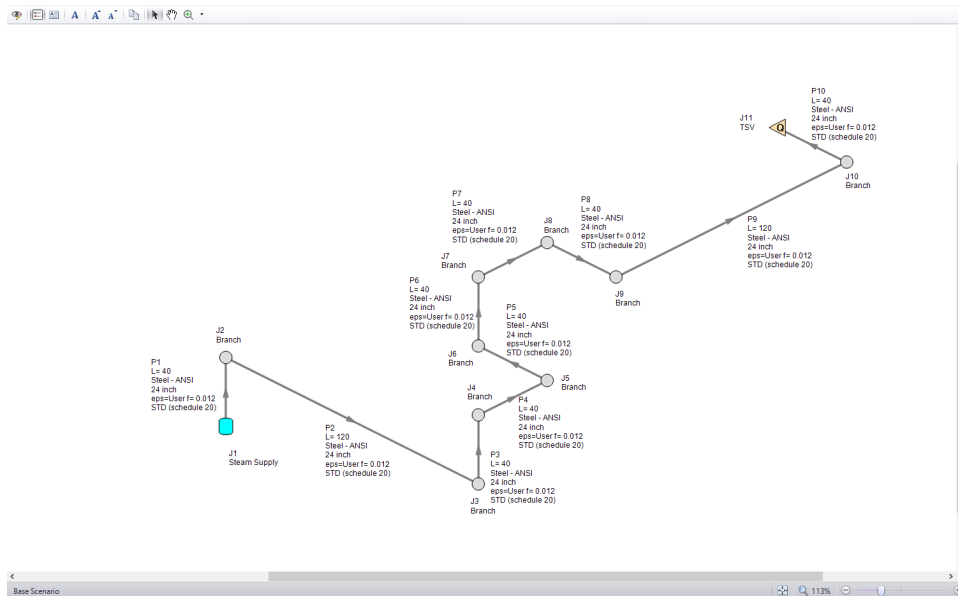


Figure 1: Layout of Tutorial Example

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.

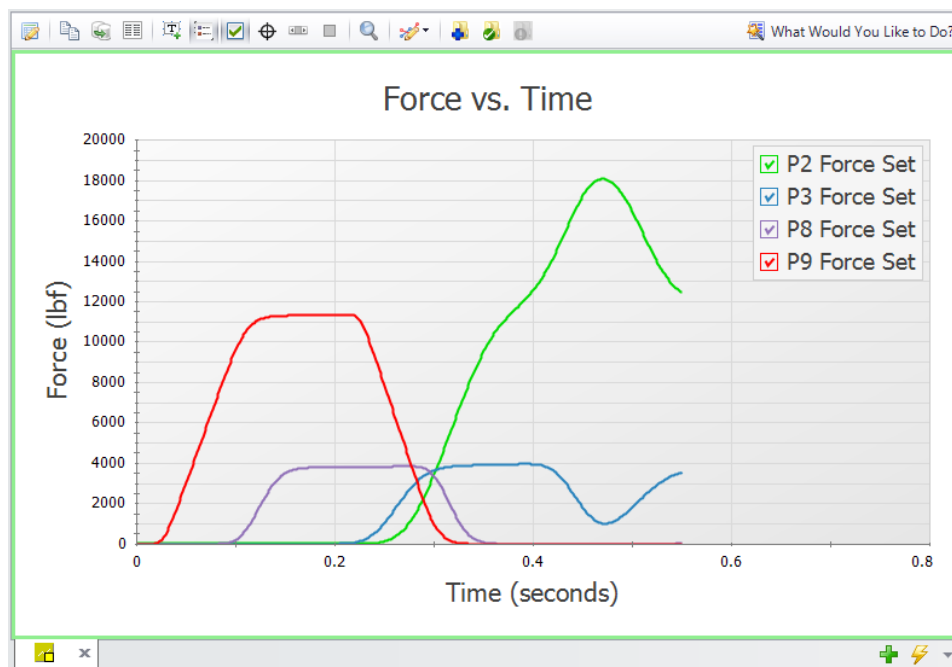


Figure 2: Force Results for AFT Standard, 10 Minimum Sections, Standard Maximum Mach Number in Pipe

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.

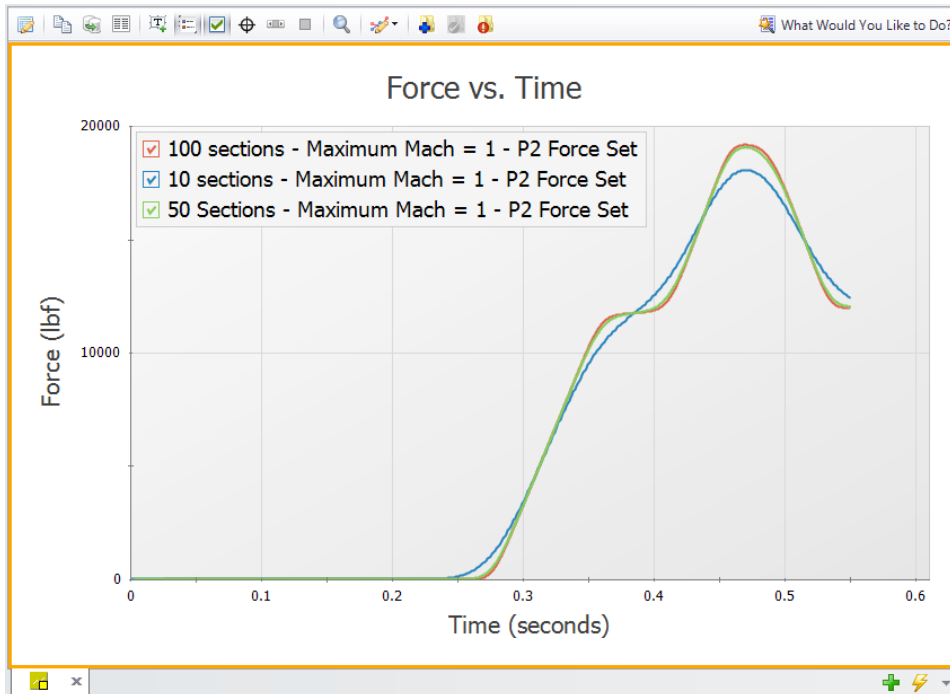


Figure 3: Effect of Sectioning on Forces

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.

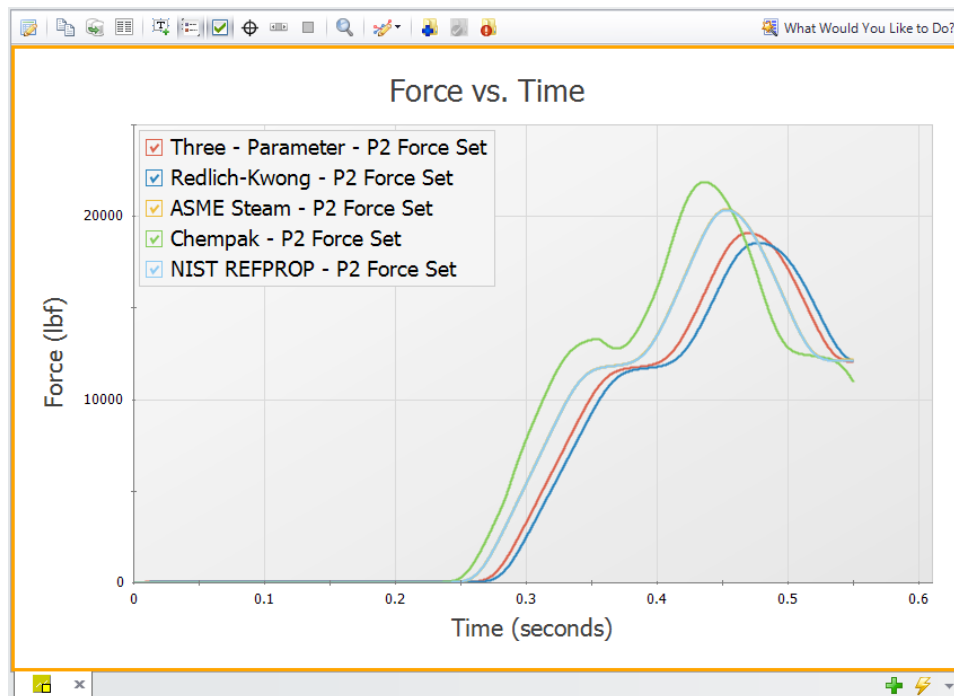


Figure 4: Effect of library on Forces (minimum 50 sections used in each pipe, ASME Steam has indistinguishable differences from NIST REFPROP and thus cannot be seen in this figure)

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.

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.

Interpreting Transient 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.

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	0.06006	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

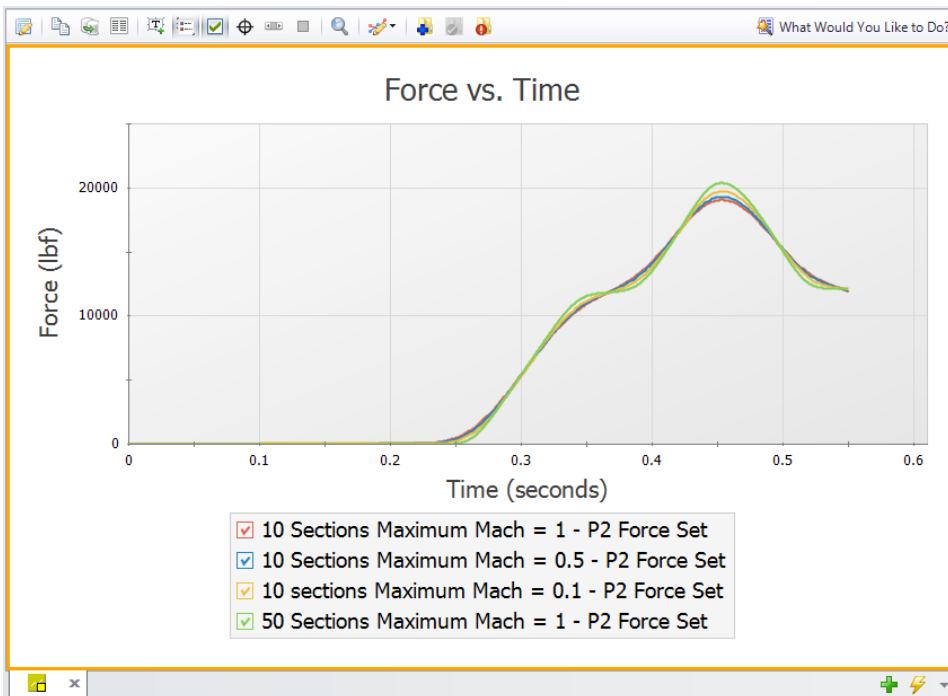


Figure 6: Effect of Maximum Pipe Mach Number During Transient on Forces

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.

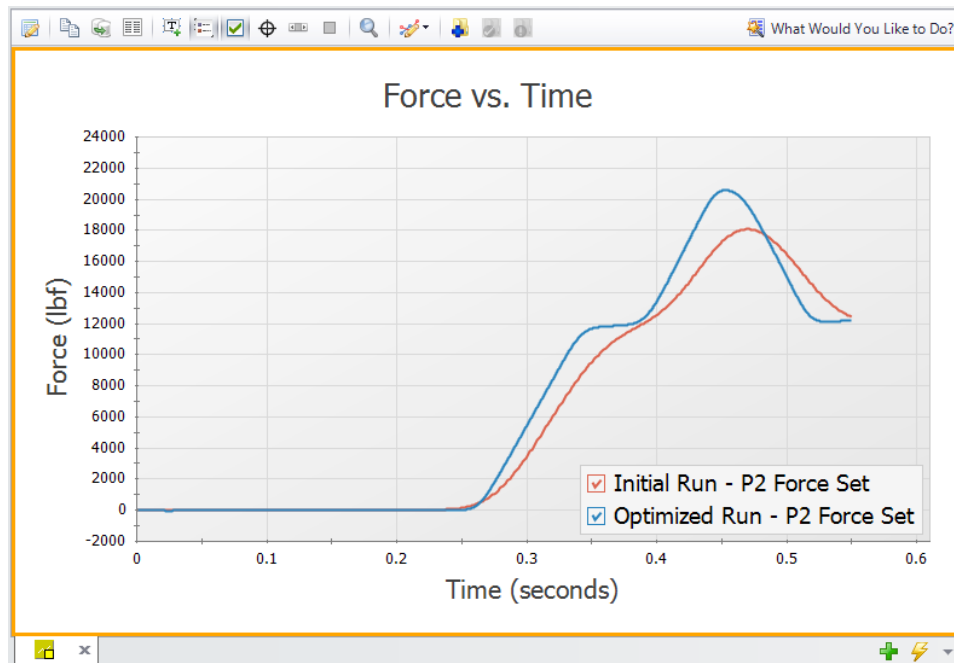


Figure 7: Initial Run vs Modified Run

Related Topics

Transient Forces.....	43
Force Definitions Panel.....	117

Fluid Panel.....	87
Equation of State Models.....	94
Pipe Sectioning - Introduction to Method of Characteristics.....	38
Pipe Sectioning and Output Group.....	111
Force Theory.....	47
Animating Graph Results.....	51
Animation Layers.....	55
Transient Result Accuracy.....	59
Method of Characteristics.....	490
Minimizing Run Time.....	41
Reducing Run Time.....	572

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

Pipes and Junctions	69
Features for Modeling Irrecoverable Losses	70
Reference Positive Flow Direction	70
Philosophy of Computer Modeling	71
Role of Pressure Junctions and How They Work	72
Open vs. Closed Systems	75
Curve Fitting	77
Stagnation vs. Static Pressure Boundaries	79

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 Basis Area	221
Pressure Drop in Pipes - Detailed Discussion	212

Reference 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 Fathom assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the [Arrange menu](#) or selecting the reverse direction button on the [Toolbar](#). With exception to some junctions, the reference positive flow direction is used for reference purposes only and does not need to correspond with the real direction of flow calculated by the Solver (the part of AFT xStream that applies the governing incompressible flow equations to obtain a solution to the pipe system). If the flow direction obtained by the Solver is opposite of the reference positive flow direction, then the output will show the flow rate as a negative number.

Note: The reference positive flow direction is important for compressors because the pipe directions will define which side is suction vs discharge. Furthermore, reverse flow through components may result in a [Caution message](#) which may or may not be valid depending on the intended direction of flow.

Convention for Flow Entering and Exiting

Many of the junction types permit gas 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 and temperature 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/system gains the flow) and flow out is negative (the object/system 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.....	33
Primary Window Overview.....	309
Overview of Fluid Transients.....	37

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.

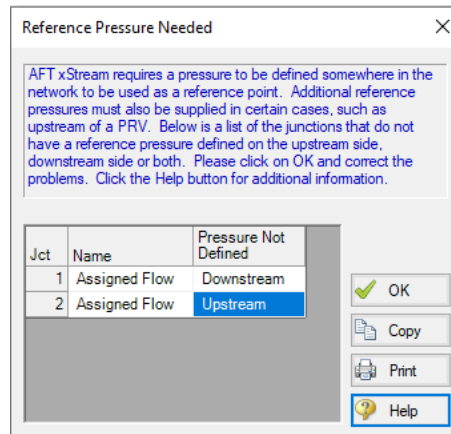


Figure 2: AFT xStream message when you try to run a model without sufficient reference pressures

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.

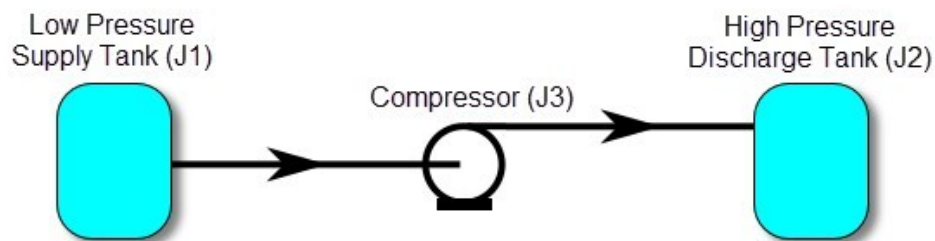


Figure 1: Physical compressor system to transfer air from low pressure tank (J1) to high pressure discharge tank (J2)

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.

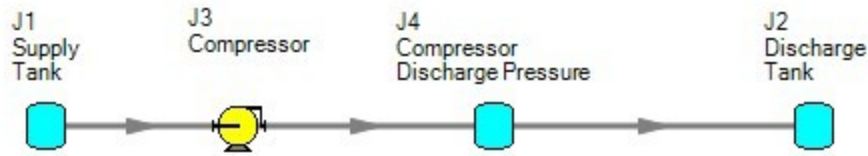


Figure 2: Model to size compressor attempting to represent system in Figure 1, but actually represents the system in Figure 3

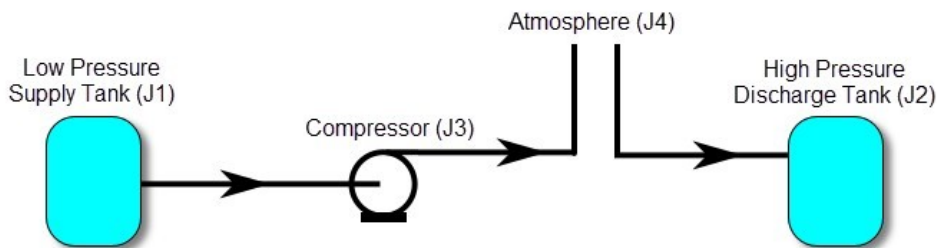


Figure 3: Physical system represented by model in Figure 2

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.

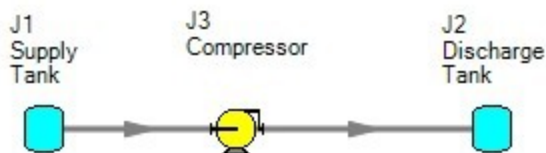


Figure 4: Proper model of physical system shown in Figure 1

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).....	535
Assigned Pressure.....	246
Tank.....	289

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.

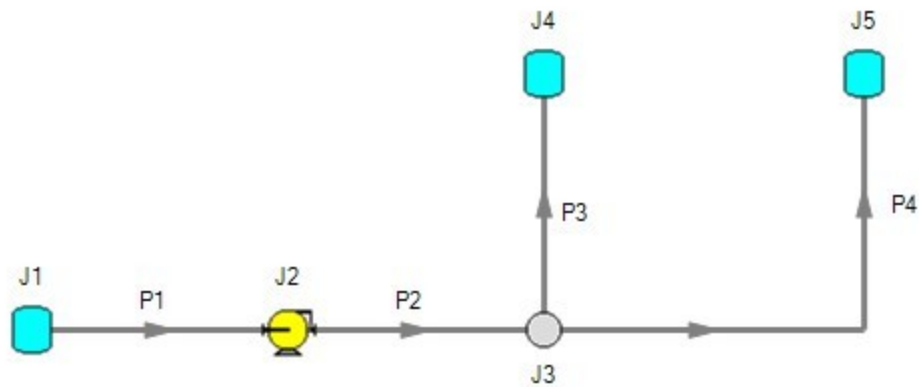


Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5

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.

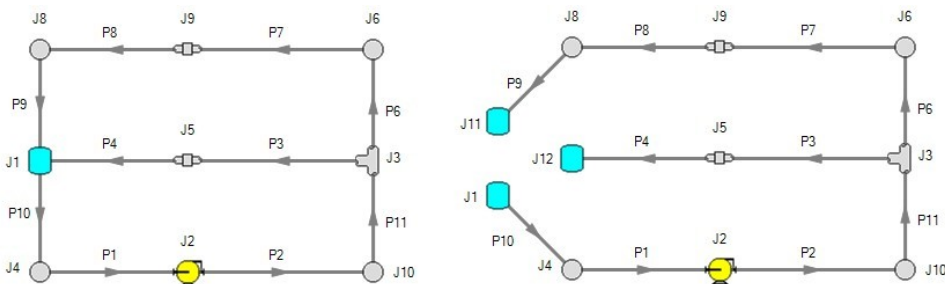


Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two system will appear identical to AFT xStream

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 could include heat exchanger input and heat transfer to or from pipes, as well as any heat of compression effects from compressors.

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](#).

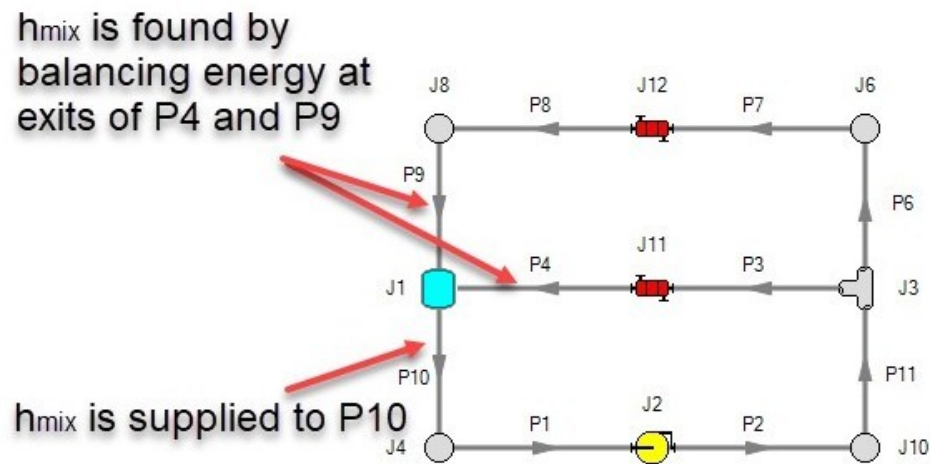


Figure 1: Example of balancing energy in a closed system

Curve Fitting

There are many places within the interface where the user may be asked to define how a property changes in relation to another. Some examples include:

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

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

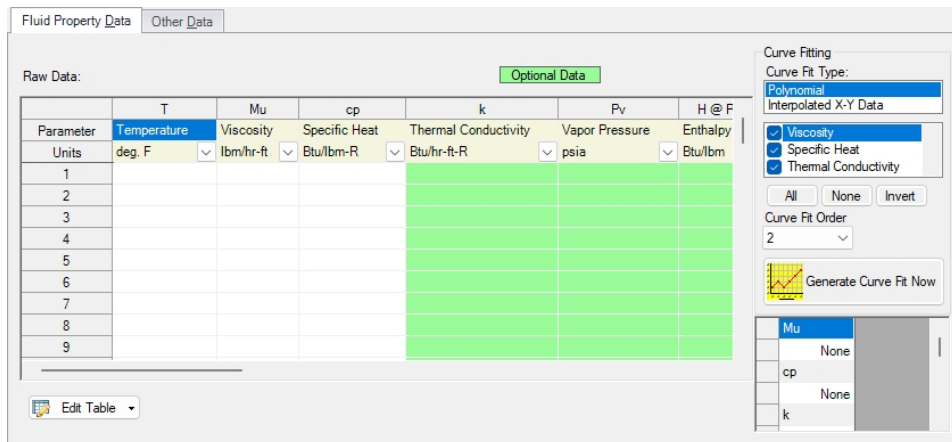


Figure 1: Custom Fluid Definition

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 (a liquid) 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:

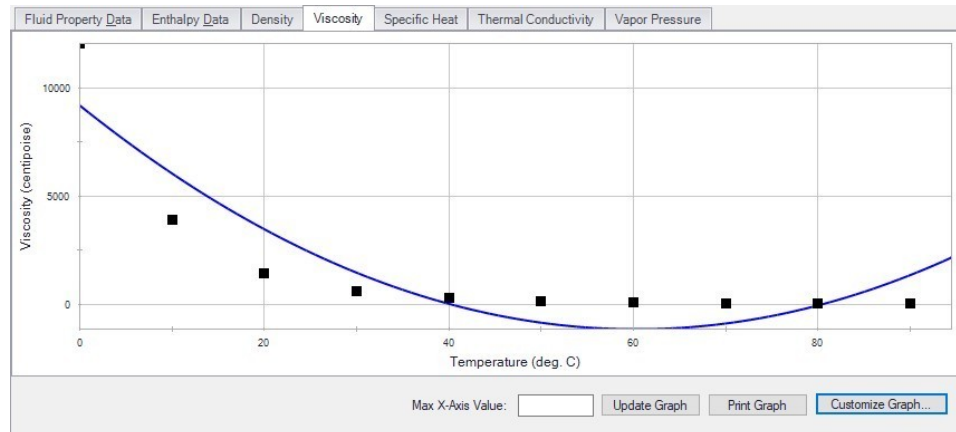


Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data

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.

Related Topics

Compressor Configuration	259
Resistance Curve Loss Model	231
Fluid Library	447
Pipe Material Library	448
General Component	281
Heat Exchanger	282
Reference Density	221

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.

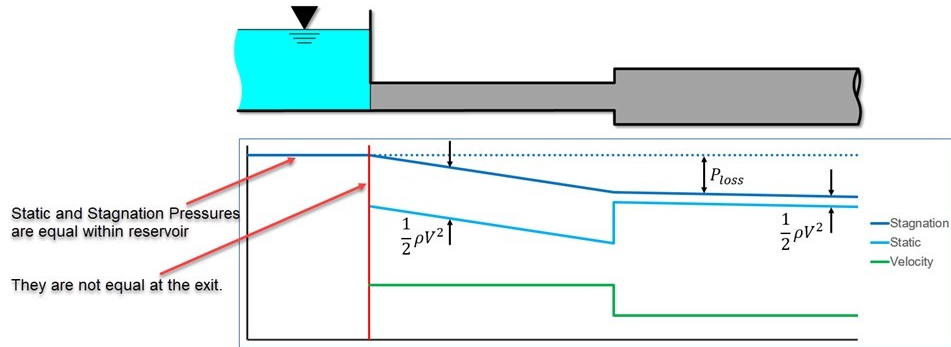


Figure 1: Pressure profile along horizontal pipe shown in Figure 1 (incompressible flow)

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.....	465
Stagnation Properties.....	510

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](#)

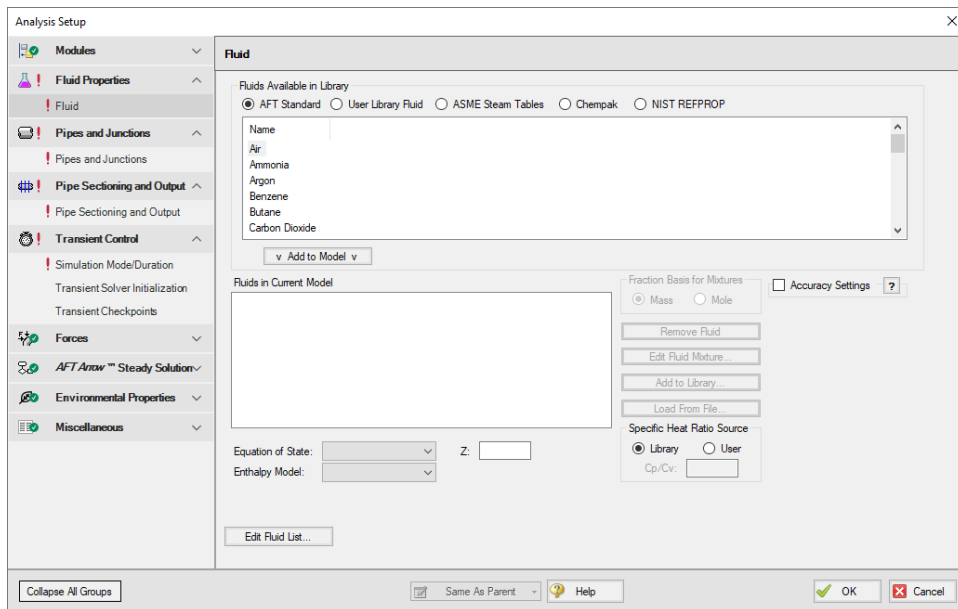


Figure 1: Default state of the Analysis Setup window in a new blank model

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.

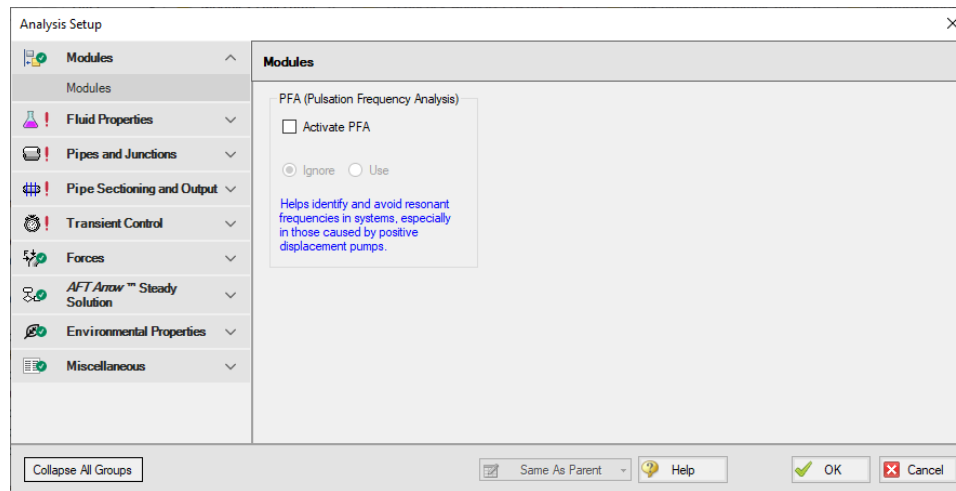


Figure 1: The default state of the Modules panel

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 Dis-abled	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

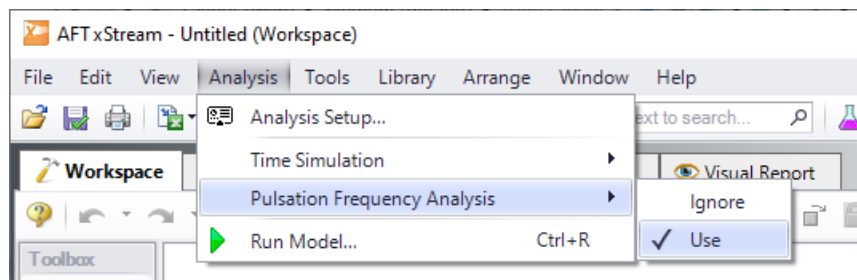


Figure 2: Select Use from the Pulsation Frequency Analysis in the Analysis menu to perform pulsation analysis

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.....516

Related Blogs

[Get More Flexible With Your Fluids](#)

Fluid Panel

The Fluid panel controls the properties of the fluid in the model. A default fluid is not selected - one must be defined to run any simulation unless one was previous defined on the [Startup Window](#).

The first selection that must be made is what Fluids library to use. The available [Fluid Libraries](#) are:

- **AFT Standard** - Common fluids from various industry standard sources have been curve fit to optimize calculation speed. This fluid library offers the best balance between accuracy and computation time. Saturation line data was added for AFT Standard fluids in xStream 2.
- **ASME Steam Tables** - Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation (1997). Saturation line data is included.
- **NIST REFPROP** - The Reference Fluid Thermodynamic and Transport Properties (REFPROP) Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via Equations of State. REFPROP supports user-specified fluid mixtures. For more info see the following sources: [Program Summary](#), [REFPROP FAQ](#), [NIST Chemistry WebBook](#), [NIST Chemical Name Search](#).
- **Chempak Fluid** - Optional add-on for purchase which contains data for several hundred fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.
- **User Library Fluid** - If the desired working fluid is not available via the other sources, custom fluid properties can be entered and saved to the User Library for repeated use in the future. Fluids can be created using the [Fluid Libraries](#) menu.

Select a library by clicking the radio button next to the name of the library. 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.

Note: All fluids in xStream are treated as compressible and superheated as stated in the [Engineering Assumptions](#).

When determining which fluid library 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*
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 use an AFT Standard fluid (even if the real system fluid is a mixture), and the run time recorded. This run time 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 resistance curve loss models, and the presence of sonic choking.

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

AFT Standard Fluid Library

The AFT Standard Library offers several [equation of state models](#) and [enthalpy models](#). These models account for both temperature and pressure dependence on density and enthalpy.

Transport properties (specific heat, dynamic viscosity, and thermal conductivity) and Saturation Pressure are based on polynomial curve fits which are functions of temperature only. The assumption that these properties are only functions of temperatures is generally a good assumption at low pressure. However, at high pressures (especially above the Critical Pressure), this assumption starts to break down.

To add a fluid into your model, select it from the list at the top and choose the Add to Model button. The fluid will appear in the lower list of Fluids in Current Model. You can have only one AFT Standard fluid in your model, as there is no mixing support for Standard fluids.

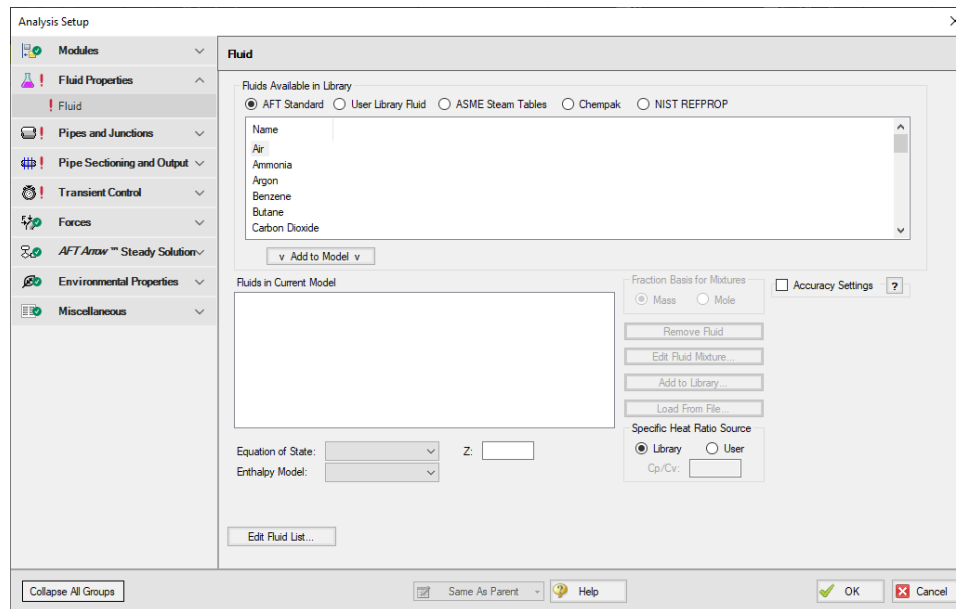


Figure 1: Fluid panel in Analysis Setup with the AFT Standard fluid library selected

ASME Steam Tables

Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam (ISPWS-IF97), see [ASME Press in References](#).

This library is selected by choosing the option ASME Steam Tables at the top of the Fluid panel. After this selection has been made, "ASME '97 Steam" is displayed in the Fluids in Current Model area at the bottom of the Fluid panel.

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](#).

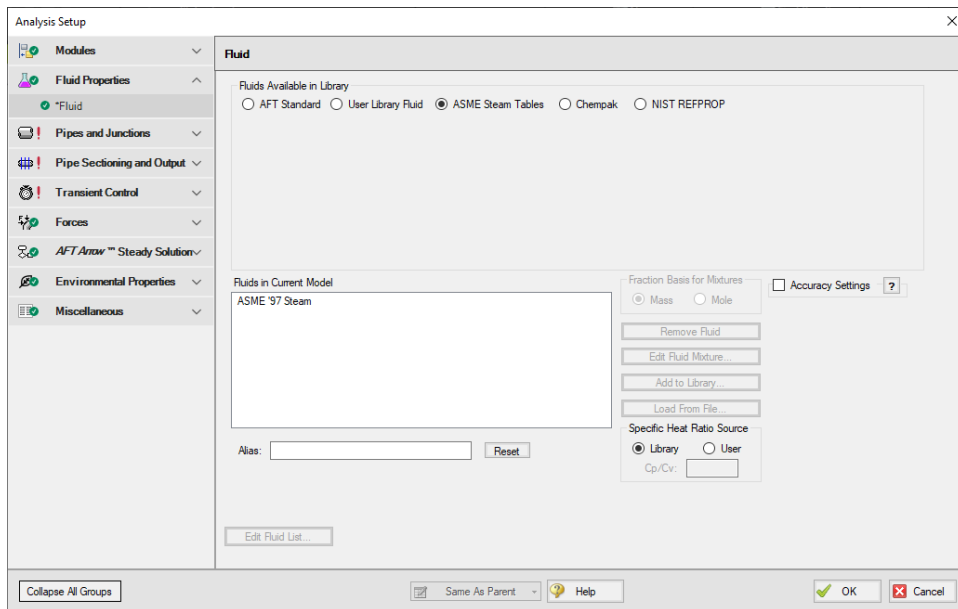


Figure 2: Fluid panel in Analysis Setup with the ASME Steam fluid library selected

NIST REFPROP Library

The Reference Fluid Thermodynamic and Transport Properties (REFPROP) Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via Equations of State (generally the Helmholtz Free Energy Equation of State). The data comes from Version 10 of the library. For more info see the following sources: [Program Summary](#), [REFPROP FAQ](#), [NIST Chemistry WebBook](#), [NIST Chemical Name Search](#).

A NIST REFPROP fluid can be added to the model in the Fluid panel by selecting the NIST REFPROP radio button. Add a fluid from the list by selecting it and clicking Add to Model, as shown in Figure 3.

The NIST REFPROP library allows for the calculation of [gas mixtures](#), as well as dynamic mixing in the model, but note that no chemical reactions are modeled as listed in the [Engineering Assumptions](#). Saturation line data is also available.

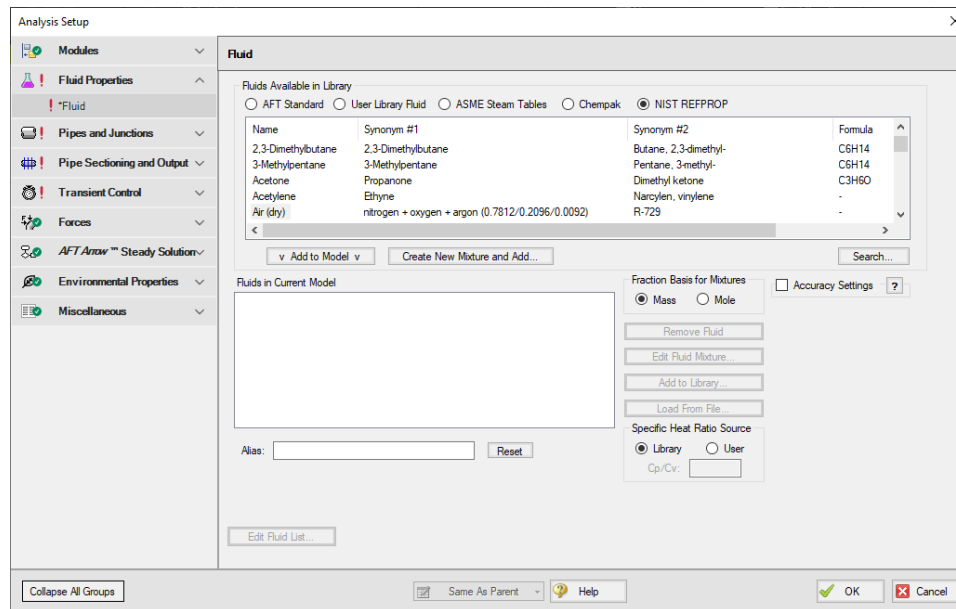


Figure 3: Fluid panel in Analysis Setup with the NIST REFPROP fluid library selected

Chempak Library

Optional add-on for purchase which contains data for several hundred fluids. Chempak is licensed from Madison Technical Software.

Fluids from the Chempak library support [gas mixtures](#) and dynamic mixing, but no chemical reactions are modeled as listed in the [Engineering Assumptions](#).

To add a Chempak fluid, select Chempak Fluid at the top of the Fluid panel. From here, select the appropriate fluid from the fluid list 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.

The equation of state is that of Wu & Stiel which is an extension of Lee Kesler to cover polars as well as non-polars. Properties are derived by interpolation between a Simple Fluid (mainly Argon), a Reference Fluid (mainly Octane) and a Polar Fluid (Water) using Acentric Factor and Stiel Y Factor as interpolating variables. These compound parameters (OM, the Acentric Factor, and YFACT the Stiel Y Factor) are derived from the compound vapor pressure curve. For more information see 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](#).

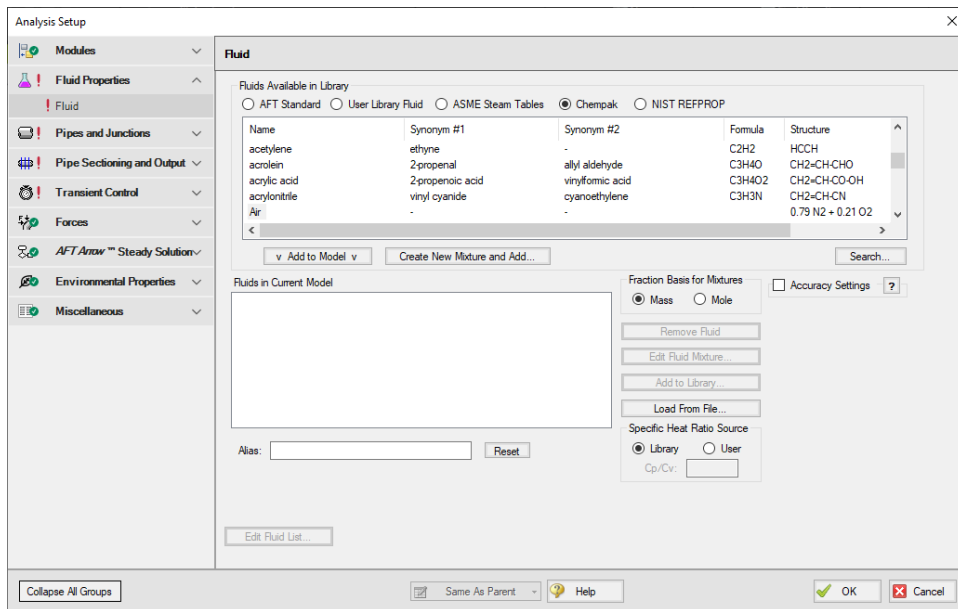


Figure 3: Fluid panel in Analysis Setup with the Chempak fluid library selected

Fluid Accuracy Settings

Several advanced settings are available to fine-tune the accuracy of the fluid property calculations and depend on which fluid library is selected. These settings are discussed in the [Fluid Accuracy Settings](#) topic.

Related Topics

References.....	516
Use an AFT Standard Fluid.....	586

Creating Mixtures

You have the option of creating a mixture of fluids with the NIST REFPROP or Chempak fluid libraries.

To create a mixture, first select NIST REFPROP or Chempak Fluid on the [Fluid panel](#) in Analysis Setup. Once this has been selected, the option to Create New Mixture and Add button becomes available. Click this button, then locate and select the first fluid in the mixture you wish to specify (the Search button can be used to search for a name or synonym). 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 at the bottom of the Fluid panel below Fraction Basis for Mixture. Note that this option is only available after you specify the model fluid as a mixture.

The mixtures you create here are referred to as **static mixtures** because you are specifying the composition before the model is run. Any of these gases or mixtures shown as Fluids in Current Model can be assigned to boundary junctions such as [Tanks](#) or [Assigned Flows](#).

Assigning different fluids or mixtures to different junctions will (usually) result in mixing flow streams where the composition will change. The relative percentage of gases in the mixed composition depends on the flow solution. Hence, these mixture compositions change as the simulation converges on a flow balance. For this reason these mixtures are called **dynamic mixtures**.

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

The 'Create Mixture' dialog box contains the following elements:

- Fluids Available In Library:** A table with three columns: Name, Synonym #1, and Synonym #2. The table lists various chemical species such as Ethane, Ethanol, Ethylbenzene, Ethylene, Ethylene glycol, and Ethylene oxide. Below the table is a 'v Add Fluid to Mixture v' button and a 'Mass Fraction: 0 %' input field.
- Mixture Properties:** A section for defining the mixture's name and composition.
 - New Mixture Name:** A text input field containing 'New Mixture 1'.
 - Composition:** A table with three columns: an index (1-8), 'Components in Mixture', and 'Mass Fraction (%)'. The first two rows are populated with '1 Methane 80' and '2 Ethane 20'. Below the table is a 'Total Fraction: 100 %' input field.
 - Buttons:** A vertical stack of buttons on the right side: 'Set as Remainder', 'Ratio Fractions to Total', 'Remove Component', 'Clear All Components', 'Import Mixture from File...', and 'Export Mixture to File...'.
- Footer:** 'OK', 'Cancel', and 'Help' buttons.

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

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.

ω 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

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

$$b = 0.08664 \frac{RT_c}{P_c}$$

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 ω 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 ω 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 ω is the [acentric factor](#).

Related Topics

Nomenclature.....	463
References.....	516

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 463

Fluid Accuracy Settings

Several advanced settings are available to fine-tune the accuracy of the fluid properties for the xStream simulation. These options are enabled on the [Fluid panel](#) by selecting the Accuracy Settings check box. Once enabled the different Accuracy Setting options will be selectable using the available radio buttons.

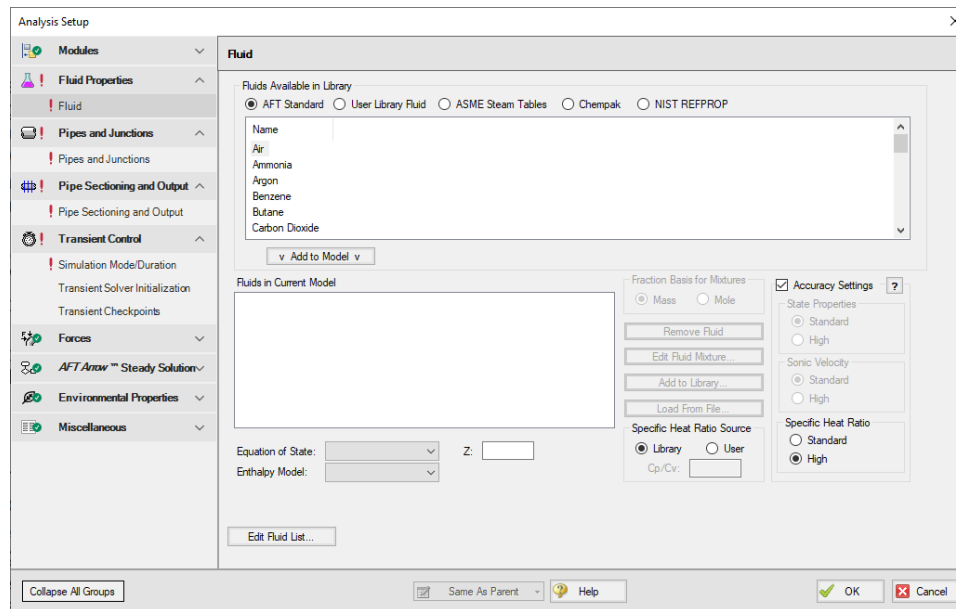


Figure 1: Fluid panel in Analysis Setup with the Fluid Accuracy Settings enabled

State Properties

This option is applicable only to Chempak fluids.

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 (bilinear) or 16 point (bicubic) 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.

Sonic Velocity

This option is applicable only to Chempak fluids.

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.

Specific Heat Ratio

This option is applicable to all of the fluid libraries.

There is a choice of standard or high accuracy for the specific heat ratio. The standard accuracy uses [Equation 80](#) for gamma (γ). 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.

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.

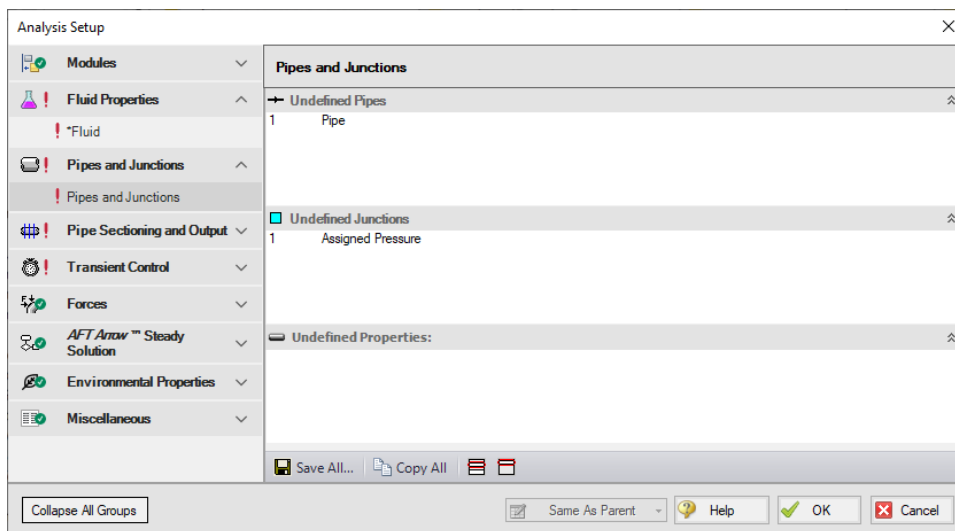


Figure 1: Default state of the Pipes and Junctions panel

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.

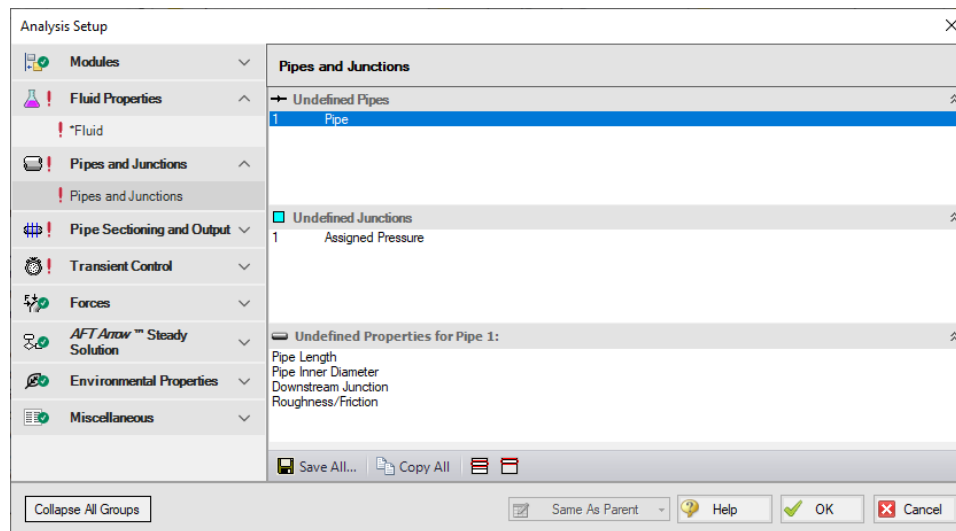


Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed

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](#).

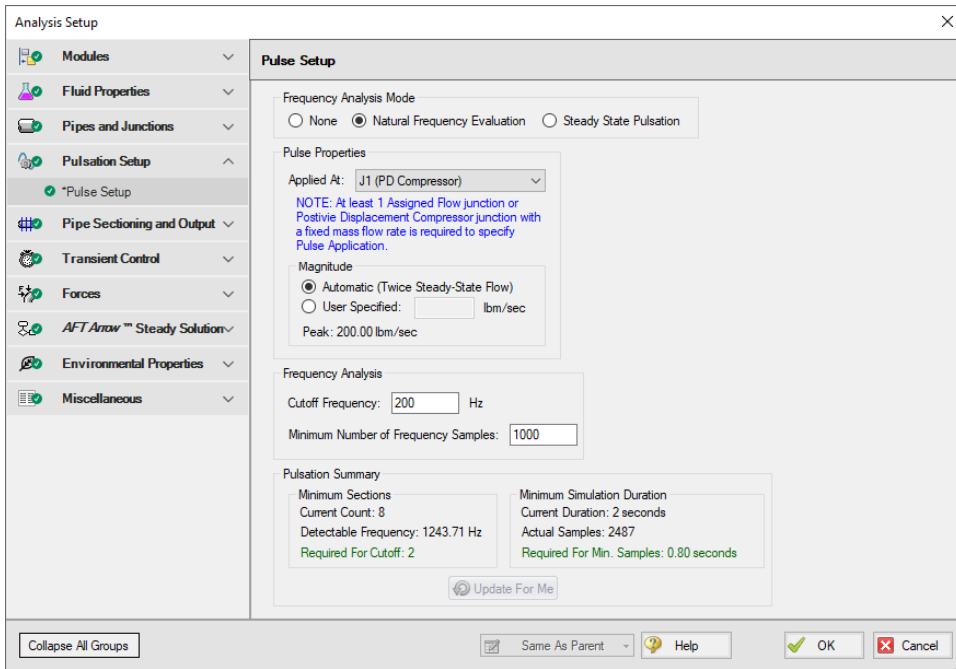


Figure 1: Pulse Setup panel in the Analysis Setup window with Natural Frequency Evaluation selected

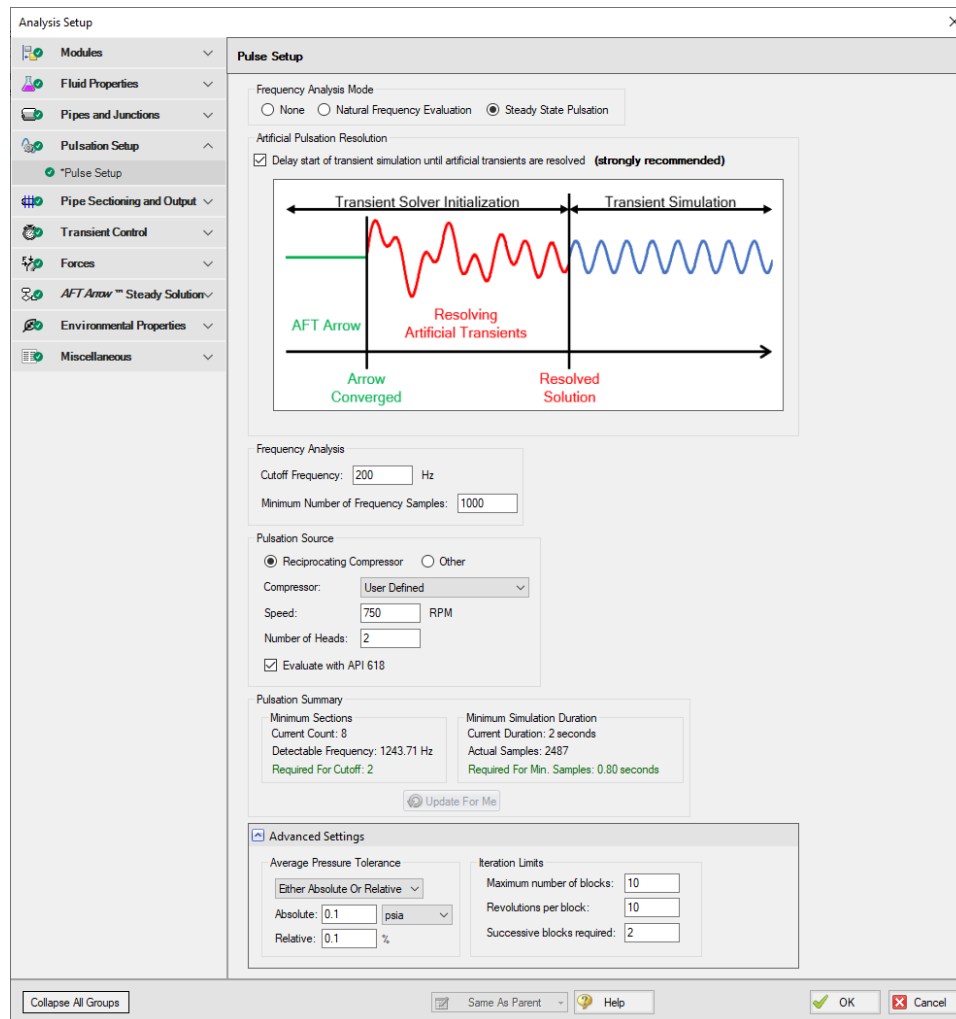


Figure 2: Pulse Setup panel in the Analysis Setup window with Steady State Pulsation selected

Frequency Analysis Mode

At the top of the window the Frequency Analysis Mode can be specified. The options include: Natural Frequency Evaluation, Steady State Pulsation, and None. The Natural Frequency Evaluation will produce a single transient pulse at a specified assigned flow or fixed flow rate positive displacement compressor junction so as to determine the frequencies that would excite the system. The Steady State Pulsation option will determine how a pulsation source, such as a reciprocating compressor, would interface with a system in steady operation.

Natural Frequency Evaluation

When Natural Frequency Evaluation is selected, 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 or [fixed flow rate positive displacement compressor](#).

Steady State Pulsation

The Steady State Pulsation analysis mode allows for the evaluation of pressure amplitude for each frequency at each pipe station in the model when these frequencies are excited by a reciprocating compressor. The reciprocating compressor can be modeled explicitly using the [reciprocating positive displacement compressor](#) junction, or approximated by applying a sinusoidal flow transient to an assigned flow junction. In practice, the natural frequencies of the system may be used along with harmonic multiples to identify compressor RPMs of particular concern.

The Steady State Pulsation analysis mode also offers acoustical evaluation of the reciprocating compressor and system interaction based on the API 618 standard.

Artificial Pulsation Resolution

It is highly recommended that the option to delay transient simulation until artificial transients are resolved be left enabled. This resolution method is based on comparison of average pressures between blocks of compressor rotations, and operating without this method will result in less accurate frequency evaluation. Pressure is averaged over the course of each block, and this value is compared to the previous block until it meets a specified tolerance. This method is distinct from the one used in the [Transient Solver Initialization Panel](#), which is not used and will not appear when using Steady State Pulsation analysis.

The parameters for pulsation resolution can be modified in the Advanced Settings section at the bottom of the Pulse Setup panel.

- **Average Pressure Tolerance** - Similar to solver tolerance settings, this defines the pressure tolerance from block to block that must be achieved for a block to pass tolerance requirements. This can be defined using absolute tolerance, relative tolerance, or any combination thereof.
- **Iteration Limits**
 - **Maximum number of blocks** - The maximum number of blocks that may be evaluated without success before the artificial transients are considered to be unresolved.
 - **Revolutions per block** - This determines the number of time steps per block based on number of crankshaft rotations, the time step size, and compressor RPM.

Note: If the default value of 10 is not sufficient, try increasing this value to 20.

- **Time steps per block (Other Pulsation Source)** - When using Other as the pulsation source, this parameter replaces revolutions per block.

Successive blocks required - This defines the number of consecutive blocks that must meet tolerance requirements before artificial transients are considered resolved.

Pulsation Source

The pulsation source can be set to Reciprocating Compressor or Other. The Other option allows a user to excite the system with any arbitrary pulsation source. In practice this takes the form of cyclical user-defined [Junction Transient Data](#) applied to one or multiple junctions in the system. The Reciprocating Compressor option allows a reciprocating positive displacement compressor junction or assigned flow junction to be selected, and has the following parameters.

- **Compressor**
 - **Reciprocating Compressor** - Select an existing reciprocating compressor from the model listed by junction number.

Note: If there are multiple reciprocating compressors with the same speed and number of heads, any may be selected. If these parameters differ between compressors, use the Other option for Pulsation Source.

- **User Defined** - This option is selected when using one or more assigned flow junctions to represent reciprocating compressors. The appropriate transient must be defined in the Transient tab of the assigned flow junction, typically using the periodic flow feature. The Speed and Number of Heads parameters do not define this transient flow, but are used for Artificial Pulsation Resolution and API 618 and should match the assumptions that were used to define the transient flow.
- **Speed** - The current operating speed of the compressor in RPM. If a reciprocating compressor junction is selected, this will display the value defined in the Compressor Properties window.
- **Number of Heads** - This is the number of compression chambers in the compressor, which is the number of cylinders in a single-acting compressor and twice the number of cylinders in a double-acting compressor. If a reciprocating compressor junction is selected, this parameter will be determined from the compressor properties.
- **Evaluate with API 618** - When this option is checked, an API 618 acoustical analysis will be performed. More detail about this analysis is provided in the section below.

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:

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

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.

Note: In Steady State Pulsation, the cutoff frequency should be at least six times the base pulsation frequency.

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, under the Steady State Pulsation selection, 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 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.

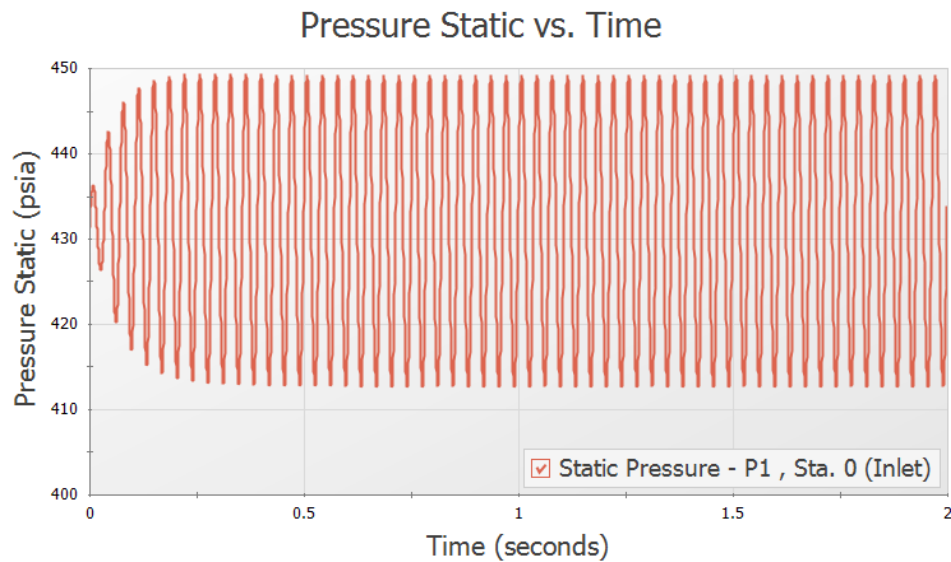


Figure 3: Example pressure results at the outlet of the reciprocating compressor with an oscillating flow transient specified

With output an Acoustical Frequency Analysis can be graphed. When API 618 is being evaluated, the limit and the frequency will be graphed together as shown in Figure 4.

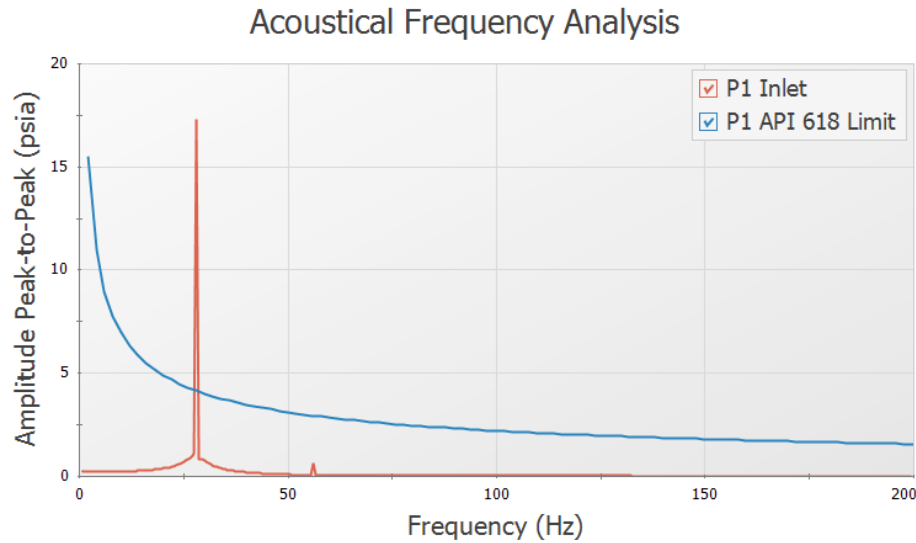


Figure 4: Example Acoustical Frequency Analysis Plot including the API-618 Limit

The equations for the API 618 peak to peak pressure level limits are as follows ([American Petroleum Institute 2007](#), Clause 7.9):

$$P_1 (\%) = \sqrt{\frac{a}{1150}} \left(\frac{300}{(P_L ID f)^{0.5}} \right) \quad US \ Units$$

$$P_1 (\%) = \sqrt{\frac{a}{350}} \left(\frac{400}{(P_L ID f)^{0.5}} \right) \quad SI \ Units$$

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), f is the pulsation frequency in hertz, and a is the speed of sound inside the pipe in feet per second (meters per second).

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 Topics

Reciprocating Compressor.....	271
Positive Displacement Compressor.....	269
Defining Pulsation Settings.....	434
The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X locations.....	560
Pulsation peak-to-peak pressures are within allowable API 618 levels.....	570

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.....	490
--------------------------------	-----

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.

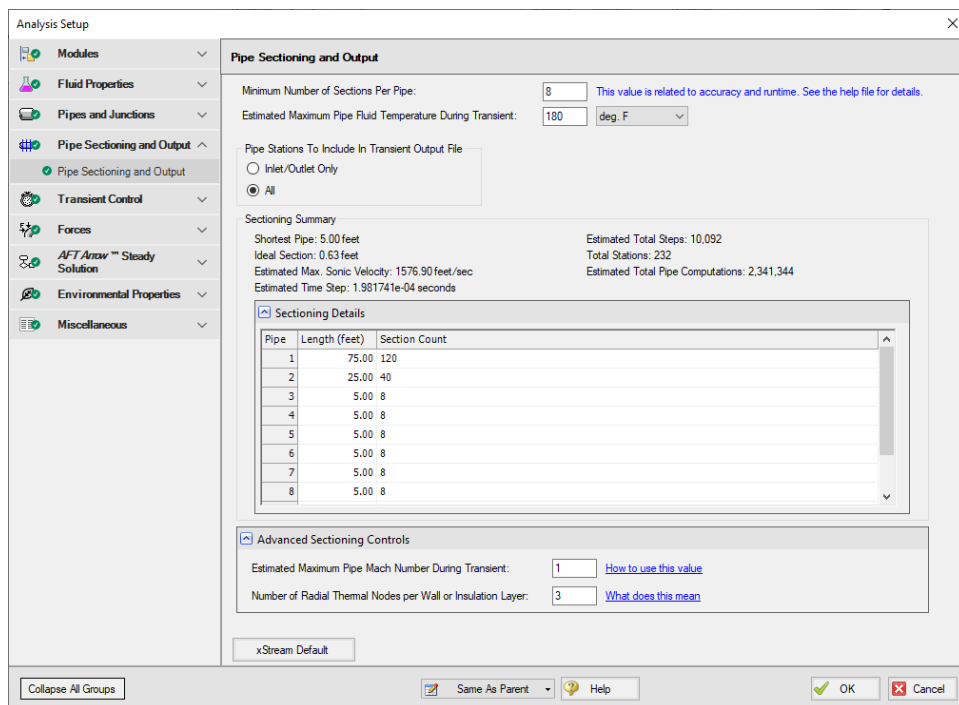


Figure 1: The Sectioning panel in the Analysis Setup window

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.

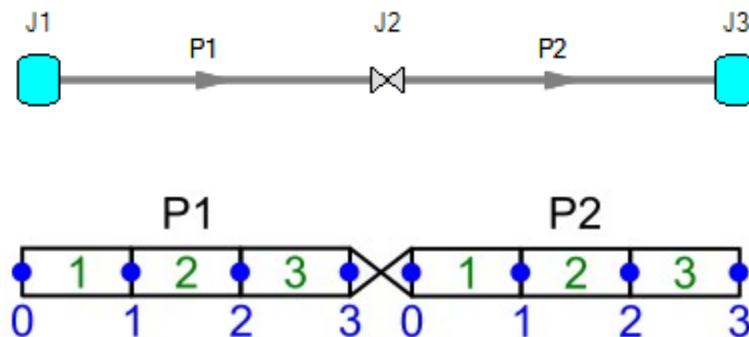


Figure 2: Computing Stations (blue) exist at the boundary of every section (green)

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.

Transient Control Group

The Transient Control Group in [Analysis Setup](#) allows the user to specify transient parameters such as the simulation duration, transient solver initialization, and transient checkpoints. Only the Simulation Mode/Duration panel is required to run the model.

- [Simulation Mode/Duration panel](#) - Required panel. Contains the controls for simulation type, simulation duration, and options to save less output to file.
- [Transient Solver Initialization panel](#) - Optional panel. Allows Specification of how the solver will resolve artificial transients.
- [Transient Checkpoints panel](#) - Optional panel. Allows specification of when to create checkpoint files used to continue transient simulation.

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.

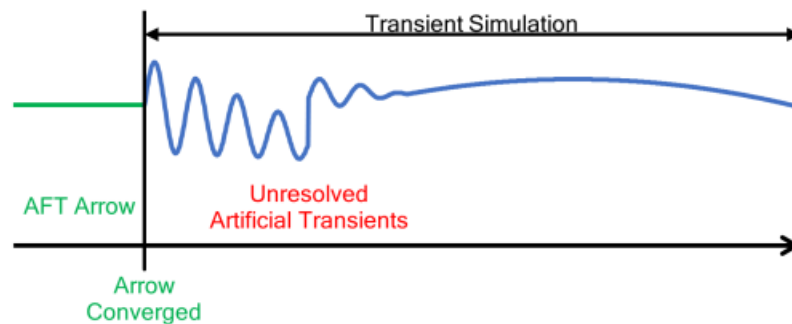


Figure 1: Illustration of the Transient Solver Initialization process without artificial transient resolution

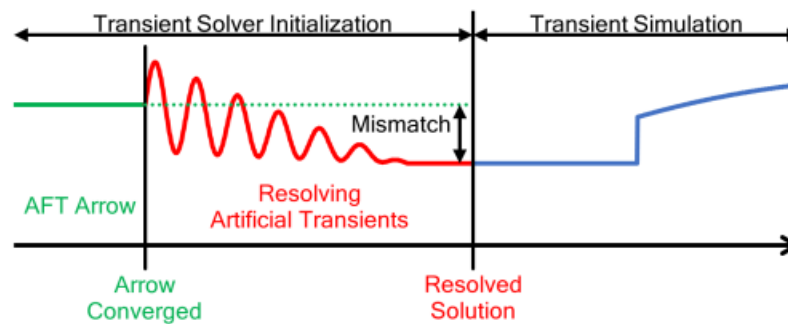


Figure 2: Illustration of the Transient Solver Initialization process with artificial transient resolution

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.

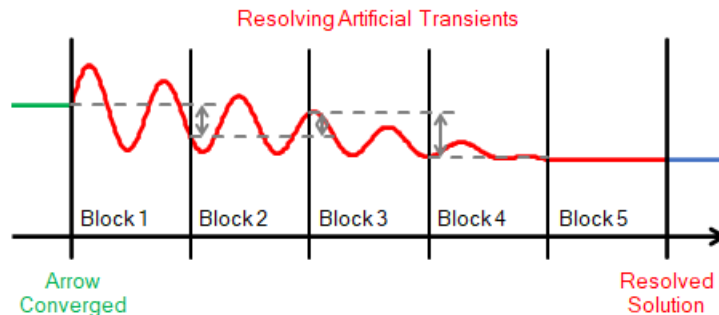


Figure 3: Example of blocks used during artificial transient resolution period

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 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.....	59
Method of Characteristics.....	490

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.....	43
-----------------------	----

Force Definitions Panel

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

Note: As AFT xStream is only a pneumatic solver, only pneumatic 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. Pneumatic force files can be generated for [import into software](#) designed for force analysis.

Defining Force Sets from Analysis Setup

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

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 Pipe Forces from Turbine Shutdown Example.

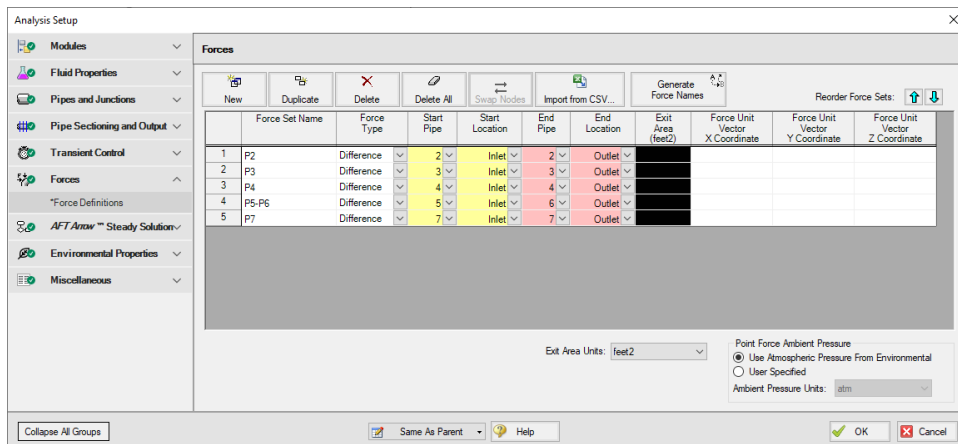


Figure 1: Force set data is defined on the Force Definitions panel in the Analysis Setup window

- **New** - Add a new force set to the table.
- **Duplicate** - Copy an existing force set.
- **Delete/Delete All** - Remove the selected force set / remove all 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 panel](#) value, but can be changed here.

Each individual force set has a number of required parameters:

- **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 node (for Difference force types). 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.
- **Exit Area** - For Exit force types, 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 Force Sets 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.

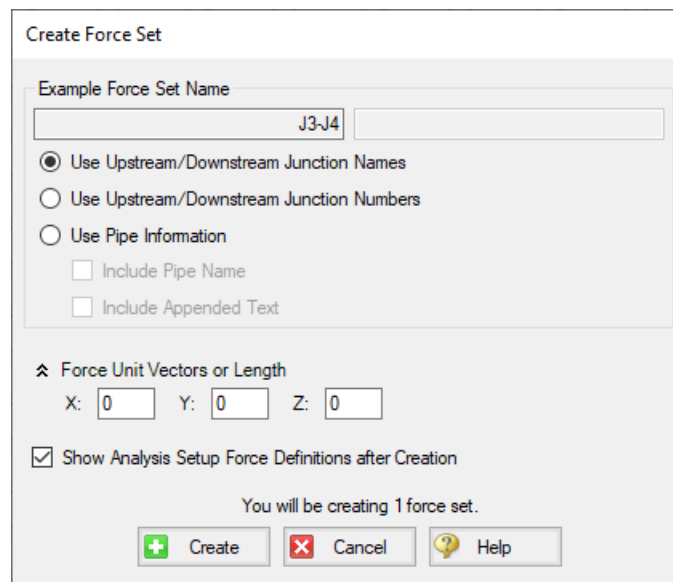


Figure 2: Create Force Set window in the Workspace

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

Related Topics

Transient Forces.....43

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.....516

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.

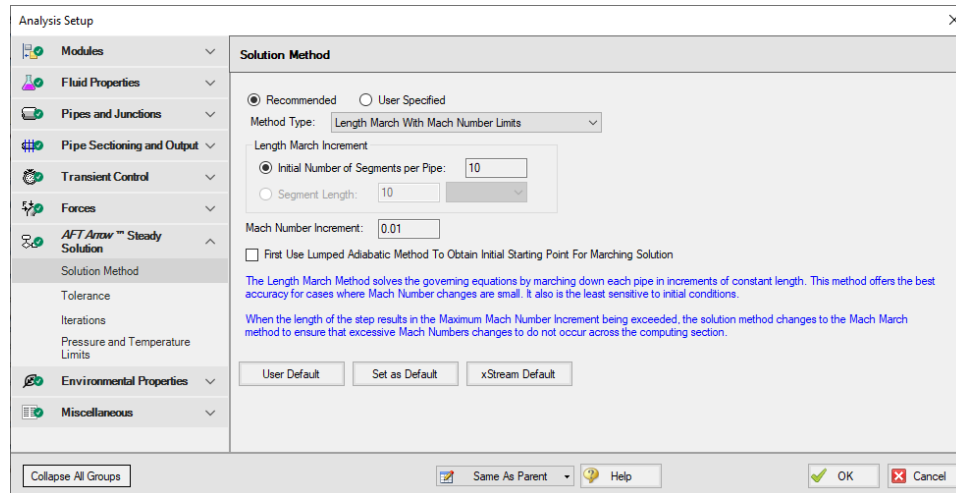


Figure 1: Default settings on the Solution Method panel

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.

		Pipes		Axial Points		Heat Transfer				
		Pipe -- Point	x (feet)	Mach #	Mass Flow (lbm/sec)	P Static (psia)	Rho Static (lbm/ft ³)	Velocity (feet/sec)	H Stag. (Btu/lbm)	T Static (deg. F)
Length March	<input type="checkbox"/>	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
Mach Number March		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.

Related Topics

Transient Result Accuracy.....	59
--------------------------------	----

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](#)

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

Integrating Equation 25 yields:

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

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

$$P_{o2} = P_{o1} e^{-C}$$

$$C = \frac{\gamma \bar{M}^2}{2} \left[\ln \frac{T_{o2}}{T_{o1}} + \ln \frac{Z_2}{Z_1} + \ln \frac{\gamma_2}{\gamma_1} + \frac{f(x_2 - x_1)}{D} \right] + \frac{g \sin \theta (x_2 - x_1)}{R \bar{Z} \bar{T}} \quad (27)$$

Example

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

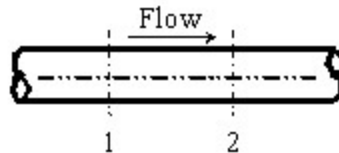


Figure 1: Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations.

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](#):

$$h_{o_2} = h_{o_1}$$

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:

$$\dot{m}_2, x_2, h_{o_2} \quad (\textit{nonisothermal})$$

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

\dot{m}_2, x_2, T_2 (*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 relates 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.

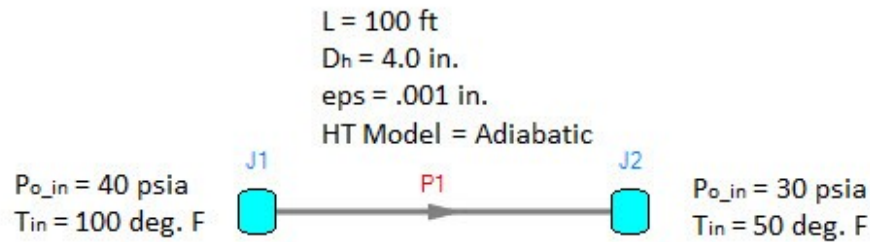


Figure 2: Model of an air system dropping from 40 to 30 psia.

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 ³)
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.

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	38.138676	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:

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

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

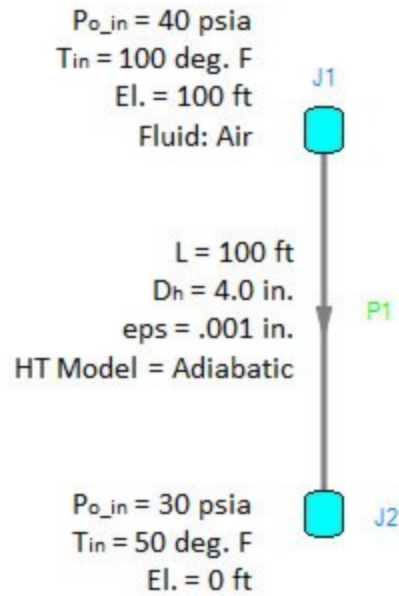


Figure 5: AFT xStream model solved in Figure 1 and verified in Table 1. Fluid is air. File is Verify_Governing_Equations.xtr.

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

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

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

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

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	
x (feet)	P Static (psia)	Rho Static (lbm/ft3)	Velocity (feet/sec)	H Stag. (ft)	f	ρ (lbm/ft3)	V (feet/sec)	d/ (ft)	dh/dx (ft/ft)	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	0.1866	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	-0.87144	-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	-4.66E-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-08	-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).

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 ³)	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 463

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:

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

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

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

$$F_g = 1 + \frac{\gamma - 1}{2} M^2 \quad (18)$$

Using the four new parameters in Equation 18 simplifies to

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

Integrating Equation 19 yields

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

Equation 20 can be solved for x_2 :

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

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:

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

Example

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

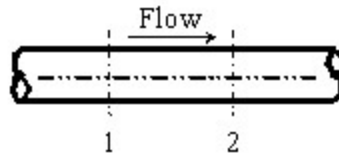


Figure 1: *Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations*

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 following 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., Δx) for which we are solving. What is the reason for trying to solve for Δx 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 ΔM 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 (*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 $h_{o,2}$ 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 Δx , 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 ΔM . 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 ΔM , 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 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.

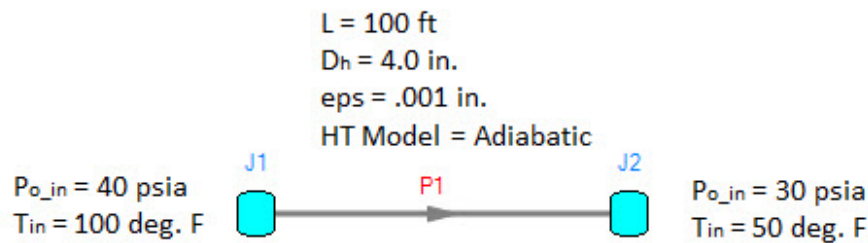


Figure 2: Model of an air system dropping from 40 to 30 psia.

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/ft3)
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.

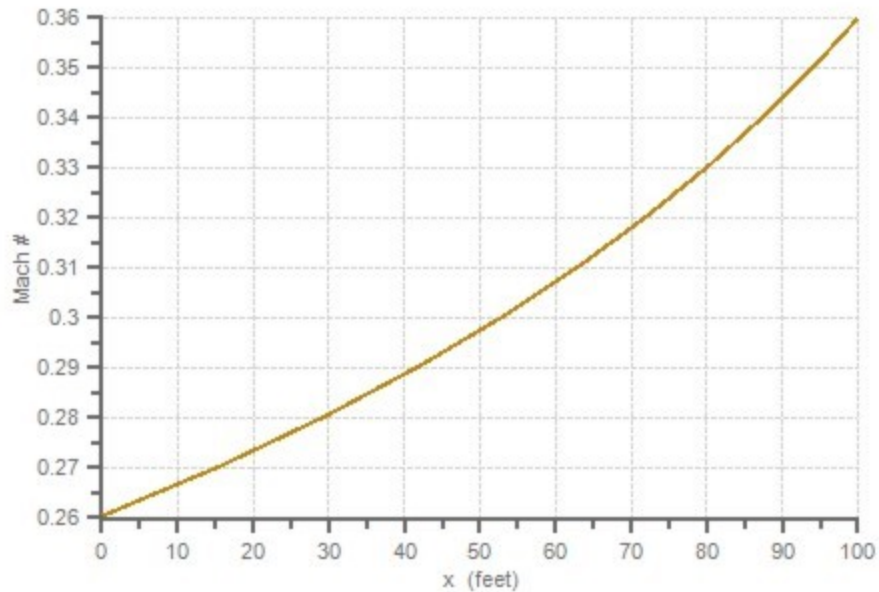


Figure 4: The Mach number increases with length in this plot of the Figure 3 results

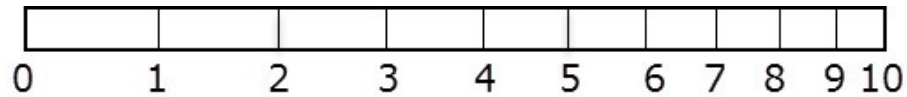


Figure 5: Relative pipe section locations for Figure 3 results. Notice how computing sections get closer as a result of gas acceleration as the end of the pipe is approached

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.

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	
x (feet)	P Static (psia)	Rho Static (lbm/ft3)	Velocity (feet/sec)	H Stag. (Btu/lbm)	f	ρ (lbm/ft3)	V (feet/sec)	d/dx (feet/feet)	dho/dx (Btu/lbm/feet)	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	307.056	-11.373	-0.0200	-15.60	15.60	1.42117	-1.30300	-0.13815	0.01985	-0.00012	1.54E-09	-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
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
52.87	33.014	0.1622	346.745	202.246	0.0149	0.1650	0.00547	341.088	-11.314	-0.0142	-11.06	11.06	1.15068	-1.02599	-0.13743	0.01267	-0.00006	-1.11E-07	-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
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
79.80	29.974	0.1478	380.582	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
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
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
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
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

Related Topics

Nomenclature 463

Lumped Adiabatic Method - Detailed Discussion

When this [AFT Arrow Steady Solution Method](#) is used, all heat transfer data entered for individual pipes is neglected. The other pipe data is left as entered by the user in case the user switches back to one of the marching methods.

Elevation data is also 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:

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

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 .

Note: In [Saad 1993](#) on page 209 equation 5.27 (Equation 43 above) the square brackets are not included. They have been added above to clarify that the natural log applies to all of the encapsulated terms. Furthermore, the left-hand side in Saad contains a multiplier of 4 since the Fanning friction factor was used, whereas Equation 43 above used in AFT software uses the Darcy friction factor.

Related Topics

References.....516

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 γ 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.....516

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.

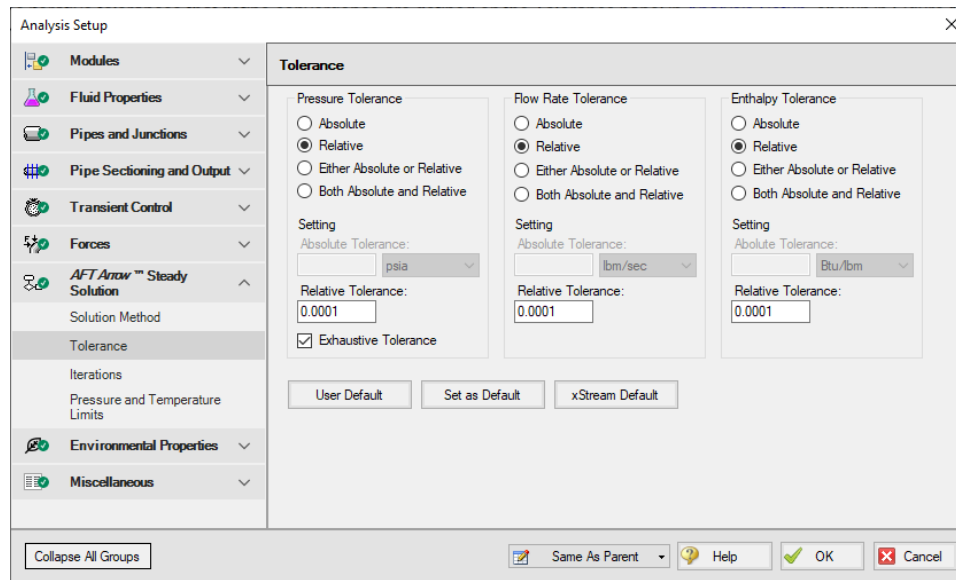


Figure 1: Tolerance panel in Analysis Setup

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 ± 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	0.6666667	8.0000000	NO	NO	NO	NO
3	14.0000	0.142857	2.0000000	NO	NO	NO	NO
4	13.5000	0.037037	0.5000000	NO	NO	NO	NO
5	13.6000	0.007353	0.1000000	NO	NO	NO	NO
6	13.5900	0.000736	0.0100000	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.

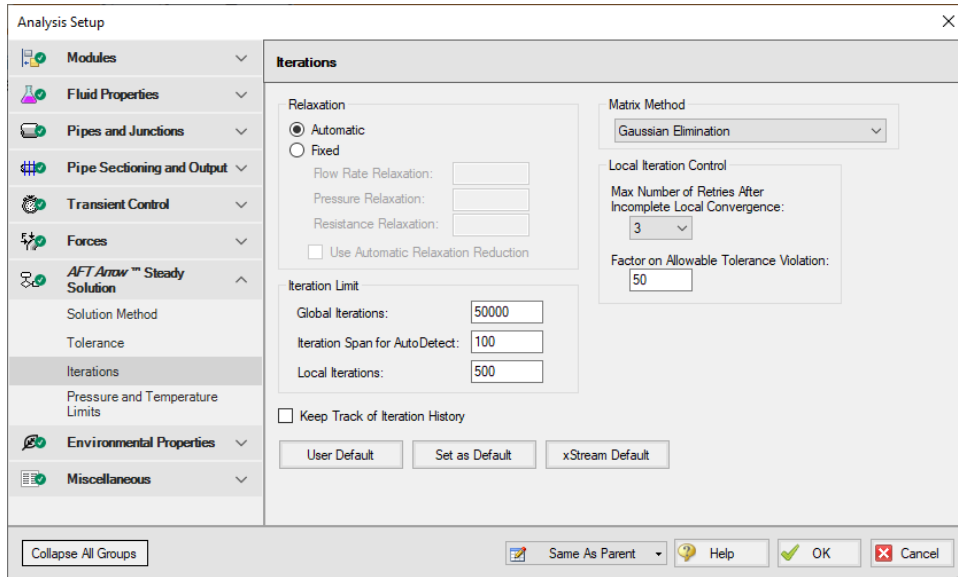


Figure 1: Iterations panel in Analysis Setup

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

The Pressure and Temperature Solver Limits panel allows users to set boundaries, temperature and pressure, within the solver. These limits are intended to help with convergence and prevent the solver from providing unrealistic results. These limits are intended to keep the solver within physical restraints, such as temperatures that can't be produced on Earth. 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.

For design requirements and limitations, use [Design Alerts](#) to signify when the solvers values are beyond a desired range yet is still physically possible to occur.

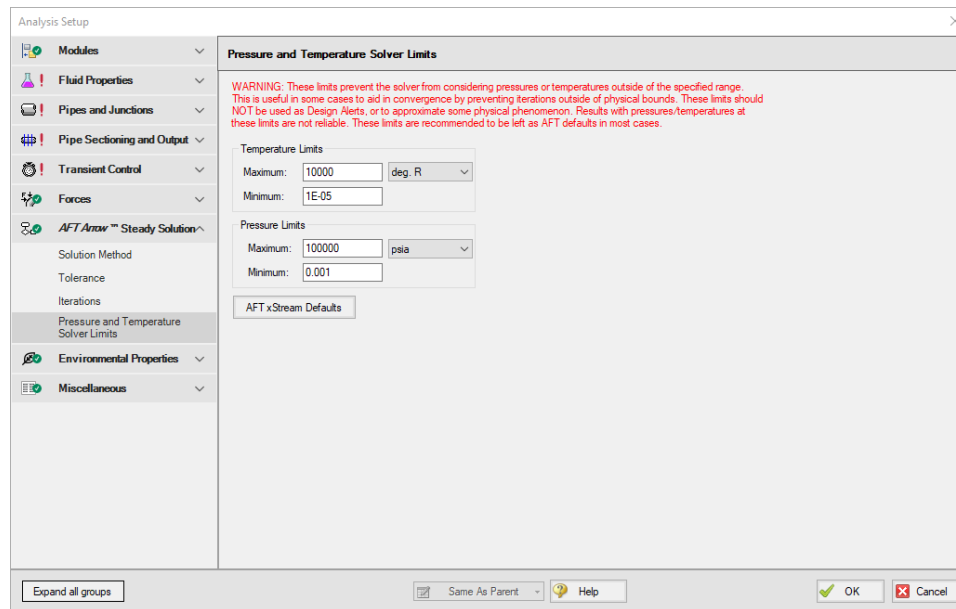


Figure 1: Pressure and Temperature Solver Limits panel within Analysis Setup

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.

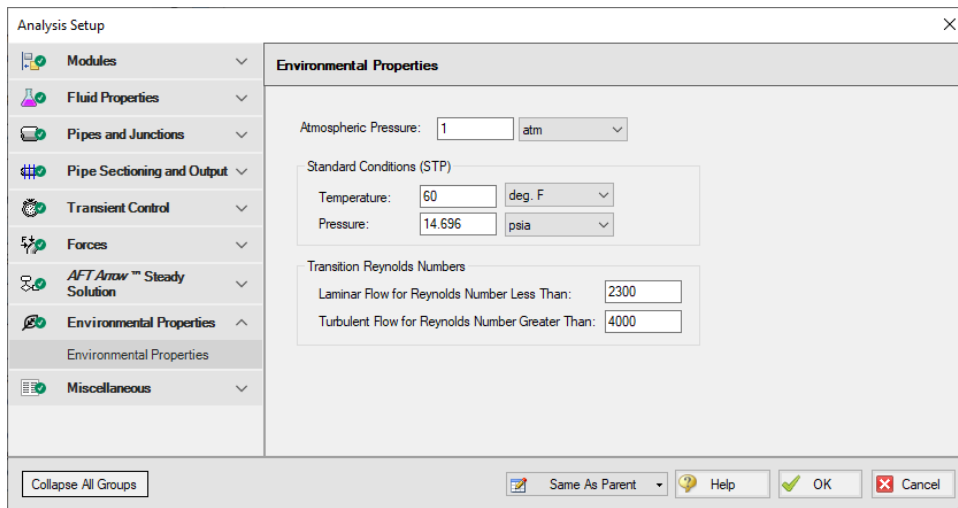


Figure 1: Environmental Properties panel in the Analysis Setup window

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.

Transient Solver Advanced Parameters

The Transient Solver Advanced Parameters panel is located in the Miscellaneous group of Analysis Setup.

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

Selecting and Finding Objects	159
Pipe Drawing Modes	163

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	159
Moving Objects	179
Editing Objects	180
Defining Objects	180

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 Layer Settings 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.....	159
Moving Objects.....	179
Editing Objects.....	180

Defining Objects..... 180

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.....	359
Global Pipe Editing.....	171
Global Junction Editing.....	172
Force Set Types.....	46

Pipe Drawing Modes

AFT Impulse has a selection of pipe drawing modes. To select or change a pipe drawing mode, navigate to Arrange -> Pipe Drawing Mode. These pipe drawing modes are:

- [2D Freeform](#)
- [2D Orthogonal](#)
- [Isometric](#)

A second function called **Snap to Grid** is highly integrated with Pipe Drawing Modes and will alter how each pipe drawing mode works. To select or change a Snap to Grid option, navigate to Arrange -> Snap to Grid. The Snap to Grid options for the 2D modes are on and off. Snap to Grid options for Isometric are more extensive. These snap to grid options are:

- Snap to Isometric Grid Lines
- Snap to Grid Points
- Do Not Snap

2D Freeform

This Pipe Drawing Mode allows placing pipes or junctions anywhere on the Workspace, with pipes at any angle. Snap to Grid can be turned on with this mode to force the junctions to be at grid intersection points.

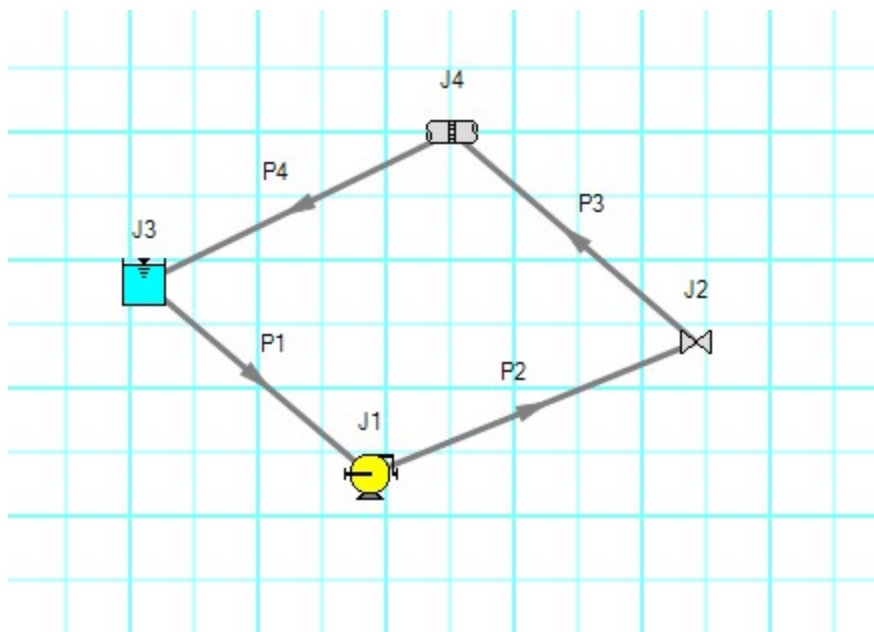


Figure 1: 2D Freeform with Snap to Grid off

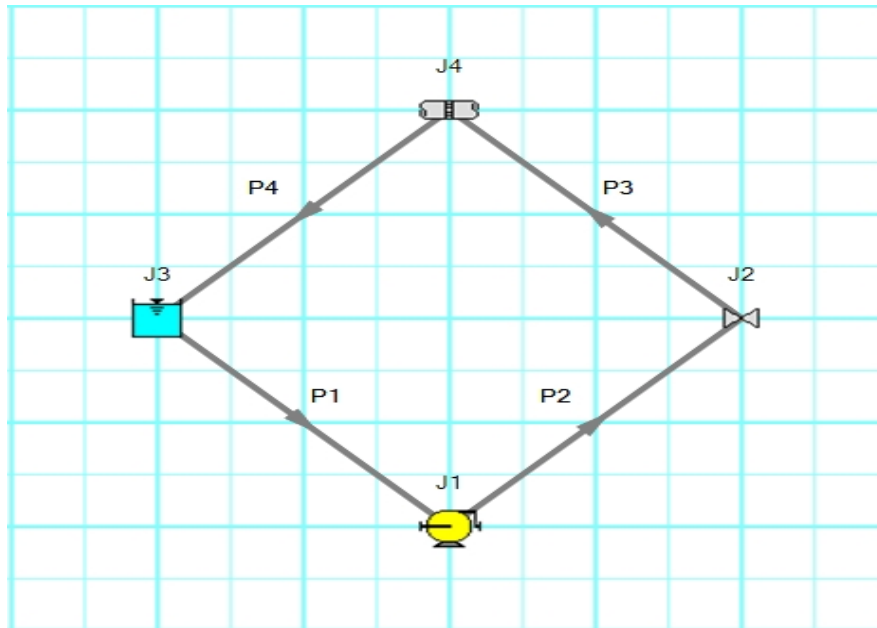


Figure 2: 2D Freeform with Snap to Grid on

2D Orthogonal

The 2D Orthogonal Pipe Drawing Mode forces all pipes drawn in the model to be either horizontal or vertical. Segments are automatically inserted into pipes to enforce drawing only along the horizontal and vertical directions.

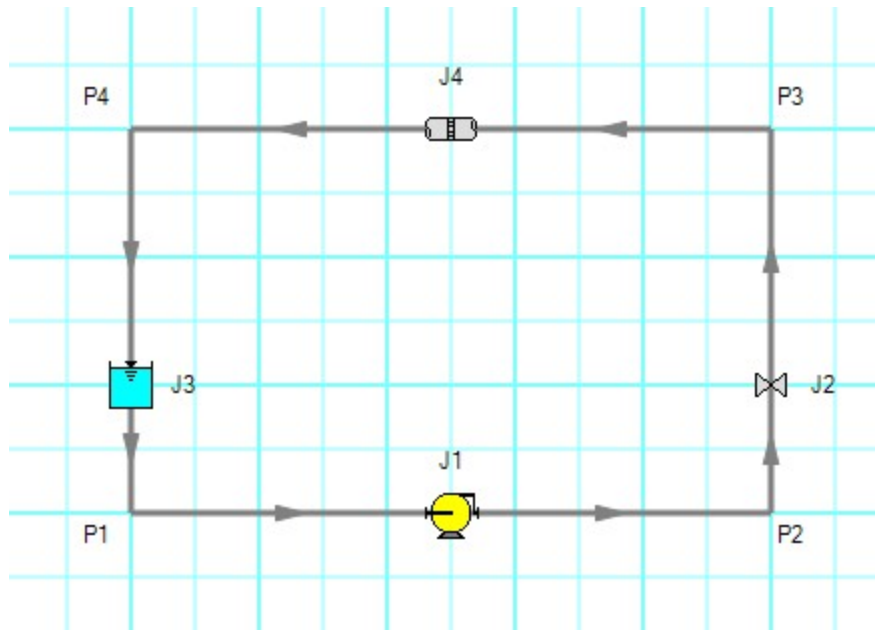


Figure 3: Model in 2D Orthogonal mode with Snap to Grid on

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 segments
- Use a [fine grid](#) and the 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 using the [Workspace Layers](#)

Isometric

When modeling, it may be convenient or desirable to demonstrate the three dimensional nature of a system. A typical 2D grid has two sets of grid lines representing the x and y axes. An isometric grid has three sets of grid lines that are offset by 60°, representing the x, y, and z axes. It is ideal to set the grid type desired before building a model to avoid the pipe and junction locations from being rearranged in undesired ways. It is also very helpful to show the Grid (Arrange -> Show Grid) when building an isometric model.

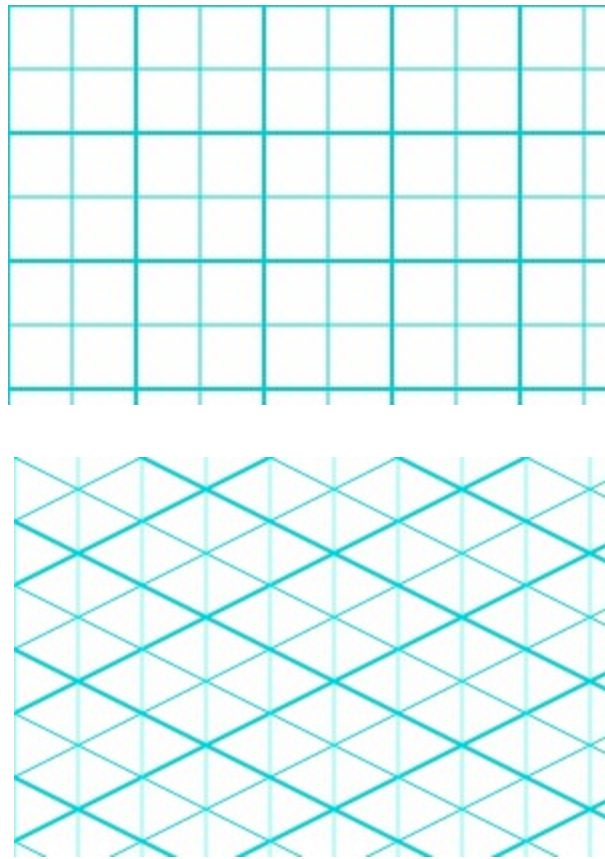


Figure 4: 2D (left) and isometric (right) grids

Drawing Pipes in Isometric Mode

How pipe drawing in isometric mode functions will change depending on the selected snap to grid option. When Snap to Isometric Grid Lines is selected, the pipes are automatically segmented to lie on the grid lines 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 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.

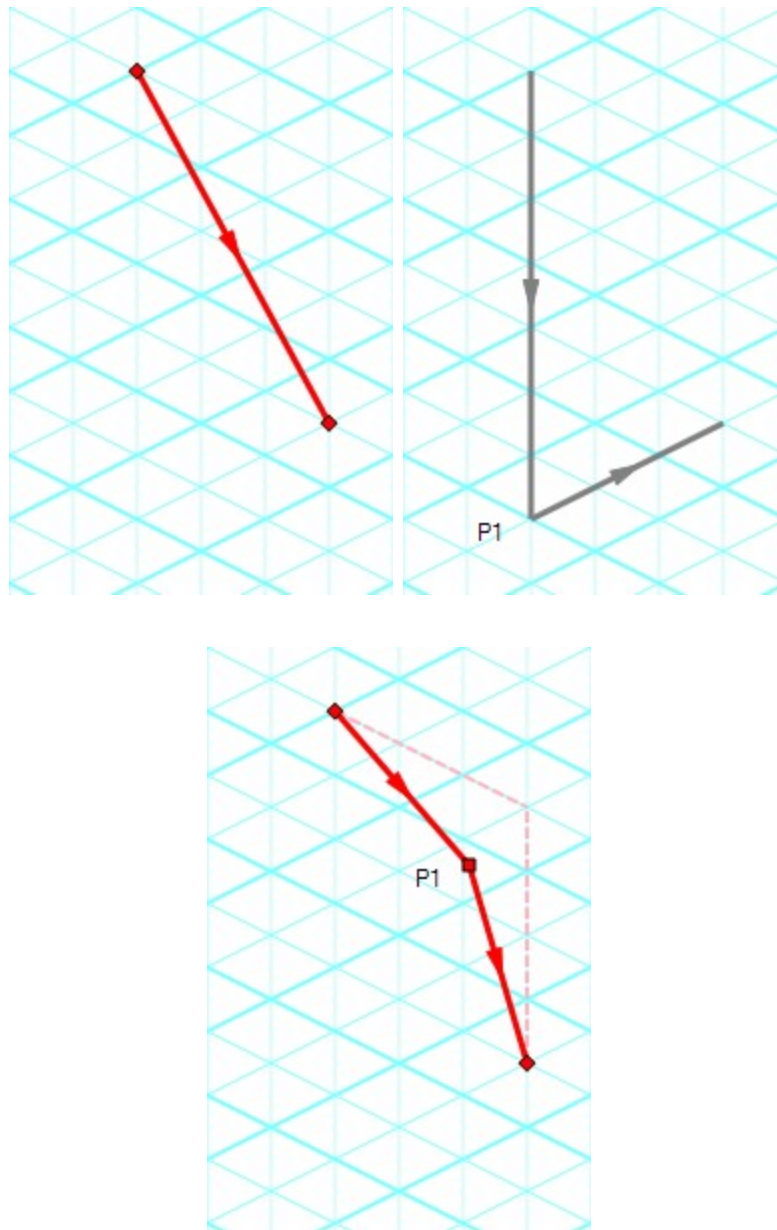


Figure 5: Automatic layout of isometric pipes

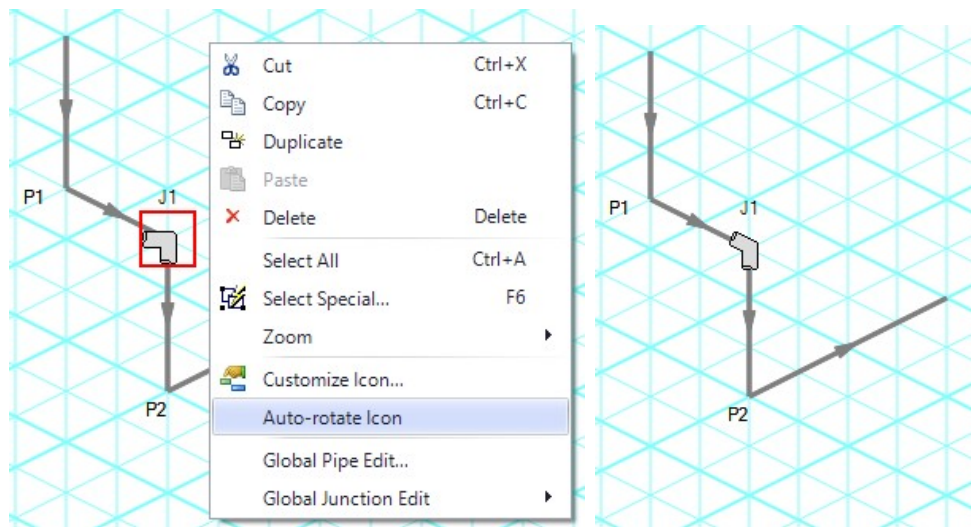
The Snap to Grid Points option will force the end points of the pipes to snap to the points where the grid lines intersect without adding segments to it. This is akin to the 2D Freeform mode while Snap to Grid is on.

The Do Not Snap option will allow the end points of the pipes to be placed anywhere within the Workspace.

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 Impulse will rotate the icon automatically following the Workspace arrangement. When pipes are connected to icons already placed on the Workspace, right-click and select Auto-rotate Icon to adjust the icon. For manual control of icon appearance, right-click the junction and select Customize Icon to choose the desired appearance.

Note: The Rotate button in the Customize Icon window can be used to find the appropriate icon orientation.



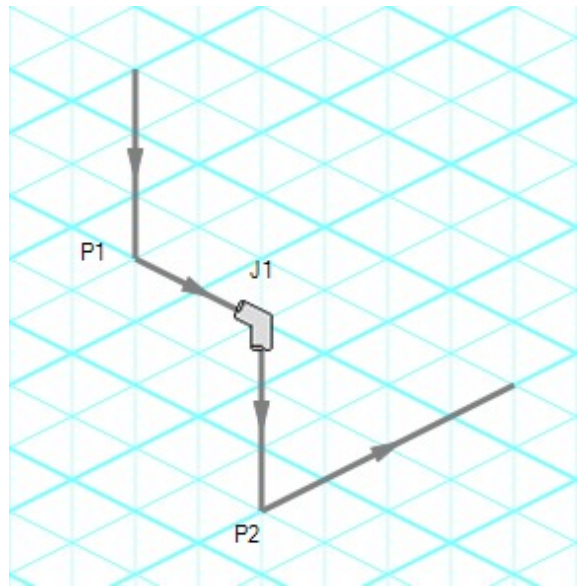
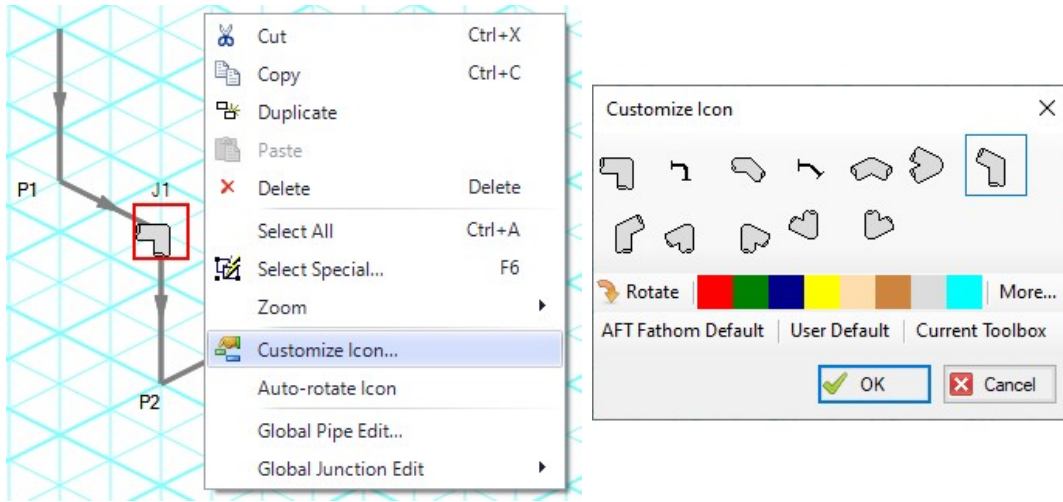


Figure 6: Icon rotations in isometric space

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 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.

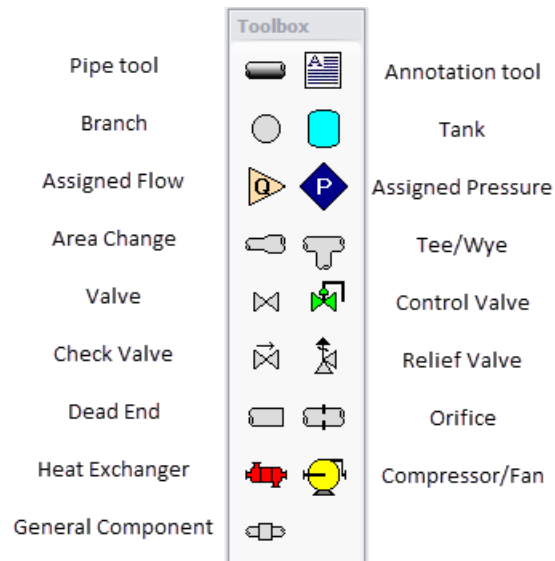


Figure 1: Toolbox items

Custom Junction Icons

Right-clicking a junction icon in the Toolbox will open the Customize Icon window. A different icon can be selected from the predefined assortment, or a custom image can be uploaded using the Add Image button. This functionality enables you to use standard P&ID icons or custom images to visually represent your equipment. By using recognizable icons, you can improve the clarity and comprehension of model results, especially for those who may not be familiar with the default AFT junction icons.

Note: The [Annotation Tool](#) also supports triangle and cloud shapes which can be used in combination with the Notes section on the Scenario tab of the [Quick Access Panel](#) to designate revisions of the model file, similar to a typical P&ID drawing.

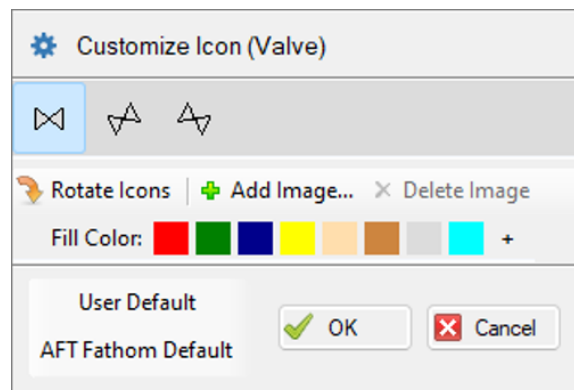


Figure 2: The Customize Icon window allows a custom image to be applied to any junction

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.....340

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 lose data when combined, or if they can't be combined.

If any junctions are selected that would lose 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.....340

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.....	340
---------------------------------------	-----

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.....	340
Selecting and Finding Objects.....	159
Selecting Objects.....	159
Editing Objects.....	180
Defining Objects.....	180

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.....	340
Selecting and Finding Objects.....	159
Selecting Objects.....	159
Moving Objects.....	179
Defining Objects.....	180

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.....	340
---------------------------------------	-----

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 displays data associated with a Workspace object in read-only fashion via a pop-up overlay (see Figure 1). Inspecting is faster than opening the Properties window if you only want to review the data and no edits need to be made. Inspecting can be done directly from the Workspace and provides a quick summary of the parameters that appear on the [Model Data Primary Window](#).

To inspect an object, position the mouse pointer on the object and hold down the right mouse button.

The Inspection feature is also available *inside* each pipe/junction's Properties window. This is useful if you want to check the input data of objects connected to the one you are viewing. The upstream/downstream pipes or junctions are listed, and holding the right mouse button on the listed ID number will open the associated inspection overlay. For example, when working in an [Orifice 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.

Judicious inspecting can save you time when assembling or debugging a model.

Note: Output data will be shown on the inspection overlay only if a valid [Output File](#) is present for the current scenario.

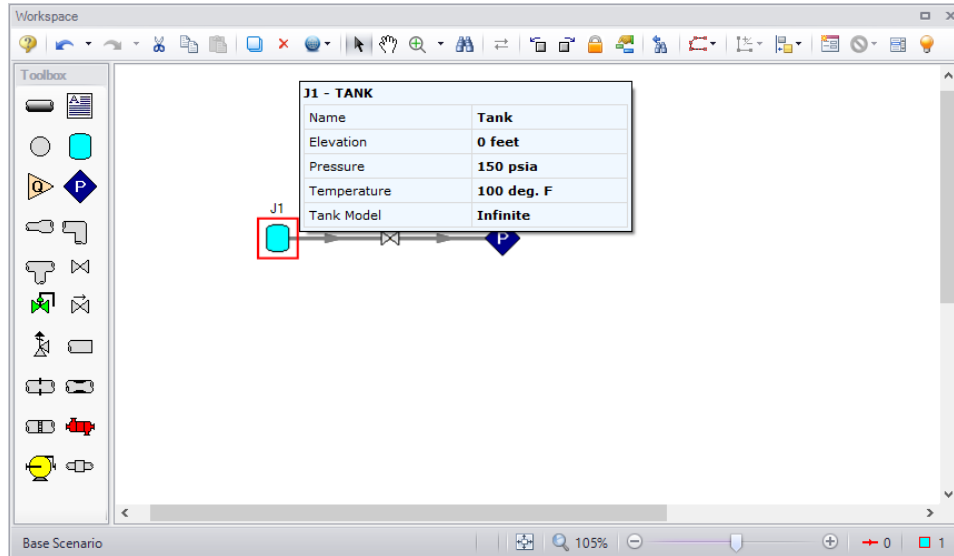


Figure 1: Holding the right mouse button on a Workspace object displays the associated data

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.....	187
General Junction Information.....	215

Status


The status tab lists all undefined parameters that are required to define the pipe or junction.

Related Topics

Pipe Properties Window.....	187
General Junction Information.....	215

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.....340

Moving Pipe and Junction Labels

Each pipe and junction on the [Workspace](#) also displays a label. A label refers to any text associated with an object - pipe numbers, names, flow rates, and so forth. The label will display a unique combination of defined display text items from each visible [Workspace Layer](#). This label can be dragged to improve model aesthetics. The relative location of the label to the object is linked 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
- Click Reset Label Locations in the Label Control group of the [Layer Settings window](#) to reset all label locations

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 [User Options](#).

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 User Options.

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

Pipe Properties Window	187
Common Pipe Input Parameters	188
Pipe Model	189
Fittings & Losses	190
Heat Transfer in Pipes Overview	192
Insulation for Pipes	210
Optional Input for Pipes	210
Add/Remove Pipe Segment	211
Reverse Direction	212
Pressure Drop in Pipes - Detailed Discussion	212

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.

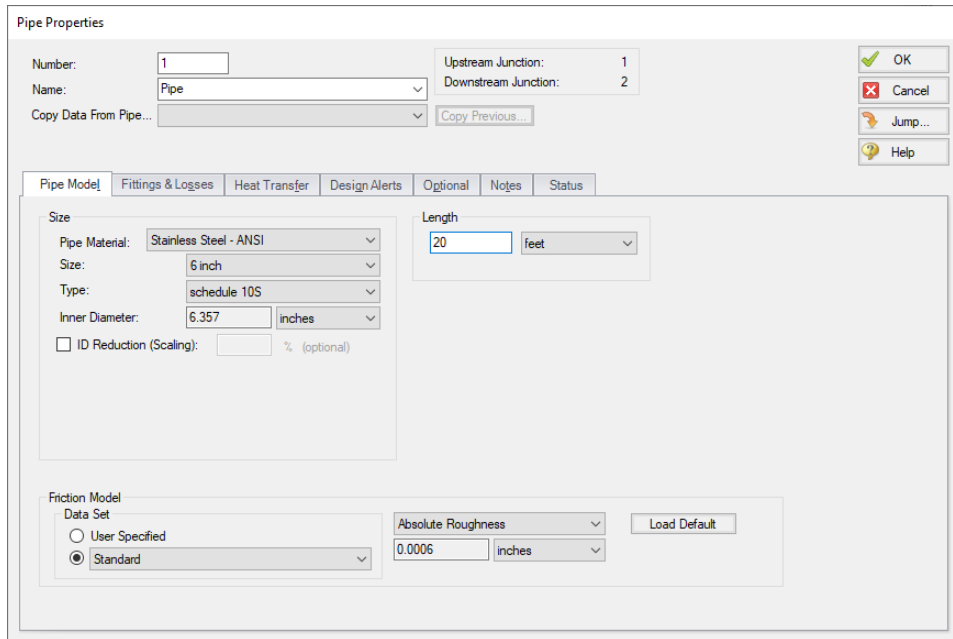


Figure 1: Fully Defined Pipe Properties Window

Related Topics

Notes.....	183
Status.....	183

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 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 losses, open the Pipe Properties window, click the Fittings & Losses tab, then 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 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.

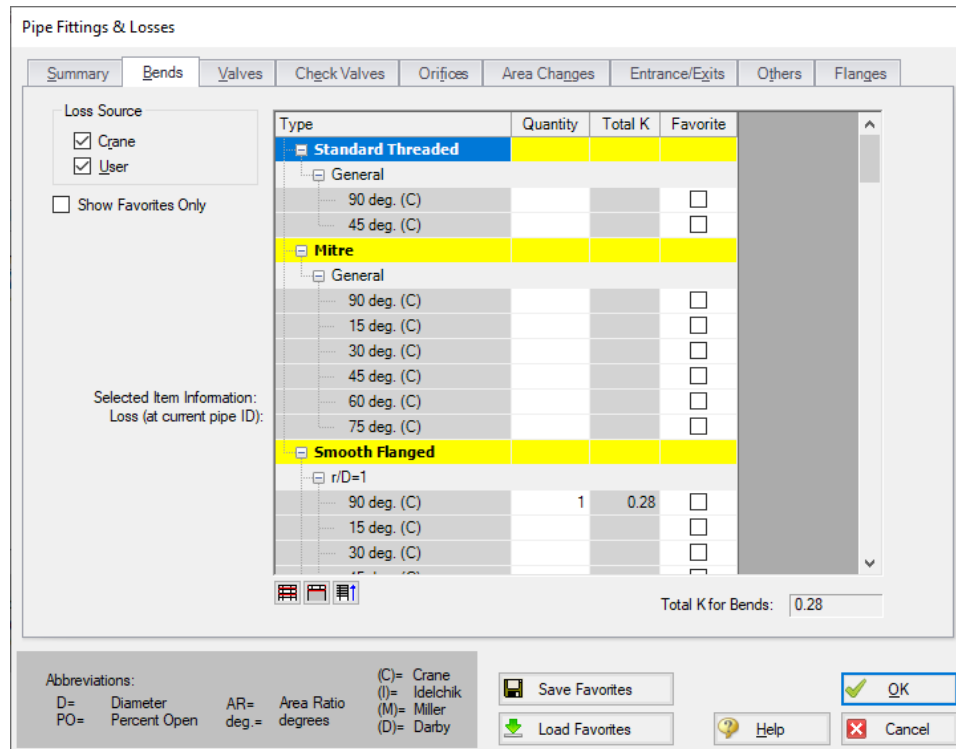


Figure 1: Pipe Fittings & Losses window

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.....	451
Specifying Losses.....	220
References.....	516

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.

Related Topics

References.....516

Constant Heat Transfer

There are several thermal models for pipes in AFT xStream which fix the thermal behavior of the pipe to some constant value:

- **Adiabatic At Internal Surface** - The heat transfer between the pipe and the environment is fixed to zero, and no heat is stored in the pipe walls. The entire length of the pipe will have a stagnation enthalpy equal to that of the upstream junction. The actual fluid temperature depends on the system.

The above models follow a common format on the Pipe's Heat Transfer tab, as shown below. There is only one required value input unless insulation is added. One layer of insulation (external or internal) can be added, with conductivity values from a library or User Specified in this window.

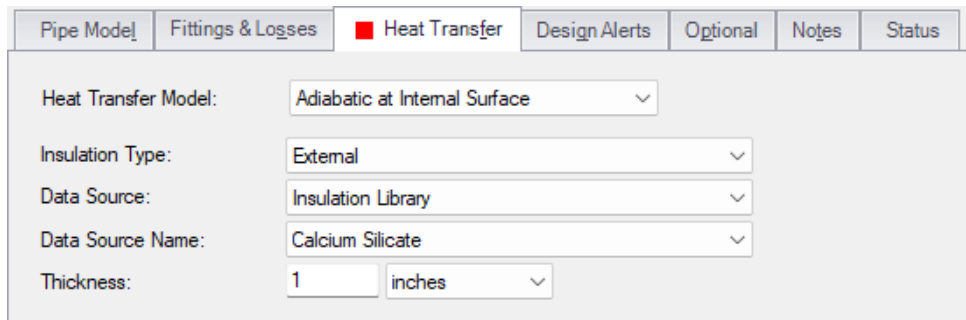


Figure 1: Heat Transfer tab of the Pipe Properties window

Related Topics

Heat Transfer in Pipes Overview..... 192
 Convective Heat Transfer.....194
 Heat Tracing Flux Based On Wire Turns and Length.....197
 Steady State Heat Transfer In Pipes..... 201

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 the [NIST REFPROP fluid library](#).
- **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** - Multiplier for the value of surface area used in the heat transfer calculations (default value of 1). In general, this represents any "extra" heat transfer area that may exist. For example, cooling fins on the outside of the pipe could be modeled by entering a value larger than 1 on the External resistance row.
 - **Convection Coefficient** - Applies to Fluid Internal and External. Both values can be either Correlation or User Specified.

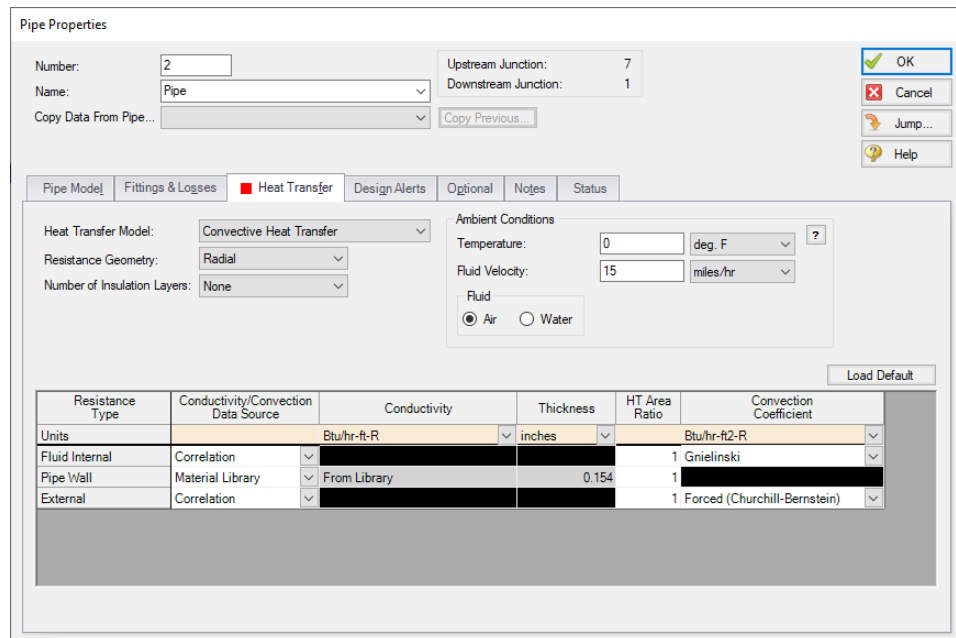


Figure 1: Convective Heat Transfer Layout

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 Topics

Heat Transfer in Pipes Overview.....	192
Constant Heat Transfer.....	194
Heat Tracing Flux Based On Wire Turns and Length.....	197
Steady State Heat Transfer In Pipes.....	201

Heat Tracing Flux Based On Wire Turns and Length

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.

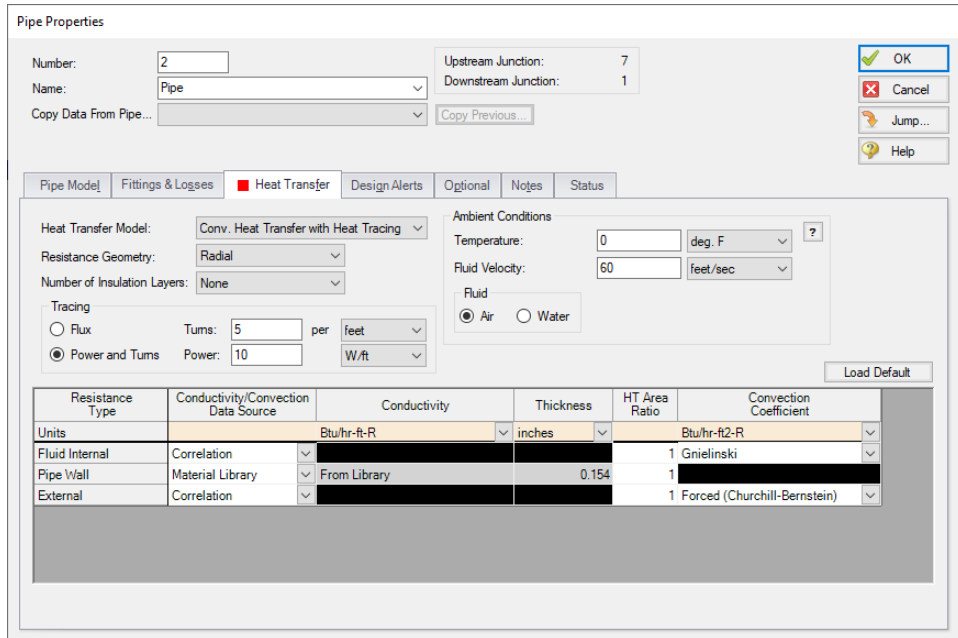


Figure 1: Pipe Properties window with the Conv. Heat Transfer with Heat Tracing model enabled

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.

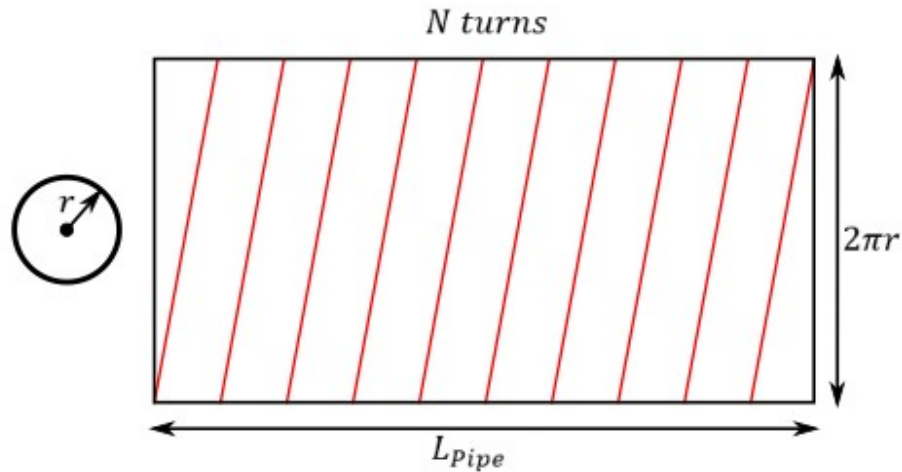


Figure 2: Wire tracing geometry

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:

$$Q'' = p \sqrt{\frac{1}{(2 \pi r)^2} + n^2}$$

Related Topics

Heat Transfer in Pipes Overview.....	192
Constant Heat Transfer.....	194
Convective Heat Transfer.....	194
Steady State Heat Transfer In Pipes.....	201

Steady State Heat Transfer In Pipes

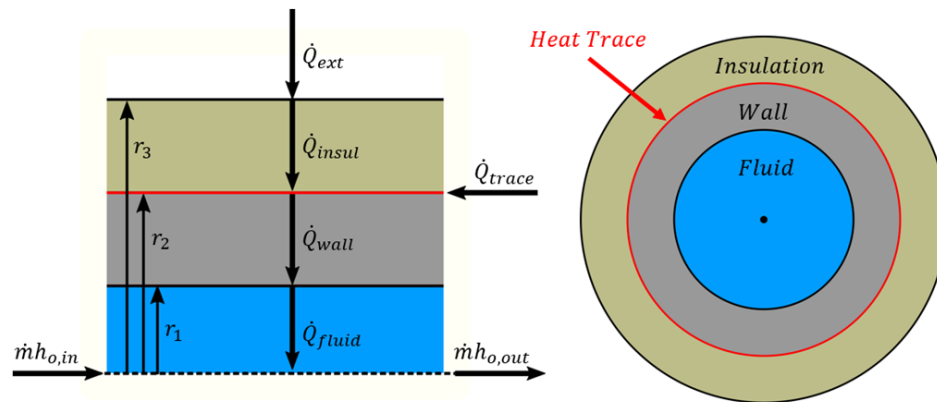


Figure 1: Diagram of a pipe with Heat Tracing and Insulation

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:

$$\begin{aligned} \dot{Q}_{fluid} &= \dot{Q}_{trace} + \dot{Q}_{ext} && CV \text{ around wall \& insulation} \\ \dot{Q}_{wall} &= \dot{Q}_{fluid} && CS \text{ at inside of wall} \\ \dot{Q}_{ext} &= \dot{Q}_{insul} && CS \text{ at outside of insulation} \end{aligned}$$

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 ΔT_{fluid} used for determining \dot{Q}_{fluid} , the correct solution has been found. If they are different, ΔT_{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

$$Nu = 0.023 Re^{4/5} Pr^n \quad \text{where } \left(\begin{array}{l} n = 0.4 \text{ for heating} \\ n = 0.3 \text{ for cooling} \end{array} \right) \quad \text{valid for } \left(\begin{array}{l} 0.6 \leq Pr \leq 160 \\ Re \geq 10^4 \end{array} \right)$$

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.....	516
Heat Transfer in Pipes Overview.....	192
Constant Heat Transfer.....	194
Convective Heat Transfer.....	194
Heat Tracing Flux Based On Wire Turns and Length.....	197

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.

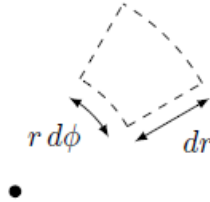


Figure 1: Cylindrical control volume for heat rate calculation through the pipe wall/insulation, with dz extending out of the page

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.

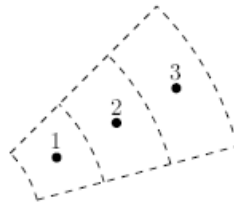


Figure 2: Partial pipe cross-sectional area representing the radially sectioning for heat transfer calculations

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](#).

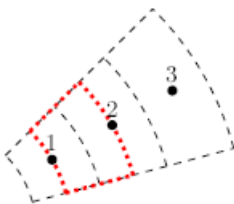


Figure 3: Partial pipe cross-sectional area with the staggered control volume outlined in red

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.

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 fluid temperature in the pipe to the specified value for the initial steady-state solution, but only if the flow is initially zero.


If the temperature override option is not used, then 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 for Junctions.....	220
Make Initial Flow Rate Guesses for Pipes.....	585

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.....	340
---------------------------------------	-----

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.

$$f = \frac{64}{Re} \quad \text{Laminar, } Re < 2300 \text{ (default)}$$

$$f = \left[1.14 - 2 \log \left(\frac{\epsilon}{D} + \frac{9.35}{Re \sqrt{f}} \right) \right]^{-2} \quad \text{Turbulent, } Re > 4000 \text{ (default)}$$

- **Absolute Roughness (default)** - The absolute average roughness height ϵ is specified directly.

- **Relative Roughness** - The roughness is specified as a ratio ϵ/D .
- **Hydraulically Smooth** - The ratio ϵ/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.....	70
Conventions for Specifying Basis Area.....	221
Nomenclature.....	463

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

General Junction Information	215
Junction Loss Models	224

General Junction Information

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.

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.....	183
Status.....	183
Common Junction Input Parameters.....	219

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.

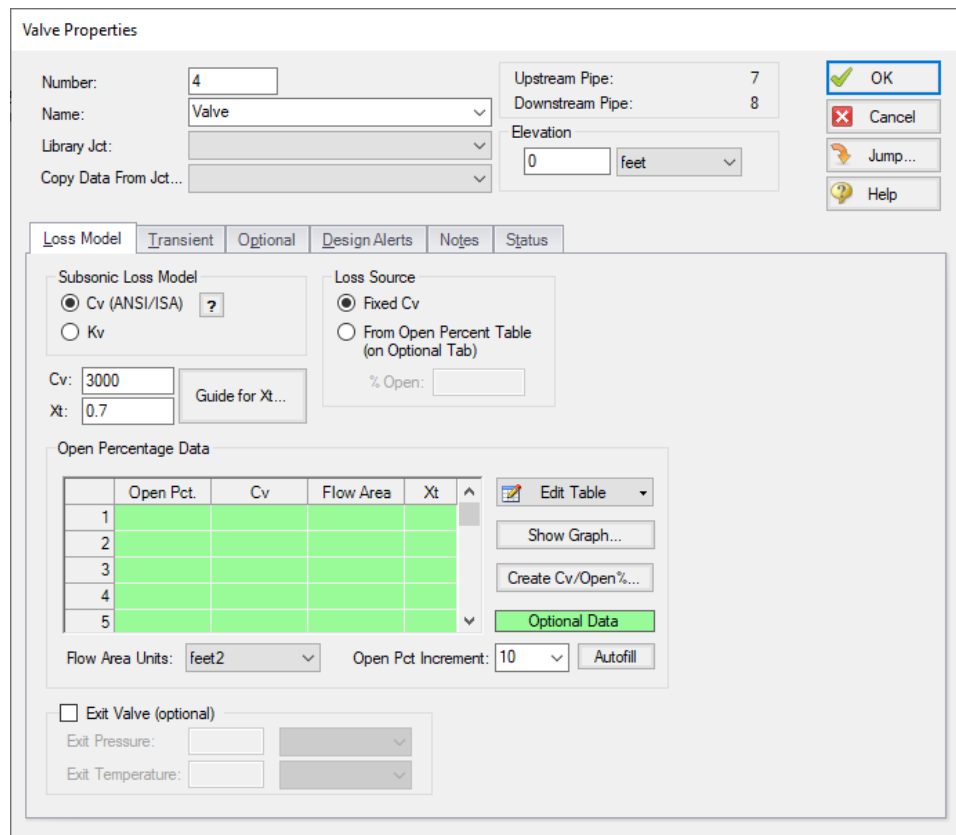


Figure 1: Properties window format #1 (fixed connecting pipe display).

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.

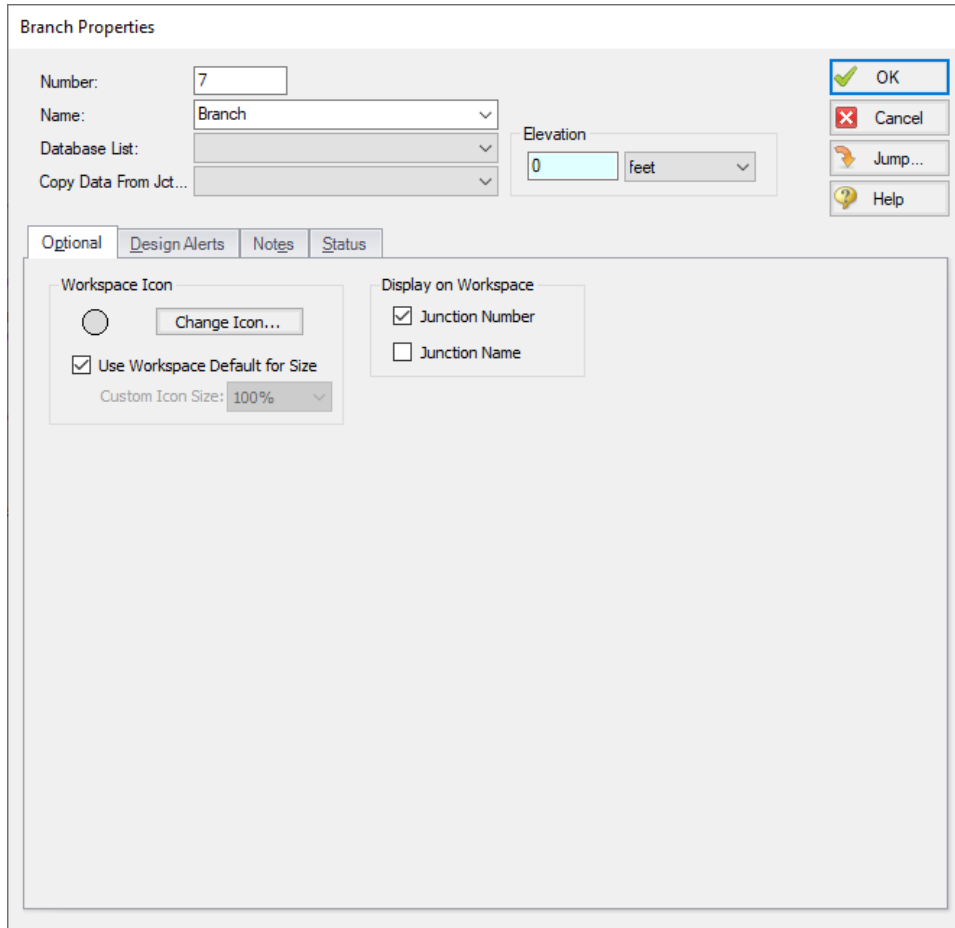


Figure 2: Properties window format #2

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 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.....	183
Status.....	183

Optional Input for Junctions

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

The junction icon can be changed to better represent a specific junction type. This is accomplished by clicking the Change Icon button. Custom images can be added by clicking the Add Image button. The size of the junction can be changed via [Workspace Layers](#).

Related Topics

Optional Input for Pipes.....	210
-------------------------------	-----

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.....	451
Fittings & Losses.....	190
Discharge Coefficient Loss Model.....	224
K Factor Loss Model.....	225
Cv/Kv and XT Loss Model.....	228
Resistance Curve Loss Model.....	231

Conventions for Specifying Basis 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 basis area is always the upstream pipe flow area. However, this can be modified by the user. In most junction Properties windows (on the Loss Model tab), the basis area can be specified as the upstream pipe, downstream pipe, user specified area, or user specified diameter.

However, some junction types, especially those that allow three or more connecting pipes, specify loss factors based on each connecting pipe flow area, whether upstream or downstream. An example of this is a [Wye](#) in which each of the connecting pipes has a different diameter. The idea of specifying a loss factor based on upstream flow area loses meaning because there may be more than one upstream pipe. In these cases the loss factor is referenced to the base area of the actual downstream pipe(s) where it connects to the junction. The types of junctions that do not always follow the upstream pipe area convention are the [Tee/Wye](#), [Branch](#), [Tank](#), [Assigned Pressure](#), and [Assigned Flow](#). Other exceptions are [Valves](#) and [Relief Valves](#) specified as exit valves, and [Orifices](#) specified as an exit.

Related Topics

Features for Modeling Irrecoverable Losses.....	70
Pressure Drop in Pipes - Detailed Discussion.....	212

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.

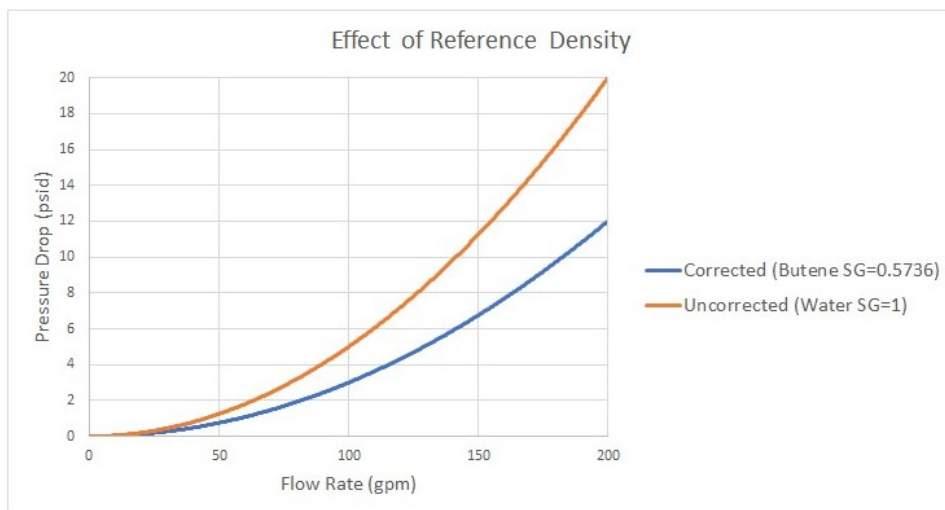


Figure 1: Visual comparison of Pressure Loss Curves for differing densities. Head curves would be identical for this component

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.

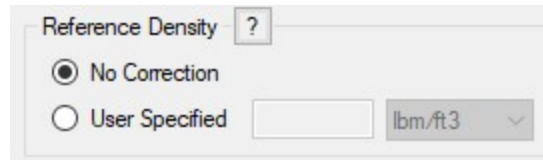


Figure 2: Reference Density Selection

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.....77

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:

- τ = 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.....	277
Valve.....	293

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.....	282
Relief Valve.....	285
Subsonic vs. Sonic Pressure Losses.....	509

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.....	243
General Component.....	281
Heat Exchanger.....	282
Control Valve.....	277
Relief Valve.....	285
Valve.....	293
Orifice Loss Model.....	283

Transient Sonic Choking	497
Sonic Choking Detailed Description	504
References	516

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.

$$(C_{path} + B_{path} P_{in}) A_{in} \left(\frac{P_{in} - C_{wave}}{B_{wave}} \right) = C_V N_6 F_P \left(1 - \frac{1}{3 X_{choked}} + \frac{1}{3 X_{choked}} \frac{P_{out}}{P_{in}} \right) \sqrt{(C_{path} + B_{path} P_{in}) (P_{in} - P_{out})}$$

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.....	463
Valve.....	293
Control Valve.....	277
Check Valve.....	254
Method of Characteristics.....	490
Sonic Choking.....	503
Sonic Choking Detailed Description.....	504
References.....	516

Related Blogs

[Making Sense of xT – Valve Loss in Compressible Flow](#)

Resistance Curve Loss Model

The resistance curve loss model is available in many different junctions. It is useful when measured flow vs pressure loss data is available for a component.

Clicking Enter Curve Data will open a new window where the input can be entered into a table. Several options for the Flow Parameter are available using the dropdown in the column header, and the appropriate units can be selected.

Fill As Quadratic (Using a Single Known Data Point)

It is not uncommon to obtain only a single data point from an equipment manufacturer. For example, the data might be 3 psid pressure drop at 1 lbm/sec. In this case, the known data point can be entered into the first row of the data table. Then, selecting Fill As Quadratic will automatically insert another two data points to be used with a quadratic curve fit. The first data point will be zero pressure loss at zero flow which is physically realistic for a static component. Note that elevation differences within a junction are accounted for externally by the Solver. The last data point will be a quadratic extrapolation using the zero point and the originally provided data point. Clicking Generate Curve Fit Now will create the curve fit constants and complete the resistance curve definition.

This method is often a reasonable approach to define a static component because the pressure drop characteristics of most equipment follows the square law. As the flow rate increases, the pressure drop increases with the square of the flow rate. Defining the component with a curve fit provides a robust solution where the junction can still provide a reasonable pressure loss even if the flow rate in the model is changed. This also aids in numerical convergence when running a transient simulation.

In our example, an increase in the flow rate to 2 lbm/sec would result in a pressure drop of 12 psid. Although this may not be exactly correct depending on the component, it is much better than assuming a flat 3 psid drop for all flow rates, which we know for certain is not correct.

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

Curve Fitting

[Curve fitting](#) can be performed via a polynomial or interpolated data. The Curve Fit Order can also be changed to achieve a better fit to the data, and the curve fit constants are displayed in a table.

Note: Unrealistic behavior could occur if the curve fit does not pass through the origin of the graph. This may negatively impact convergence when running transient simulations that have zero flow through the component at a point in time. See the [Troubleshooting Resistance Curves](#) topic for more information.

Reference Density

Options to correct for a reference density are available when using mass or pressure parameters. This is useful for when the pressure loss data was measured with a different working fluid than what is used in the model. See the [Reference Density](#) topic for more information.

Related Topics

Curve Fitting	77
Troubleshooting Resistance Curves	583
General Component	281

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.

Valve Properties

Number: 7 Upstream Pipe: 12
Name: Valve Downstream Pipe: None
Library Jct.: Elevation: 0 feet
Copy Data From Jct.:

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		

Repeat Transient

Xt Source
 User
 Open % Table
Fill Xt from table

OK Cancel Jump... Help

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.....	244
Assigned Pressure.....	246
Valve.....	293
Heat Exchanger.....	282
Compressor/Fan.....	255
Control Valve.....	277
Check Valve.....	254

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.

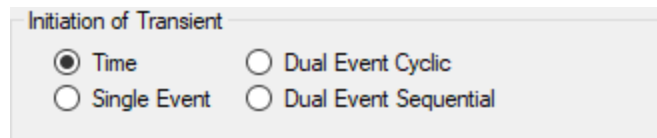


Figure 1: Type of transient initiation

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.

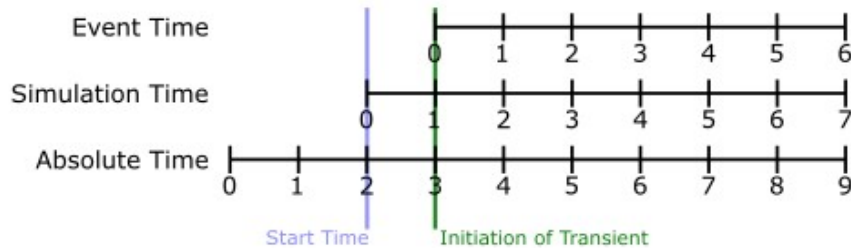


Figure 2: Time basis

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.

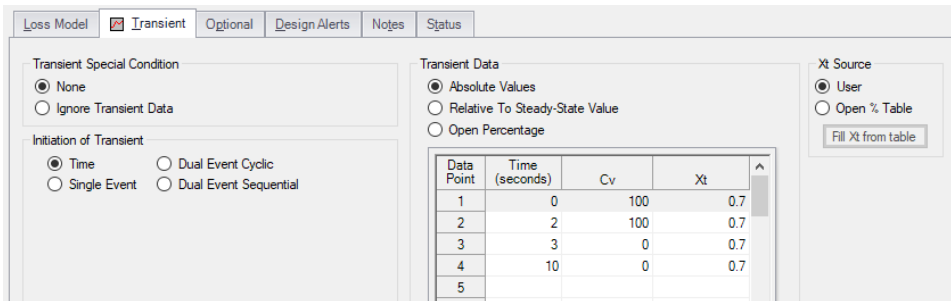


Figure 3: Defining a valve closure at a specific time

Single Event

Single Event initiated transients do not begin until a certain condition is met.

To completely define an Event, four parameters are required:

- **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.

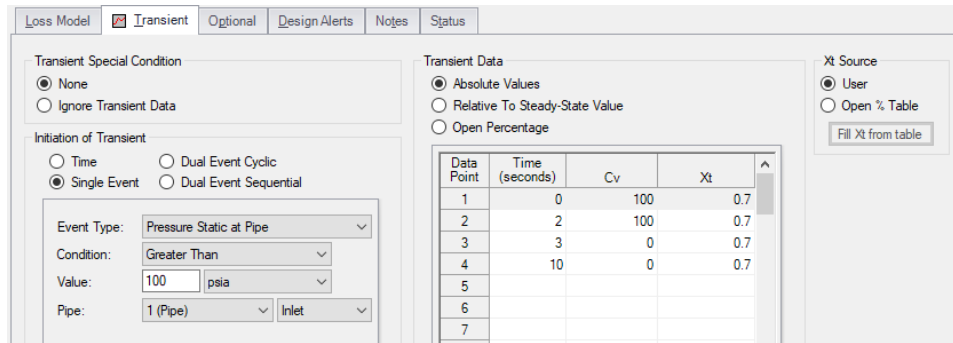


Figure 4: Event definition

Table 1: Event types

** Multi-Condition **	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.

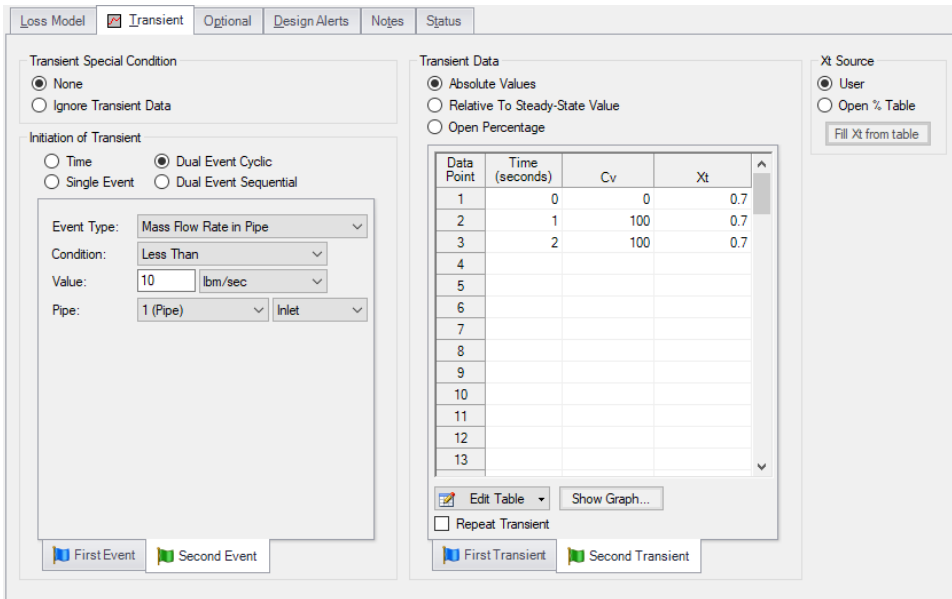


Figure 5: Dual Event set up. Note that a Second Event requires additional data for the Second Transient

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.

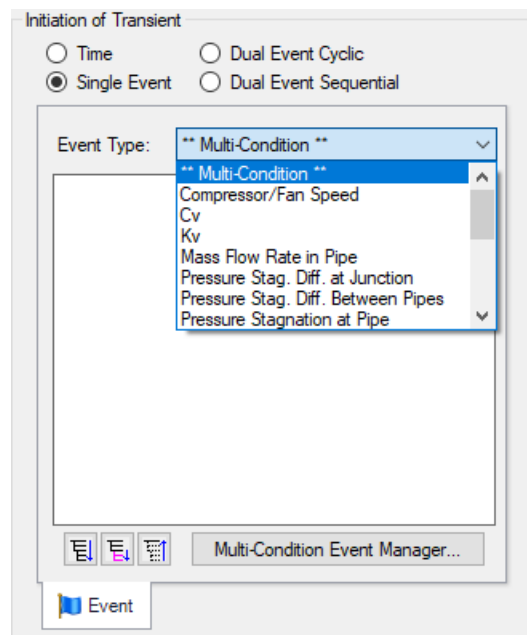


Figure 1: Selecting Multi-Condition Event Type

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.

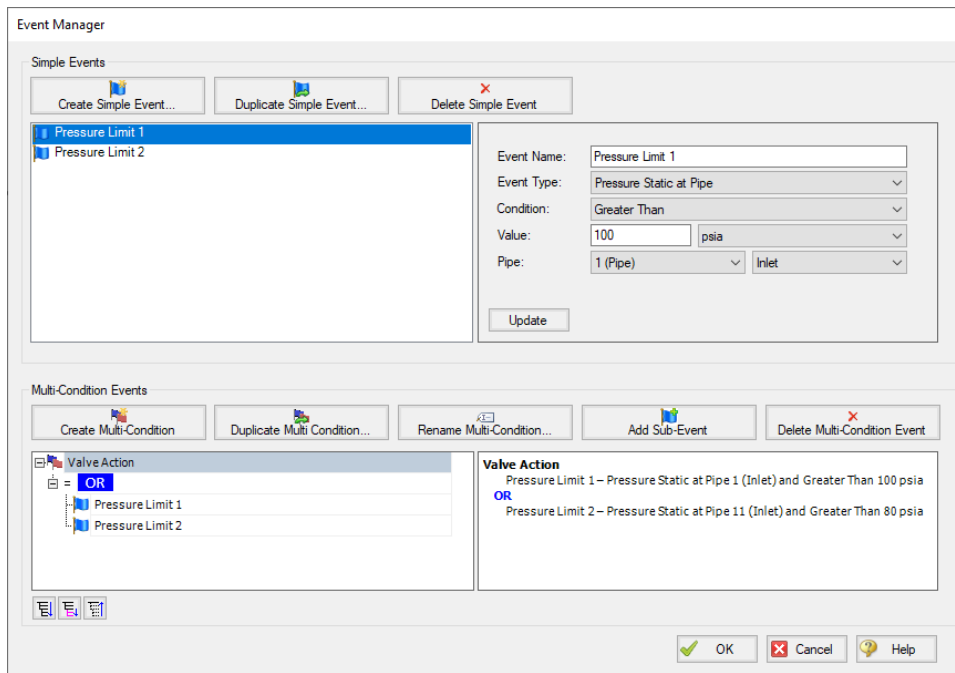


Figure 2: The Event Manager

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.

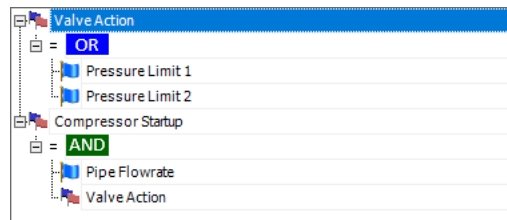


Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))

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.

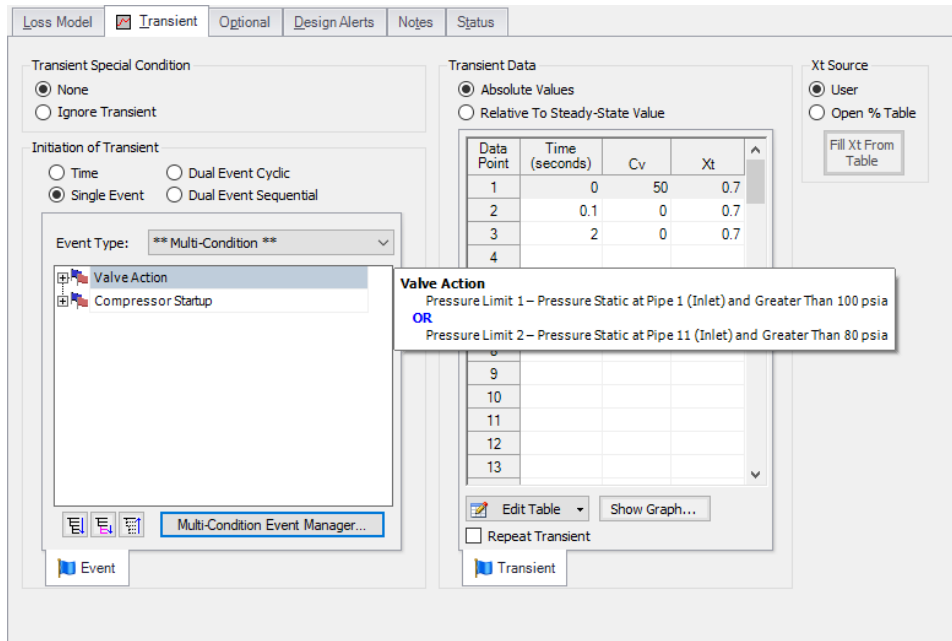


Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event

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.....219

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.....	463
K Factor Loss Model.....	225
References.....	516

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.....	219
Junction Transient Data.....	233

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.....	463
Method of Characteristics.....	490
Sonic Choking Detailed Description.....	504

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).....	535
Role of Pressure Junctions and How They Work.....	72
Tank.....	289
Common Junction Input Parameters.....	219
Junction Transient Data.....	233

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](#).

$$V_{i,new} = \frac{P_{known} - C_{\bar{wave}}}{B_{\bar{wave}}}$$

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.....	463
Method of Characteristics.....	490
Sonic Choking Detailed Description.....	504

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

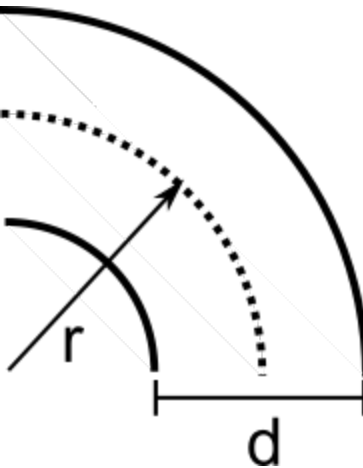
90 degree bends

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 **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:

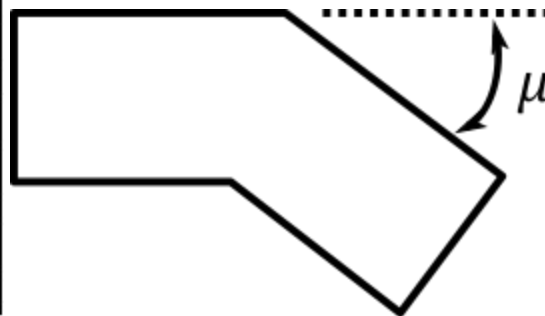
$$K_B = (n - 1) \left(0.25 \pi f_T \frac{r}{D} + 0.5 K \right) + K$$

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

μ (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.....516

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.....219

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_0 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.....	463
Method of Characteristics.....	490
Sonic Choking Detailed Description.....	504

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 modeling of the pressure added to a system by a compressor, and modeling many transient events involving compressors/fans. Because all compressors/fans are different, AFT xStream does not provide preset configurations for centrifugal compressors/fans. For these junctions, 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 several compressor models available:

- [Centrifugal \(Rotodynamic\)](#) - Represents a typical centrifugal compressor or fan where the head added to the fluid is related to the flow through the compressor via a compressor curve.
- [Positive Displacement \(Fixed Flow Rate\)](#) - 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.
- [Positive Displacement \(Reciprocating\)](#) - Represents a compressor in which pistons connected to one or more crankshafts are used to pressurize a volume of gas. Cylinders can be arranged by single cylinder, opposed cylinders, inline, and vee cylinder options. Single and double-acting configurations are available.

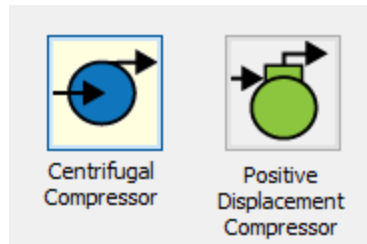


Figure 1: Compressor/Fan Models

Variable Speed

This allows the speed of the centrifugal compressor to vary.

- **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

Centrifugal compressors/fans and fixed flow rate positive displacement compressors allow for transients. This allows the user to directly specify how the [speed of the centrifugal compressor/fan](#) or the [flow rate of the fixed flow rate positive displacement compressor](#) 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](#).

Reciprocating compressors operate at constant speed and do not allow Speed vs Time transients.

Optional

There are several [optional parameters](#) that can be defined for a compressor/fan. The Workspace Icon can be adjusted, as is the case for most junctions.

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.

Related Topics

Common Junction Input Parameters	219
Junction Transient Data	233
Initiation of Transients	235

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.

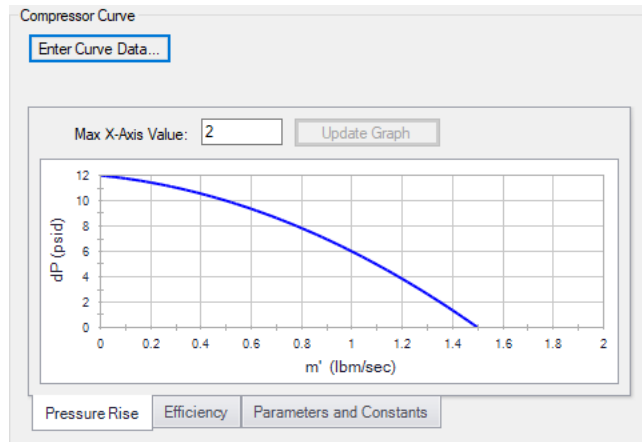


Figure 1: Standard Compressor Curve

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 isentropic and 100% efficient. If the compression process is isentropic, 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 Topics

Affinity Laws.....	263
Multiple Configurations.....	264
Transient - Speed vs. Time.....	268
Compressor/Fan Efficiency Data.....	276
Compressor Configuration.....	259

Related Blogs

[The Truth About Air Compressors](#)

[Modeling Fans, Blowers, and Compressors - Oh My!](#)

Compressor 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.

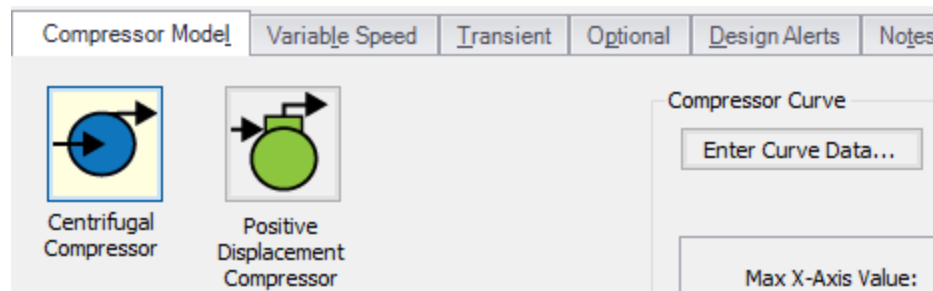


Figure 1: Enter Curve Data

Configuration Method

By default, a compressor curve is defined using the Simple configuration. This means that the user inputs only one compressor 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 at different speeds. See a detailed description in [Compressor/Fan Multiple Configuration](#). This feature is especially useful when multiple curves for a given compressor 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.

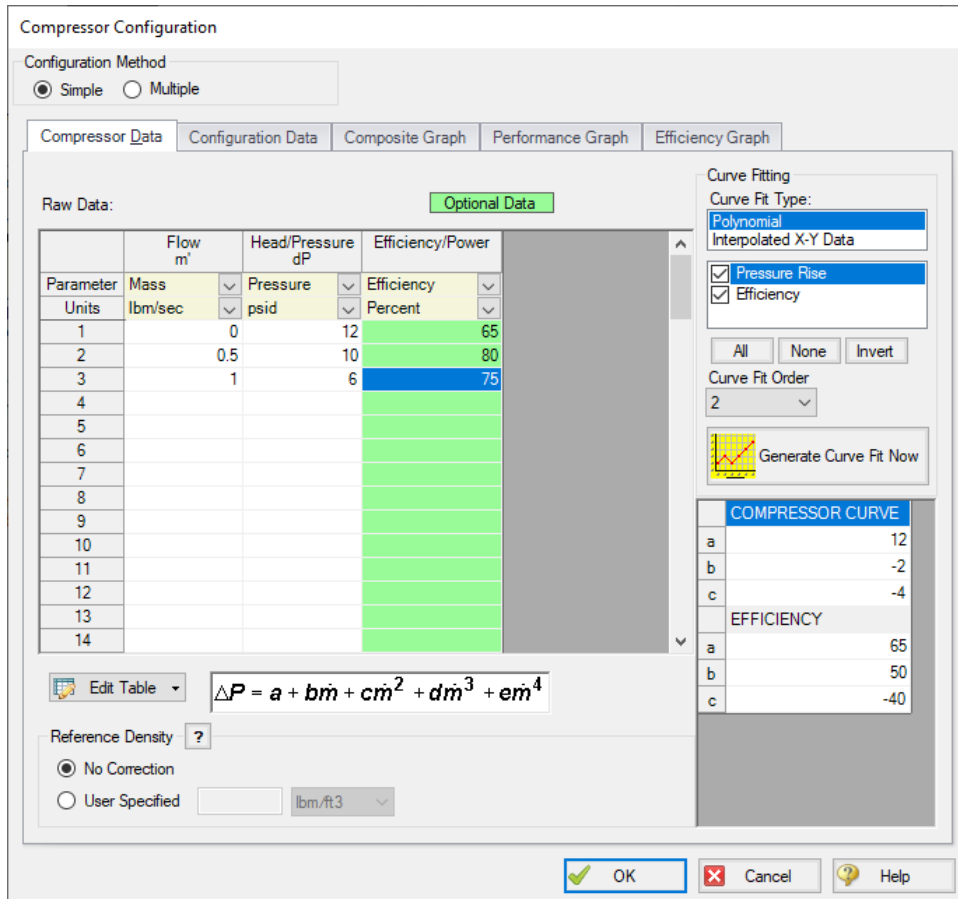


Figure 2: Compressor Data

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.

Compressor Configuration

Configuration Method
 Simple Multiple

Compressor Data Configuration Data Composite Graph Performance Graph Efficiency Graph

Best Efficiency Point (BEP)
 Calculate from Efficiency/Power Data
 User Specified

End Of Curve Flow Rate (optional)
 lbm/sec

Preferred Operating Region (POR)
 None
 User Specified
 Low End % of BEPQ
 High End % of BEPQ

Allowable Operating Region (AOR)
 None
 User Specified
 Low End lbm/sec
 High End lbm/sec

OK Cancel Help

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.
- **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.

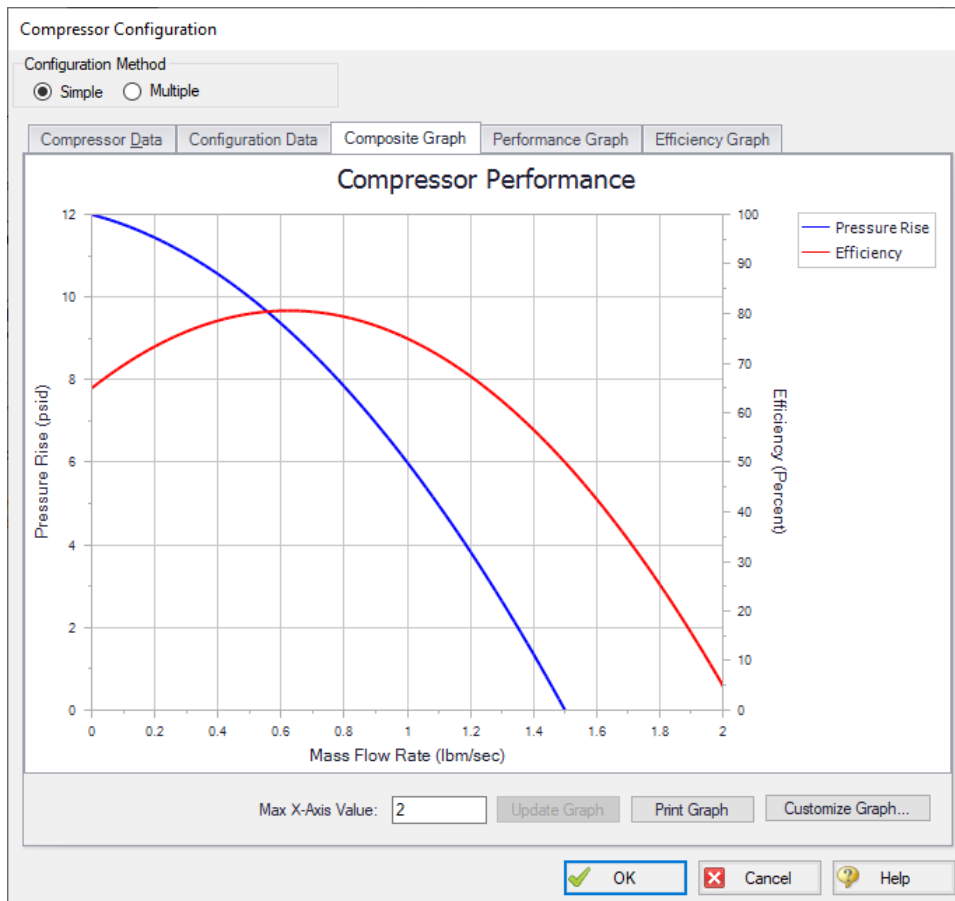


Figure 4: Composite Graph

Related Topics

Curve Fitting.....	77
Reference Density.....	221
Centrifugal Compressor.....	257
Affinity Laws.....	263
Multiple Configurations.....	264
Transient - Speed vs. Time.....	268
Compressor/Fan Efficiency Data.....	276

Related Blogs

[Modeling Fans, Blowers and Compressors - Oh My!](#)

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 Topics

Centrifugal Compressor.....	257
Multiple Configurations.....	264
Transient - Speed vs. Time.....	268
Compressor/Fan Efficiency Data.....	276
Compressor Configuration.....	259

Related Blogs

[To Affinity and Beyond!](#)

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.

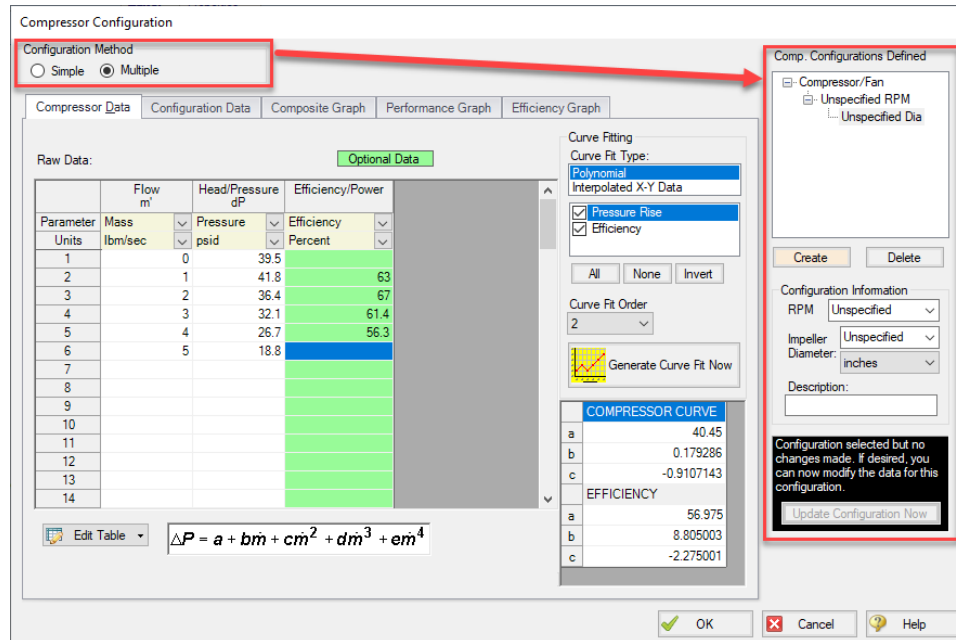


Figure 1: Multiple Configuration Method Selected

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:

Defining Junction Transients

Compressor Configuration

Configuration Method
 Simple Multiple

Compressor Data Configuration Data Composite Graph Performance Graph Efficiency Graph

Raw Data: Optional Data

Parameter	Flow m ³	Head/Pressure dP	Efficiency/Power
Units	Mass lbm/sec	Pressure psid	Percent
1	0	39.5	
2	1	41.8	63
3	2	36.4	67
4	3	32.1	61.4
5	4	26.7	56.3
6	5	18.8	
7			
8			
9			
10			
11			
12			
13			
14			

Curve Fitting
 Curve Fit Type:
 Polynomial
 Interpolated X-Y Data
 Pressure Rise
 Efficiency
 All None Invert

Curve Fit Order
 2

Generate Curve Fit Now

COMPRESSOR CURVE

a	40.45
b	0.179286
c	-0.9107143

EFFICIENCY

a	56.975
b	8.805003
c	-2.275001

Configuration Information
 RPM: 1100
 Impeller Diameter: 30 inches
 Description:

Configuration completed but not yet updated. Click the 'Update Configuration Now' button to keep this data.

Update Configuration Now

OK Cancel Help

Figure 2: Update Configuration

Clicking Update Configuration Now adds the Configuration to the Compressor Configurations listed in the top-right.

Compressor Configuration

Configuration Method
 Simple Multiple

Compressor Data Configuration Data Composite Graph Performance Graph Efficiency Graph

Raw Data: Optional Data

Parameter	Flow m ³	Head/Pressure dP	Efficiency/Power
Units	Mass lbm/sec	Pressure psid	Percent
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			

Curve Fitting
 Curve Fit Type:
 Polynomial
 Interpolated X-Y Data
 Pressure Rise
 Efficiency
 All None Invert

Curve Fit Order
 2

Generate Curve Fit Now

COMPRESSOR CURVE

a	40.45
b	0.179286
c	-0.9107143

EFFICIENCY

a	56.975
b	8.805003
c	-2.275001

Comp. Configurations Defined

- Compressor/Fan
 - 1100 RPM
 - 30 inches

Create Delete

Configuration Information
 RPM:
 Impeller Diameter: inches
 Description:

No configuration selected. Select an existing configuration or click the Create button to create a new one.

Update Configuration Now

OK Cancel Help

Figure 3: Current Compressor Configurations

To create an additional configuration, click Create.

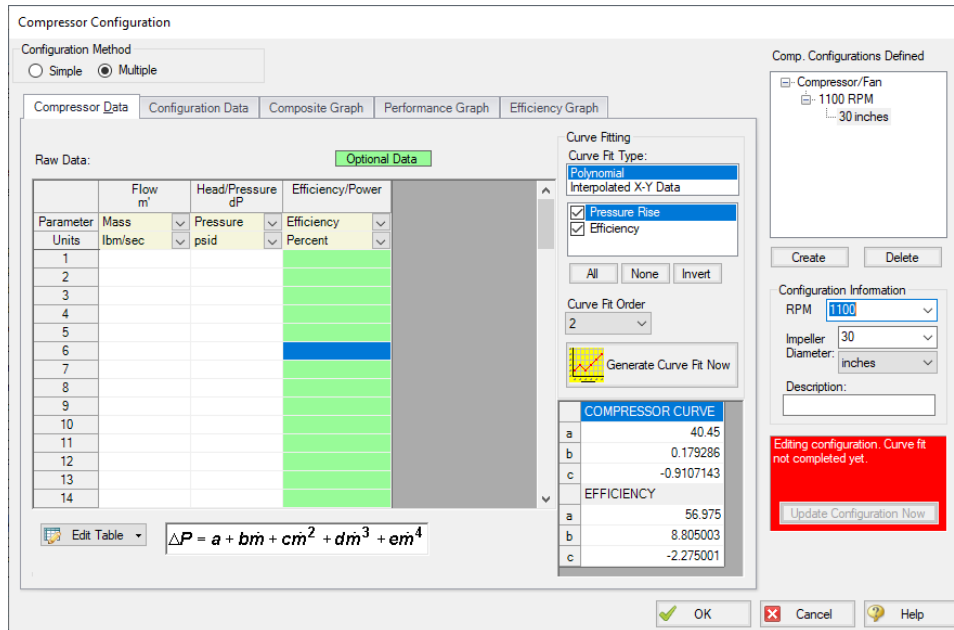


Figure 4: Creating a new Configuration

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

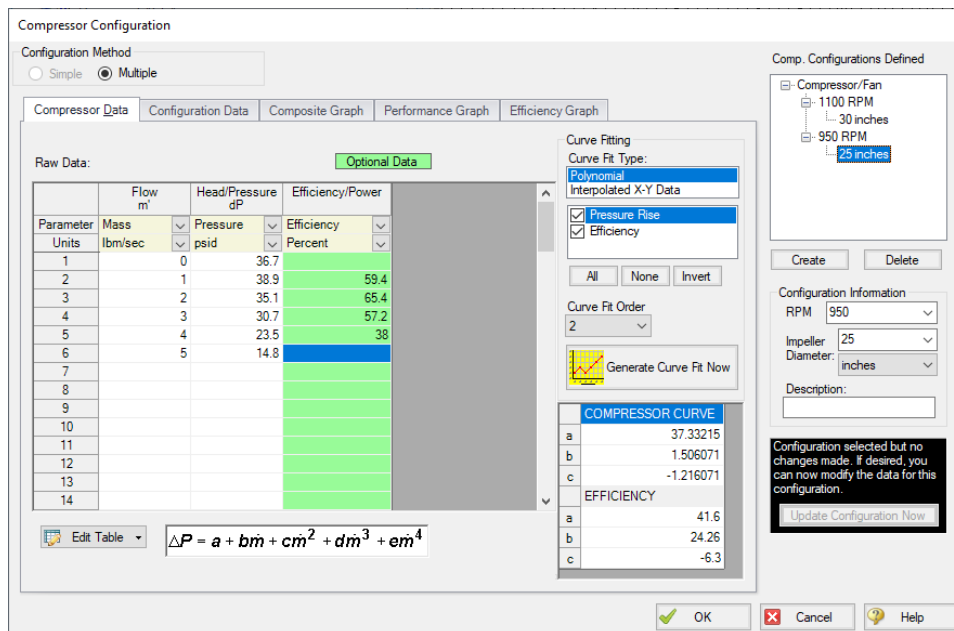


Figure 5: Editing a Configuration

Now that multiple configurations are available, they can be selected on the Compressor Model tab:

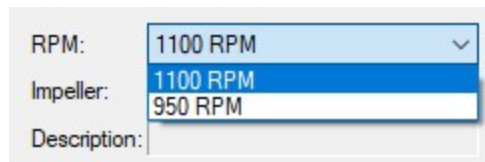


Figure 6: Selection of a Configuration

Related Topics

Centrifugal Compressor.....	257
Affinity Laws.....	263
Transient - Speed vs. Time.....	268
Compressor/Fan Efficiency Data.....	276
Compressor Configuration.....	259

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](#).

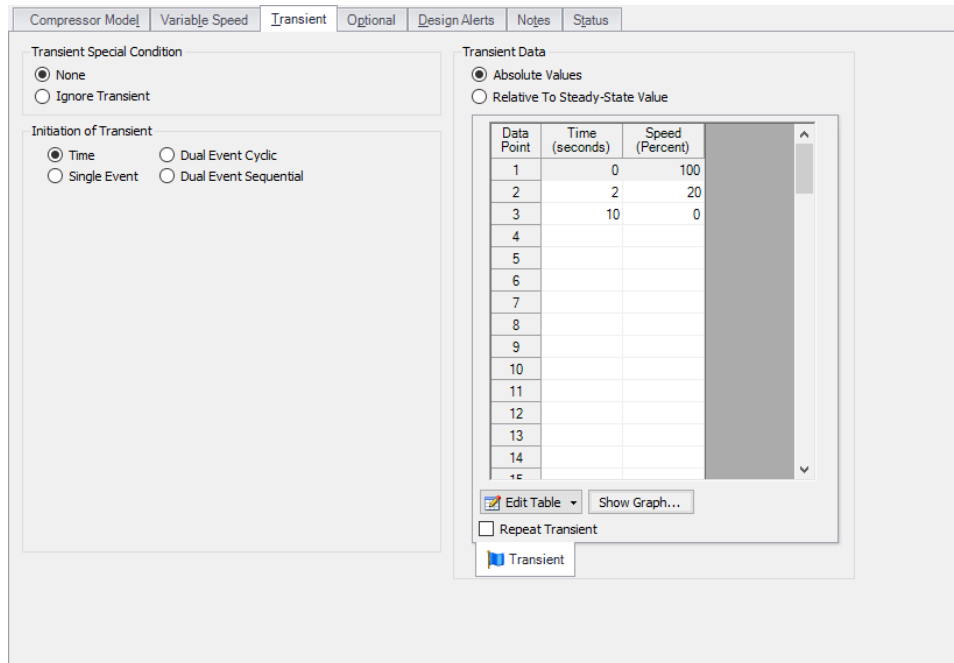


Figure 1: A Speed vs. Time transient

Related Topics

Initiation of Transients.....	235
Centrifugal Compressor.....	257
Affinity Laws.....	263
Multiple Configurations.....	264
Compressor/Fan Efficiency Data.....	276
Compressor Configuration.....	259

Positive Displacement Compressor

To model a positive displacement compressor, select the Positive Displacement option on the Compressor Model tab of the [Compressor/Fan Properties window](#). There are two subtypes of positive displacement compressors. Generally, the Fixed Flow Rate model is used for positive displacement compressors that do not function with a piston mechanism, or when configuration information for a reciprocating compressor is unknown.

- [Fixed Flow Rate Compressor](#) - 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.
- [Reciprocating Compressor](#) - Represents a compressor in which pistons connected to one or more crankshafts are used to pressurize a volume of gas. Cylinders can be arranged

by single cylinder, opposed cylinders, inline, and V cylinder options. Single and double-acting configurations are available.

Related Topics

Reciprocating Compressor.....	271
Pulse Setup Panel.....	103
Fixed Flow Rate Compressor.....	270
Transient - Setpoint vs. Time.....	275

Fixed Flow Rate Compressor

Fixed Flow Rate Compressors are a subtype of Positive Displacement Compressors that provide a fixed flow for steady-state analysis. This allows for a fixed flow to be defined in a straightforward manner. With a Fixed Flow Rate Compressor, a transient can be defined in terms of [flow setpoint varied over time](#).

Parameter

- **Volumetric Flow Rate** - Fix the volumetric flow rate at the inlet of the compressor. Due to compressibility effects, the outlet volumetric flow rate may differ.
- **Mass Flow Rate** - Fix the mass flow rate through the compressor.

Fixed Condition

- **Fixed Flow Rate** - 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.



Figure 1: Fixed Flow Rate Positive Displacement Compressor model

Related Topics

Positive Displacement Compressor.....	269
Reciprocating Compressor.....	271
Transient - Setpoint vs. Time.....	275

Related Blogs

[Modeling Fans, Blowers and Compressors - Oh My!](#)

Reciprocating Compressor

The reciprocating compressor is a subtype of [Positive Displacement Compressor](#) used to represent a compressor with flow driven by a crankshaft and pistons. This can be used to model both single and double-acting compressors with one or multiple cylinders in single, inline, V, or opposed configurations. The model incorporates the effects of cylinder geometry, crankshaft angles, and valve opening and closing angles into a thermodynamic model for gas expansion and compression. Gas compression and expansion within each cylinder is assumed to be isentropic and the valves are assumed to be lossless and open or close instantaneously. The options for configuring a reciprocating compressor are shown in Figure 1 below.

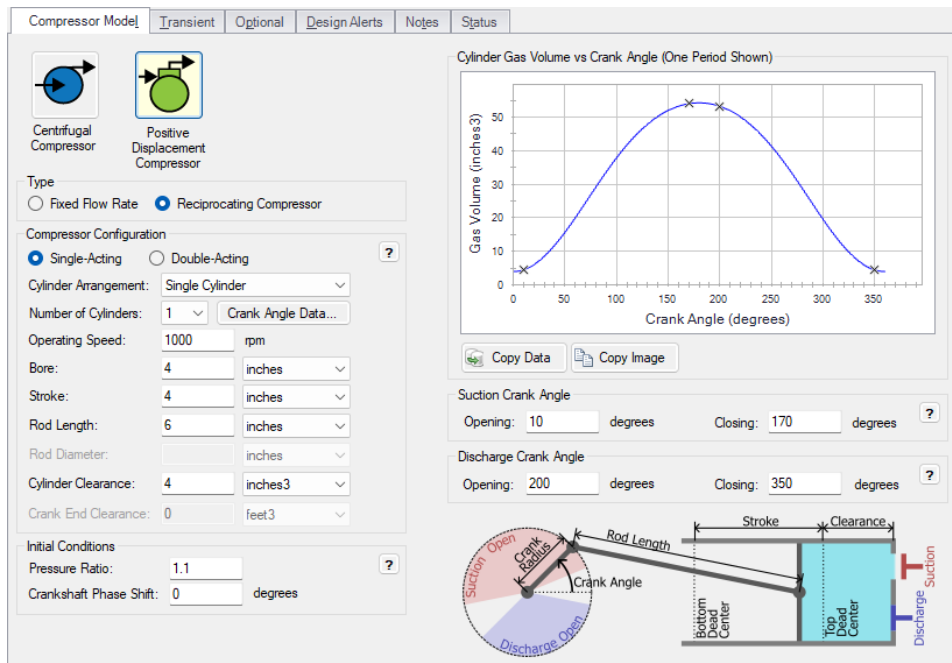


Figure 1: The Compressor Model tab of the Compressor/Fan Properties window for a reciprocating compressor

Compressor Configuration

The Compressor Configuration section of the Compressor Model tab has the following parameters:

- **Action**
 - **Single-Acting** - This is used for compressors with a single compression chamber per piston where the connecting rod is outside the compression chamber.
 - **Double-Acting** - This is used for compressors with two compression chambers per piston where the piston rod passes through the chamber that is on the side of the crankshaft. The connecting rod connects the piston rod to the crank arm. The volume of the piston rod is subtracted from its associated chamber volume. See the Rod Diameter parameter below.
- **Cylinder Arrangement**
 - **Single Cylinder** - The single cylinder option is used for single piston compressors or used when modeling each cylinder of a compressor separately, such as in multiple stage compressors or when building a model to capture manifold effects.
 - **Opposed** - An opposed cylinder arrangement has two banks of cylinders with a shared crankshaft separated from each other by 180 degrees. The cylinders in each bank operate in phase with their counterpart, and do not share crank arm angles between cylinder banks.

- **Inline** - Inline cylinders share a crankshaft and move in parallel to one another.
- **V** - A vee cylinder arrangement has two banks of cylinders with a shared crankshaft separated by an angle of 90 degrees. Crank arm angles are the same for each cylinder pair, resulting in a 90 degree phase offset between the first bank and the second bank.
- **Operating Speed** - The operating speed is the crankshaft speed in RPM for the compressor in the current scenario.
- **Bore** - The bore is the diameter of each cylinder.
- **Stroke** - The stroke is the distance traveled by a piston between TDC and BDC.
- **Rod Length** - The rod length is the length of the connecting rod between the rotational axes of the connections at the piston and the crank arm.
- **Rod Diameter (Double-Acting Only)** - The rod diameter is the diameter of the piston rod, which connects the connecting rod to the piston in a double-acting compressor. This term is used to calculate the volume of the piston rod, which is subtracted from the gas volume for the chamber nearest the crankshaft.
- **Cylinder Clearance** - The cylinder clearance is the volume of the compression chamber at TDC. In a double-acting compressor, this is for the compression chamber facing away from the crankshaft.
- **Crank End Clearance (Double-Acting Only)** - In a double-acting compressor, this is the volume of the crank end compression chamber when the piston is at BDC.

The **Crank Angle Data** button lists the crank angle for each cylinder relative to the compressor's reference angle. For opposed and V cylinder arrangements, cylinder numbers alternate between banks so that even-numbered cylinders and odd numbered cylinders are in separate banks. The cylinder offsets for single-acting compressors are shown in Table 1.

Table 1: List of crank angle offsets for each single-acting cylinder in order of cylinder number based on the cylinder arrangement and number of cylinders

Cylinder Arrangement	2	4	6
Opposed	0,0	0,0,90,90	0,0,120,120,240,240
Inline	0,0	0,180,180,0	0,120,240,240,120,0
V	0,90	0,90,180,270	0,90,120,210,240,330

Note: The head ends of double-acting compressors have the same offsets, except each cylinder also has a crank end that is offset from its head end by 180 degrees.

Initial Conditions

- **Pressure Ratio** - The pressure ratio is the ratio of the discharge stagnation pressure to the suction stagnation pressure for the reciprocating compressor. This is only used

directly for steady-state calculations, and must be greater than one.

- **Crankshaft Phase Shift** - This essentially offsets the phase of the crankshaft by shifting the reference crank angle. This can be used for offsetting the timing of multiple compressors from each other, which is useful when dealing with cylinder timings other than the defaults, for modeling reciprocating compressors with multiple compression stages, or modeling cylinders individually when modeling manifolds for a reciprocating compressor. This must be a positive number.

While the pressure ratio is not directly used in the transient solution, the steady-state solution is used to initialize the Transient Solver. The transient pressure ratio is dependent on the mass and energy entering the reciprocating compressor, the compressor configuration, and the stagnation pressures upstream and downstream of the compressor.

Disagreement between the transient and steady-state pressure ratios can contribute to artificial transients at the initial time steps of a model with reciprocating compressors.

Note: This can be reduced by observing the stabilized transient stagnation pressures and adjusting the steady-state pressure ratio accordingly. However, this may result in a lower pressure ratio than the real system due to reduced valve losses and will impact the accuracy of steady-state results.

Cylinder Gas Volume vs Crank Angle

The Cylinder Gas Volume vs Crank Angle graph shows one period of each compression chamber in the model, which is one period per cylinder in a single-acting compressor. This graph provides quick visual confirmation that the compressor is set up properly. The volume at each crank angle is determined from the cylinder and crankshaft geometry that are defined in the Compressor Configuration section, while the shift between the start of each compression chamber's cycle is determined by the Crank Angle Data values. Valve opening and closing events are displayed on each plot with X marks. The Crankshaft Phase Shift, if applicable, is shown as a vertical line at the crank angle defined by the shift.

The X-Y Data of the plot can be copied and will contain a series for each compression chamber and one series for valve timing events. An image of the plot can also be copied.

Crank Angles

Crank angles for valve events are defined relative to a crank angle of 0 degrees at TDC so that the suction stroke occurs before the discharge stroke, as shown in Figure 2.

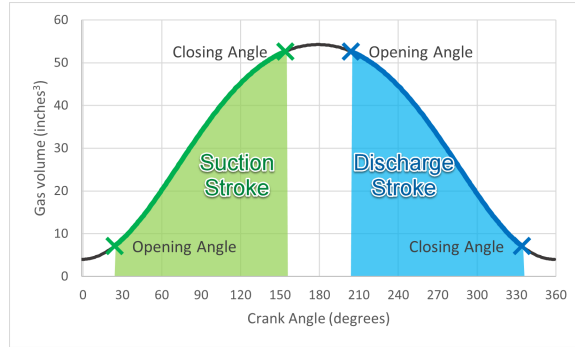


Figure 2: Illustration of suction and discharge strokes with crank angles

All cylinders within a compressor are assumed to obey the same valve timing relative to their individual TDC point (or BDC for the crank end of double-acting cylinders), and the valve angle set points must therefore be between 0 and 360 degrees.

Related Topics

Pulse Setup Panel.....	103
Positive Displacement Compressor.....	269
Fixed Flow Rate Compressor.....	270
Transient - Setpoint vs. Time.....	275

Transient - Setpoint vs. Time

When modeling a [Fixed Flow Rate 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.

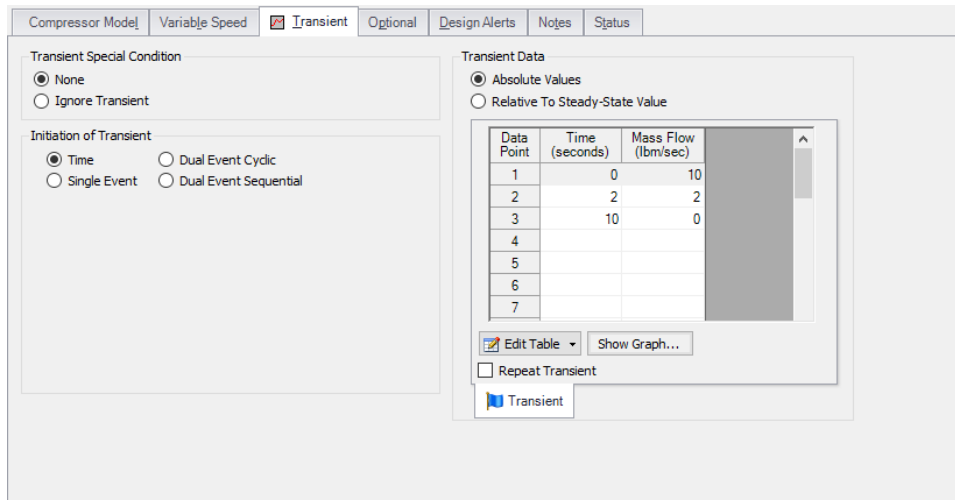


Figure 1: A Setpoint vs. Time transient

Related Topics

Initiation of Transients.....	235
Positive Displacement Compressor.....	269
Fixed Flow Rate Compressor.....	270
Reciprocating Compressor.....	271

Compressor/Fan Efficiency Data

Note: All efficiency data in the Compressor junction is treated as an isentropic efficiency.

Best Efficiency Point (BEP)

By defining a power or efficiency curve, the Best Efficiency Point of the compressor/fan can be found. Optionally, the calculated value can be overridden with a user-defined value on the Configuration Data tab on the [Compressor/Fan Configuration](#) window. It is generally desirable to operate compressors/fans as close to the BEP as possible. The Compressor/Fan Summary tab in the Output can display what the BEP is, as well as how far off (in % of BEP) of BEP the compressor/fan is operating in the model.

Nominal Efficiency

When modeling a fixed flow or fixed rise compressor/fan, you can optionally input a nominal efficiency as an approximation. This can be useful for sizing or design purposes, although more precise efficiency data should be entered when modeling a compressor/fan curve.

Related Topics

Centrifugal Compressor.....	257
Affinity Laws.....	263
Multiple Configurations.....	264
Transient - Speed vs. Time.....	268
Compressor Configuration.....	259

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 C_v , K_v , or a K factor (not applicable for PDCV valve types). When C_v , or K_v , is selected, the option becomes available to pull the value from an open percentage table on the Optional tab.

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.

Open Percentage Table

The Optional tab allows data to be entered for special control valve characteristics. Specifically, the valve C_v 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 C_v , 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 the following [Special Conditions](#): Fully Open - No Control, or Closed.

- **Closed** - No flow is allowed through the Control Valve.
- **Fully Open - No Control (can regain control)** - Starts in the Fully Open position and uses the Loss When Fully Open value from the Control Valve Model tab of the Properties window (or Control Setpoint for the PDCV valve type). Throughout the simulation, the Control Valve can regain control if applicable.
- **Fully Open - Never Control** - Remains in the Fully Open position for the entire simulation and uses the Loss When Fully Open value from the Control Valve Model tab of the Properties window (or Control Setpoint for the PDCV valve type).

Related Topics

Common Junction Input Parameters.....	219
K Factor Loss Model.....	225
Cv/Kv and XT Loss Model.....	228
Junction Transient Data.....	233

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.....	463
Method of Characteristics.....	490

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.....	219
---------------------------------------	-----

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.

$$P_{i,new} = C_{\bar{wave}}$$

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.

$$P_{i,new} = C_{path}^+ + B_{path}^+ P_{i,known}$$

Related Topics

Nomenclature.....	463
Method of Characteristics.....	490
References.....	516

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.....	77
Common Junction Input Parameters.....	219
K Factor Loss Model.....	225
Resistance Curve Loss Model.....	231

Troubleshooting Resistance Curves	583
Reference Density.....	221

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 Topics

Curve Fitting	77
Common Junction Input Parameters.....	219
K Factor Loss Model.....	225
Junction Transient Data	233
Reference Density.....	221

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.....	219
Discharge Coefficient Loss Model.....	224
Subsonic vs. Sonic Pressure Losses.....	509

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):

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}} \right)^2 \left[0.707 \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375} + \left(1 - \frac{A_{orifice}}{A_{down}} \right) \right]^2$$

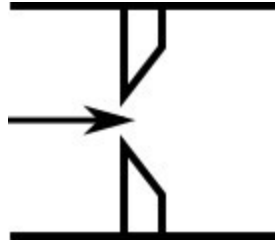


Figure 1: Sharp-edged orifice (in this case $A_{up} = A_{down}$)

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)

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}} \right)^2 \left[K' \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.75} + \left(1 - \frac{A_{orifice}}{A_{down}} \right)^2 + 2 \left(1 - \frac{A_{orifice}}{A_{down}} \right) \sqrt{K' \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375}} \right]$$

where

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge} / D_{orifice}}$$

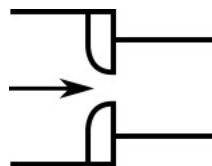


Figure 2: Round-edged orifice with different upstream and downstream pipe areas

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

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge} / D_{orifice}}$$

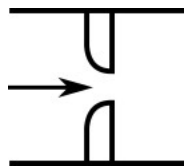


Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

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.....	225
References.....	516

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.....	219
Discharge Coefficient Loss Model.....	224
Subsonic vs. Sonic Pressure Losses.....	509
K Factor Loss Model.....	225
Junction Transient Data.....	233
Initiation of Transients.....	235
References.....	516

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.....	490
--------------------------------	-----

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.....	219
Subsonic vs. Sonic Pressure Losses.....	509

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):

$$K_{up} = 1.3 \left(1 - \frac{A_{screen}}{A_{up}} \right) + \left(\frac{A_{up}}{A_{screen}} - 1 \right)^2$$

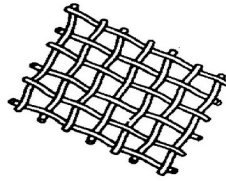


Figure 1: Circular metal wire screen

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 \(2007\)](#).

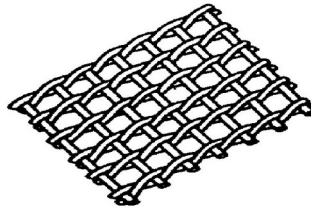


Figure 2: Silk thread screen

Related Topics

References.....516

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).....	535
Role of Pressure Junctions and How They Work.....	72
Assigned Pressure.....	246
Common Junction Input Parameters.....	219

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.

$$\rho_o, h_o = f(P_o, T_o) \quad (1)$$

$$m = \rho_o V \quad (2)$$

$$E = V (h_o \rho_o - P_o) \quad (3)$$

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.....	463
Method of Characteristics.....	490
Sonic Choking Detailed Description.....	504

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.....219

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..... 490

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. 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. You also have the ability to change the loss with time to open or close the valve in full or in part.

Loss Source

In order to completely define a valve junction, you must specify the valve loss source by clicking on the appropriate radio button under Loss Source:

- **User Specified** - This is the best option if manufacturer data is known. Manually input the loss information using one of the Loss Models listed in the next section.
- **From Characteristic** - This option is useful if no information about the valve is known aside from the type, and an estimate needs to be made. Automatically calculate C_v based on the selection of a common valve geometry type and a Full Open C_v value. With the exception of the User Defined characteristic from the dropdown, the window will automatically populate an Open Percent vs C_v data table by scaling the shape of the characteristic curve to fit the provided Full Open C_v . Then, choose an Open Percentage under the Loss Parameters area to automatically calculate a C_v . The data for the characteristic shapes comes from [Val-Matic 2009](#).
- **Handbook** - Uses standard valve loss models from [Idelchik, Miller, and Crane](#).

Loss Model

Depending on the selection of the Loss Source, various loss models will be displayed:

- [Flow Coefficient](#) - C_v for English units or K_v for Metric units, and X_T (the terminal pressure drop ratio) will be required to check for [sonic choking](#)
- [K Factor](#)
- [Resistance Curve](#)

A special **Reduced Port** configuration is available when using the Handbook loss source, and requires the selection of a valve type and several geometric inputs. Once specified, a K Factor from the Handbook source will be calculated.

Exit Valves

Valves that are used as exit flow control mechanisms are specified as Exit Valves. These valves require an associated ambient pressure definition (also called a back pressure). Exit valves can only be connected to a single upstream pipe. To specify an Exit Valve, select the Exit Valve checkbox, and input the required Exit Pressure and Exit Temperature.

Sonic CdA & Optional Losses

The optional inputs in this expandable section depend on which loss model is selected:

- **Sonic CdA** - Optional input that is only available for the K Factor, Resistance Curve, and Reduced Port loss models. This parameter describes the effective area restriction in the valve for the purpose of calculating [sonic choking](#). In most cases, information on the C_dA must be obtained from the manufacturer or test data.
- **Basis Area** - See the topic on [Specifying Basis Area](#).

Open Percentage Data

Optional data may be entered for special valve characteristics. Specifically, the Open Percentage vs C_v , X_T , and Flow Area can be specified. This data does not affect the Solver'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 C_v , and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point. A graph of the open percentage, C_v , and flow area data can be displayed by selecting the Characteristic Graph tab below the table.

The Characteristic Source parameters can be used to create an Open Percent vs Cv curve that follows the shape of several common valve types including Ball, Butterfly, [Equal Percentage](#), Globe, Linear, Plug, and Quick Opening.

Transient Data

On the Transient tab, a transient C_v and X_T vs time or K Factor and C_dA vs time table can be entered to simulate the opening or closing of the valve. If the optional open percentage table is

entered, it can also be used to specify an open percent vs time transient. For a closed valve, the *K* Factor is infinite. To account for this, a *K* Factor of -1 (negative one) can be used in the transient data table. For more information on transient data, including event transients, see the [Junction Transient Data](#) topic.

Special Conditions

The [Special Condition](#) on the Optional tab allows the valve to be specified as Closed, which is useful to turn off a valve while retaining the input data for future use.

Related Topics

Common Junction Input Parameters.....	219
K Factor Loss Model.....	225
Cv/Kv and XT Loss Model.....	228
Junction Transient Data.....	233
References.....	516

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.....	219
Subsonic vs. Sonic Pressure Losses.....	509

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 Primary Window tabs, and descriptions of the customizable parameters in the User Options menu.

Contents

Menus and Toolbars	297
Printing	308
Primary Window Overview	309
Status Bar	311
Quick Access Panel	311
Model Status Light	322
Solution Progress Window	322
Workspace Window	322
Model Data Window	343
Output Window	347
Graph Results Window	359
User Options Overview	379

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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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 Only) - Copies selected [Workspace](#) 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 [Joining 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.
- **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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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 an 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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

Tools Menu

- **Scenario Manager** - Opens the [Scenario Manager](#) window.
- **Last Scenario** - Switches back to the previously loaded [scenario](#).
- **Compare Scenarios** - Quickly sets up a [comparison](#) between the current scenario and a related scenario.
- **Scenario Comparison Tool** - Opens the [Scenario Comparison Tool](#) window, allowing users to compare differences between any two or more scenarios in a model.
- **Integration Query** - Opens the 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 what content can be printed from the [Output](#) window.
- **Output Language** - Selects the display language for certain phrases in the Output window, Graph Results window, and the labels in Workspace Layers.
- **Model Data Control** - (Model Data Only) - Opens the [Model Data Control](#) window.
- **Model Data Print Content** - (Model Data Only) - Configures what content can be printed from 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.
- **Design Alerts** - Opens the [Design Alerts](#) window.
- **Multi-Condition Events** - Opens the [Event Manager](#) window which allows users to make transients that are triggered by more than one parameter condition in the model.
- **Fluid Property Viewers** - Gives options to open either the [SteamCalc Viewer](#) or [Chempak Viewer](#). These are separate executables bundled with the software. Chempak Viewer requires a license.
- **Activate Modules** - Opens the [Modules panel](#) in Analysis Setup where optional add-on 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 but depends on the specific company license.
- **Model Diagnostics** - Opens the [Model Diagnostics](#) window which provides troubleshooting information.
- **Custom Units** - Allows custom unit names to be enabled or disabled. Custom unit names can be defined in the User Options under the [Unit System](#) section.
- **User Options** - Opens the [User Options](#) window.

Related Topics

Workspace Window.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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).

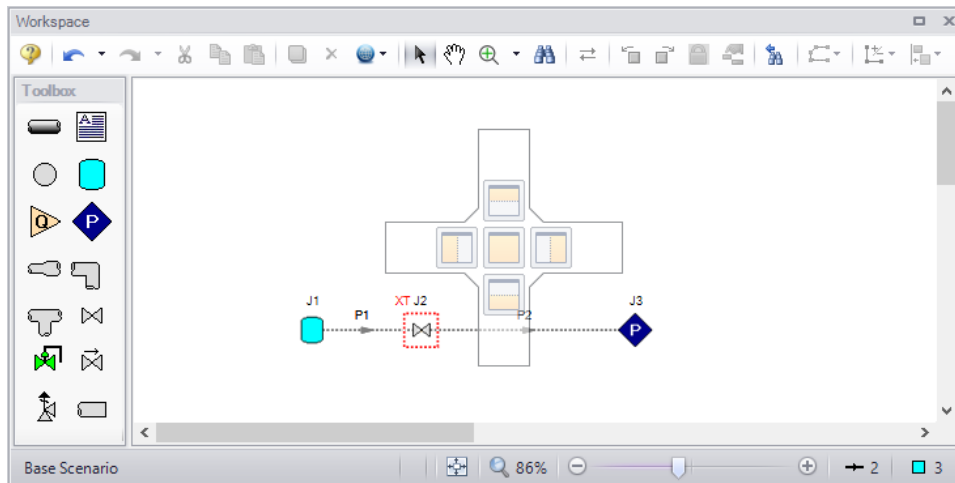


Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time

- **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.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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.

Related Topics

Workspace Window.....	322
Model Data Window.....	343
Output Window.....	347
Graph Results Window.....	359

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
- **Quick Start Video** - Opens ~15 minute quick start video on aft.com 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.....	322
Model Data Window.....	343

Printing

Printer Page Layout

The Printer Page Layout tool enables users to control how a model on the Workspace will be split across pages in the Print Preview window. To access this tool click the printer icon in the bottom-right of the Workspace. The Printer Page Layout options should appear.

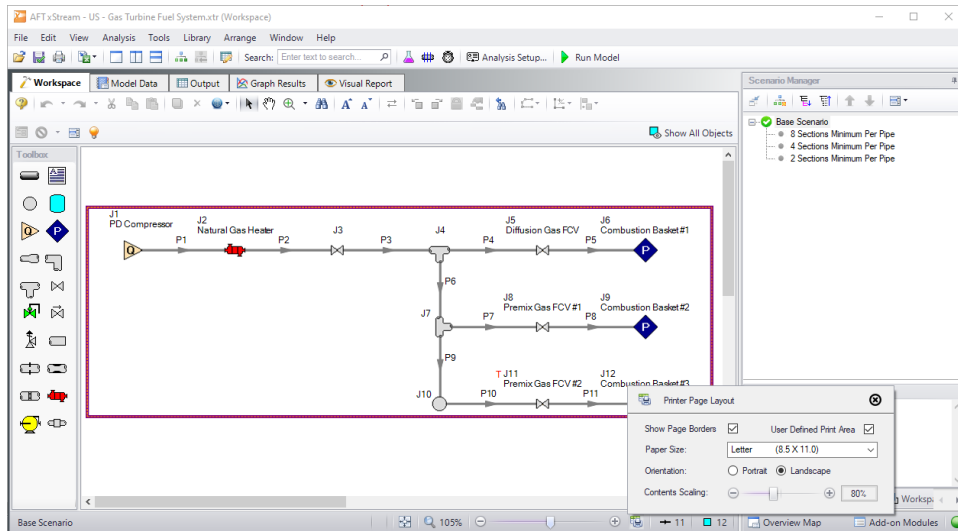


Figure 1: The Printer Page Layout options control how the model is split between pages on the Print Preview

- **Show Page Borders** - When checked, a dotted yellow outline will appear around sections of the model. If the dotted yellow lines form a single box around the model, this means the entire model will fit on a single page. If the dotted yellow lines form multiple boxes, regardless of the size of the boxes, the model will be divided such that everything in a box will appear on its own page once the Print Preview is used.
- **User Defined Print Area** - When checked, a purple box will be drawn around the entire model. Clicking on this box will cause sizing handles to appear. Using these sizing handles along with the "Show Page Borders" enables users to customize more precisely what appears on each page. Additionally, a user can shift the model on the Workspace in order to place specific sections of the model on specific pages.
- **Paper Size** - The paper size is a drop down with a selection of page dimensions based on the page sizes. The list of page sizes is determined by whatever printer page sizes are supported by your printer. Selecting one of these sizes will affect the size of the paper the model will be printed on. As such, the size of the boxes formed by the dotted yellow lines from the "Show Page Borders" will adjust in size.
- **Orientation** - The portrait setting will flip the page such that the longest side from the paper size will align vertically on the page. The Landscape setting will flip the page such that the longest side from the paper size will align horizontally. As such the boxes created

from the "Show Page Borders" option will follow suit.

- **Contents Scaling** - This feature determines how large the model will appear on each page. As the slider moves, it will appear as though the boxes from the "Show Page Borders" are shrinking or growing. This does not affect the dimensions of the pages. Instead, when the Print Preview is used the page size will remain the same, but the objects in each box will fit to the pages as displayed on the Workspace.

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 window to be compressed onto the specified number of pages. This is a convenient feature for creating reports of your model. The Print Preview supports whatever printer page sizes are supported by your printer, so the 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. The "Selected on Workspace" option works with Workspace printing by selectively printing the pipes and junctions currently selected on the Workspace.

By default, the junction icons on the Workspace will be printed with the same colors as displayed on the screen. It also prints the background color as specified in the User Options > Workspace > Colors and Fonts. 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 picture from the [File menu](#), this picture will be included in the printout.

Finally, if you have a grid displayed on the Workspace, the grid will be included in the printout.

Page options allow the use of a company logo, user comments, titles, and different options for borders.

Primary Window Overview

AFT xStream has several Primary Windows that work in an integrated fashion. Each is located on a separate, moveable tab. You work exclusively from one of these windows at all times. For this reason, they are referred to as *Primary Windows*.

You can change between the Primary Windows by selecting them from the [Window menu](#) or by clicking the appropriate tab within the AFT xStream window.

- **Workspace**

The Workspace Primary Window is where the user creates the graphical representation of the system. Engineering data is input on pipe and junction property windows which are opened by double-clicking the objects.

The Workspace Layers are also located on this Primary Window and allow the user to display both input and output data superimposed on the pipe system schematic.

- **Model Data**

The Model Data Primary Window summarizes the engineering input data from the Workspace into categorized tables. This allows for easy review of the system inputs, and double clicking cells will open the pipe or junction properties for quick editing.

- **Output**

The Output Primary Window offers a powerful and diverse range of features for reviewing analysis results which are organized into categorized tables and lists. The Output allows the user to easily compare multiple design scenarios with color coded rows, details errors or design requirement violations, and prepare the results for documentation.

- **Graph Results**

The Graph Results Primary Window allows the user to create a wide array of graphs with various parameters which are integral to gaining a deeper understanding of the pipe system's behavior. The user can adjust various formatting settings such as axis ranges, labels, colors, and create dual y-axis or stacked graphs. The settings can be saved into templates called Graph List Items so that the graphs can be easily reproduced.

Primary Window	Can Display Input	Can Display Output
Workspace	✓	✓
Model Data	✓	
Output		✓
Graph Results		✓

Related Topics

Overview of AFT xStream.....	17
xStream Learning Resources.....	33
Overview of Fluid Transients.....	37
Philosophy of Computer Modeling.....	71

Status Bar

AFT xStream's Status Bar displays contextual information about the current scenario or operation.

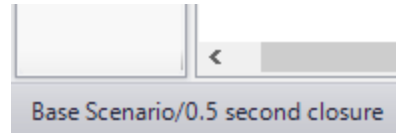


Figure 1: Current scenario displayed in the Status Bar

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.

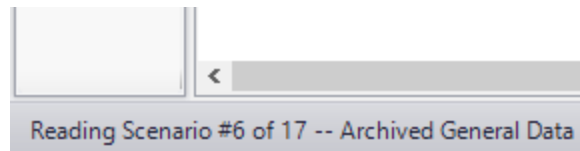


Figure 2: Status Bar displaying progress when opening a model

Quick Access Panel

The Quick Access Panel, as shown in Figure 1, contains several useful features:

- **Scenario Manager** - Configure the scenarios and variations of the model
- **Properties tab** - Review pipe/junction inputs or output, and edit annotations
- **Workspace Layers** - Superimpose input and output parameters on top of the Workspace model
- **Overview Map** - Easily navigate large models
- **Activate Add-on Modules** - Shortcut to the Add-on Modules group in the Analysis Setup
- **Model Status Light** - Green/red light indicating if all required input is present and if the Solver is ready to run

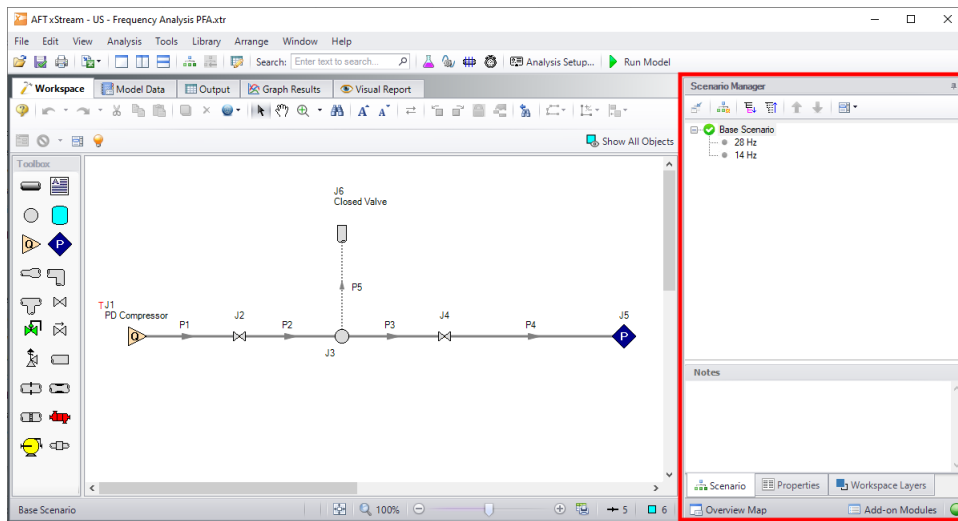


Figure 1: The Quick Access Panel

Scenario Manager

By clicking on the Scenario tab in Figure 2 below, you have the full scenario customization ability as you would with the [Scenario Manager](#). You can create child scenarios, expand or collapse various families of scenarios, and use the many other Scenario Manager features here as well.

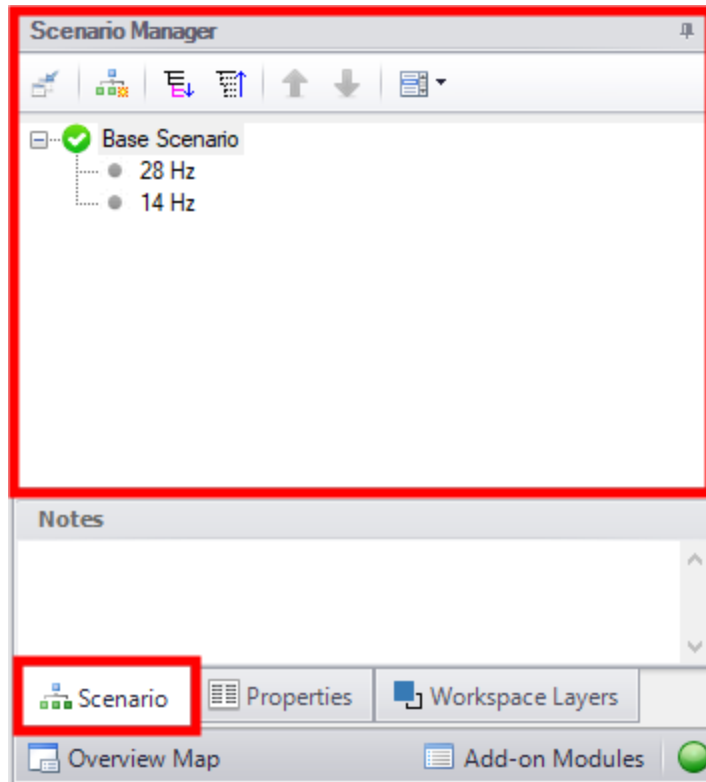


Figure 2: The Scenario Manager

Properties Tab

As seen in Figure 1 or 2, 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 the "Properties" tab. The tab's name will dynamically update depending on which pipe/junction is selected. This will display the Object Property data as well as any Output data if the model has been run. Figure 3 below shows an example of what the Properties tab would look like when clicking on a pipe on the Workspace after it has been defined and output was created.

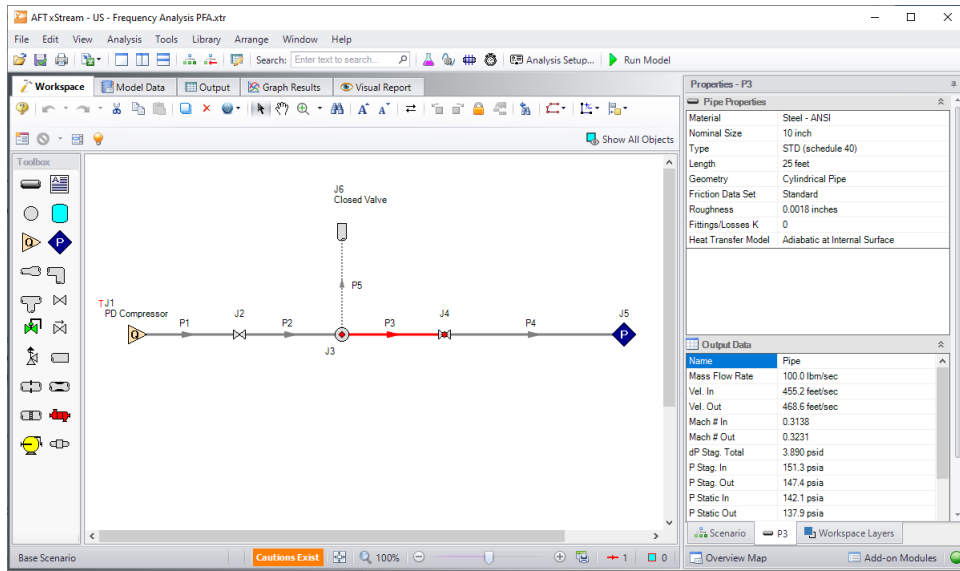


Figure 3: Viewing the Pipe/Junction Properties and Output Data from the Quick Access Panel

[Annotations](#) can be edited by selecting an annotation on the Workspace which will bring up their properties in this tab where they can then be edited.

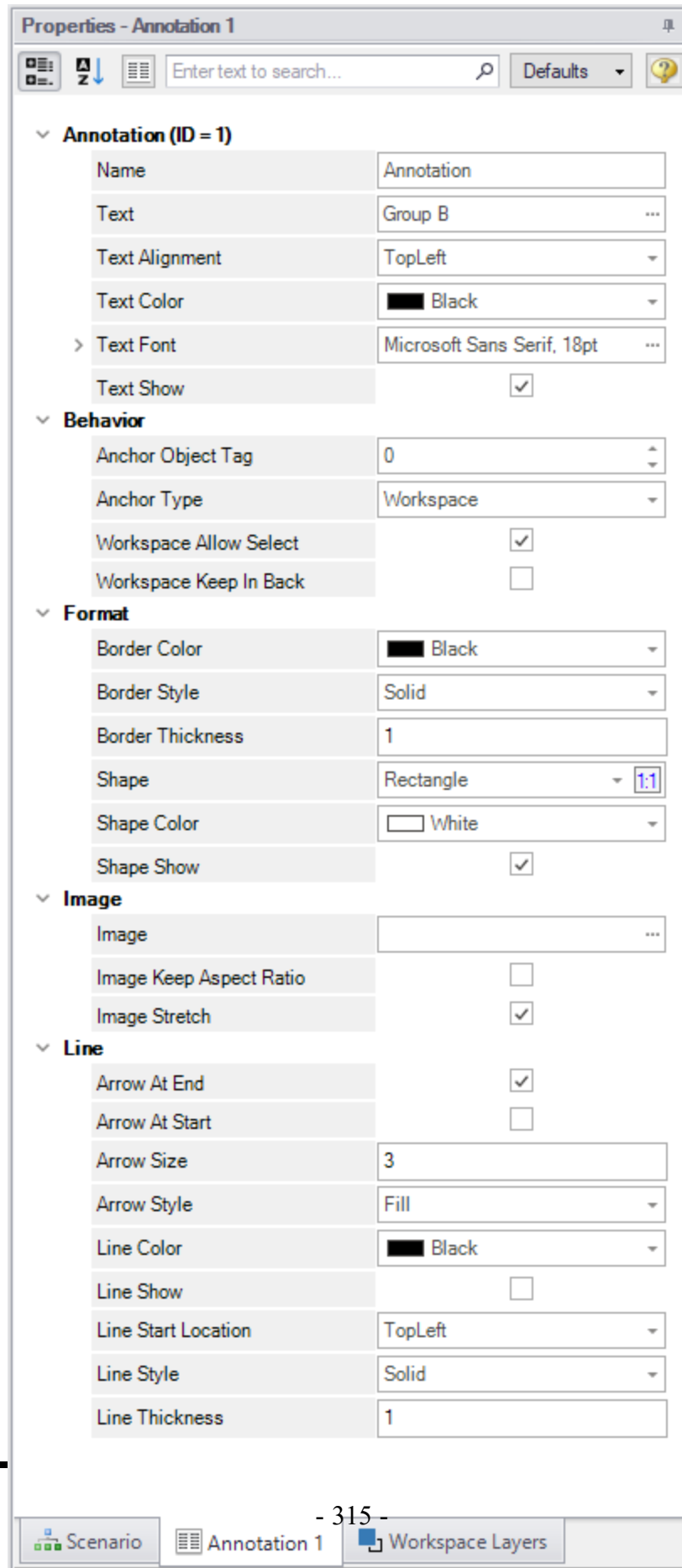


Figure 4: Annotation view for the Properties in the Quick Access Panel

Workspace Layers Tab

The [Workspace Layers](#) tab contains settings relating to object visibility, labels, superimposing input/output data, and color maps.

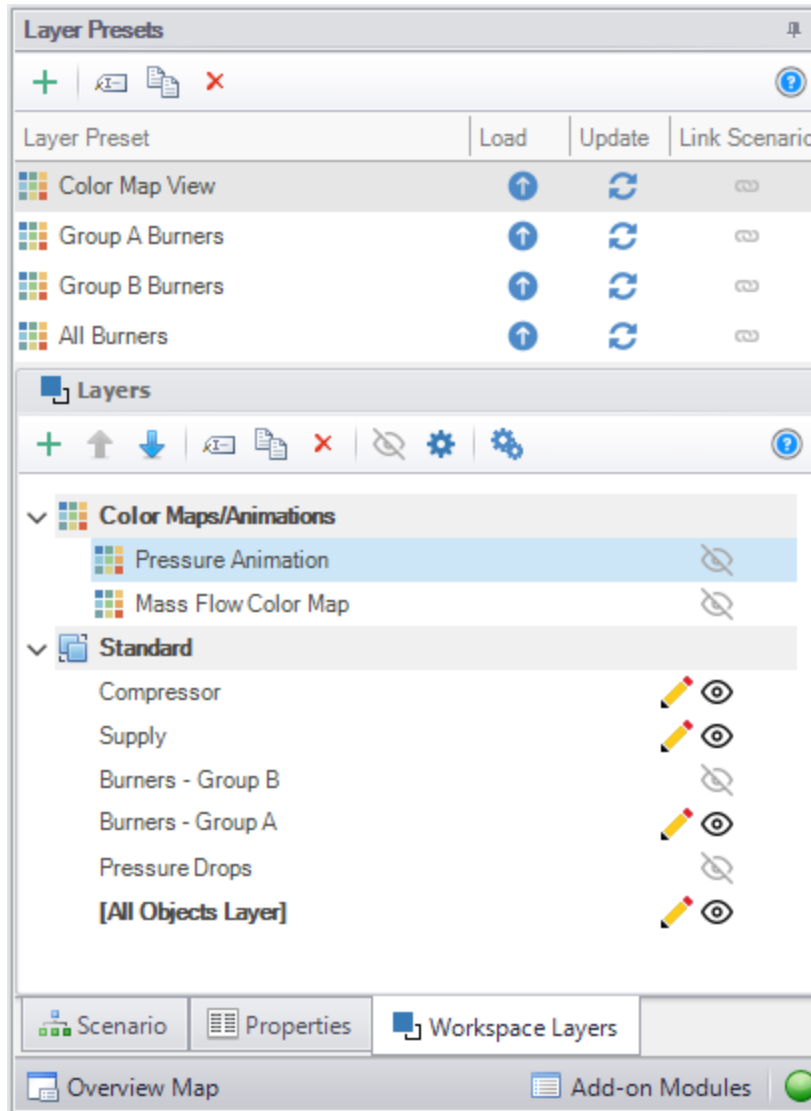


Figure 5: View of the Workspace Layers panel structure

Overview Map

When a model is very 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 6, 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. You can also zoom in and out using the mouse wheel.

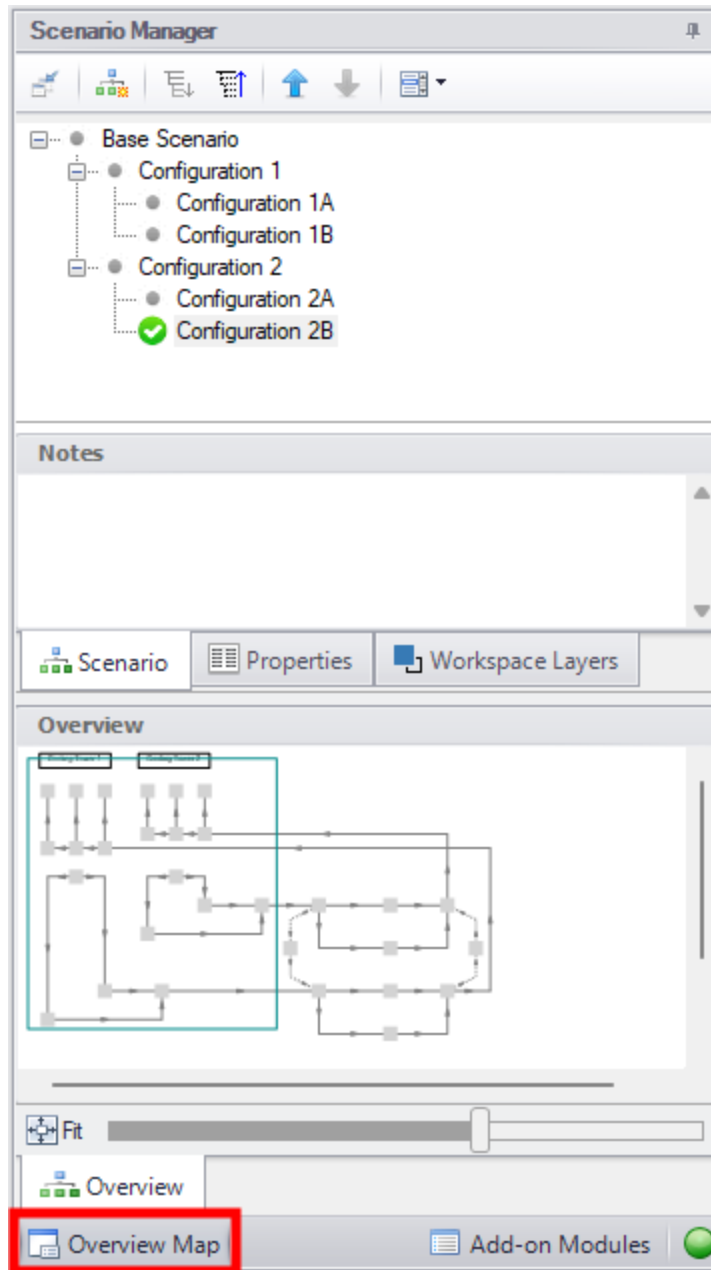


Figure 6: Workspace Overview Map

Activate Add-on Modules

Clicking the Modules button in the bottom right of the Quick Access Panel in Figure 7 launches Analysis Setup to the Modules panel to activate or deactivate add-on modules.

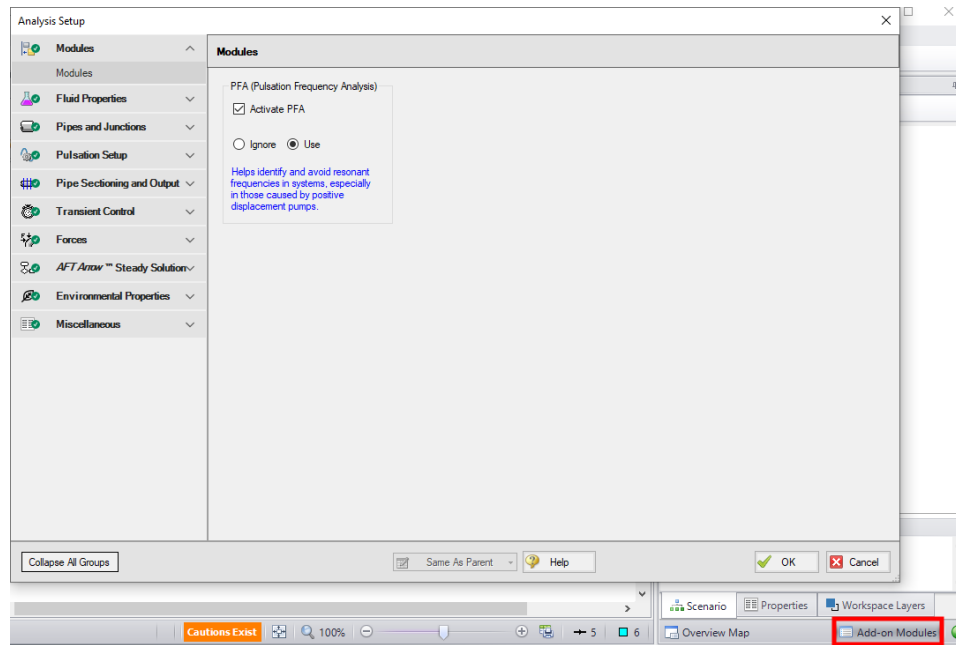


Figure 7: Activating the Modules from the Quick Access Panel

Model Status Light

The Model Status Light is located in the bottom-right of the Quick Access Panel (Figure 8). Clicking the Status Light will open Analysis Setup. When each item in Analysis Setup is fully defined, it will have a green check mark. When all necessary items in Analysis Setup are completed, the Status Light will turn from red to green, indicating that the model can be run.

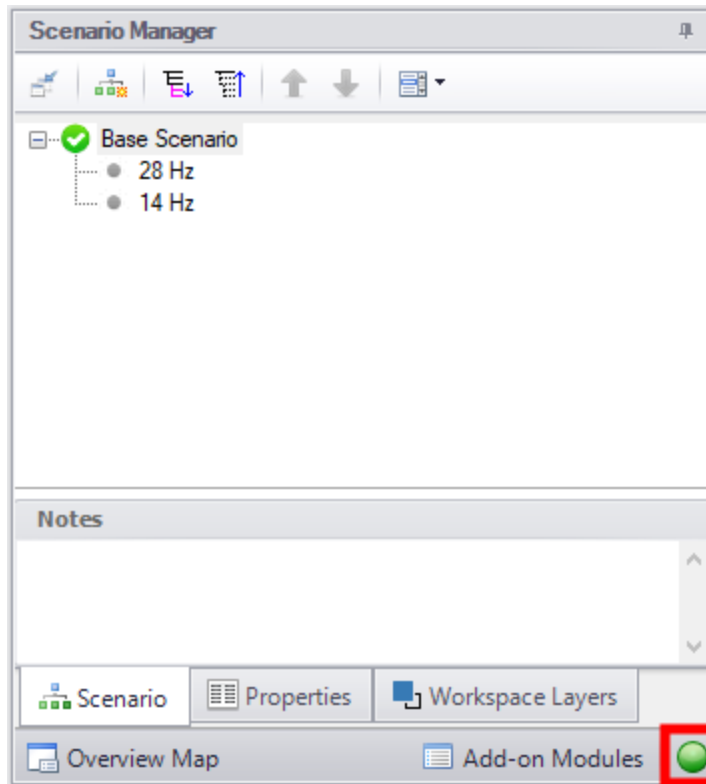


Figure 8: The Model Status Light is located at the bottom-right of the Quick Access Panel

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 9 shows the AFT xStream window with the Quick Access Panel initially expanded. By clicking on the thumbtack pin button (highlighted below with the red box), the Quick Access Panel will then become collapsed as shown in Figure 10. After the Quick Access Panel is collapsed, it can be temporarily expanded again simply by hovering over the "Scenario" sidebar with the mouse.

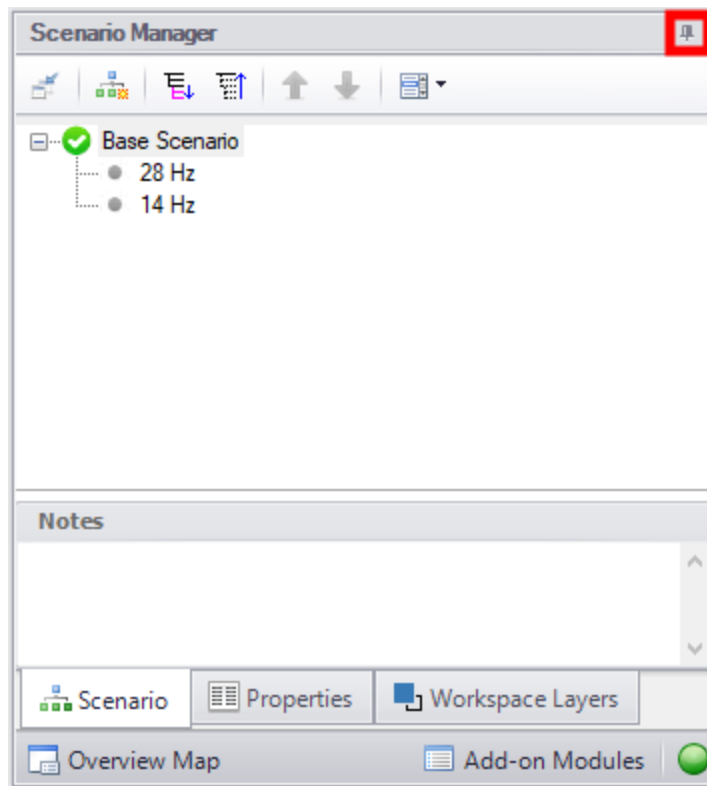


Figure 9: The Workspace with the Quick Access Panel expanded



Figure 10: The Workspace with the Quick Access Panel collapsed

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. 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 the [Output](#), [Graph Results](#), and [Workspace](#) become available.

Other Actions - Pausing the simulation allows the modification of certain [AFT Arrow Steady Solution](#) parameters during the run, as well as viewing the iteration history if it was enabled via the Keep Track of Iteration History setting on the [Iterations panel](#).

Retain Partial Transient Output - Canceling the run allows for output data to be kept up to the point at which the run was canceled. The [Output File](#) will be finalized and the partial output can be viewed. A caution will be shown in the Output stating the last valid time step. This can be used in combination with [Transient Checkpoint Files](#) to continue the run later.

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).

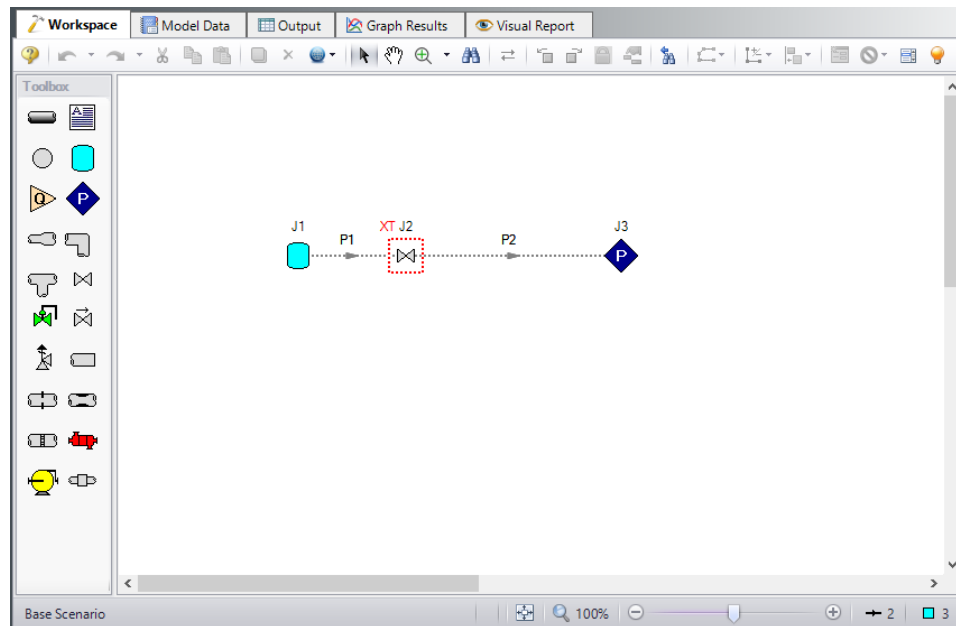


Figure 1: The Workspace window

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 certain selected Workspace objects.

Related Topics

Workspace Layers.....	324
Creating Pipes - the Pipe Drawing Tool.....	175
Annotations.....	335
Optional Input for Junctions.....	220
Workspace.....	382

Workspace Layers

Fundamentally, Workspace Layers are used to control the visibility of [Workspace](#) objects and labels. Layers allow custom views to be created in which results can be displayed without needing to leave the Workspace window. A group of layers functions like a stack of transparencies, with each layer displaying specific, customizable information. The view on the Workspace is a composite of the information presented by all visible layers, but all objects exist regardless of visibility for the purposes of calculations and object connections.

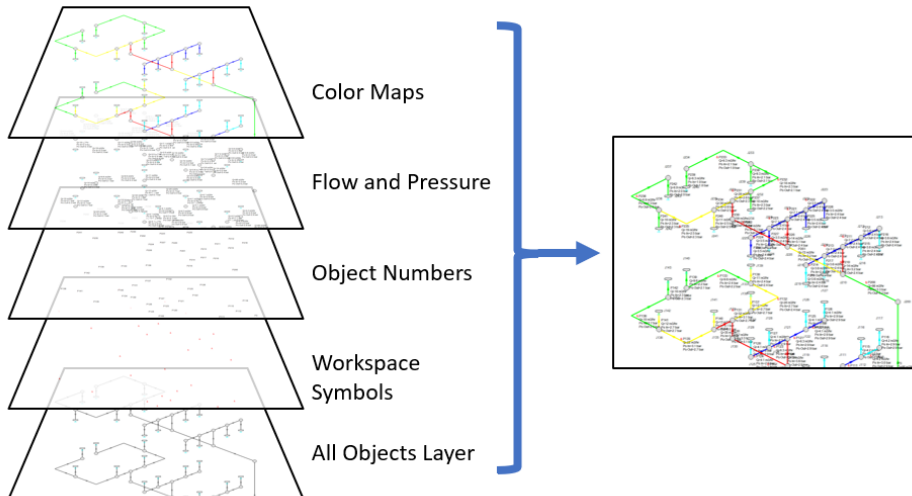


Figure 1: Information from all currently visible layers combines to form the view displayed on the Workspace

Layers can be shown or hidden to create different views of the Workspace. The visibility of objects on the Workspace depends on the layers that are shown, as objects on hidden layers are not shown on the Workspace unless present on another layer. If an object exists on multiple layers, only the top-most representation is visible. The order of the layers is controlled by the order in which they are listed in the Workspace Layers panel.

Workspace Layers Panel

The Workspace Layers panel is accessed via the Workspace Layers tab in the [Quick Access Panel](#). There are two areas within the Workspace Layers panel to control Layer Presets and Layers.

In the Layers area, new layers can be created, and any selected layers can be renamed, duplicated, and toggled in visibility. The Global Layers Settings menu is accessed here, and the column of visibility icons on the right shows the current visibility of each layer and can be clicked to toggle visibility. Hovering over a layer, a gear icon appears that links to the layer's Layer Settings options for customization.

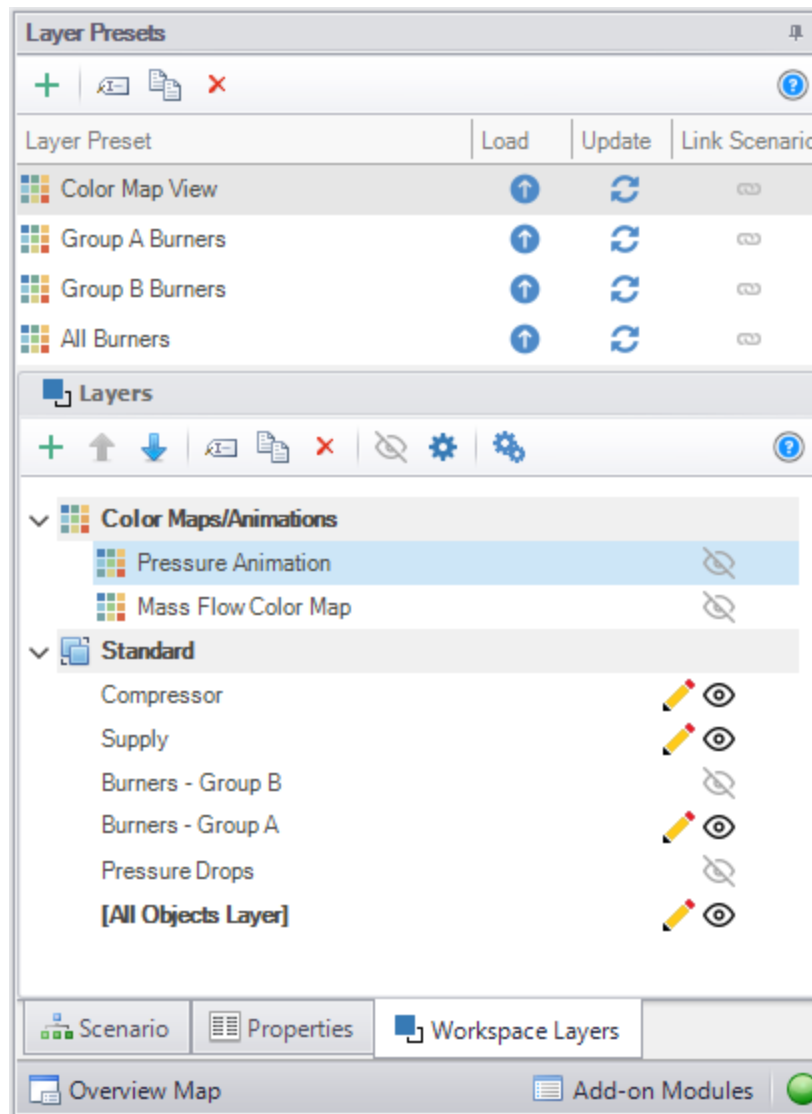


Figure 2: View of the Workspace Layers panel structure

Standard layers control pipe and junction styles, labels, and visibility, along with the visibility of any annotations in the model. Among the standard layers is the All Objects Layer. The All Objects Layer exists in all models and contains all pipes, junctions, and annotations in the model. This serves as the primary layer, provides a central location for users to quickly see all objects in the model, and is always the bottommost layer. When creating a view where some objects are hidden, the All Objects Layer can be toggled in visibility so that only objects visible in other layers are shown on the Workspace.

Other standard layers can be reordered up or down the layer stack using the areas at the top of the Layers area. For pipe and junction labels, the order of displayed text items is affected by the order of the layers containing label text.

Standard layers can be created from the New Layer drop-down as a Blank Layer or From Workspace Selections. Layers created from Workspace selections have the selected objects and their labels visible in the new layer.

By default, new objects are visible only on the All Objects Layer. However, if layers are set as Active Layers, then any new objects that are created and their labels will be visible within all Active Layers. Active Layers are set by hovering over a layer and clicking the pencil icon and any currently Active Layers are denoted by the pencil icon.

Layer Presets

Layer Presets allow customized Workspace views to be retained by saving the status (visibility) of each layer. There are four columns in the Layer Preset area. One shows the name of the preset, the Load Preset option allows the previously saved layer states to be loaded onto the Workspace, the Update Preset option saves the current layer states to that Layer Preset, and the Link Preset option links that preset to the currently active scenario.

By linking a preset to a scenario, it will automatically load whenever that scenario is loaded in the future. Individual layers, by contrast, are not tied to a specific scenario and are defined globally and immediately available in all scenarios. A layer can only show objects that exist in the active scenario, and changes to the style of an object in a layer will affect all scenarios.

Global Layer Settings

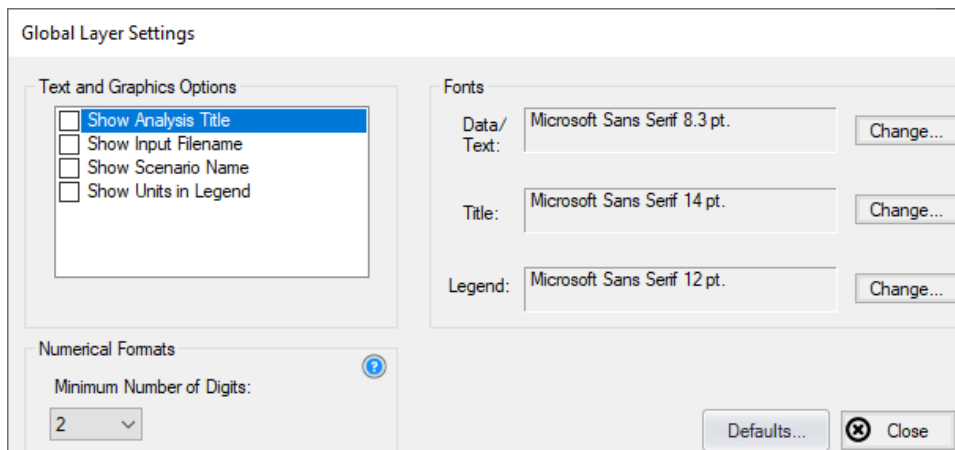


Figure 3: Global Layer Settings window

The Global Layer Settings are accessed from the double gear icon on the Layers Toolbar in the Quick Access Panel. The properties that are controlled by Global Layer Settings are not defined for individual layers – changes to these settings affect all layers globally.

Text and Graphics Options allow the analysis title, input filename, scenario name, and units in the legend to be toggled.

Note: The analysis title is set in the general tab of [Output Control](#).

The Fonts setting controls the fonts for Data/Text (e.g. labels), the Title text (the analysis title, input filename, and scenario title if visible) and the text in the legend.

The Numerical Formats option determines the minimum number of digits shown for input and output data displayed within label text.

Layer Settings - Pipes & Junctions

The Layer Settings window can be accessed by double-clicking the layer, from the single gear icon on the right of each layer, or by selecting the layer and clicking the single gear icon on the Layers Toolbar.

The layout of the Layer Settings window is similar to Analysis Setup, with a number of *groups* listed on the left side of the window. Each group contains *items* that when selected change the content of the main panel area on the right.

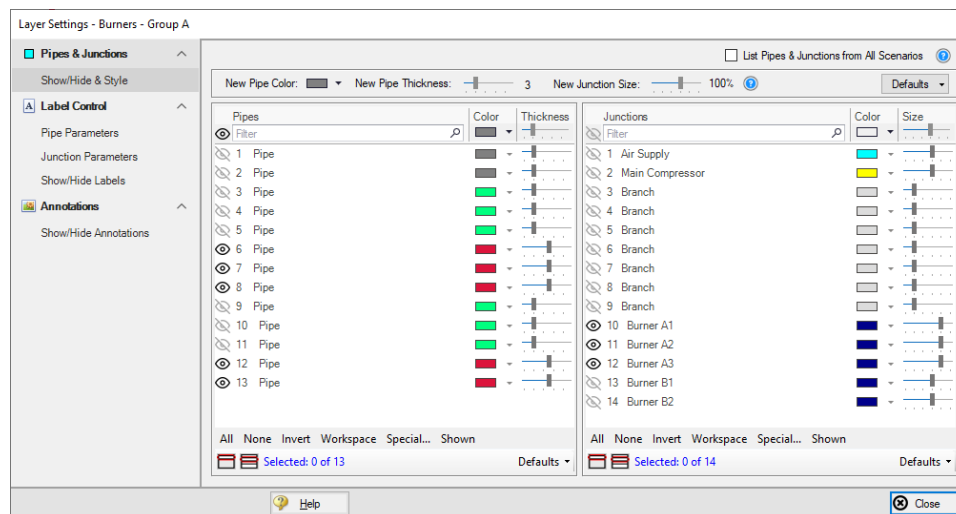


Figure 4: The Show/Hide & Style panel allows the appearance and visibility of pipes and junctions to be managed

The Pipes & Junctions group of the Layer Settings window allows the pipe and junction color and size/thickness to be defined for a given layer, along with which pipes and junctions are visible. Note that changes to this panel are made live in real time, and no Apply button is needed to apply the changes that you make. Pipes and junctions are shown in two separate lists, which can be filtered.

Multiple pipes or junctions can be edited at a time using the Color, Size, and Thickness modifiers at the top of each table, or set to default options with the Defaults buttons towards the bottom.

The usual tools for selecting pipes and junctions, including Workspace select and [Select Special](#), are available here. Pipes and junctions may also be edited individually with the visibility, color, and size options in each row.

The appearance of new pipes and junctions is layer specific, and can be set to default options with the top Defaults button or modified with the New Pipe Color, New Pipe Thickness, and New Junction Size options. Note that color and icon style for new junctions are set via the [Toolbox](#) as they are specific to each type of junction.

List Pipes & Junctions from All Scenarios is toggled off by default to reduce unintended changes to items in other scenarios, but this can be toggled on to include objects from all scenarios in the lists.

Layer Settings - Label Control

A label is text that is associated with a pipe or junction on the Workspace. Labels include both input and output parameters along with object names, numbers, and workspace symbols. While there is no separate output mode, output parameters are only displayed when the current scenario has output data available. If no output is present, the parameter in the list to the right will appear grayed out. Input parameters will always be shown when added to the list on the right. The label shown for an object on the Workspace is a combination of defined label items from each visible layer, and the order of the items is determined by both the layer order and the order that parameters are listed within each layer.

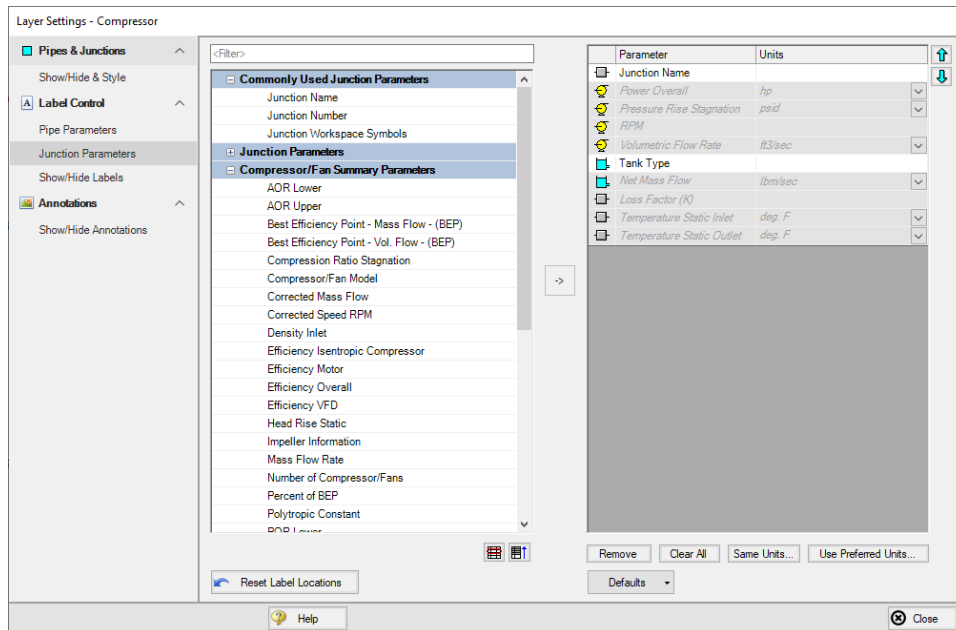


Figure 5: The Junction Parameters panel within the Label Control group of Layer Settings

Within the Label Control group, the Pipe Parameters and Junction Parameters items allow for the parameters associated with a layer to be customized. When there is no output available, output parameters will be grayed out within the list, but their list position and units may still be edited.

Label locations are defined globally, so the Reset Label Locations option will apply to all labels in all layers and all scenarios.

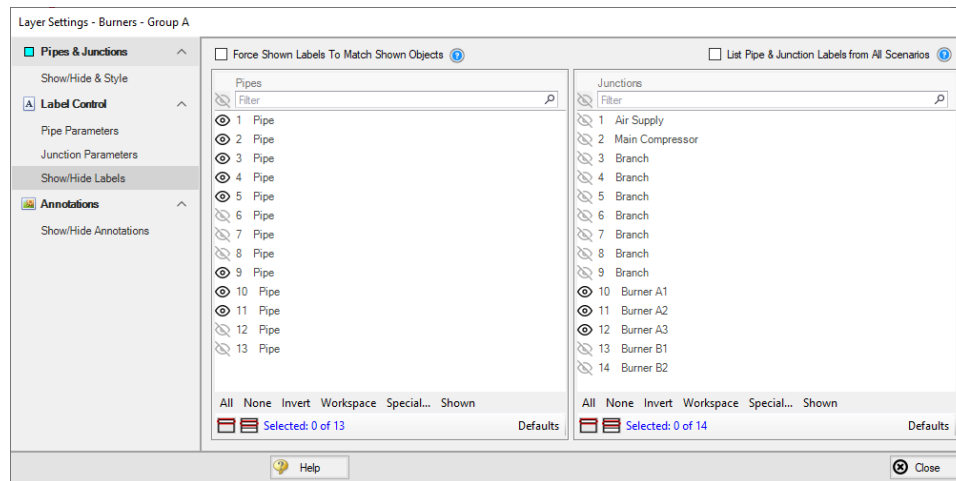


Figure 6: The Show/Hide Labels panel determines which objects will have visible labels for a given layer

The Show/Hide Labels panel can be used to toggle the visibility of labels for each pipe and junction within a layer. This allows a user to create a layer that shows only the relevant parameters for the relevant objects in the model. Visibility can be modified for one or several objects at a time via the same methods as pipe and junction visibility.

The Force Shown Labels to Match Shown Objects option forces the visibility of each label to match its object visibility as defined in the Pipes & Junctions group of the Layer Settings. The Defaults options at the bottom of the tables simply toggle this option on if it had been previously unselected.

Layer Settings - Annotations

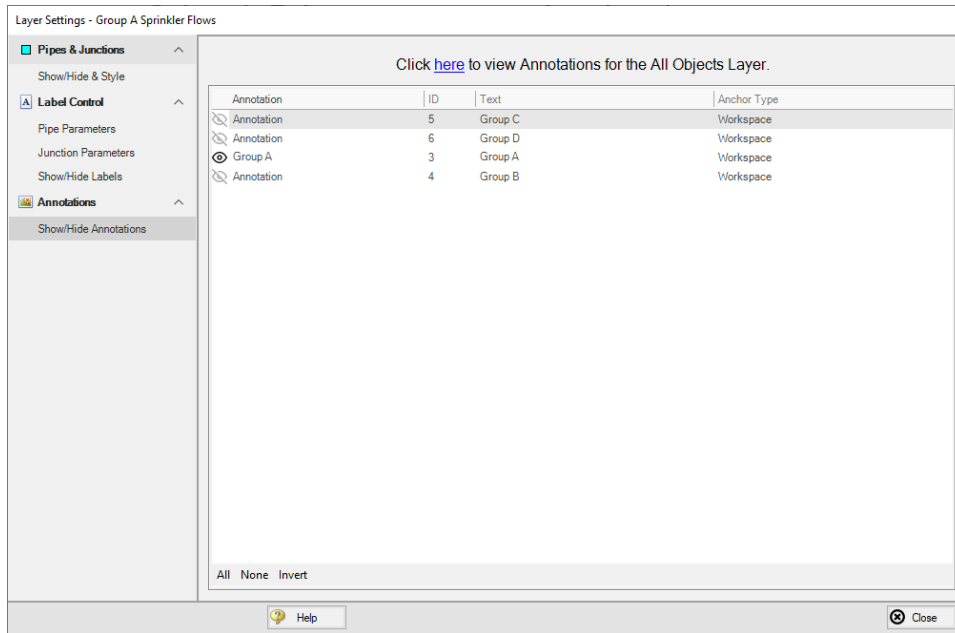


Figure 7: The Show/Hide Annotations panel shows all annotations that exist in the model file and allows their visibility to be controlled on a per-layer basis

The Show/Hide Annotations panel allows the visibility of annotations to be defined for the current layer and displays the visibility of each annotation in the current layer. While annotations exist globally, in concert with Layer Presets this menu allows annotations to be controlled so that they are only visible in the relevant scenarios. Annotations can be edited from this list view by double-clicking them in the list to open the annotation in the Annotation Editor.

Color Maps and Animations

A Color Map layer allows you to visualize magnitudes of various parameters by coloring the pipes. Color Maps are a special layer type, and are shown in a separate group on the Layers panel. There can be only one special layer visible at a time, and they override the behavior of standard layers in the model - in this case superseding any colors assigned to pipes by other layers.

Example use cases for Color Maps includes visualizing hot/cold loops, locations of high velocity/reverse flows, or pipe diameters/lengths.

To create a Color Map layer, click the new layer icon on the Layers Toolbar, select a Color Map layer type, and name the new layer. To edit the layer, go to its Layer Settings menu, accessed in the same manner as a standard layer.

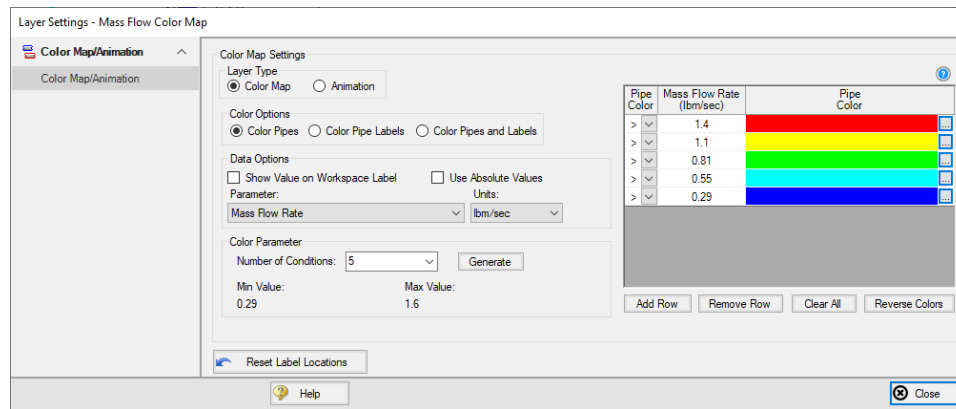


Figure 8: Layer Settings window for a Color Map

In the Color Map Layer Settings window, choose a parameter and units for which all pipes will be categorized. Then choose the Number of Conditions (number of colors) and click Generate. The operators (i.e. >, >=, =, <=, <) and values can be modified. Values may be categorized on an absolute value basis by selecting the Use Absolute Value checkbox. The Show Value on Workspace Label option shows the evaluated parameter next to each visible pipe in the model, and combines with any other label text items from other layers.

The order of entry is not important. Arrow will sort the entries first by value and then by the operator, with the greater than (>) operator being above the less than (<) operator. 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 defined four colors using the following inequalities,

- >=60
- <60
- >=20
- <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 color 2.

To avoid overlapping colors, only use less than (<) as the last entry. A better map would be the following:

- >=60
- >=20
- <20

To delete a map value, select the row from the Color Map Grid and click the Remove Row button. Instead of coloring the pipes, it is possible to color just the pipe numbers. This is done by choosing the Color Pipe Labels radio button in the Color Options area. Alternatively, the Color Pipes and Labels radio button allows both to be colored.

Color choices from the Color Map Grid will appear in a legend without units shown by default. The legend may be repositioned by dragging it to any location on the [Workspace](#). Units may be displayed in the legend using the Show Units in Legend setting within Global Layer Settings.

If there is no output for the current scenario, any Color Map layer that displays an output parameter will have a red exclamation mark next to it on the Layers panel to warn the user that there is no output. The model must run before such Color Map layers can be displayed.

Color Maps can be animated for transient data using [Animation Layers](#). The interface for animating a color map layer is shown in Figure 9.

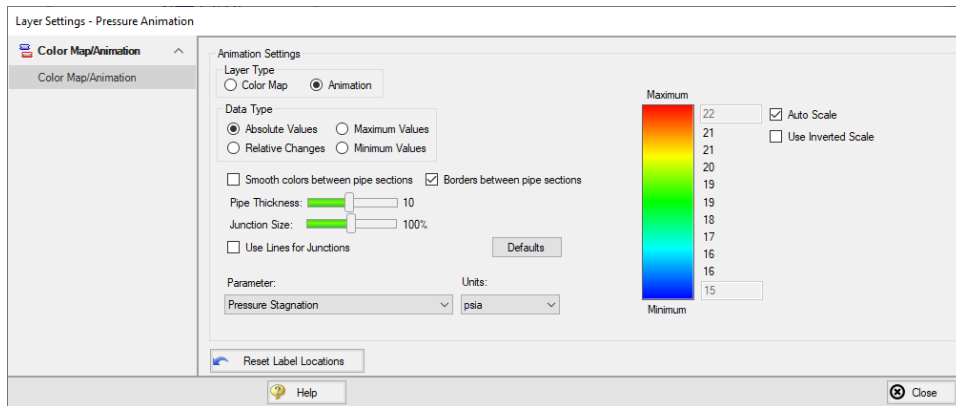


Figure 9: Layer Settings window for an animated color map layer

Workspace Layers Import Assistant

The Workspace Layers Import Assistant can be used when importing a model from Arrow 9 or earlier. By default, when opening a model from these previous versions, layers are created as applicable for object numbers, workspace symbols, and object names. This is done regardless of whether or not the Workspace Layers Import Assistant is run.

The purpose of the Workspace Layers Import Assistant is to import visual styles of pipes and junctions and to import labels from the deprecated Visual Report Window as Workspace Layers so that previous Visual Reports can be recreated. This is a step by step guided procedure that will attempt to bring these elements into Workspace Layers while creating as few new layers as possible. These layers are created on a per-scenario basis.

If a model has a high number of scenarios with similar Visual Reports, it may be preferable to recreate the Visual Report manually as a new layer that can be used with any scenario. If the chosen scenarios for the Import Assistant share many of the same visual settings, the Import Assistant will create all relevant layers for each, even if those layers are identical to each other.

The Workspace Layers Import Assistant will allow you to select which scenarios to use to create the new layers. Once these are selected, the next page of the Import Assistant will allow you to see how many layers and Layer Presets will be generated. Once satisfied with the proposed Workspace Layers and Layer Presets, click Import Layers to complete the import process.

Related Topics

Animation Layers.....	55
-----------------------	----

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).

Note that annotations, layers, and layer presets are not imported from the second model.

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

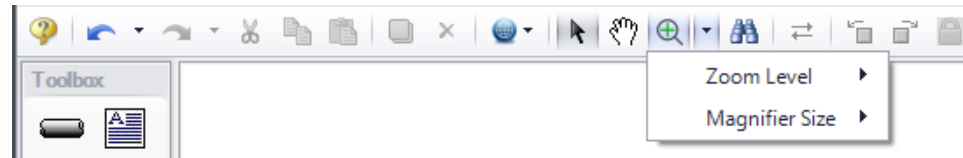


Figure 1: Magnifier tool on the Toolbar

Related Topics

Toolbars 307

Annotations

Annotations are objects that can be placed on the [Workspace](#) to display customized text, express ideas via shapes or lines, or even to insert pictures into the Workspace. These are purely visual and do not affect results; their purpose is to communicate conceptual information about a system. They appear both in the Workspace and when printing from the Workspace.

New annotations are created using the Annotation Tool in the [Toolbox](#). The Annotation Tool is located to the right of the Pipe Drawing Tool. With this tool, the user can draw a line or a shape on the Workspace. Shapes can display an image or text on the Workspace or can be used without text to display boundaries around various regions of the model.

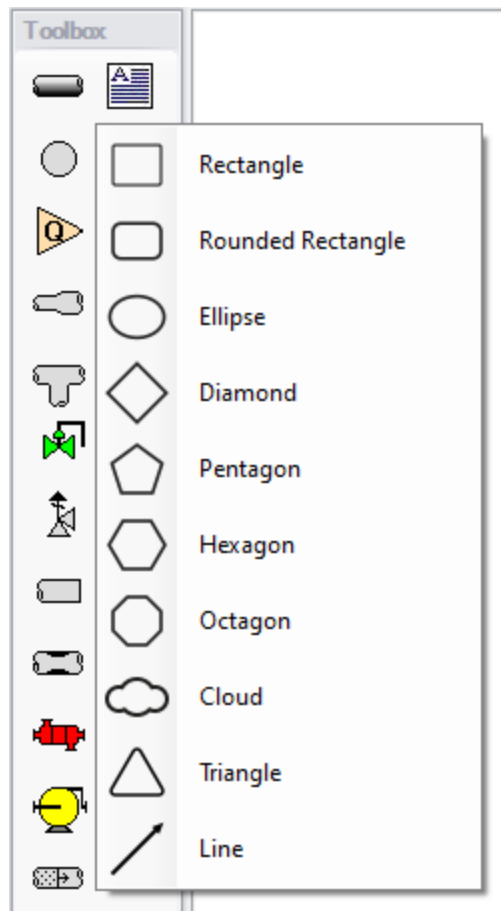


Figure 1: After clicking the Annotation Tool, select the shape for the new annotation

Properties

Properties - Annotation 1

Enter text to search... Defaults ?

- Annotation (ID = 1)**
 - Name: Annotation
 - Text: Group B
 - Text Alignment: TopLeft
 - Text Color: Black
 - Text Font: Microsoft Sans Serif, 18pt
 - Text Show:
- Behavior**
 - Anchor Object Tag: 0
 - Anchor Type: Workspace
 - Workspace Allow Select:
 - Workspace Keep In Back:
- Format**
 - Border Color: Black
 - Border Style: Solid
 - Border Thickness: 1
 - Shape: Rectangle (1:1)
 - Shape Color: White
 - Shape Show:
- Image**
 - Image:
 - Image Keep Aspect Ratio:
 - Image Stretch:
- Line**
 - Arrow At End:
 - Arrow At Start:
 - Arrow Size: 3
 - Arrow Style: Fill
 - Line Color: Black
 - Line Show:
 - Line Start Location: TopLeft
 - Line Style: Solid
 - Line Thickness: 1

Scenario Annotation 1 Workspace Layers

Figure 2: Annotation view for the Properties in the Quick Access Panel

After an annotation is placed on the Workspace, the properties of the annotation will be displayed in the Properties tab of the [Quick Access Panel](#) where they can be edited. For existing annotations, it can be accessed quickly by double-clicking on the annotation in the Workspace. Annotation styles can be set as default or restored to default settings using the Defaults drop-down menu in the upper right of the Annotation properties area.

- **Annotations** - The name of the annotation, its text content, text style, and text visibility can be edited within the Annotations section. The name of the annotation is not displayed on the Workspace but is shown within the [Layers Settings](#) where it can be used to communicate the purpose of an annotation.
- **Behavior** - The Behavior section allows the behavior of the annotation to be customized. 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.

Note: Whether an annotation appears behind or in front of another object is usually determined by the order of the Workspace Layers in which they are visible, but the toggle Workspace Keep in Back forces the annotation to appear below pipes and junctions regardless of layer order.

- **Format** - The Format section controls the shape as well as the shape's style and visibility.
- **Image** - The Image section allows an image to be displayed within the annotation and controls its appearance.
- **Line** - The Line section controls the appearance of lines and only applies when the shape is set to be a line in the Format section. Similarly, the shape color and border properties from the Format section are not used for lines.

Interaction with Scenarios and Layers

Annotations are not scenario-specific and exist globally within a model. Their visibility is associated with layers and can be controlled for each layer in the [Layer Settings](#). While annotations exist globally, their visibility for a given scenario can be controlled using [Layer Presets](#).

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.

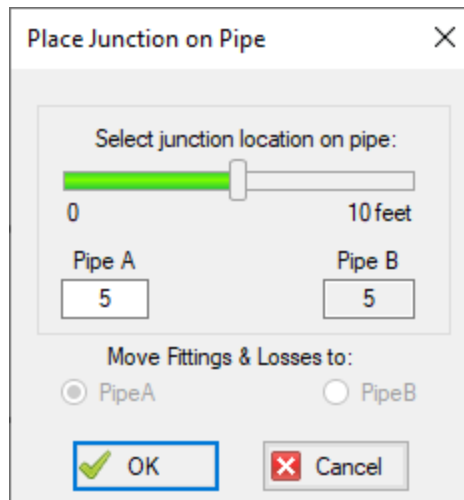


Figure 1: Split pipe keyboard shortcut

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

- Open the Keyboard Shortcuts Help Page (this page!): CTRL + Q
- 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
- Duplicate: CTRL + D
- Delete selected pipes and junctions: DEL
- Reverse pipe direction: F3
- Lock Object: CTRL + L
- Toggle special condition of pipes/junctions between open and closed: F4
- Show object status: CTRL + H
- Other available shortcuts can be seen listed by their command on the menus

Related Topics

Add/Remove Pipe Segment.....	211
Joining or Splitting Pipes.....	176
Creating Junctions.....	178
Defining Objects.....	180
Editing Objects.....	180
Morphing Junctions.....	180
Moving Objects.....	179
Creating Pipes - the Pipe Drawing Tool.....	175
Workspace Pan.....	184

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

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.

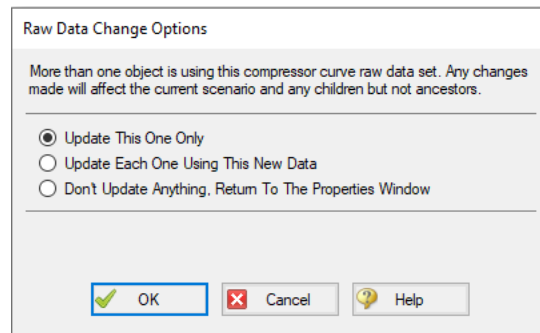


Figure 1: Raw Data Change Options

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.
- **Copy Layer Style Settings From Original Objects** - Duplicates [Workspace Layers](#) style settings (visibility, thickness, color, label parameters, etc.) for selected objects.

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.

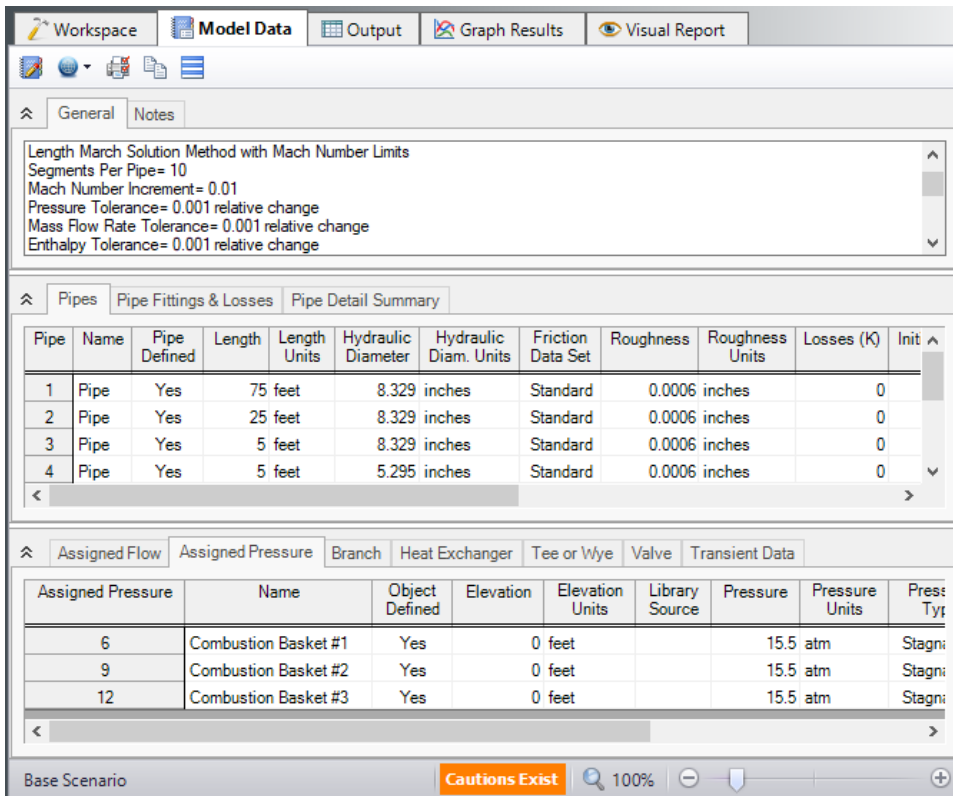


Figure 1: The Model Data window displays input in text form

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.

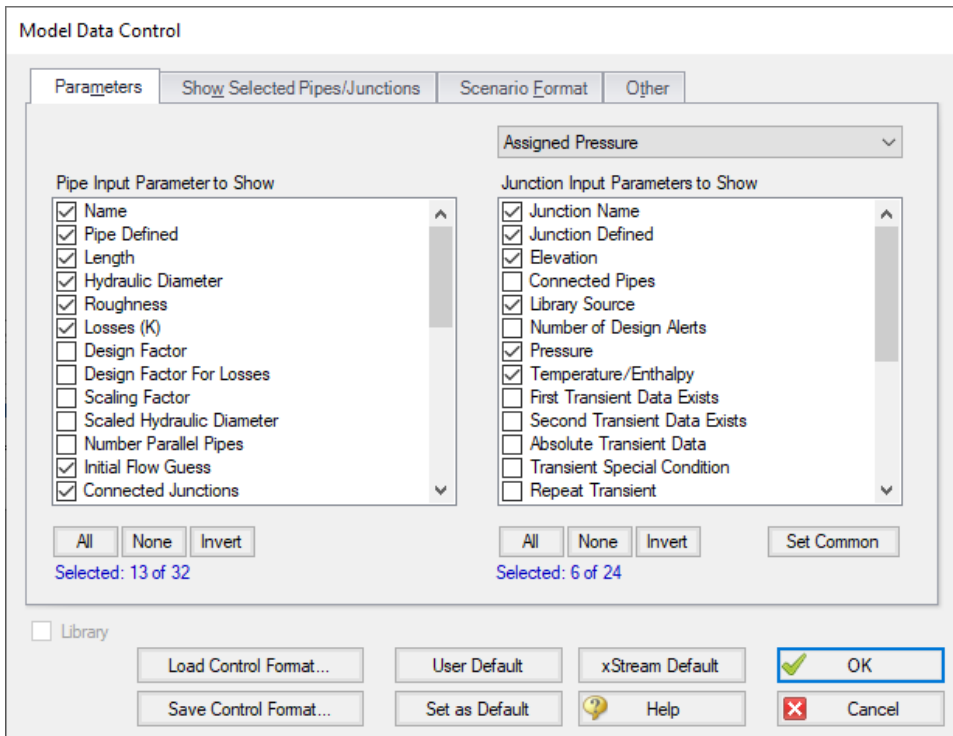


Figure 1: The Model Data Control window allows selection of the parameters to show in the Model Data window

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.

Interface and User Options

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:

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

Pipes 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	0.02556	40
3	268.8	2.000	0	242.0	0.000	8	25.004	0.000	0	15.910	0.02180	8

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.

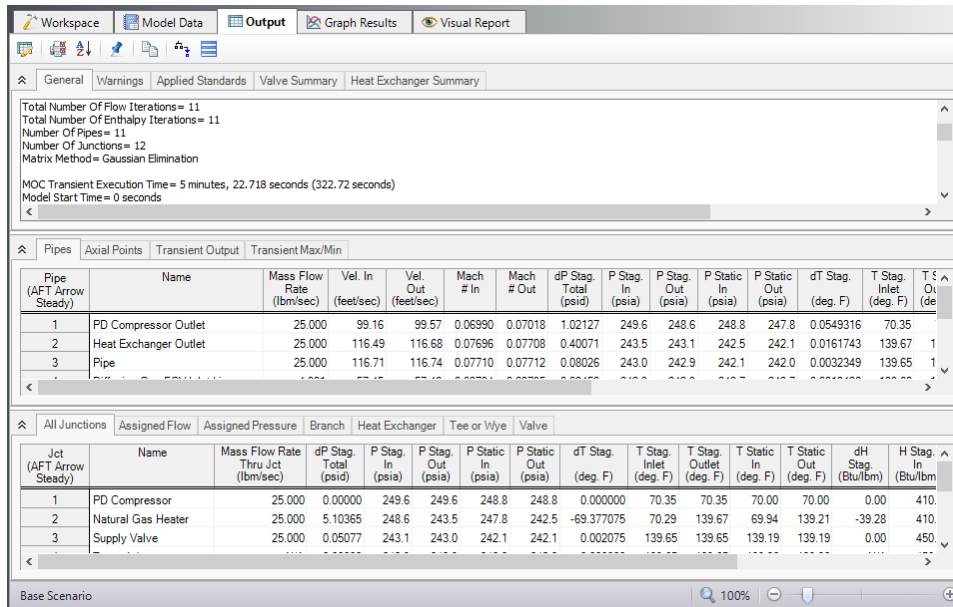


Figure 2: The Output window shows steady flow results in the Pipes and Axial Points tables and the various junction tables. Transient data is shown on the transient tabs.

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.

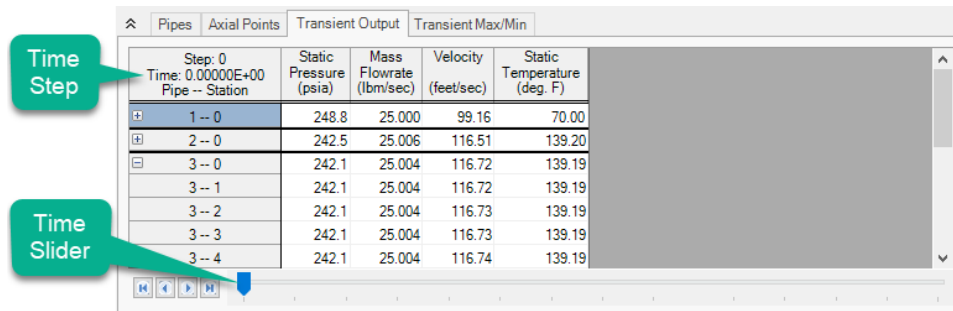


Figure 3: The Transient Output table in the Output window shows output results for pipe stations and at specified times

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)	Min. Velocity (feet/sec)	Min. 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	0.000	0.000
1 -- 118	273.9	2.000	247.8	0.000	25.006	0.000	17.3353	0.04459	99.59	0.000	0.000	0.000
1 -- 119	273.9	2.000	247.8	0.000	25.006	0.000	17.3234	0.04419	99.59	0.000	0.000	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	0.000	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	0.000	0.000
2 -- 1	269.2	2.000	242.5	0.000	25.005	0.000	17.7107	0.04340	116.52	0.000	0.000	0.000
2 -- 2	269.2	2.000	242.5	0.000	25.005	0.000	17.9505	0.04261	116.52	0.01684	0.000	0.000
2 -- 3	269.1	2.000	242.5	0.000	25.005	0.01605	17.9636	0.04162	116.53	0.01625	0.000	0.000

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	7
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	4.931	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.

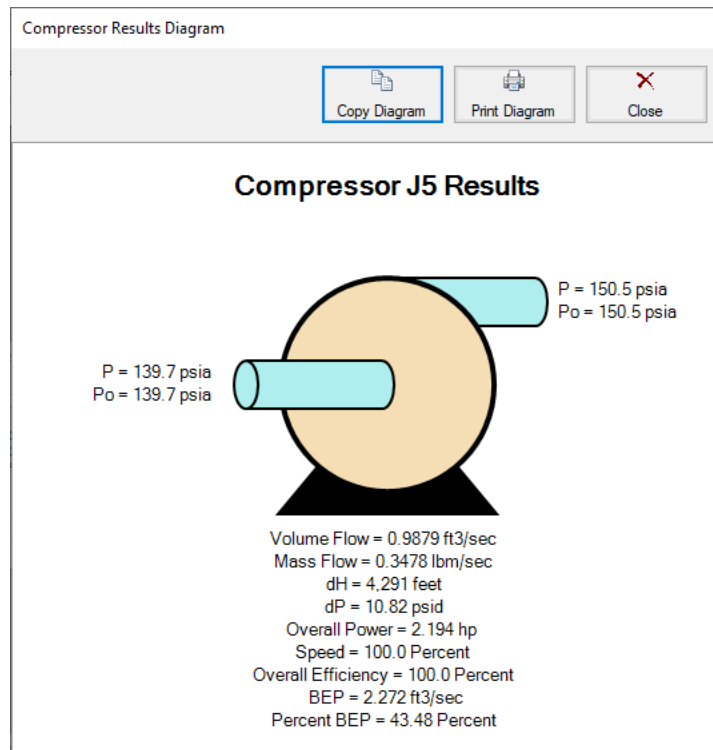


Figure 6: Compressor/Fan Results Diagram

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	46
Output Messages & Warnings	547
Applied Standards	358

Output Control

Output Control (see Figure 1) allows you to customize the parameters shown on the [Output Window](#).

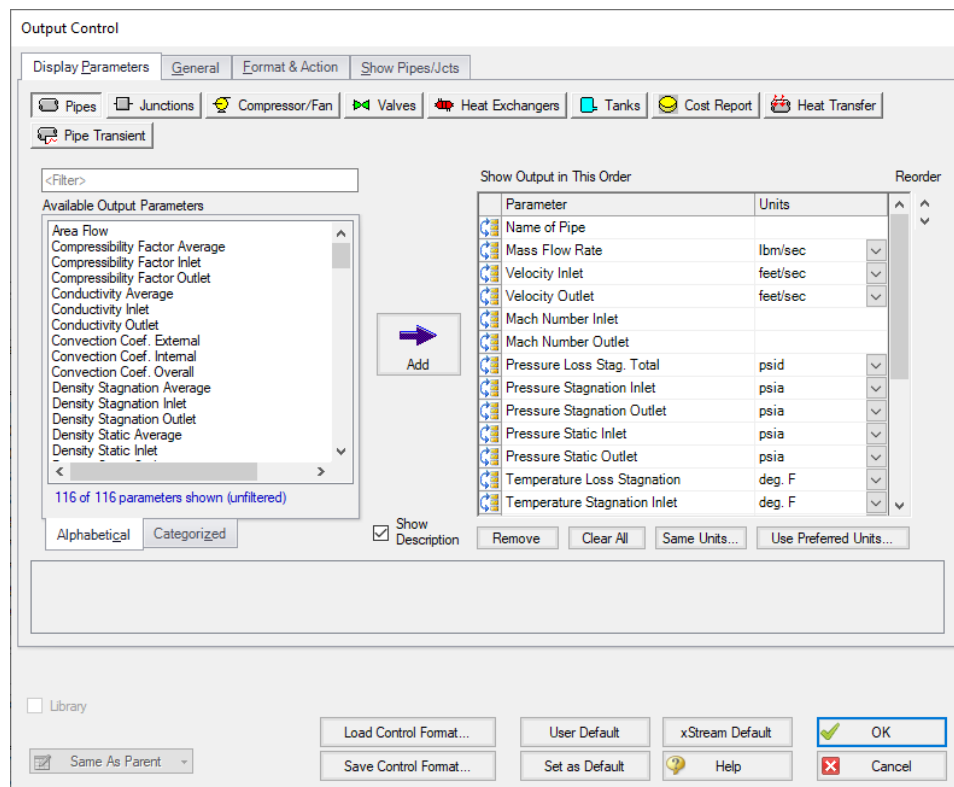


Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe Output parameters are shown. The other output parameters function the same way

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 [Workspace](#) 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.

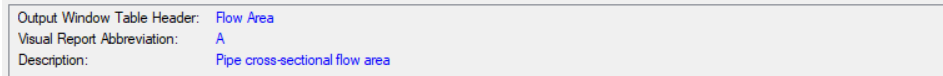


Figure 1: A description is provided at the bottom of Output Control for the selected parameter

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.....516

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.

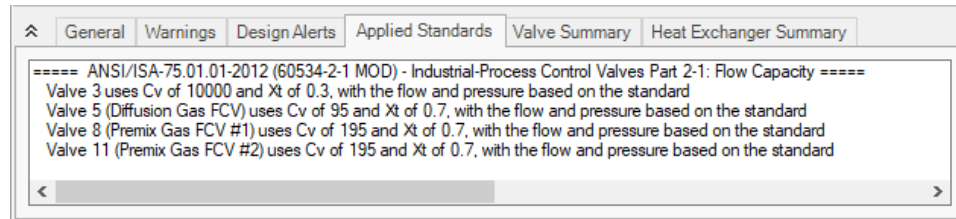


Figure 1: Applied Standards when applied for Cv/Xt loss at a valve

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.

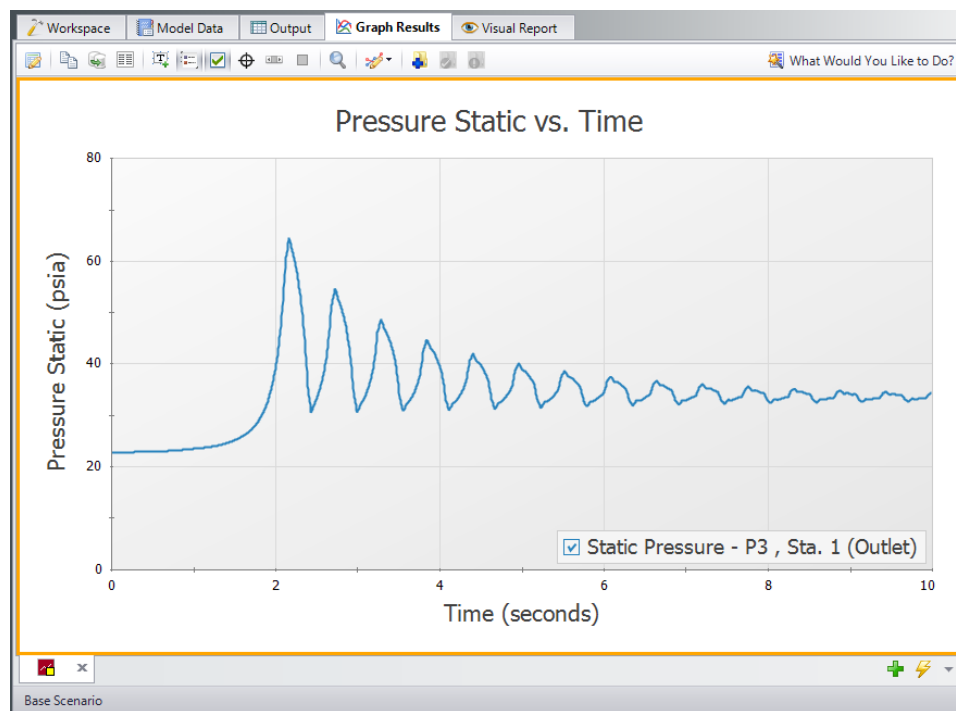


Figure 1: The Graph Results window offers full-featured plot generation

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 51

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 access, add it below the parameter that will be on the primary access, 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".

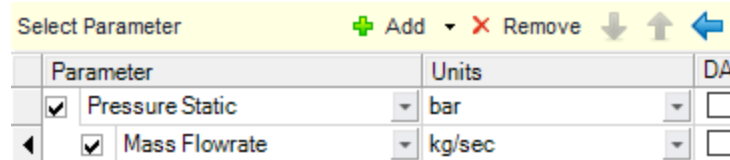


Figure 1: Creating a graph with a parameter on the secondary axis

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.

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.

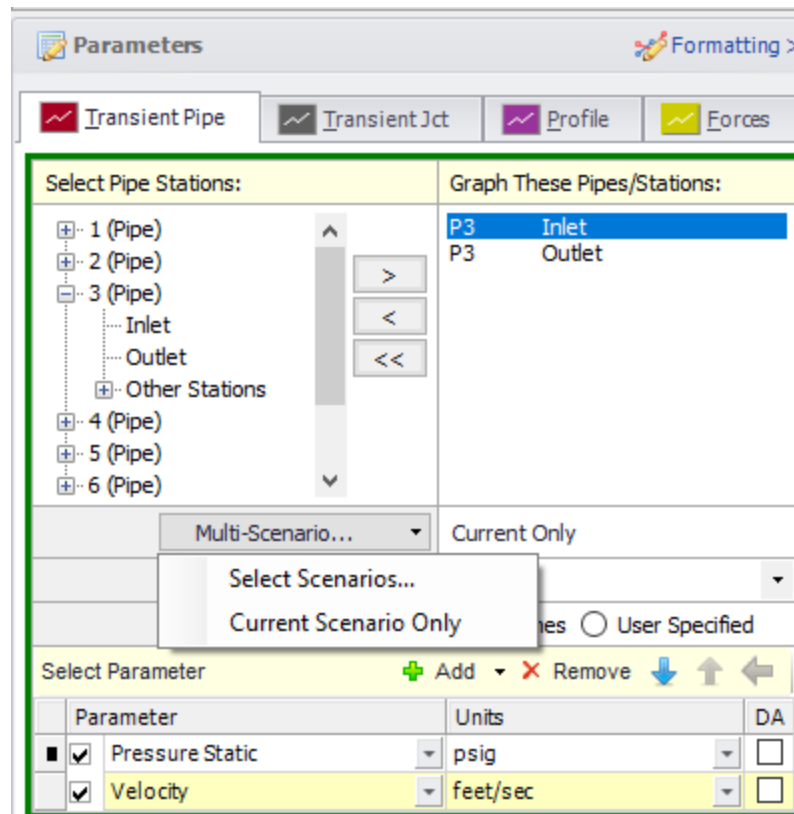


Figure 2: Graph Parameters window showing Multi-scenario selection options

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 [\(return to top\)](#)

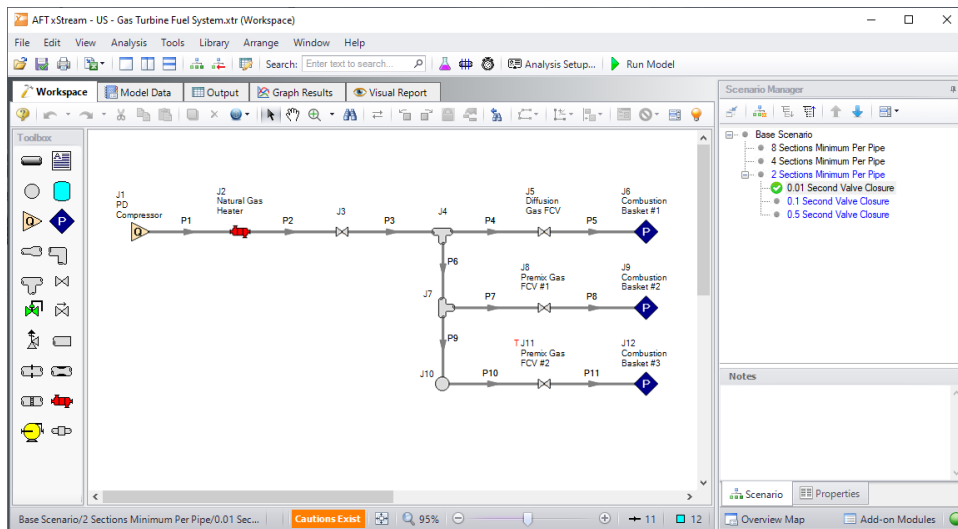


Figure 1: High Pressure Steam Example

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 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.

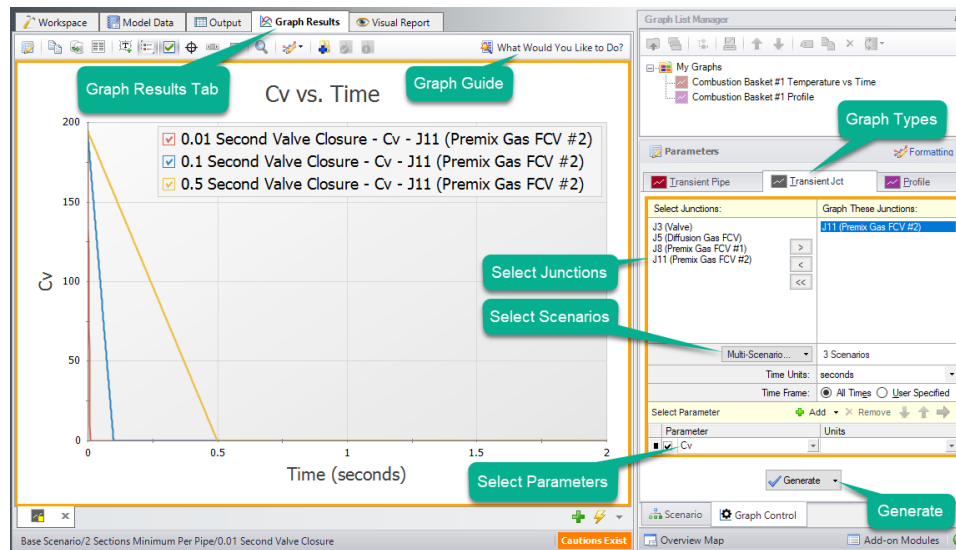


Figure 2: Valve Cv vs. Time

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.

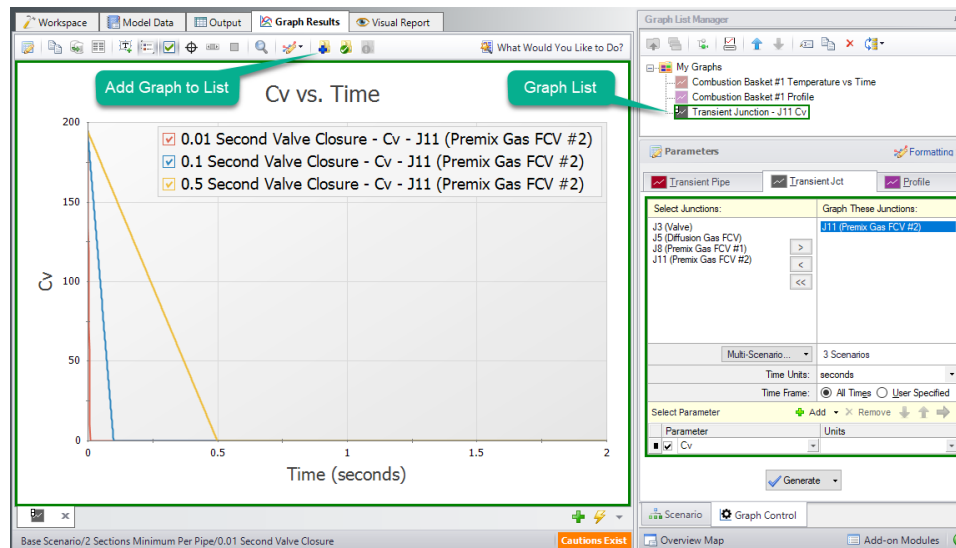


Figure 3: Adding a Graph to My Graphs

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.

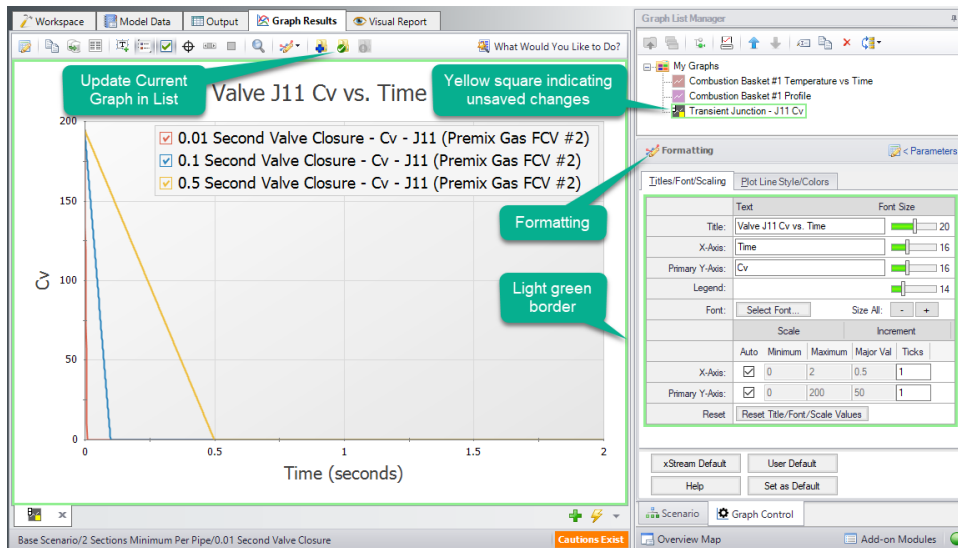


Figure 4: Update Formatting Changes to Graphs

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).

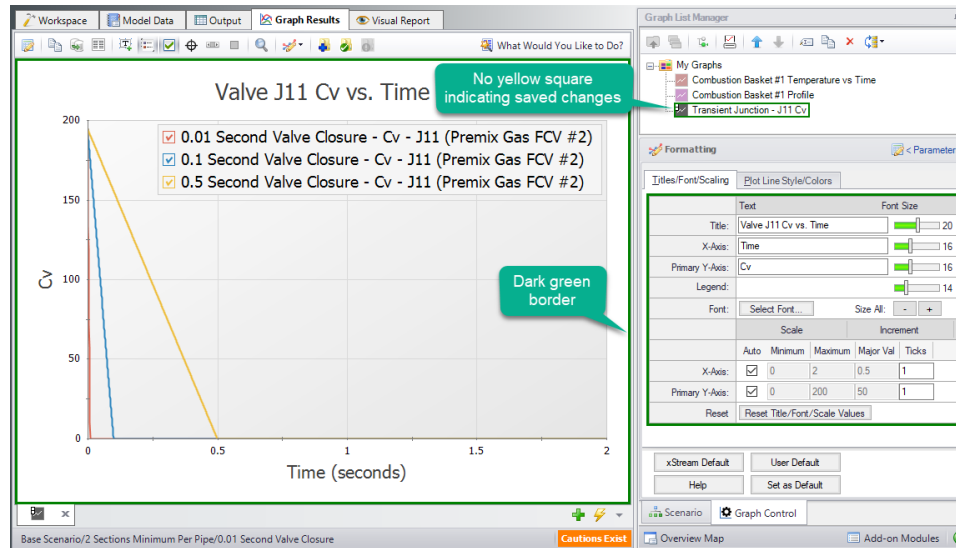


Figure 5: Valve Cv graph with updated formatting changes

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.

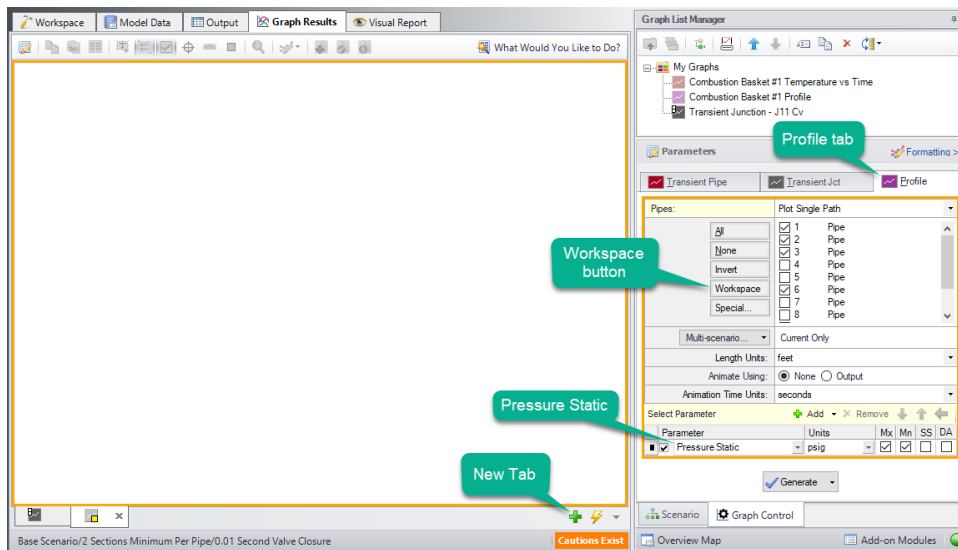


Figure 6: Generating a Profile plot with Min and Max Pressure Static

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.

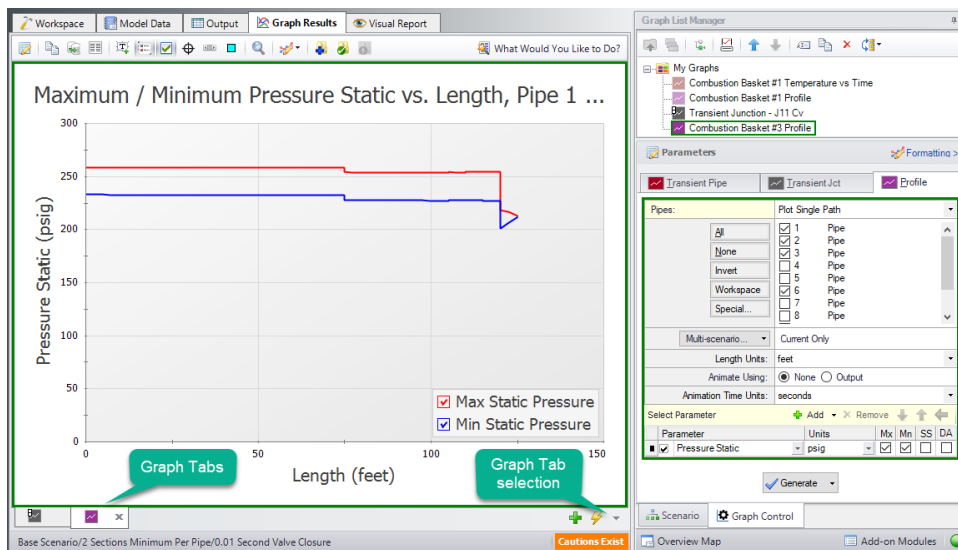


Figure 7: Pressure Static vs. Flow Length Profile added to Graph List

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.

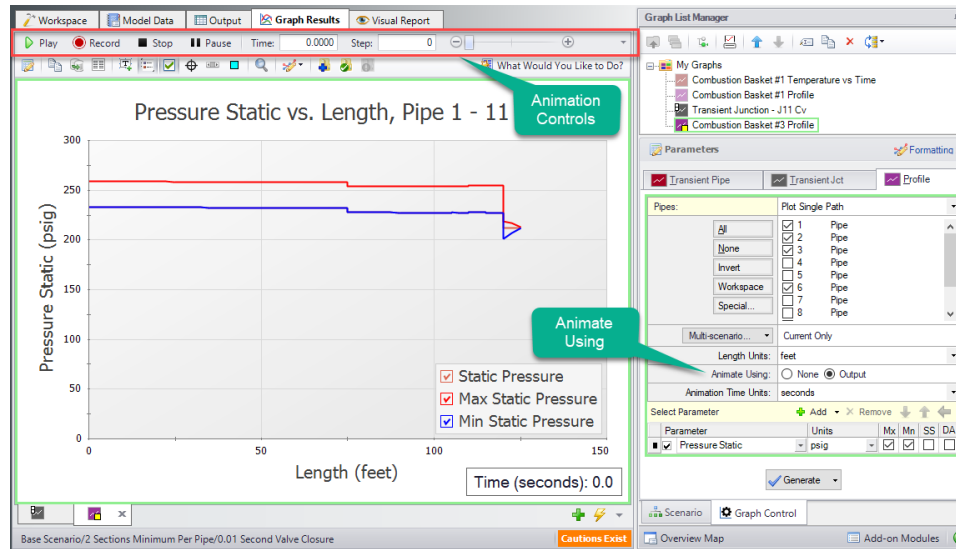


Figure 8: Max/Min Profile Graph with Animation

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.

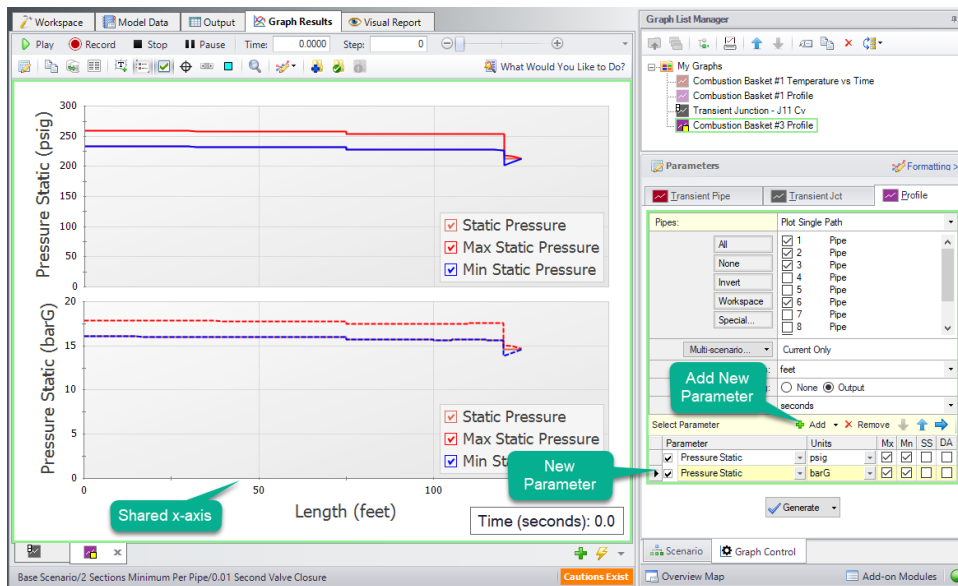


Figure 9: Stacked Pressure Profile with different units

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.

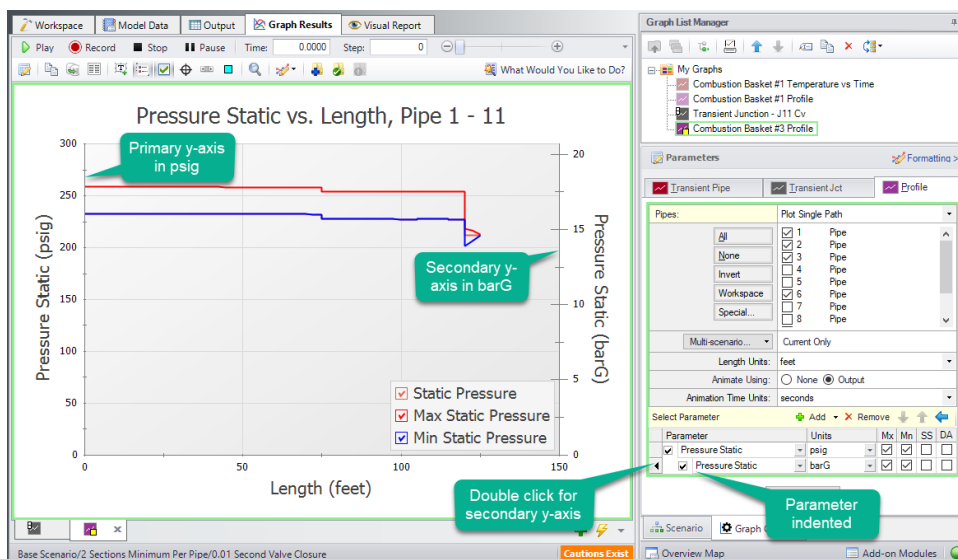


Figure 10: A Single Graph with Alternate Units on a Secondary Y-Axis

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.

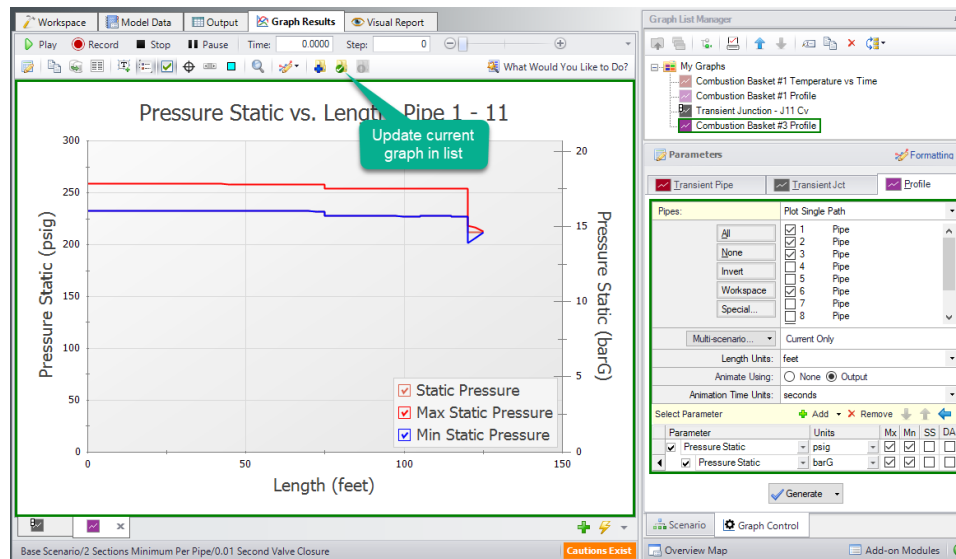


Figure 11: The Resulting Graph when it is Updated

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.

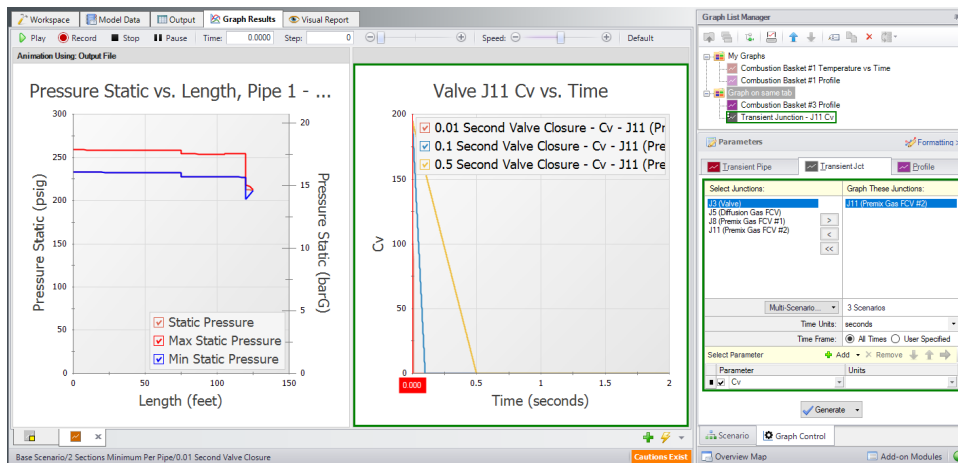


Figure 12: All graphs loaded into the same tab

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.

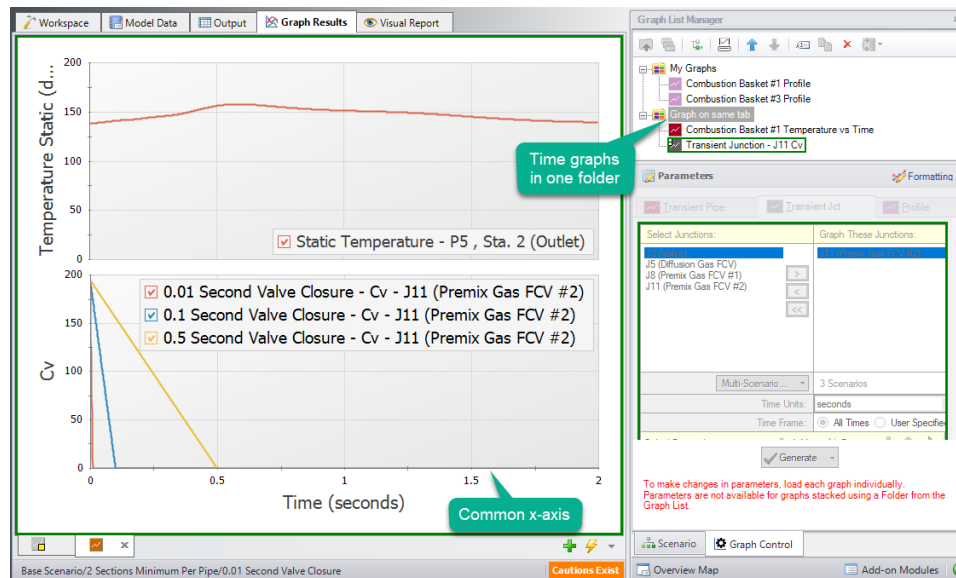


Figure 13: Load Current Folder Graphs in Same Tab - Stack Same x-Axis

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”.

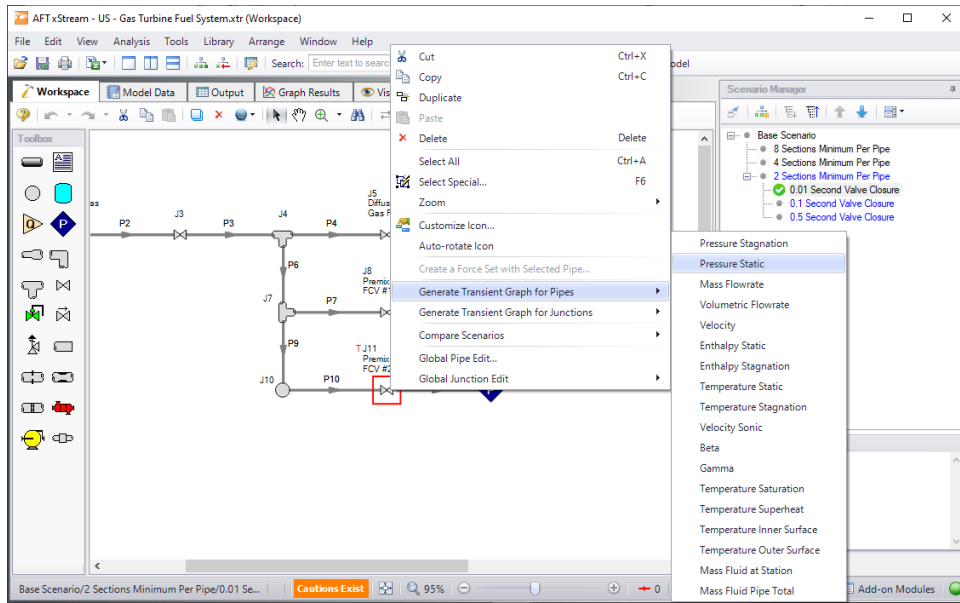


Figure 14: Creating Transient Graphs of the Inlet/Outlet Pipes Connected to a Junction

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.

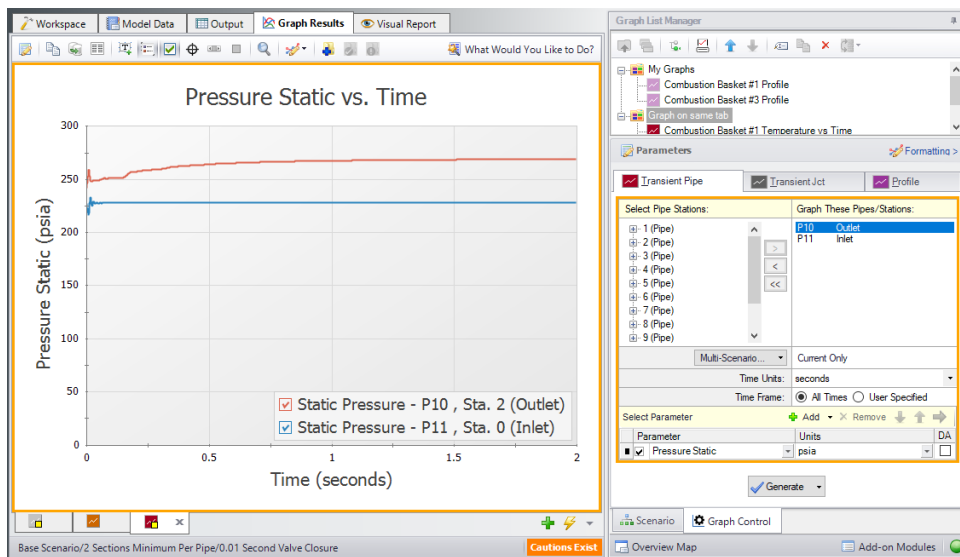


Figure 15: The Pressure Static vs. Time of the Two Pipes Connected to Valve J11

Make one more Graph from the Workspace by right-clicking pipe 10, selecting “Create Transient Graph for Pipes”, “Mass Flowrate”, and then “Outlet Only”.

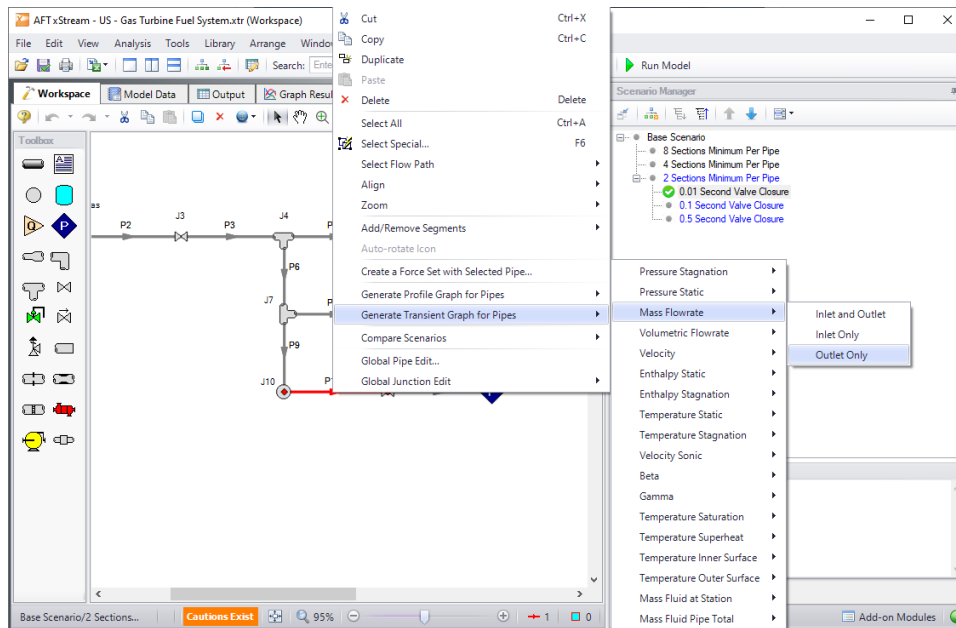


Figure 16: Graph the Changes to Mass Flowrate vs. Time from the Workspace

This will add a Graph of the Mass Flowrate at the Outlet Station of Pipe 10 as is shown below in Figure 17.

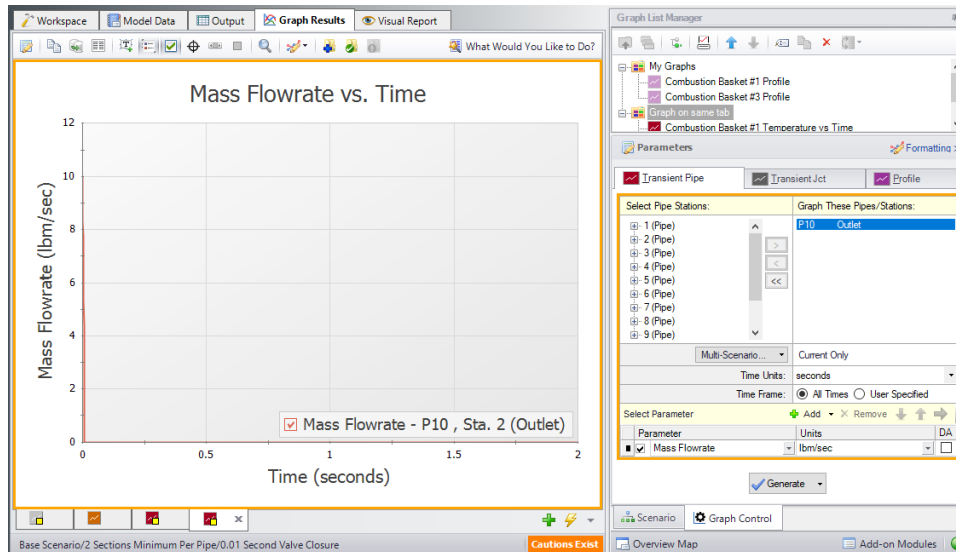


Figure 17: Mass Flowrate vs. Time at the Outlet of Pipe P10

User Options Overview

An array of User Options are provided in a single window that can be found in the [Tools menu](#).

The General section of the User Options window provides the following options:

- Choose to show the Startup Options window at startup
- Choose to show the New Version Welcome window at startup
- Enabling orthogonal drawing tips
- 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 several buttons at the bottom of the User Options window. There are built-in AFT Default parameters, units, and settings which you can choose by clicking the Reset to xStream Defaults button. You can also develop your own settings, tailored to your project or industry, and have these used by default. To make your own default, first select the parameters, units, and settings you would like to use, then click the Save All 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 All User Defaults button. The default settings are updated only when you click Set All 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.

Note: Regions outside the United States that use commas as the decimal separator may have *.csv files that are delimited with semicolons (this was a change made by Microsoft). AFT software requires true comma delimited files. A workaround for such regional users in Excel is listed below.

Workaround for regional users with alternate decimal separators (e.g. Europe):

1. In Excel go to File > Options > Advanced
2. Under the Editing Options heading, uncheck the box for "Use system separators"
3. Change the "Decimal separator" to a period symbol (.)
4. Change the "Thousands separator" to a comma (,)
5. Save the *.csv file
6. Re-check the "Use system separators" option to revert to the default Excel settings

Workspace

In the Workspace group of the User Options window, you can change the symbols used, default 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 displayed on the workspace for modeling convenience based on the [Pipe Drawing Mode](#) that is selected. 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.

The Pipe Drawing Mode and the toggle to Show Grid can also be found 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.

The default pipe line thickness for new layers can be adjusted by selecting the desired pixel value in the box next to New Layer Default Pipe Line Thickness (pixels).

You can close a pipe using the [Special Conditions](#) feature which closes the pipe to flow at both ends. From the User Options window, you can select how you would like a closed pipe to be displayed on the Workspace. The options include a solid line, dashed line, a dotted line, and dashed-dotted combinations. The color of the line is still determined by the visible [Workspace Layers](#).

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.

Under the Pipe Direction Arrows section 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 positive flow direction](#) 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.

Using the default transparent icon style, the appearance and color of icons for new junctions can be set in the Toolbox, and icon size and color for existing junctions can be set in [Layer Settings](#). Junction icon and rotation can be edited for existing junctions by right clicking the junction on the Workspace and selecting Customize Icon.

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 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 when it is closed with Special Conditions. If this option is disabled, no dotted box will appear.

Labels

When a pipe or junction is inserted onto the Workspace, the next whole number available at the increment specified in this panel 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.

Whether pipe/junction names and/or numbers are displayed is controlled ultimately by [Workspace Layers](#), but the Pipe/Junction Default Display setting allows the user to set their preferred default behavior.

Users 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 with the Allowable Workspace Label Movements setting. 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 item 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 Labels
- Pipe Labels
- New Layer Default Pipe Color
- 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.

To have the background color of the children and grandchildren scenarios maintain the same color as the base scenario, click on the Set All Background Colors to Base Color button under the Parameter/Color list.

Other

Workspace Symbols representing pipe or junction special conditions and inflow or outflow are listed in the Other item within the Workspace group in the User Options window. These symbols can be disabled by unchecking the box on the left of the list. Users can also change what symbol will be used by clicking in the Symbol box on the right and typing the new desired symbol.

If there is a background image in a model, the image size can be controlled from this panel. 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.

Graphs from Workspace

Various parameters can be graphed quickly by right-clicking on a continuous selection of pipes from the Workspace and choosing the Generate Profile Graph for Pipes option. The graph parameters that are available from the right-click menu can be customized.

Parameter Type

Parameter Types, located within the Parameters section of the User Options window, allows you to specify the flow rate type, head/pressure type and Efficiency/Power types you would like as the default option wherever one of those parameter types is defined in the model. The default flow rate type can be set to either mass flow or volumetric flow, the default head/pressure type can be set to either head or pressure, and either efficiency or power can be selected as the default for efficiency/power. For example, when defining a resistance curve to describe the hydraulic pressure loss behavior through a general component, you are required to enter data that describes pressure loss at given flow rates. When defining the pressure loss behavior here, you can either define the flow rate as a mass or volumetric flow rate, and you can define the head/pressure loss as either a head or pressure loss. Adjusting the default option of these parameter types to fit your available data not only helps to make data input more efficient by preventing you from having to constantly change parameter type, but it also helps to minimize input errors by adjusting the required input to the form of your available data.

Files

The Files section of the User Options window allows you to specify file locations for saving models, library 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 Files & Auto Save

A Backup File is a copy of a previously saved version of your model file (*.xtr) 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.

The Number of Backup Files (i.e. the number of previously saved versions of the model) and the frequency that the model is saved can be adjusted. You can opt to save no Backup Files by choosing "0" in the Number of Backup Files Created dropdown.

After the model has been saved initially, subsequent instances of saving the model will create a new Backup File until the specified maximum has been reached. The most recent Backup File will always contain the file extension *.bakX-001. Larger numbers appended to the file name indicate older backups:

- Model File.xtr - Current and most recent version of the model file
- Model File.bakX-001 - Latest backup (the second most recent version of the model file)
- Model File.bakX-002 - Older backup
- Model File.bakX-003 - Oldest backup

If you need to revert to a Backup File and convert it back into a standalone model, add the extension .xtr to the Backup File (the file name should now appear as Model File.bakX-001.xtr), and open the file in xStream. From there you can re-save the file or overwrite your existing model file.

The model file can be saved periodically by enabling the checkbox for Automatically Save Model and specifying a frequency in the dropdown box.

Create Recovery File

A Recovery File (*.Recv-XTR) is similar to a Backup File but is saved periodically at the specified interval. It allows the user to recover unsaved changes in the model file prior to the application crashing or closing abnormally. The Recovery File is written to the same location as the model file, and each time the specified interval passes it will be overwritten to contain the most recent state of the model. If the application is closed normally the Recovery File will be deleted. A Recovery File can only be generated for existing model files that are opened.

If a Recovery File exists, the next time the model file is opened a prompt will be shown asking if you want to use the Recovery File:

"A recovery file for this model was found which may indicate that the application closed unexpectedly. Would you like to open this recovery file?"

- **Yes** - This will copy the previously saved model file to a new file with "(Original)" appended to the front, and the Recovery File containing the latest unsaved changes will be opened.
- **No** - The Recovery File will be deleted, and the previously saved model file will be opened. This will open the model file in its last saved state, which may not include the latest unsaved changes if any are stored in the Recovery File.

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

Importing Piping Component Files	389
Importing GIS Shapefiles	393
Importing Caesar II Netral Files	395
Importing and Exporting Epanet Files	397
Importing and Exporting from Other AFT Products	402
Exporting to Excel	403
Importing Excel Data	407

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 groups of pipes from the .pcf file can be associated with a pipe material, size, and type in the AFT Pipe Material library. Pipes can be grouped by file schedules (i.e. item codes) or diameters, and are sorted by length within each group. 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) and can be edited individually or as a group.
 - **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 imported as junctions, or as a visual change in direction of the pipe. Importing bends as pipes is identical to using [segments](#) with assigned fittings and losses to show piping layout. To account for losses, bends will be matched to smooth flanged bends in the AFT Internal [Fittings](#)

[& Losses Library](#) if an exact match for angle and r/D ratio is available and assigned to the pipe automatically. Losses that are not assigned automatically can be set up manually using junctions or [fittings & losses](#). This option is helpful to reduce unnecessary model clutter and input **if bend losses are negligible**.

Note: In most cases, it is recommended to import bends as bends rather than pipes. Bends imported as bends will always use smooth bends with r/D and/or angle imported from the .pcf file and basis area inherited from the upstream pipe.

- **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.

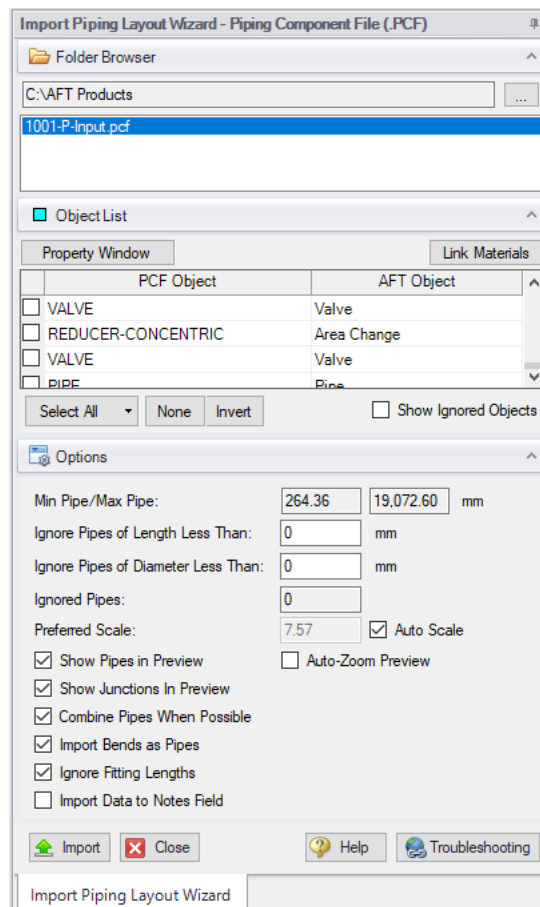


Figure 1: Piping Component File Import

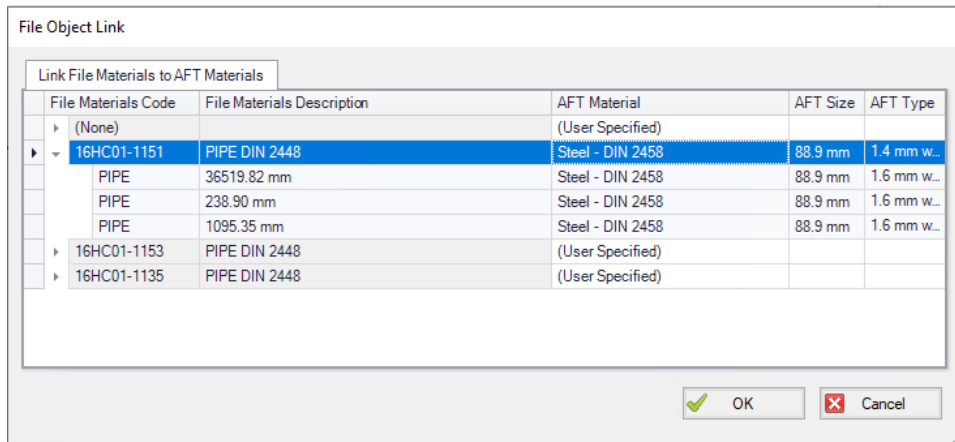


Figure 2: File Object Link window for Piping Component File Import

General Recommendations

Importing piping layouts from external file types such as Piping Component Files is intended to save time by allowing a means to quickly achieve a foundational layout. A .pcf file does not provide all information needed to create a working compressible flow model of a system and the import process will not produce a completed model or a perfectly accurate representation of the .pcf model. There will be missing data that must be assigned prior to analysis, such as fluid properties and boundary conditions. However, there are import practices and preparations that can be made to the .pcf file that will make creating model from an imported piping layout go more smoothly.

- **Workspace** - Piping Component Files are a three-dimensional file format, whereas model layouts in the AFT Workspace are a two-dimensional representation of a system. Because the Workspace is a purely visual tool independent of actual system dimensions, the [Scale/Flip Workspace](#) feature can be useful for achieving a readable model when working with imported piping layouts.
- **Object Positioning** - It is recommended to have some distance between most objects (this is not relevant to welds, branch/tap connections, gaskets, and supports). This aids in pipe connections, as objects that are very close together may have trouble connecting automatically. This is especially true of non-isometric models. Spacing between objects also streamlines the process of scaling objects on the Workspace after import.
- **Bends** - By default, Import Bends as Pipes is unchecked, and importing bends as bends is recommended for the sake of model readability and a more flexible approach to incorporating bend losses. Bends that have common angles and r/D ratios present in the internal AFT library can be imported as [Fittings & Losses](#) automatically when importing bends as pipes, but other bends will need to be added manually as junctions, or have their losses specified using the Additional K Factor option in the Pipe Fittings & Losses window or as user-specified fittings & losses using the [Fittings & Losses Library](#). When bends are imported as bends, they will be treated as smooth bends, the upstream pipe determines the cross-sectional flow area, and the r/D ratio and/or angle is imported from the Piping

Component File. If only the r/D ratio or angle is specified, the other parameter will be calculated using the endpoint coordinates.

Note: Bends modeled explicitly can be joined with adjacent pipes as using [Join Pipes/Junctions](#) later if desired. Modeling bends as explicit junctions is advised for models containing bends that are not smooth flanged or have an uncommon angle or r/D ratio. Modeling bends as explicit junctions is recommended in xStream for more accurate modeling of compressible flow except when doing so would lead to unacceptable solver run times. When calculating forces between elbow pairs, modeling bends explicitly is always recommended.

- **Pipes** - While the AFT Pipe Material Library contains many piping materials specified by ASME and ANSI, the Piping Component File format does not always contain these materials and material attributes can be defined within .pcf files in a number of drastically different ways. Because of this, materials, nominal sizes and schedules are not assigned automatically and can be assigned using Link Materials (shown above in Figure 2). Pipes can be grouped by file schedules (i.e. item codes) if they exist or by diameter so that pipe size and material can be applied to a group of pipes quickly. Pipe inner diameter is assigned from the nominal diameter defined in the pipe's coordinates within the .pcf file as a starting point, but any sizes assigned in Link Materials will override these inner diameters within the AFT model. Alternatively, pipes can be defined after import using [Global Pipe Edit](#).

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 Netral 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. Pipes are grouped by diameter by default and can also be grouped by schedule.
 - **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.

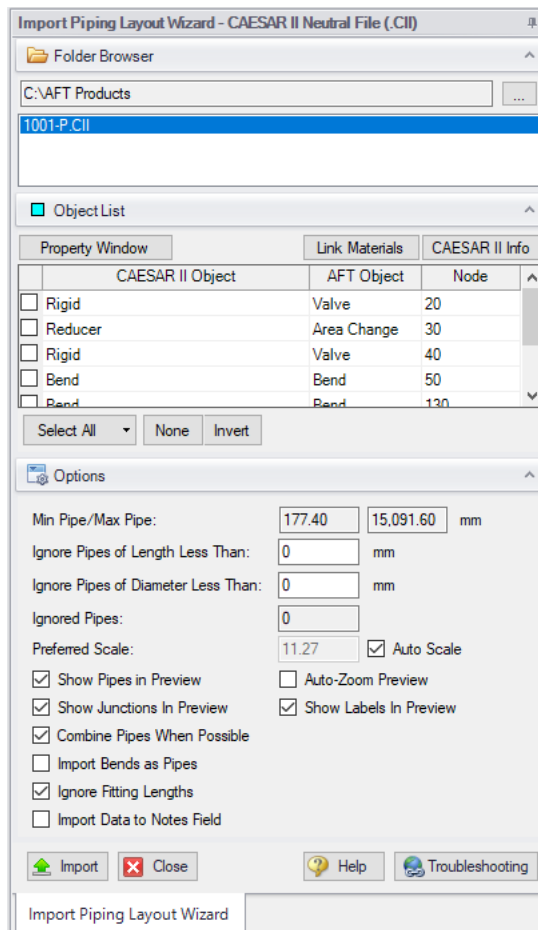


Figure 1: CAESAR II Neutral File Import

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.

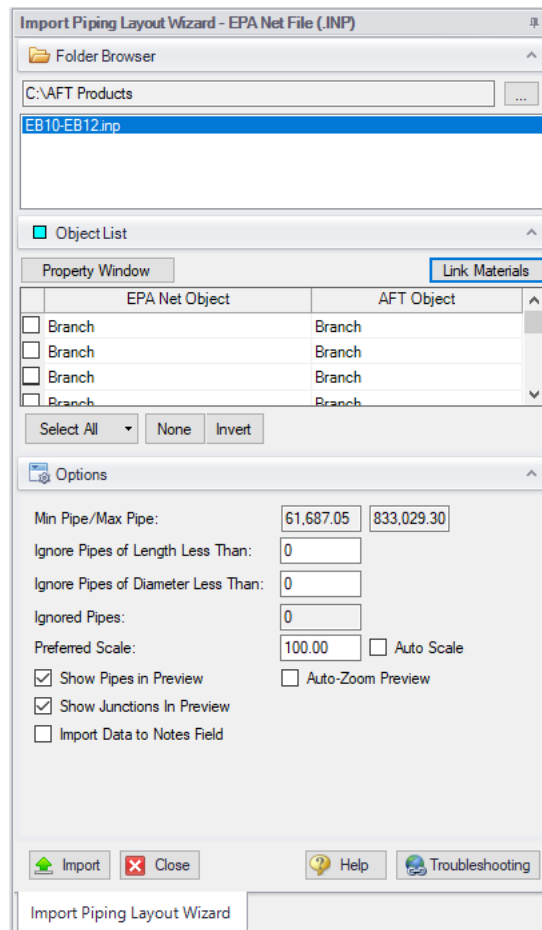
Import Piping Layout Wizard

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.



Pipes and Junctions

All pipes and junctions get imported with user-specified names from EPANET. All junctions below that have flow, elevation, length or other parameters will be imported with the same units if possible.

- **Pumps** - Pumps are imported as compressors with adiabatic compression thermodynamics. The available points on the pump curve are imported into the compressor data and used with the curve fit type set to interpolated X-Y data. If desired, polynomial curve fits can be set up within compressor configuration using this data after the import process is complete.

Note: If only one or two data points exist for the pump curve, the compressor curve will still need to be user-specified.

- **Tanks** - Finite-volume tanks will imported as finite tanks.
- **Reservoirs** - EPANET reservoirs are imported as infinite tanks in xStream.
- **Pipes** - These will use roughness, length, and inner diameter from EPANET model.

- **Junctions** - (Just called Junctions in EPANET)
 - **If one connecting pipe** - If it has a demand, it becomes an Assigned Flow with relevant inflow/outflow. Otherwise, it becomes a dead end.
 - **If multiple connecting pipes** - If it has a demand, it becomes a branch using the Flow Source or Sink optional property with the relevant inflow/outflow imported from the EPANET junction demand. Otherwise, it is imported as a normal branch.
- **Valves** - The K factor of a valve is assigned with the value specified in the EPANET model. Valves can be imported as normal valves, control valves, relief valves, or check valves depending on how they are defined within EPANET:
 - Normal valves come from GPV and TCV in EPANET.
 - Control valves come from FCV, PBV, and PSV in EPANET.
 - Relief valves come from PRV in EPANET.
 - Check valves come from CV in EPANET.
- **Heat Exchangers** - While Heat Exchangers are not an explicitly defined object type in EPANET, EPANET junctions with "Heat Exchanger" in their user-defined name are imported as Heat Exchangers in AFT software.
- **Emitters** - Emitters in EPANET are defined as a junction with a base demand and an emitter coefficient. These are imported as Spray Discharge junctions with no loss model information in AFT xStream. This is because the loss coefficient used by EPANET for emitters, K Fire Sprinkler, is only for incompressible flow. Loss model values and CdA should therefore be defined manually by the user.

Exporting EPANET Files

AFT models can be exported to an EPANET file by using the File -> Export Piping Layout To -> EPANET File (.inp)... option from the Workspace window. The EPANET Export window shown in Figure 2 below allows the user to specify the desired destination to export the file to.

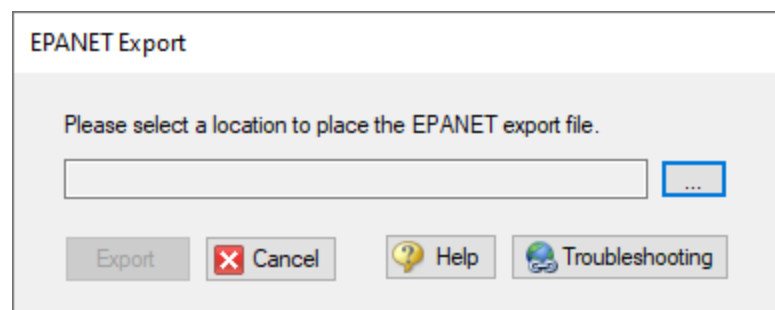


Figure 2: EPANET Import/Export Dialog

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.

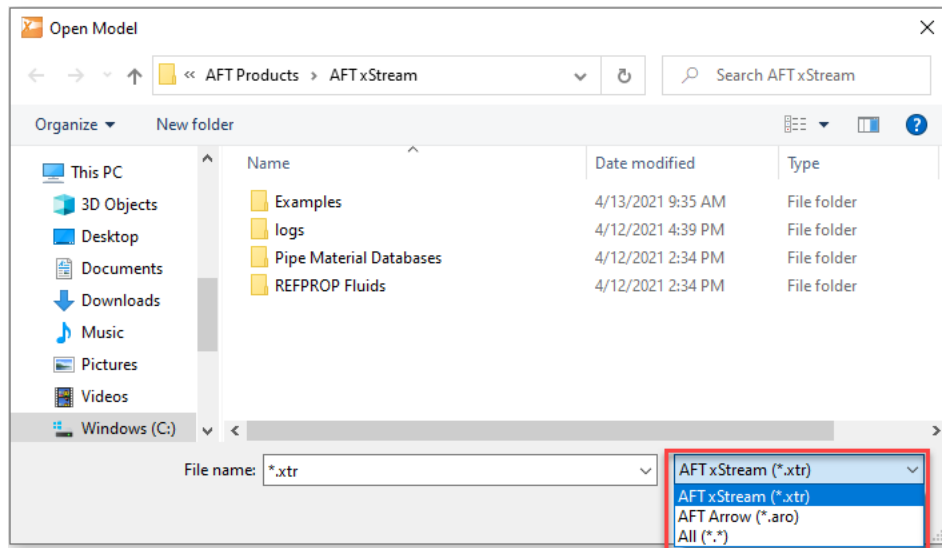


Figure 1: File extensions options for Open File window

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.

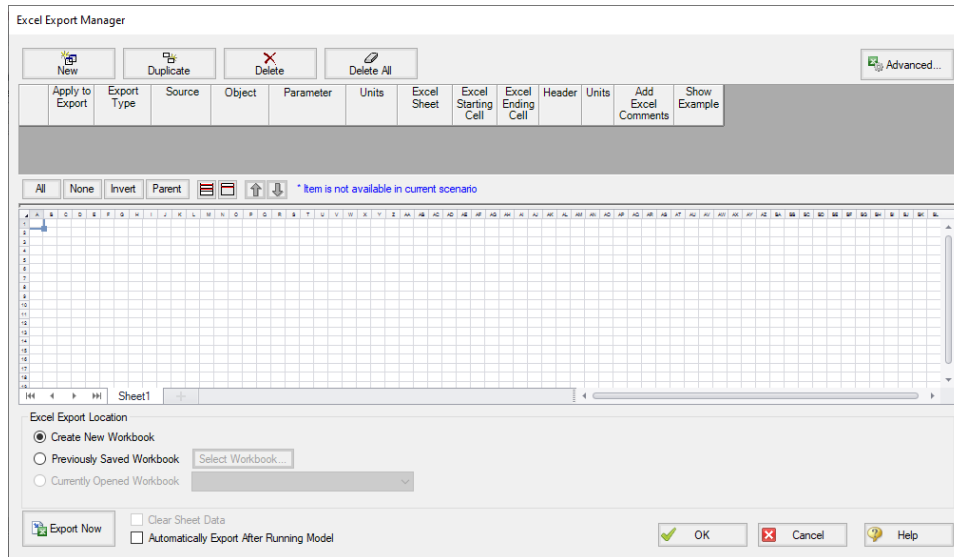


Figure 1: The Excel Export Manager

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.

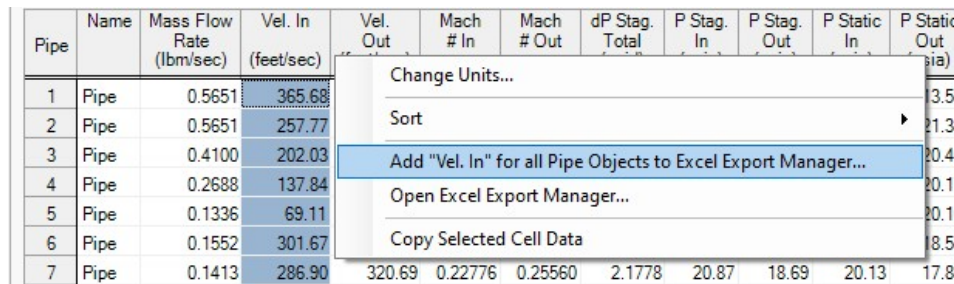


Figure 2: Adding items directly from the Output

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 con-

tains the Parameter name and unit.

- **Show Example** - Displays an example of how the exported cells will appear.

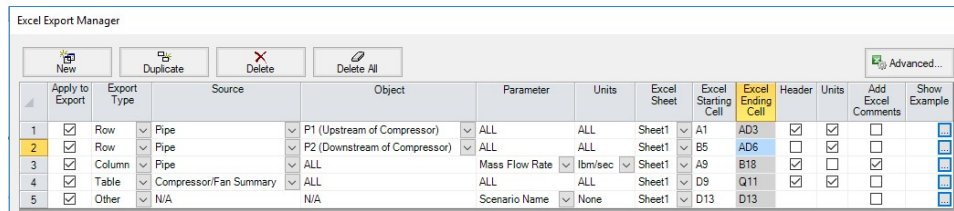


Figure 3: Example Export

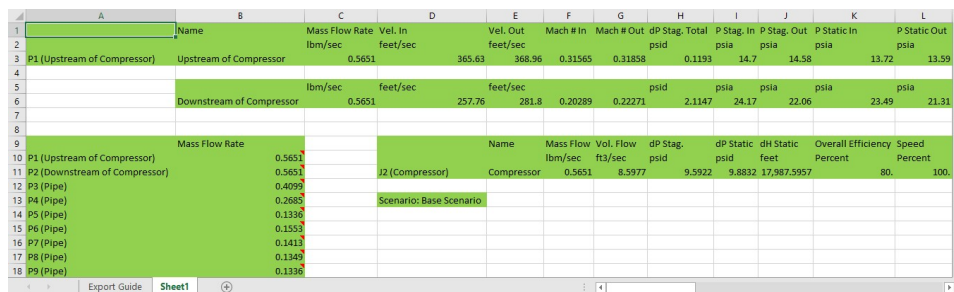


Figure 4: Result of the above example. Color added for emphasis.

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.

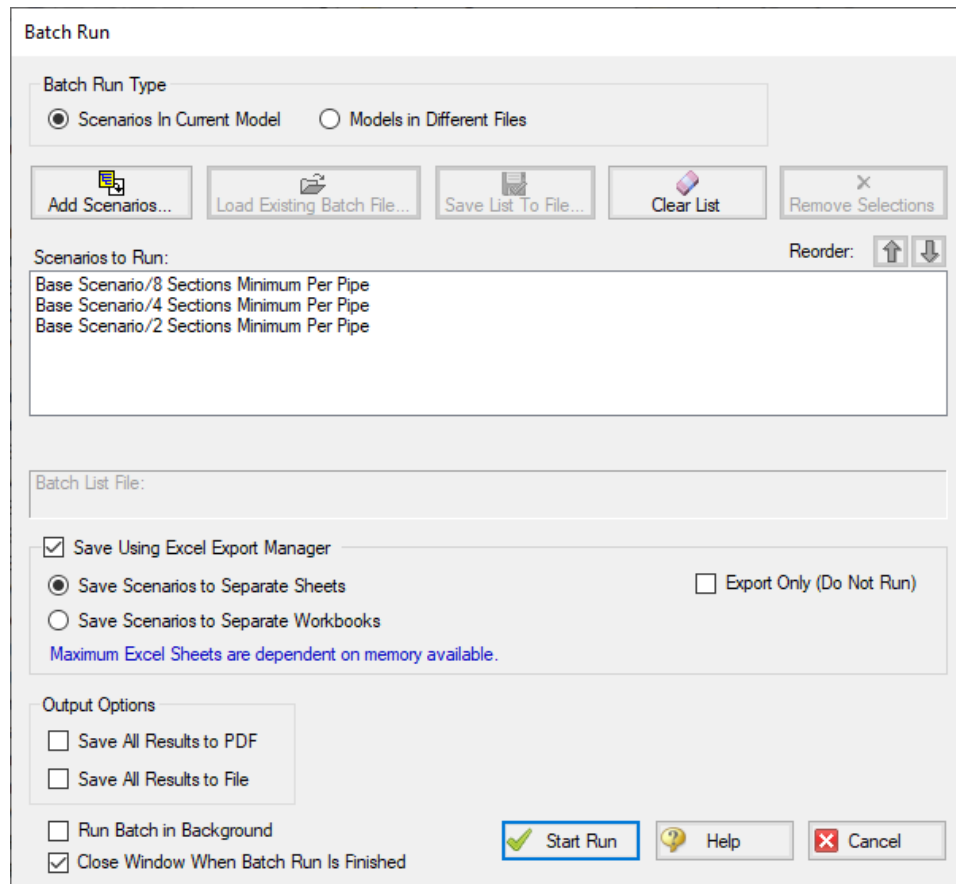


Figure 5: Saving Results From Multiple Scenarios

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

Pipe 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	Area Change S CdA (for sonic choking) S Elevation C Junction Name S Loss Value	Assigned Flow S Flow Setting S Flow Temperature S Elevation C Junction Name I Special Condition	Assigned Pressure S Elevation C Junction Name S Pressure Setting S Temperature	Branch S Elevation C Junction Name
Check Valve 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	Compressor/Fan S Assigned Flow S Elevation C Junction Name S Nominal Efficiency I Special Condition S Speed (%)	Control Valve S Control Setting S Elevation S Full Open Loss C Junction Name I Special Condition S Xt Value	Dead End S Elevation C Junction Name	General Component S CdA (for sonic choking) S Elevation C Junction Name S Loss Value I Special Condition
Orifice S Area/Diameter S CdA (for sonic choking) S Elevation S Exit Pressure S Exit Temperature C Junction Name S Loss Value	Relief Valve 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	Tank S Elevation C Junction Name S Pressure S Temperature	Tee or Wye S Elevation C Junction Name	Valve 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.

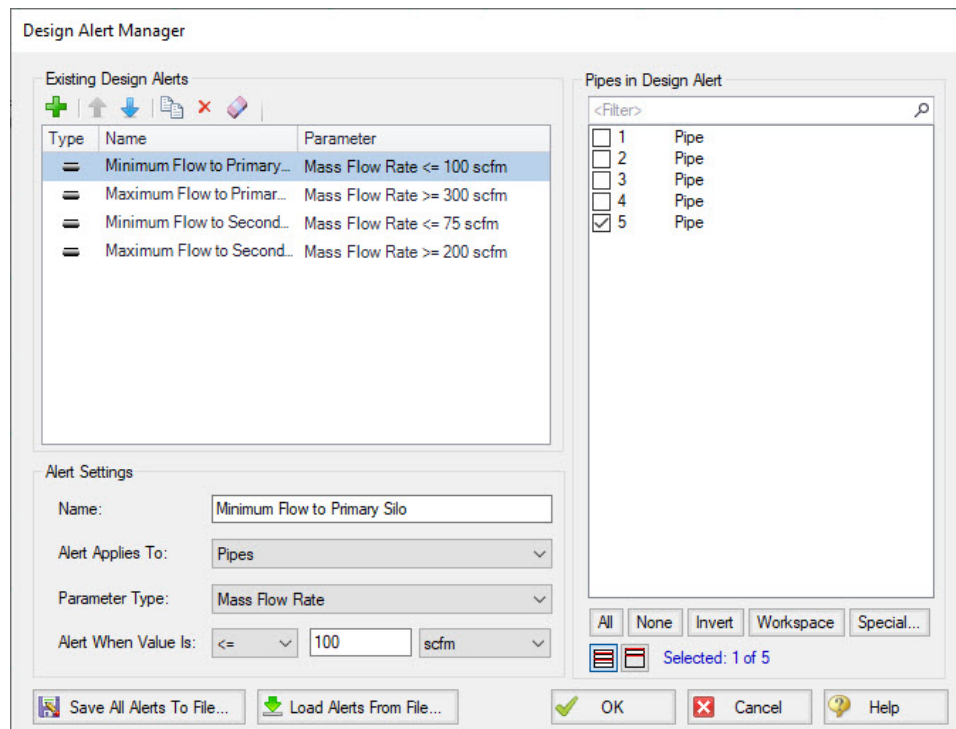


Figure 1: Design Alert Manager

Creating and Applying Design Alerts

Click the green plus sign (New Design Alert) to add a new Design Alert to the model.

A Name, Object Type, and Parameter Type and Value can be specified. Depending on the Object Type, a different selection of Parameters are available, as shown below. To add a Parameter:

Design Alerts

1. Give the Design Alert a distinct and descriptive name.
2. Select the Object Type the alert will be applied to.
3. Select the Parameter Type the alert will monitor.
4. Specify a value, unit if applicable, and companion.
 1. Use \geq when the value shouldn't rise above the specified value (i.e. temperature shouldn't exceed a maximum value)
 2. Use \leq when the value shouldn't fall below the specified value (i.e. pressure shouldn't drop below a minimum value)
5. Select which Objects from the Workspace the Design Alert will be applied to.
6. Click "OK" to save changes.

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.

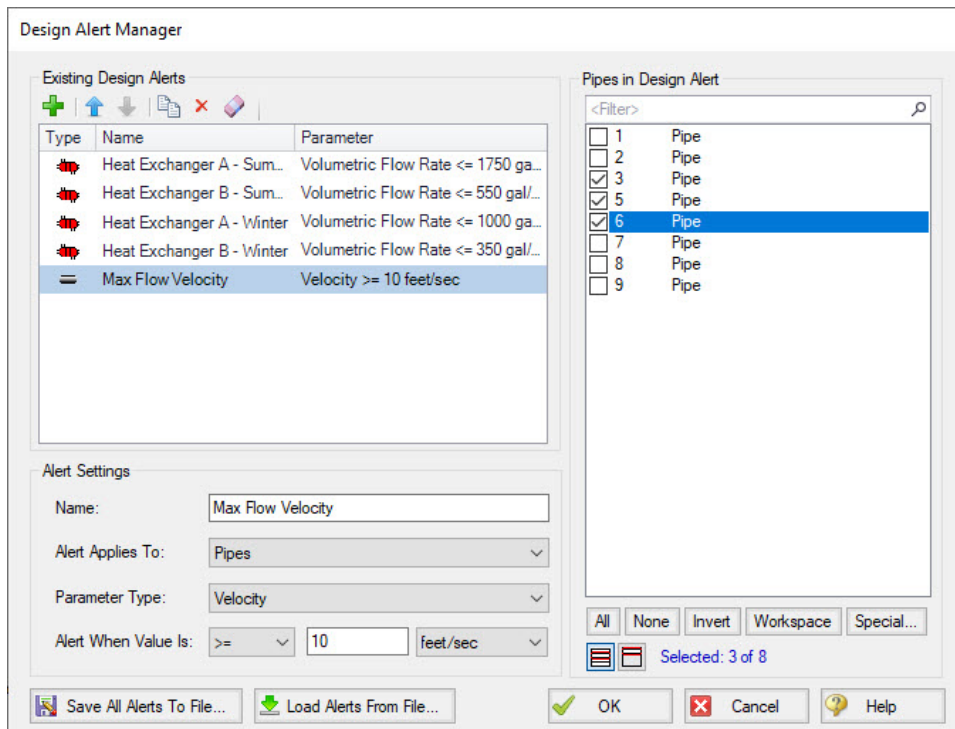


Figure 2: Creating a Design Alert

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

When running the model, the selected object's output values will be checked against the conditions that have been defined. If any of the Design Alert conditions are met, the user will receive a Design Alert [warning](#) and the violating parameters will be highlighted in the [output](#) tables. Design Alert warnings can be double-clicked to take you to that object in the Workspace.

The screenshot shows the 'Design Alerts' window with a list of alerts under the heading 'Pipe Design Alerts'. The alerts are:

- Velocity is above the maximum design alert of 12 feet/sec at Pipe 112
- Velocity is above the maximum design alert of 12 feet/sec at Pipe 113
- Velocity is above the maximum design alert of 12 feet/sec at Pipe 114
- Velocity is above the maximum design alert of 12 feet/sec at Pipe 212
- Velocity is above the maximum design alert of 12 feet/sec at Pipe 213
- Velocity is above the maximum design alert of 12 feet/sec at Pipe 214

Below the alerts is a table with the following columns: Pipe, Name, Vol. Flow Rate (gal/min), Velocity (feet/sec), P. Static Max (psia), P. Static Min (psia), Elevation Inlet (feet), Elevation Outlet (feet), dP Stag. Total (psid), and dP. Static Total (psid). The table data is as follows:

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P. Static Max (psia)	P. Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP. Static Total (psid)
109	Return 1	19,170	9.153	31.07	28.00	0.000	0.00	3.0690	3.0690
110	CT 1 Hdr A	12,609	9.529	27.95	27.80	0.000	0.00	0.1514	0.1514
111	CT 1 Hdr B	6,210	8.525	27.92	27.75	0.000	0.00	0.1738	0.1738
112	Riser 1/1	6,210	14.450	26.84	15.45	0.000	25.00	11.3934	11.3934
113	Riser 1/2	6,398	14.888	26.93	15.49	0.000	25.00	11.4353	11.4353
114	Riser 1/3	6,559	15.262	27.00	15.53	0.000	25.00	11.4708	11.4708
200	Circ Pump 2A Suction	9,542	10.519	16.30	13.96	6.000	0.00	-2.3493	-2.3493
201	Circ Pump 2A Disch	9,542	10.519	40.26	39.96	0.000	0.00	0.2943	0.2943

At the bottom of the interface, a status bar indicates 'Design Alerts Exist' and shows a search icon and a 100% zoom level.

Figure 4: 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](#) 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...

Pipe (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)	
1	Pipe	0.5586	361.03	364.23	0.31161	0.31442	0.1163	14.70	14.58	13.74	13.61	0.002258	
2	Pipe	0.5586	254.34	279.							15	21.41	0.029785
3	Pipe	0.4048	200.06	211.							16	20.58	0.017273
4	Pipe	0.2651	137.36	140.							11	20.28	0.007813
5	Pipe	0.1315	69.57	70.							11	20.27	0.002075
6	Pipe	0.1539	299.82	340.							10	18.68	0.036743
7	Pipe	0.1399	287.07	322.							15	17.98	0.031921
8	Pipe	0.1333	280.17	312.							17	17.68	0.029602

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 saved 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.

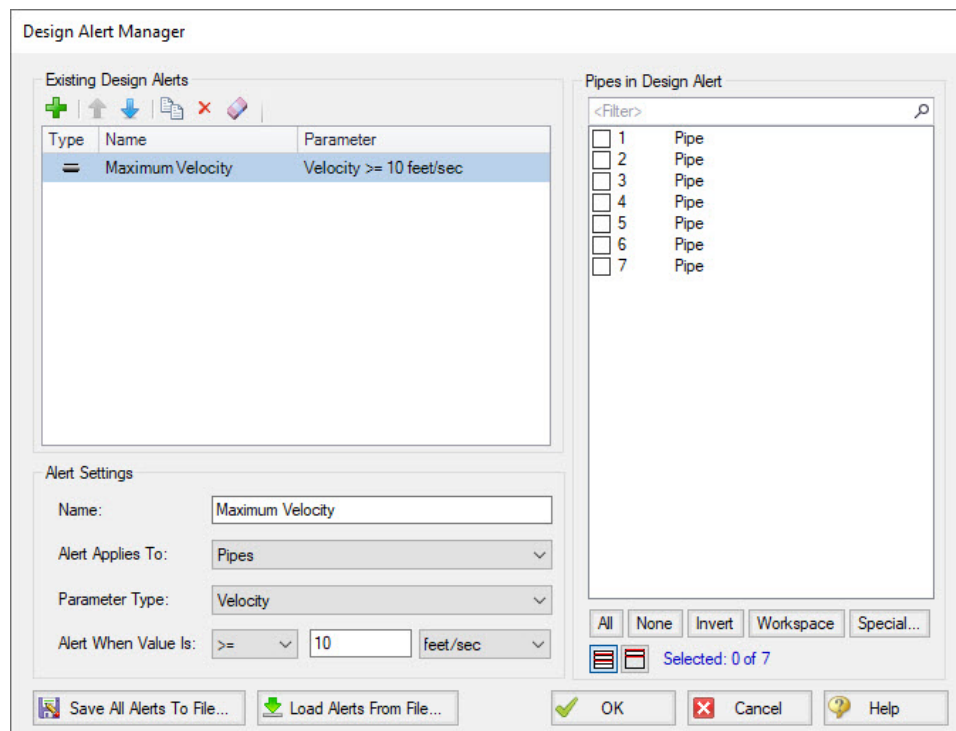


Figure 1: A design alert for velocity to be exported

2. Select "Save All Alerts To File..." and give the file a descriptive name.

Import a Design Alert

1. Open the Design Alert Manager from the Tools menu.
2. Select "Load Alerts From File...", then browse to and select 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.

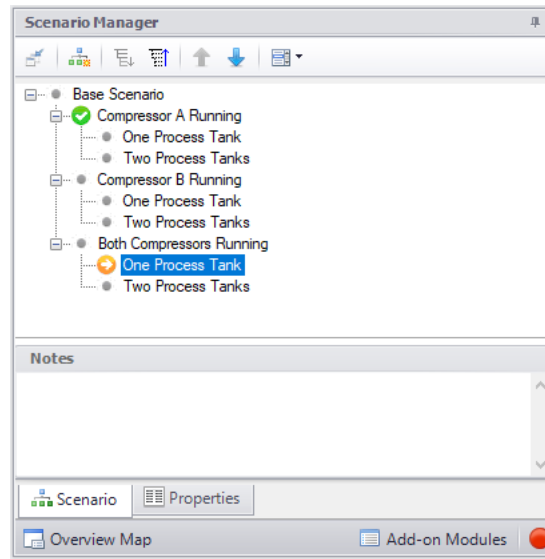


Figure 1: Scenario Manager on Quick Access Panel

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.

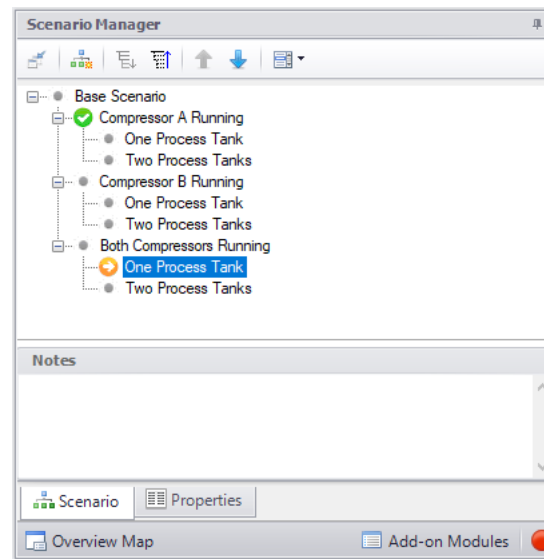


Figure 1: Scenario Manager Window

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.xtr :: 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. The Last Scenario feature is found on the [View menu](#) and [Common 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

[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, or Library Manager.

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 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.

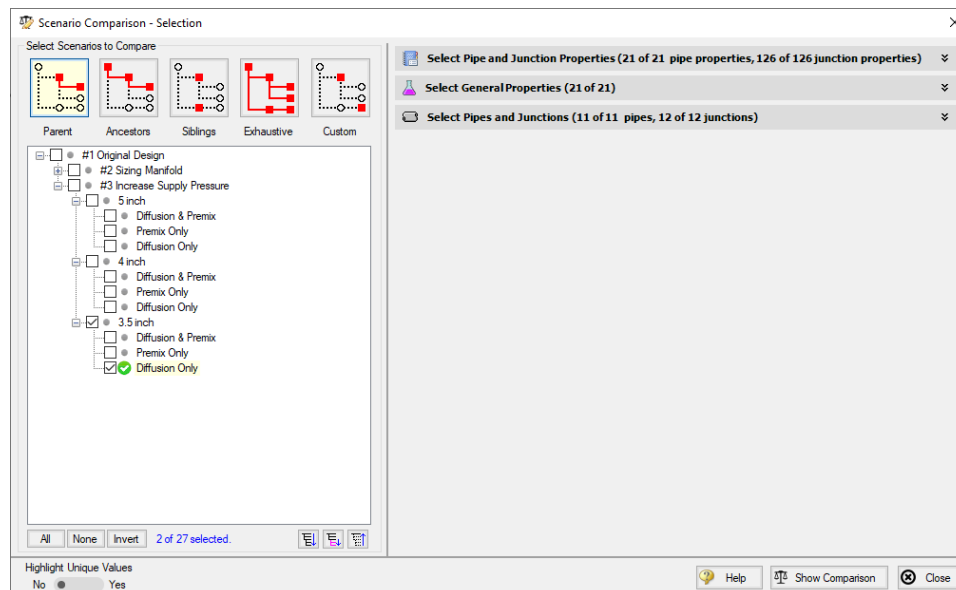


Figure 1: Scenario Comparison window options to Modify Comparison

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		
Differences <input type="radio"/> All <input checked="" type="radio"/>		Differences <input type="radio"/> All <input checked="" type="radio"/>		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
Junctions (12 of 12 compared)						
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 checked="" type="radio"/>
At Least 1 Parameter Difference	Different Values Across Scenarios	No <input type="radio"/> Yes <input checked="" 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.

Item		Parameter		Scenario		
Differences	Show	Differences	Show	All		
				3.5 inch	4 inch	Diffusion Only
P4	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
No Differences						
P7	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
No Differences						
P10	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches	3.548 inches	3.548 inches
		3-1/2 inch	4 inch	3-1/2 inch	3-1/2 inch	3-1/2 inch
Junctions (12 of 12 compared)						
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	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	Closed (stays closed)	Closed (stays closed)	Closed (stays closed)
J9 (Combustion Baskets) No Differences						
J10 (Bend) No Differences						
J11 (Premix Gas FCV 2)	Special Condition	None	None	Closed (stays closed)	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.

All Parameters Same Value	Same Value Across Scenarios	Highlight Unique Values	No <input type="radio"/> Yes <input checked="" type="radio"/>	Copy	Help	Modify Comparison	Close
At Least 1 Parameter Difference	Different Values Across Scenarios						

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 peak-to-peak pressure limits defined in the API 618 standard for [reciprocating 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

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](#).

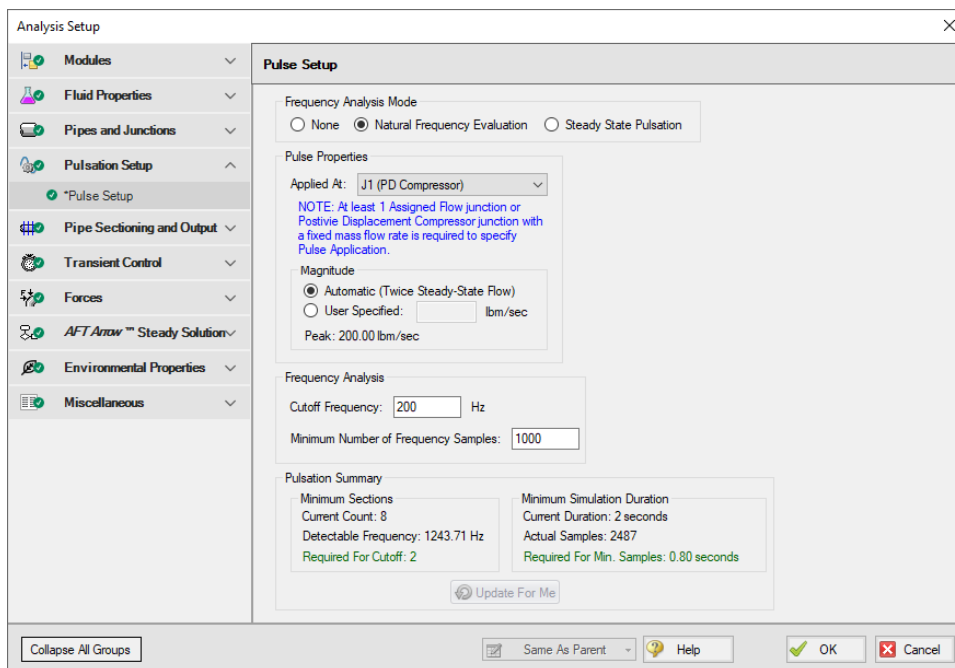


Figure 1: Pulse Setup panel in the Analysis Setup window with Natural Frequency Evaluation selected

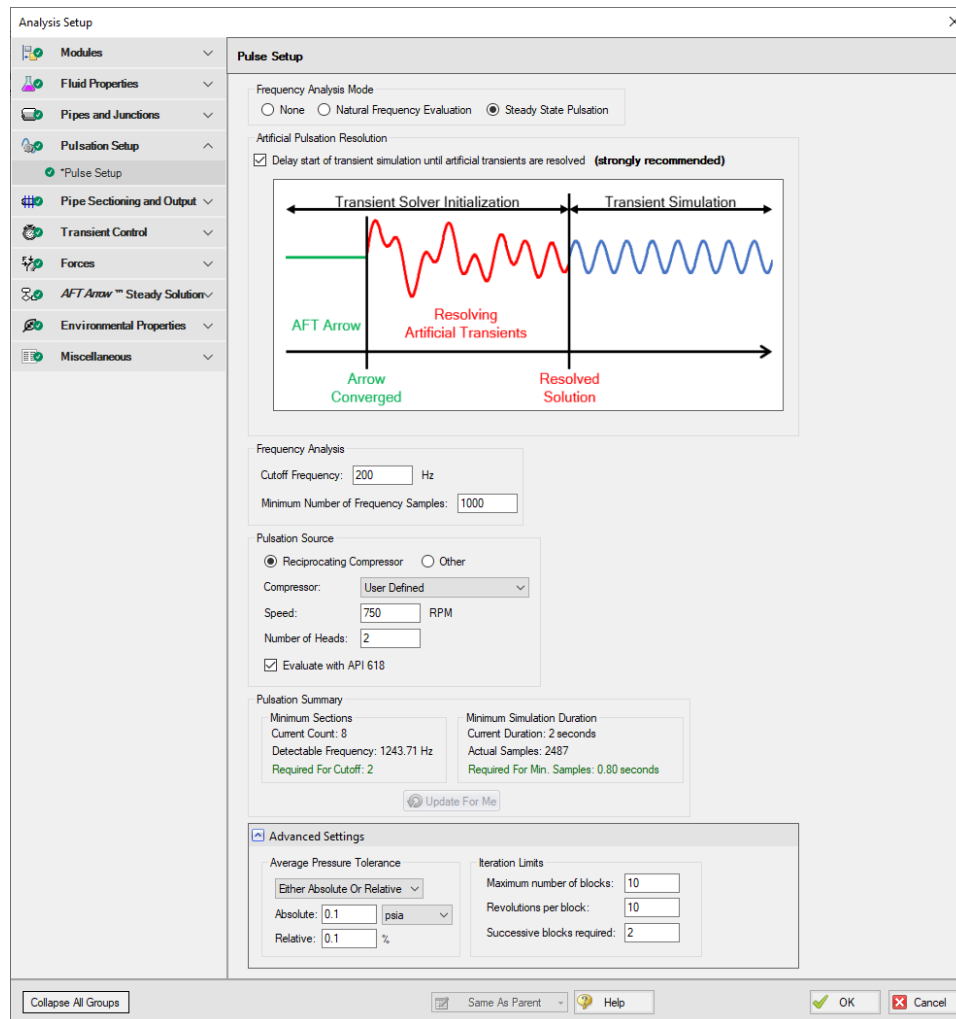


Figure 2: Pulse Setup panel in the Analysis Setup window with Steady State Pulsation selected

Frequency Analysis Mode

At the top of the window the Frequency Analysis Mode can be specified. The options include: Natural Frequency Evaluation, Steady State Pulsation, and None. The Natural Frequency Evaluation will produce a single transient pulse at a specified assigned flow or fixed flow rate positive displacement compressor junction so as to determine the frequencies that would excite the system. The Steady State Pulsation option will determine how a pulsation source, such as a reciprocating compressor, would interface with a system in steady operation.

Natural Frequency Evaluation

When Natural Frequency Evaluation is selected, 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 or [fixed flow rate positive displacement compressor](#).

Steady State Pulsation

The Steady State Pulsation analysis mode allows for the evaluation of pressure amplitude for each frequency at each pipe station in the model when these frequencies are excited by a reciprocating compressor. The reciprocating compressor can be modeled explicitly using the [reciprocating positive displacement compressor](#) junction, or approximated by applying a sinusoidal flow transient to an assigned flow junction. In practice, the natural frequencies of the system may be used along with harmonic multiples to identify compressor RPMs of particular concern.

The Steady State Pulsation analysis mode also offers acoustical evaluation of the reciprocating compressor and system interaction based on the API 618 standard.

Artificial Pulsation Resolution

It is highly recommended that the option to delay transient simulation until artificial transients are resolved be left enabled. This resolution method is based on comparison of average pressures between blocks of compressor rotations, and operating without this method will result in less accurate frequency evaluation. Pressure is averaged over the course of each block, and this value is compared to the previous block until it meets a specified tolerance. This method is distinct from the one used in the [Transient Solver Initialization Panel](#), which is not used and will not appear when using Steady State Pulsation analysis.

The parameters for pulsation resolution can be modified in the Advanced Settings section at the bottom of the Pulse Setup panel.

- **Average Pressure Tolerance** - Similar to solver tolerance settings, this defines the pressure tolerance from block to block that must be achieved for a block to pass tolerance requirements. This can be defined using absolute tolerance, relative tolerance, or any combination thereof.
- **Iteration Limits**
 - **Maximum number of blocks** - The maximum number of blocks that may be evaluated without success before the artificial transients are considered to be unresolved.
 - **Revolutions per block** - This determines the number of time steps per block based on number of crankshaft rotations, the time step size, and compressor RPM.

Note: If the default value of 10 is not sufficient, try increasing this value to 20.

- **Time steps per block (Other Pulsation Source)** - When using Other as the pulsation source, this parameter replaces revolutions per block.

Successive blocks required - This defines the number of consecutive blocks that must meet tolerance requirements before artificial transients are considered resolved.

Pulsation Source

The pulsation source can be set to Reciprocating Compressor or Other. The Other option allows a user to excite the system with any arbitrary pulsation source. In practice this takes the form of cyclical user-defined [Junction Transient Data](#) applied to one or multiple junctions in the system. The Reciprocating Compressor option allows a reciprocating positive displacement compressor junction or assigned flow junction to be selected, and has the following parameters.

- **Compressor**
 - **Reciprocating Compressor** - Select an existing reciprocating compressor from the model listed by junction number.

Note: If there are multiple reciprocating compressors with the same speed and number of heads, any may be selected. If these parameters differ between compressors, use the Other option for Pulsation Source.

- **User Defined** - This option is selected when using one or more assigned flow junctions to represent reciprocating compressors. The appropriate transient must be defined in the Transient tab of the assigned flow junction, typically using the periodic flow feature. The Speed and Number of Heads parameters do not define this transient flow, but are used for Artificial Pulsation Resolution and API 618 and should match the assumptions that were used to define the transient flow.
- **Speed** - The current operating speed of the compressor in RPM. If a reciprocating compressor junction is selected, this will display the value defined in the Compressor Properties window.
- **Number of Heads** - This is the number of compression chambers in the compressor, which is the number of cylinders in a single-acting compressor and twice the number of cylinders in a double-acting compressor. If a reciprocating compressor junction is selected, this parameter will be determined from the compressor properties.
- **Evaluate with API 618** - When this option is checked, an API 618 acoustical analysis will be performed. More detail about this analysis is provided in the section below.

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.

Note: In Steady State Pulsation, the cutoff frequency should be at least six times the base pulsation frequency.

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, under the Steady State Pulsation selection, 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 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.

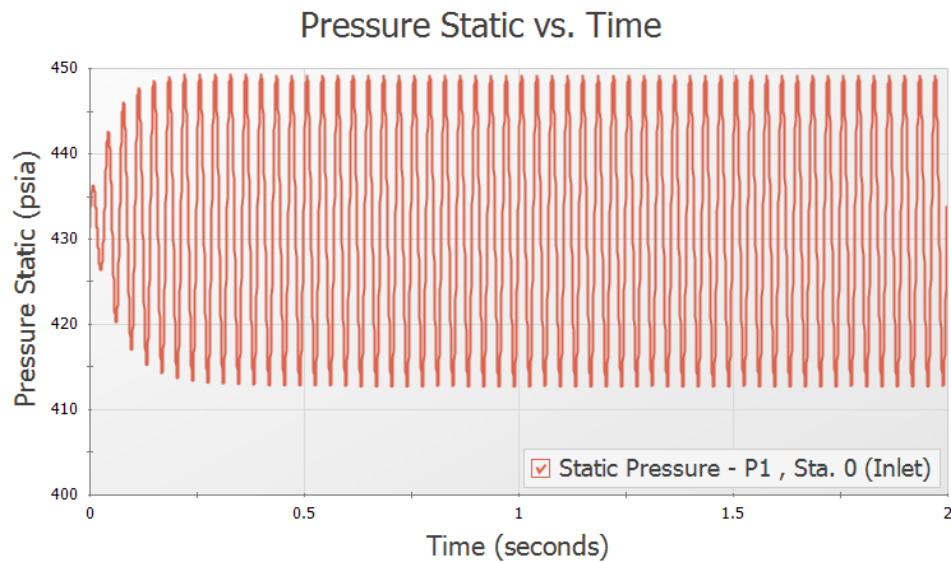


Figure 3: Example pressure results at the outlet of the reciprocating compressor with an oscillating flow transient specified

With output an Acoustical Frequency Analysis can be graphed. When API 618 is being evaluated, the limit and the frequency will be graphed together as shown in Figure 4.

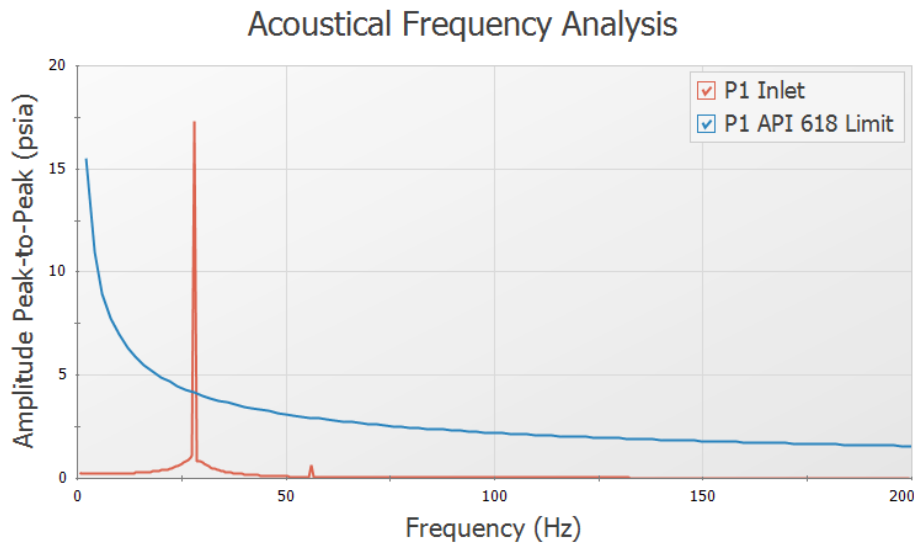


Figure 4: Example Acoustical Frequency Analysis Plot including the API-618 Limit

The equations for the API 618 peak to peak pressure level limits are as follows ([American Petroleum Institute 2007](#), Clause 7.9):

$$P_1 (\%) = \sqrt{\frac{a}{1150}} \left(\frac{300}{(P_L ID f)^{0.5}} \right) \quad \text{US Units}$$

$$P_1 (\%) = \sqrt{\frac{a}{350}} \left(\frac{400}{(P_L ID f)^{0.5}} \right) \quad \text{SI Units}$$

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), f is the pulsation frequency in hertz, and a is the speed of sound inside the pipe in feet per second (meters per second).

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 Topics

Pulse Setup Panel.....	103
The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X locations.....	560
Pulsation peak-to-peak pressures are within allowable API 618 levels.....	570

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 fre-

quencies, 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..... 490

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 (f_c)** - 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

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

Powerful customization capabilities are provided through the ability to create libraries which can store custom fluid properties, pipe materials, and all junction types. There are several categories of libraries which are discussed below.

AFT Internal Library

- **Purpose** - Stores AFT-provided library data that is always available with the default installation.
- **Filename** - xStream#.dat (where # is the corresponding version number).
- **Location** - xStream installation folder, default is C:\AFT Products\AFT xStream #.
- **Limitations** - Cannot be transferred to other users, local use only.

Local User Library

- **Purpose** - Stores user-specified data for most component types.
- **Filename** - xtr_User#.DAT (where # is the corresponding version number).
- **Location** - C:\Users\username\AppData\Roaming\Applied Flow Technology\AFT xStream\#.
- **Limitations** - Cannot be transferred to other users, local use only.

External Library

- **Purpose** - Stores user-specified data of any type: fluids, pipe materials, junctions, and fittings & losses. Multiple network or local custom libraries can be created via 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](#).
- **Filename** - User-specified filename.
- **Location** - User-specified location.
- **Limitations** - Must be connected via the Library Manager prior to use, and consideration should be given to version control or user permissions to edit.

Library Manager	444
Network Library Overview	456
Library Verification	460
Library Locations in General Preferences	462

Library Manager

Accessed from the [Library Menu](#), 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

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.

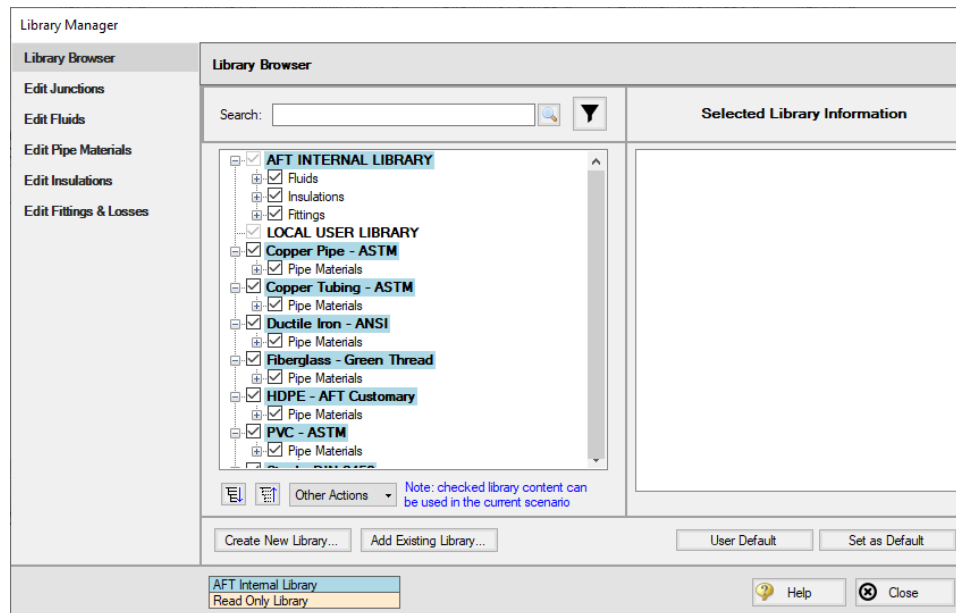


Figure 1: Library Browser in the Library Manager

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. Individuals components can

also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.

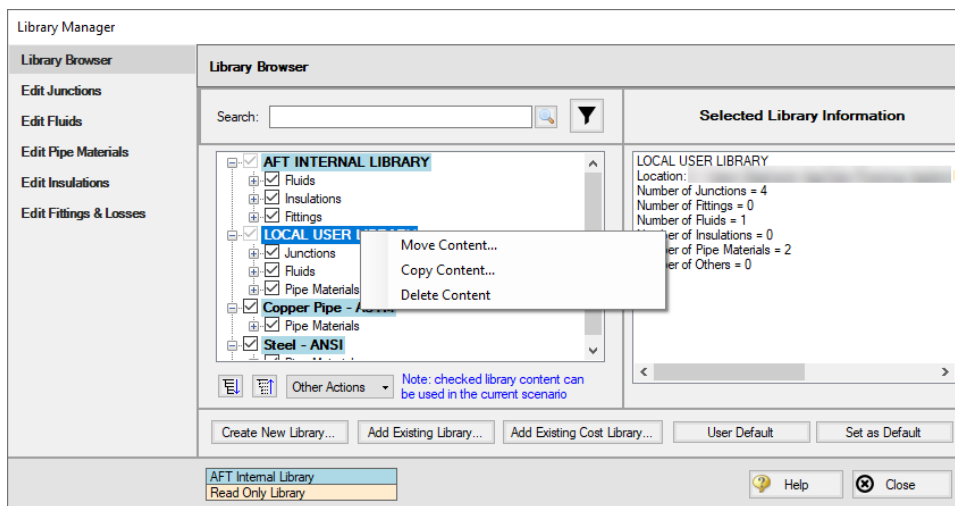


Figure 2: Library contents can be transferred or deleted from the right-click menu

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](#).

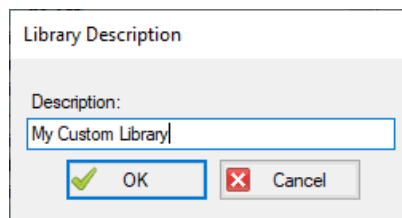


Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected

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 Jct dropdown.

AFT xStream accepts custom for all junction types.

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.

Fluid Library

AFT Arrow provides an [AFT Standard 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 reuse 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 User Library Fluid is selected.

Adding a Custom Fluid

You can add new fluids by clicking the Add New Fluid button on the Edit Fluids panel of the Library Manager. A number of fluid properties are required for a compressible flow analysis. There are also several optional properties.

Required Data:

- Dynamic Viscosity
- Specific Heat (at constant pressure)
- Reference Temperature/Enthalpy point
- Molecular Weight

Optional Data:

- Thermal Conductivity (required for heat transfer capabilities besides adiabatic/isothermal in pipes)
- Vapor Pressure (required if checks and warnings for saturation are desired)
- Compressibility Factor
- Critical property data points are used for some Equation of State and enthalpy models

Except for the constants 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 can be modeled with polynomials or interpolation, 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 xStream allow you to choose different ways of handling the pressure dependence.

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 77

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.

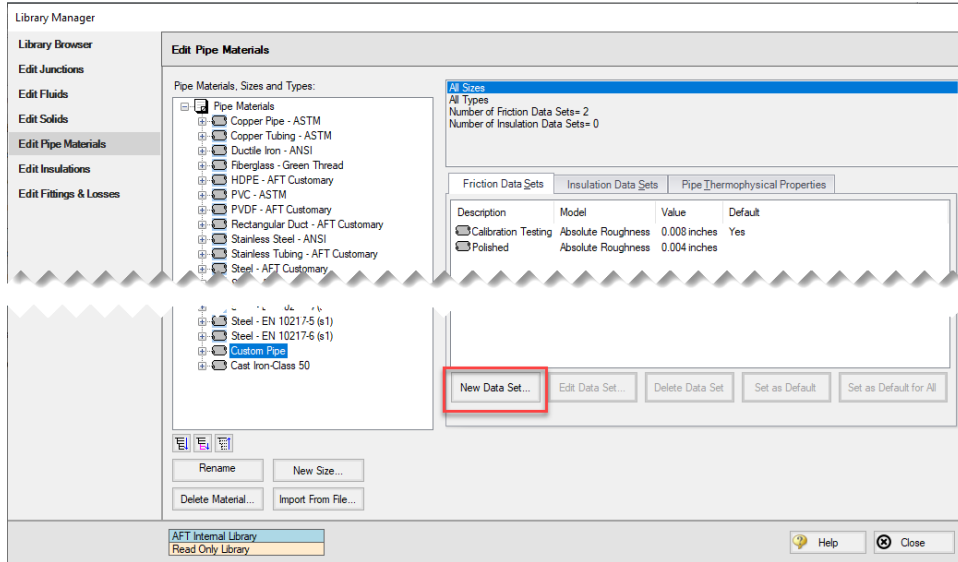


Figure 1: Friction Data Set tab on the Edit Pipe Materials panel

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.....	77
Pipe Material Library References.....	511

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			
	None			
Ball	Plane disk - AR=0.92			
Butterfly	Plane disk			
	t/D=0.1			

		<p>t/D=0.15</p> <p>t/D=0.2</p> <p>t/D=0.25</p> <p>t/D=0.3</p> <p>t/D=0.35</p> <p>Type A</p> <p>Type B</p> <p>Type C</p>		
	Cylindrical	<p>General</p> <p>Rect. Tube</p>		
	Gate	<p>Wedge/Plug</p> <p>Wedge</p> <p>Disk w/o recess</p> <p>Disk with recess</p> <p>Disk</p> <p>Wedge - Full Open 10 deg.</p> <p>Wedge - Full Open 20 deg.</p> <p>Sluice</p>		
	Globe	<p>Disk with recess</p> <p>90 degree wall</p> <p>45 degree wall</p> <p>Standard 45 deg. wall</p> <p>Standard 90 deg. wall</p>		
	Plug	<p>Standard</p>		
	Poppet	<p>Y Type</p> <p>Straight-way</p>		
		<p>W/D=.035</p>		

	Three-way	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 Nozzle Venturi 7 deg.	Area Ratio None	K (constant) k / FT (Crane)

		Venturi 3 deg. Dall Tube		
	Honeycomb cell	L/D=2		
		L/D=4		
		L/D=6		
L/D=8				
L/D=10				
Screen	L/D=12			
	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..... 190
 Specifying Losses..... 220

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 xtr_User#.DAT (where # is the corresponding version number) in the local user directory. This data is not shared and is always available to the local user for access or for editing. See the [Libraries](#) topic for more information.

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: 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
xStream Internal Libraries		
XSTREAM3.DAT	AFT Default Internal Library	AFT xStream 3 folder
XTR_USER3.DAT	AFT xStream Local User Library	AppData folder
External Shared Libraries (DATABASE.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in DATABASE.LIB
External Libraries (XTR_DBUSER.LIB)		
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_USER3.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 3\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 3\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 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.

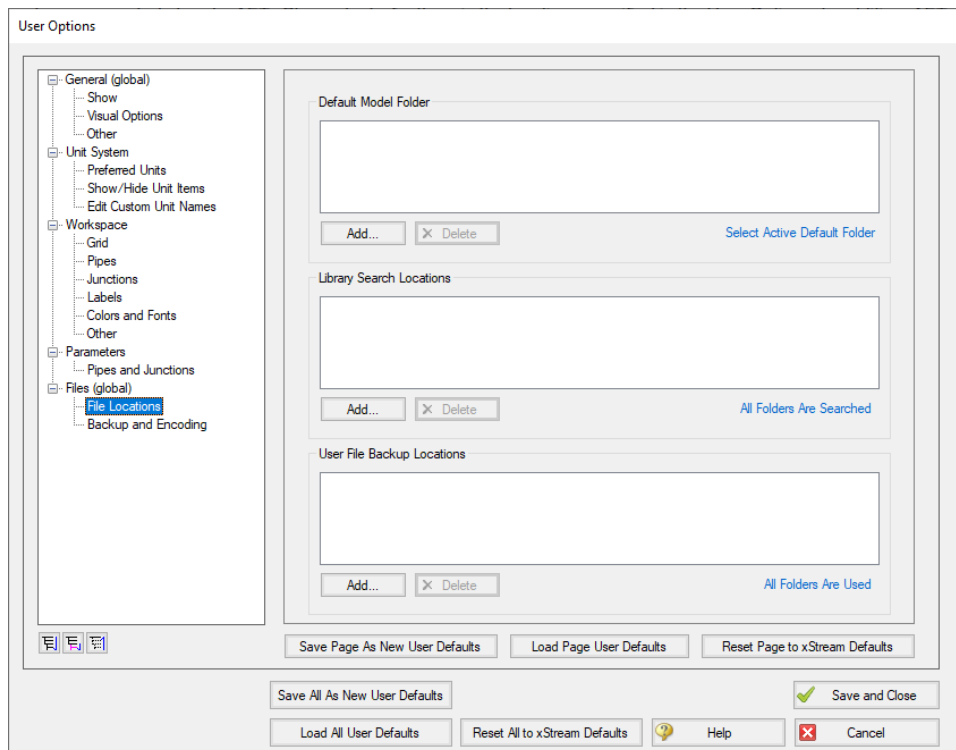


Figure 1: Library Search Locations can be defined in the User Options window

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

Nomenclature	463
Review of Compressible Flow Theory	465

Nomenclature

a	sonic speed
A	cross-sectional flow area of a pipe
B	compatibility equations parameter for method of characteristics
C	compatibility equations 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
α, μ, θ	angle
β	diameter ratio
β	coefficient of volume expansion
λ	method of characteristics multiplier
η	efficiency
ν	kinematic viscosity
ε	roughness
γ	specific heat ratio
ρ	density
τ	shear stress
ω	rotational velocity
μ	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
∞	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

$$P = \left(\frac{R}{\bar{M}} \right) \rho T$$

where R is the universal gas constant and \bar{M} is the molecular weight of the gas. These two terms are frequently combined into a constant R_g

$$P = R_g \rho T \quad (63)$$

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.

$$P = Z R \rho T \quad (64)$$

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

$$\delta q + \delta w = \delta e \quad (65)$$

For a reversible process, the work term can be represented by the following

$$\delta w = -P dv \quad (66)$$

where v is the gas specific volume (inverse of density). In addition, the incremental addition of heat is given by

$$\delta q = T ds \quad (67)$$

where s is the entropy. Substituting Equations 66 and 67 into 65 and rearranging yields

$$T ds = de + P dv \quad (68)$$

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 γ 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 γ 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 γ 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)

$$a = \sqrt{\gamma RT} \quad (86)$$

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

$$a = \sqrt{Z \gamma RT} \quad (87)$$

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.

$$M = \frac{V}{a} = \frac{V}{\sqrt{Z \gamma RT}} \quad (88)$$

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 γ , 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.

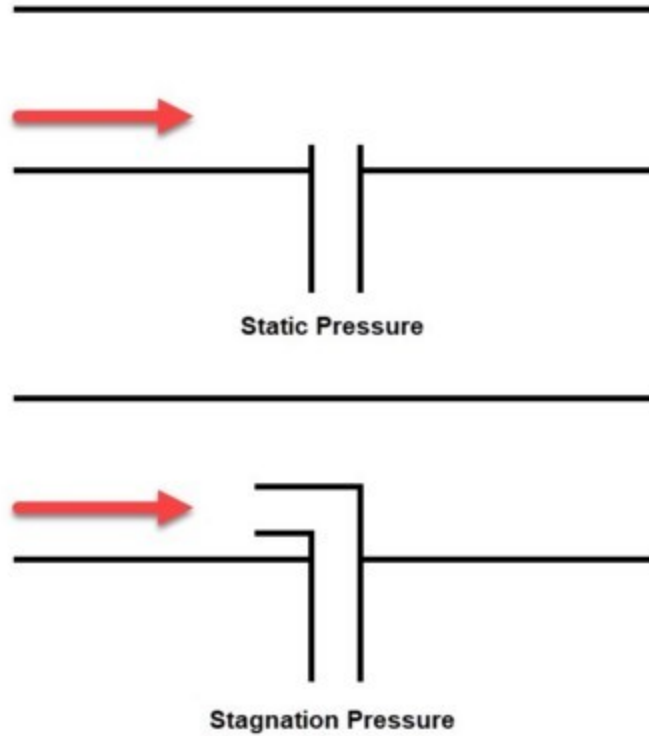


Figure 1: Comparison of static and stagnation pressure measurement at a pitot tube.

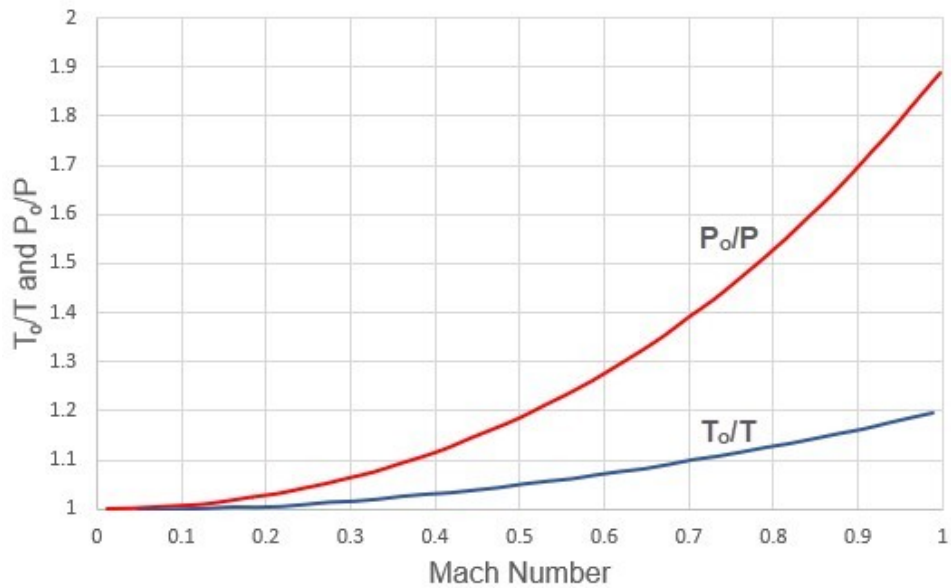


Figure 2: Variation of stagnation to static temperature ratio and stagnation to static pressure ratio with Mach number. Gamma = 1.4.

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 τ_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.

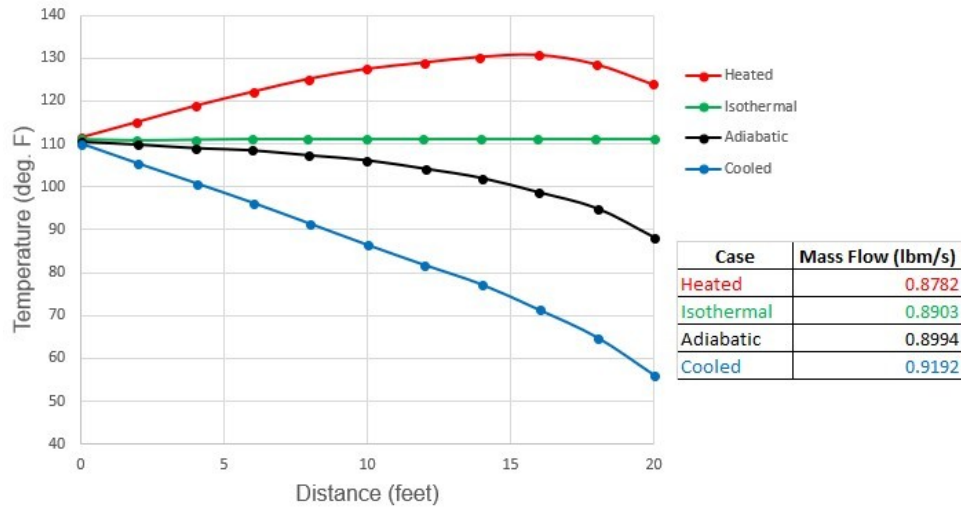


Figure 3: Comparison of temperature profiles and flow rate predictions for adiabatic, isothermal, heated and cooled air flow in a 20 ft. long 1 inch steel pipe. Inlet conditions are 100 psia, 111 °F and outlet pressure is 60 psia. Cooled flow has ambient temperature of 30 °F. Heated flow has ambient temperature of 220 °F. Both heated and cooled cases have pipe wall and insulation heat transfer coefficients which total to 100 Btu/hr-ft²-F. Note how adiabatic and isothermal cases do not bracket conditions. See [Walters, 2000](#).

Related Topics

Stagnation vs. Static Pressure Boundaries.....	79
Stagnation Properties.....	510
Nomenclature.....	463
References.....	516

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).

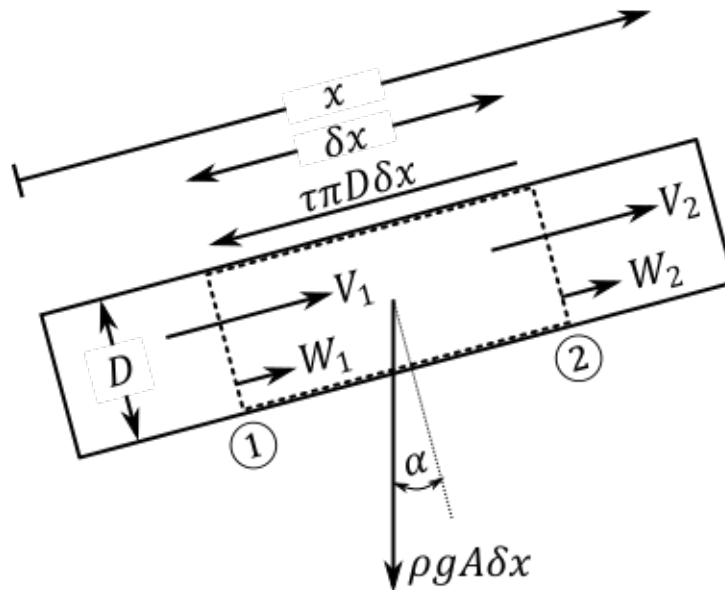


Figure 1: Control volume

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 δx

$$\frac{\partial}{\partial t} (\rho A) + \frac{\partial}{\partial x} (\rho A V) = 0$$

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 δx , 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 ρA , we find the continuity equation and can drop those terms.

$$\frac{V}{\rho A} \left(\underbrace{\frac{\partial}{\partial t}(\rho A) + \frac{\partial}{\partial x}(\rho A V)}_{\text{Continuity Eqn} = 0} \right) + \frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} = -\frac{1}{\rho} \frac{\partial P}{\partial x} - \frac{f V |V|}{2 D} - g \sin(\alpha)$$

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 ∂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.

$$\boxed{\frac{\partial h_o}{\partial t} + V \frac{\partial (h_o + gz)}{\partial x} - \frac{1}{\rho} \frac{\partial P}{\partial t} - \frac{1}{\rho} \frac{\dot{Q}}{A \partial x} - \frac{P}{\rho A} \frac{\partial A}{\partial t} = 0}$$

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.....	59
Nomenclature.....	463
Method of Characteristics.....	490

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 ρ , 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 λ :

$$\begin{aligned} & \lambda_1 \left[\frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} + \rho a^2 \frac{\partial V}{\partial x} \right] + \lambda_2 \left[\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial P}{\partial x} \right] + \lambda_3 \left[\frac{\partial \rho}{\partial t} + V \frac{\partial \rho}{\partial x} - \frac{1}{a^2} \left(\frac{\partial P}{\partial t} + V \frac{\partial P}{\partial x} \right) \right] \\ & = \lambda_1 \left[\frac{\beta a^2}{c_p} \mathbb{F} \right] + \lambda_2 \left[-g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] + \lambda_3 \left[-\frac{\beta}{c_p} \mathbb{F} \right] \end{aligned}$$

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 λ_1 , λ_2 , and λ_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 λ , 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:

$$\begin{array}{ll}
 R \left\{ \frac{dP}{dt} + \rho a \frac{dV}{dt} = \frac{\beta a^2}{c_p} \mathbb{F} + \rho a \left[-g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] \right. & \text{on } \frac{dx}{dt} = V + a \\
 L \left\{ \frac{dP}{dt} - \rho a \frac{dV}{dt} = \frac{\beta a^2}{c_p} \mathbb{F} - \rho a \left[-g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] \right. & \text{on } \frac{dx}{dt} = V - a \\
 P \left\{ \frac{d\rho}{dt} - \frac{1}{a^2} \frac{dP}{dt} = -\frac{\beta}{c_p} \mathbb{F} \right. & \text{on } \frac{dx}{dt} = V
 \end{array}$$

where

$$\mathbb{F} = \left(\frac{f \rho |V^3|}{2D} \right) + \frac{\dot{Q}}{A \partial x}$$

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*

$$dP + \rho a dV = \frac{\beta a^2}{c_p} \left[\left(\frac{f \rho |V^3|}{2D} \right) + \frac{\dot{Q}}{A \partial x} \right] dt + \rho a \left[-g \frac{dz}{dx} - \frac{fV|V|}{2D} \right] dt$$

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.

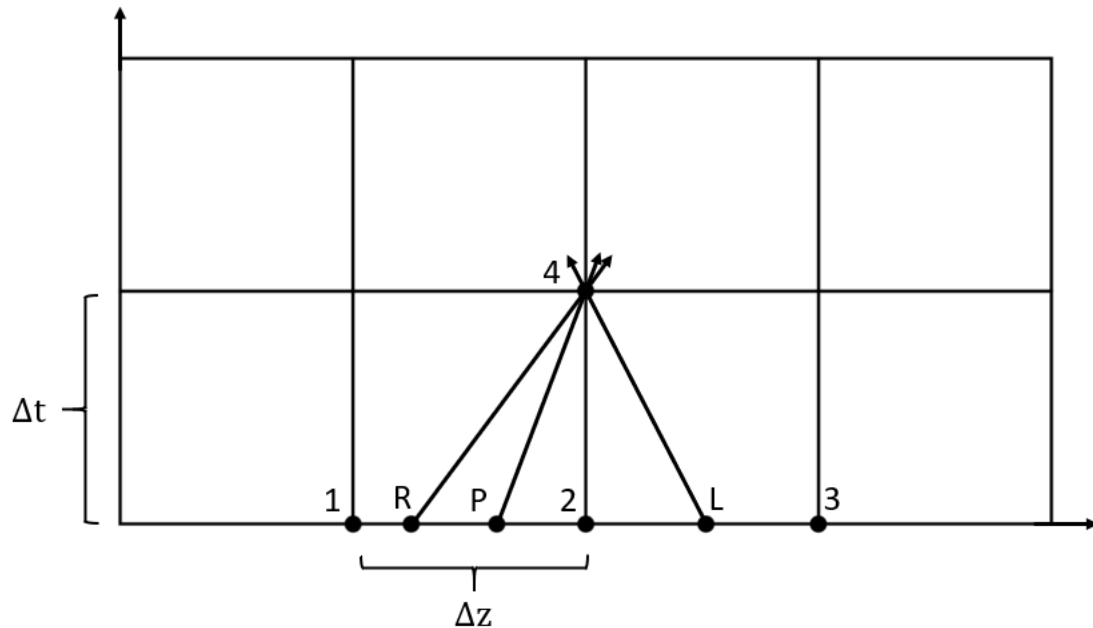


Figure 1: Characteristic grid. The point 4 sees effects from points R, L, and P after some time dt , which has equal magnitude of dx/a

Setting up the integral along the right running characteristic line:

$$\int_R^4 dP + \int_R^4 \rho a dV = \int_R^4 \left[\frac{\beta a^2}{c_p} \left(\frac{f \rho |V|^3}{2D} + \frac{\dot{Q}}{A \partial x} \right) + \rho a \left(-g \frac{dz}{dx} - \frac{f V |V|}{2D} \right) \right] dt$$

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.....	59
Nomenclature.....	463
Cv/Kv and XT Loss Model.....	228

Compatibility Equations

From the [characteristic equations](#) (repeated below), we can construct three convenient equations known as the compatibility equations.

$$(P_4 - P_R) + (\rho a)_R (V_4 - V_R) = F_R \Delta t$$

$$F = \frac{\beta a^2}{c_p} \left(\frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right) + \rho a \left(-g \frac{dz}{dx} - \frac{f V |V|}{2D} \right) \begin{matrix} \text{(Positive} \\ \text{Wave Char-} \\ \text{acteristic)} \end{matrix}$$

$$(P_4 - P_L) - (\rho a)_L (V_4 - V_L) = G_L \Delta t$$

$$G = \frac{\beta a^2}{c_p} \left(\frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right) - \rho a \left(-g \frac{dz}{dx} - \frac{f V |V|}{2D} \right) \begin{matrix} \text{(Negative} \\ \text{Wave Char-} \\ \text{acteristic)} \end{matrix}$$

$$(\rho_4 - \rho_P) - \frac{1}{a_P^2} (P_4 - P_P) = H_P \Delta t$$

$$H = \frac{\beta}{c_p} \left(\frac{f \rho V^3}{2D} + \frac{1}{A} \frac{\partial \dot{Q}}{\partial x} \right)$$

(Particle Path
Char-
acteristic)

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 endpoints 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..... 490

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.....	225
Method of Characteristics.....	490
Sonic Choking Detailed Description.....	504
Sonic Choking.....	503

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 :

$$dz = (\sin \theta) dx \quad (3)$$

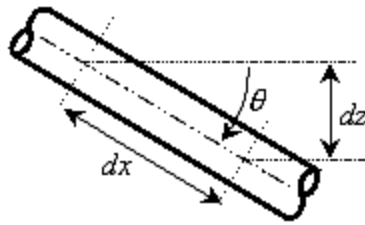


Figure 1: Relationship between distance along pipe (dx) and vertical elevation change (dz).

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/sec² or 9.81 m/sec²).

From the equation of state for a real gas:

$$P = Z \rho RT \tag{4}$$

Related Topics

Nomenclature463

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

Nomenclature463

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.....463

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.

$$h_{0,mix} = \frac{\sum_{j=1}^n \dot{m}_j^+ h_{o,j}}{\sum_{j=1}^n \dot{m}_j^+} \quad (49)$$

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.

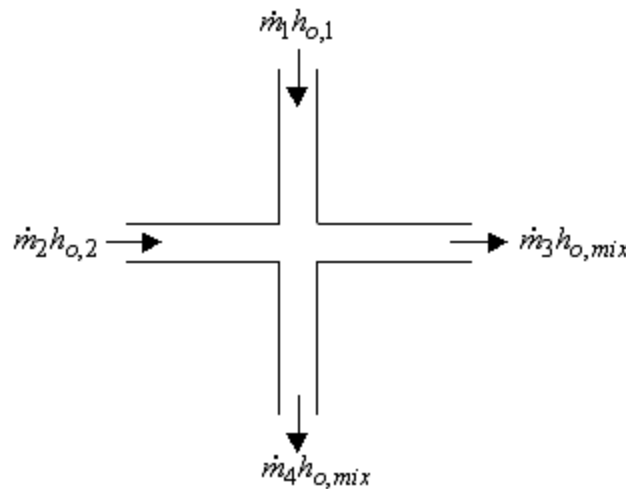


Figure 1: Flow weighted mixing and energy balance

Related Topics

Nomenclature463

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.....	228
Sonic Choking Detailed Description.....	504
Transient Sonic Choking.....	497

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.

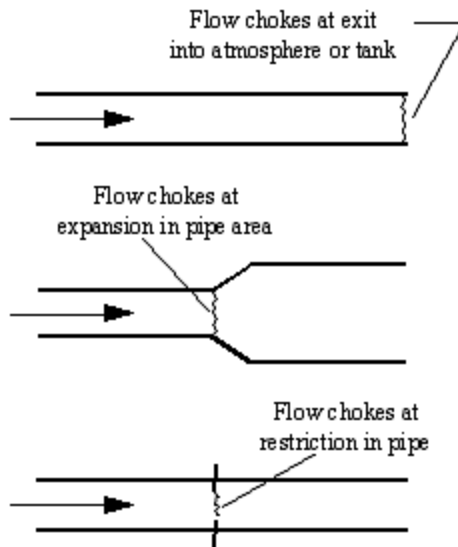


Figure 1: Three configurations which can result in sonic choking. From top to bottom, endpoint choking, expansion choking, and restriction choking

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.

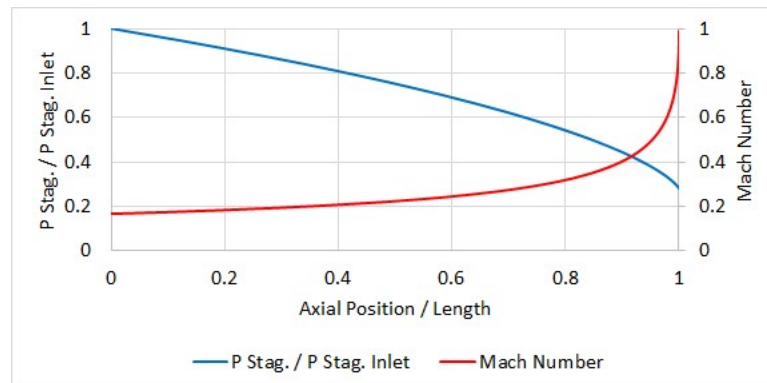


Figure 2: Example stagnation pressure and Mach number profiles for endpoint choking.

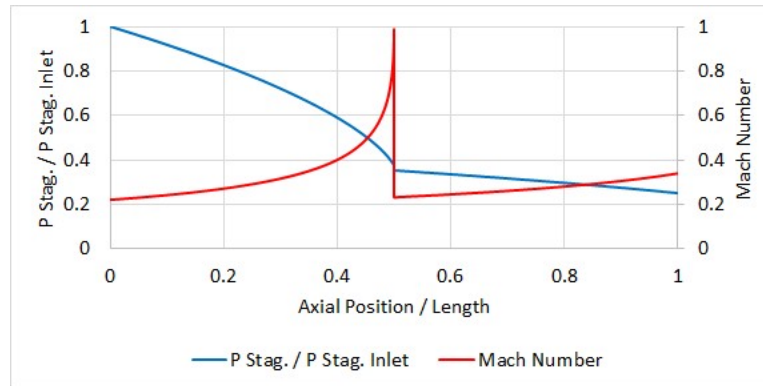


Figure 3: Example stagnation pressure and Mach number profiles for expansion choking.

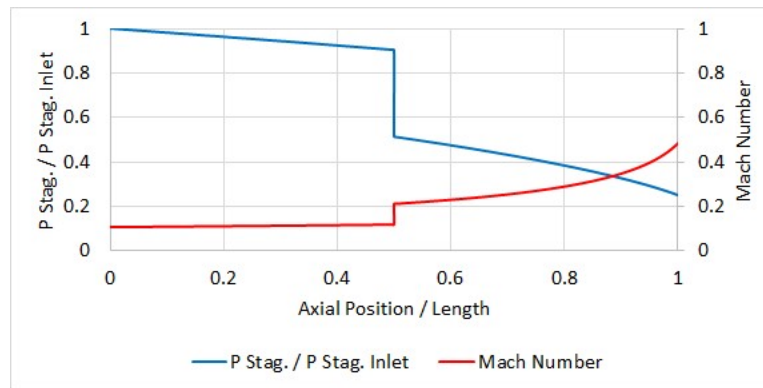


Figure 4: Example stagnation pressure and Mach number profiles for restriction choking. Note that the Mach Number reaches 1 within the restriction at 0.5.

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 endpoint 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 $C_d A$, 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 $C_d A$ would cause choking at a restriction:

$$A_{sonic} = \dot{m} \frac{\sqrt{T_0}}{P_0 f(M)_{M=1}} \quad (52)$$

A $C_d A$ 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.

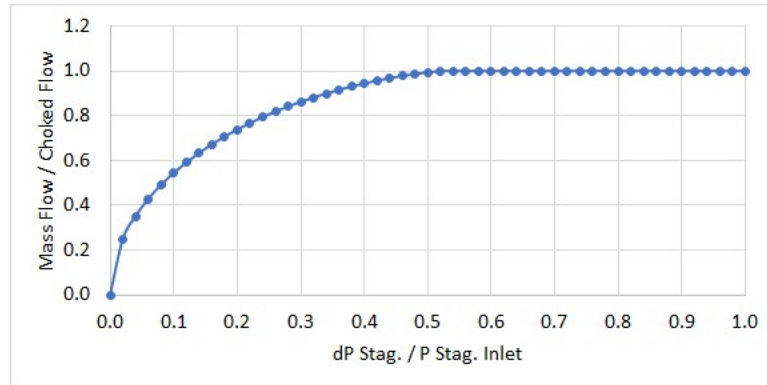


Figure 5: Example of dimensionless flow rate vs. dimensionless pressure drop. Sonic choking occurs just past the dimensionless pressure drop of ~0.5, after which the flow can no longer increase.

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.

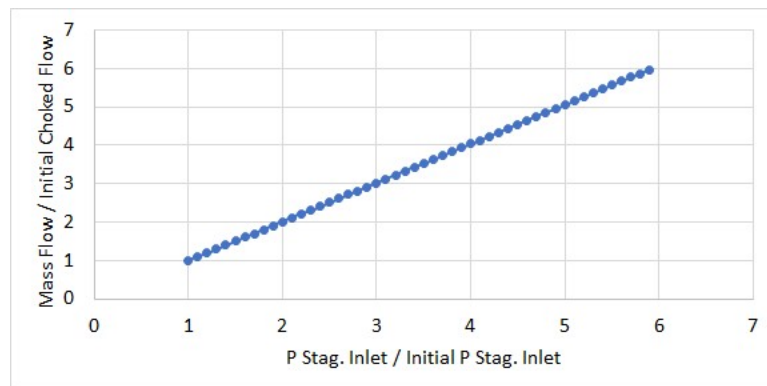


Figure 6: Example of increasing choked flow rate with supply pressure.

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.....	463
Method of Characteristics.....	490
Transient Sonic Choking.....	497
Sonic Choking.....	503

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 CdA for Sonic Choking is not entered. In this case, AFT xStream will always use subsonic loss calculations if no CdA 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.....	282
Relief Valve.....	285
Discharge Coefficient Loss Model.....	224
Screen.....	287
Venturi.....	295

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 ρ_0 , respectively), velocity (or Mach number, M), and static temperature, pressure, enthalpy, and density (T , P , h , and ρ , 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.....	79
Review of Compressible Flow Theory.....	465
Nomenclature.....	463

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, Nayar, 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, Nayar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-31; 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, Nayar, 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
 - 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 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 @ 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 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 @ 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 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 @ 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 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 @ 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 Comercial 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 Comercial 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 Comercial 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 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 @ 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 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 @ 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 @ 20oC
 - 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*, 5th Edition, American Petroleum Institute, Washington, DC, US, 2007.
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. Eckert, E.R.G., and Drake, R., *Analysis of Heat and Mass Transfer*, McGraw-Hill, New York, NY, 1972.
15. Fox, R. W. and A.T. McDonald, *Introduction to Fluid Mechanics*, 3rd edition, Wiley & Sons, New York, NY, 1985.
16. Idelchik, I. E., *Handbook of Hydraulic Resistance*, 4th edition, Begell House, Redding, CT, 2007.
17. Incropera, F.P. et. al, *Fundamentals of Heat and Mass Transfer*, John Wiley & Sons, NJ 2007
18. Ingersoll-Dresser Pumps, *Cameron Hydraulic Data*, Liberty Corner, NJ, 1995.
19. Ito, H., "Friction factors for pipe flow", *Journal of Basic Engineering*, Vol. 81, pp. 123-126, 1959.
20. Jeppson, R. W., *Analysis of Flow in Pipe Networks*, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.

-
21. Karassik, I.J., J.P. Messina, P. Cooper and C.C. Heald, Pump Handbook, 3rd Edition, McGraw Hill, New York, NY, 2001.
 22. 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.
 23. Keenan, J.H., Thermodynamics, John Wiley & Sons, Inc., New York, NY, 1941.
 24. Kittredge, C.P., Hydraulic Transients in Centrifugal Pump Systems, ASME Journal of Basic Engineering, Vol. 78, No. 6, pp. 1307-1322, Aug. 1956.
 25. Lyons, J. L., Lyons' Valve Designers' Handbook, Van Nostrand Reinhold, New York, NY, 1982.
 26. Miller, D. S., Internal Flow Systems, 2nd edition, Gulf Publishing Company, Houston, TX, 1990.
 27. Moody, Frederick J., Introduction to Unsteady Thermofluid Mechanics, John Wiley & Sons, Inc., New York, NY, 1990.
 28. Nayyar, M. L., Piping Handbook, 7th edition, McGraw-Hill, New York, NY, 2000.
 29. Pipe Line Rules of Thumb Handbook, Gulf Publishing, Houston, TX.
 30. Saad, M.A., *Compressible Fluid Flow*, 2nd Edition, Prentice-Hall, Englewood Cliffs, NJ, 1993
 31. Shapiro, A.H., *The Dynamics and Thermodynamics of Compressible Fluid Flow*, 2 vols., Ronald, New York, NY, 1953.
 32. Smith, J.M., van Ness, H.C., and Abbott, M.M., Introduction to Chemical Engineering Thermodynamics, 6th edition, McGraw-Hill, New York, NY, 2001.
 33. Swaffield, J. A., and Boldy, A. P., Pressure Surge in Pipe and Duct Systems, Avebury Technical, Hampshire, England, 1993.
 34. TAPPI, Generalized method for determining the pipe friction loss of flowing pulp suspensions, TAPPI, TIS 0410-14, Atlanta, GA, 1988.
 35. Tilton, J. N., Perry's Chemical Engineer's Handbook, Seventh Edition, Chapter 6, McGraw-Hill, New York, NY, 1997.
 36. Thorley, A.R.D., Fluid Transients in Pipeline Systems, 2nd Ed., D&L George, Ltd. 2004
 37. Thorley, A.R.D., and A. Chaudry, Pump Characteristics for Transient Flow, Pressure Surges and Fluid Transients, BHR Group, 1996.
 38. Val-Matic Valve and Manufacturing Corp., "Surge Control in Pumping Systems," Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.
 39. Walski, T. M., Analysis of Water Distribution Systems, Van Nostrand Reinhold Company, New York, NY, 1984.
 40. Walters, T.W., "Gas-Flow Calculations: Don't Choke", *Chemical Engineering*, Chemical Week Associates, Jan. 2000, pp. 70-76.
 41. 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.
 42. Watters, G.Z, Modern Analysis and Control of Unsteady Flow in Pipelines, Ann Arbor Science, Ann Arbor, MI, 1979.
 43. White, F. M., Fluid Mechanics, McGraw Hill, New York, NY, 1979.
 44. 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.

45. 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

$$\ln(P_{vr}) = (aX + bX^{1.5} + cX^3 + dX^6)/Tr$$

with

$$X = 1 - Tr$$

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:

$$\ln(P_{vr}) = B(1/Tr^m - 1) + G(Tr^7 - 1)$$

$$G = aH + bB$$

$$a = (1 - 1/Tr^m)/(Tr^7 - 1)$$

$$b = (1 - 1/Tr^m)/(Tr^7 - 1)$$

$$H = Tr \ln(P_c/P_b)/(1 - Tr)$$

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 H₂, 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),

$$m = 0.0052 M^{0.29} T^{0.72} c$$

$$G = (2.464 \cdot M) \exp(0.0000098 M T c)$$

$$B = (G - aH)/b$$

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:

$$\ln(P_{vr}) = f(0) + w f(1)$$

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

$$v_0 = a_0 k_0 \{ a_1 + a_2 T_r^n + a_3 \exp(a_4 T_r) + a_5 \exp(a_6 T_r) \} F_p^0, F_q^0$$

v_0 = low pressure viscosity in kg/ms

$$k_0 = M_0.5P_c^{0.667}T_c^{0.1667}$$

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,

$$F_{p0} = 1 \quad 0 < m_r < 0.022$$

$$F_{p0} = 1 + f(Z_c) \quad 0.022 < m_r < 0.075$$

$$F_{p0} = 1 + f(Z_c).g(Tr) \quad 0.075 < m_r$$

$$f(Z_c) = 30.55(0.292 - Z_c)^{1.72}$$

$$g(Tr) = 0.96 + 0.1(Tr - 0.70)$$

$$m_r = 52.46 m^2 P_c / T_c^2$$

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,

$$F_{q0} = 1.22 Q_0.15\{1 - 0.00385(12 - Tr)^2/M\}$$

with

$$Tr < 12$$

$$Q = 1.38 \text{ for helium}$$

$$0.76 \text{ for hydrogen}$$

$$0.52 \text{ for deuterium}$$

8.9.2 High-Pressure Equation

Define,

$$Z_1 = v_0/a_0k_0$$

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,

$$Z2 = Z1\{1 + aPre/(bPrf + 1/(1 + cPrd))\}$$

with,

$$a = (a1/Tr)\exp(a2Trm)$$

$$b = a(b1Tr - b2)$$

$$c = (c1/Tr)\exp(c2Trm)$$

$$d = (d1/Tr)\exp(d2Trp)$$

$$e = 1.3088$$

$$f = f1\exp(f2Trq)$$

$$a1 = 0.001245$$

$$a2 = 5.1726$$

$$n = -0.3286$$

$$b1 = 1.6553$$

$$b2 = 1.2723$$

$$c1 = 0.4489$$

$$c2 = 3.0578$$

$$m = -37.7332$$

$$d1 = 1.7368$$

$$d2 = 2.2310$$

$$p = -7.6351$$

$$f1 = 0.9425$$

$$f2 = -0.1853$$

$$q = 0.4489$$

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:

$$v_0 = \text{low pressure methane viscosity} = \text{Sum}(c_n T_0^{(n-4)/3})$$

$$c_1 = 2.90774E-01$$

$$c_2 = -3.31287E-01$$

$$c_3 = 1.60810E-01$$

$$c_4 = -4.33190E-02$$

$$c_5 = 7.06248E-03$$

$$c_6 = -7.11662E-04$$

$$c_7 = 4.32517E-05$$

$$c_8 = -1.44591E-06$$

$$c_9 = 2.03712E-08$$

$$H = 16.04 f_0^{0.5} / M^{0.667}$$

C_{v0} = low pressure specific heat at constant volume in J/mol.K

R = gas constant = 8.314 J/mol.K

M = 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 for T_r ≤ 2

= 2 otherwise

V* = V/V_c for 0.5 < V/V_c < 2

= 0.50 for V/V_c ≤ 0.50

= 2 otherwise

b₁ = -2.5276E-04

b₂ = 3.3433E-04

b₃ = 1.1200E+00

b₄ = 1.6800E+02

a₁ = -7.1977E+00

a₂ = 8.5678E+01

a₃ = 1.2472E+01

a₄ = -9.8463E+02

a₅ = 3.5947E-01

a₆ = 6.9798E+01

a₇ = -8.7288E+02

r₀ = 16.04 h/V

T₀ = 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,

$$P_r = P/P_c$$

$$T_r = T/T_c$$

$$V_r = P_c V / R T_c$$

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 , $P_{sr}(0)$ 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

$$T_c = \frac{\sum(x_i V_{ci} T_{ci})}{\sum(x_i V_{ci})}$$

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

$$V_c = \sum(a_i V_{ci}) + \sum(a_{ij} b_{ij}) \quad (\text{all } i, j)$$

$$a_i = x_i V_{ci}^{2/3} / \sum(x_i V_{ci})$$

$$b_{ii} = 0 \quad b_{ij} = 2c_{ij}(V_{ci} + V_{cj})$$

$$c_{ij} = A_{ij} + B_{ij}d_{ij} + C_{ij}d_{ij}^2 + D_{ij}d_{ij}^3 + E_{ij}d_{ij}^4$$

$$d_{ij} = \text{ABS}\{(V_{ci}^{2/3} - V_{cj}^{2/3}) / (V_{ci}^{2/3} + V_{cj}^{2/3})\}$$

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 ₂ or H ₂ 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

$$w_m = \sum(x_i w_i)$$

$$T_{cm} = \sum(Z_i T_{ci})$$

$$Z_i = x_i V_{ci} / \sum(x_i V_{ci})$$

$$T_{cm}^* = \sum(x_i T_{ci})$$

$$P_{cm}^* = \sum(x_i P_{ci})$$

$$P_c = P_{cm}^* [1 + (5.808 + 4.93 w_m)(T_{cm}/T_{cm}^* - 1)]$$

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,

$$w = \sum(x_i w_i)$$

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,

$$w_m = \sum(x_i w_i)$$

$$w_i = -\log(P_{vri} \text{ at } T_r = 0.7) - 1$$

$$w_m = -\log(P_{vrm} \text{ at } T_r = 0.7) - 1 = \sum(-x_i \log(P_{vri} \text{ at } T_r = 0.7) - x_i)$$

hence,

$$\log(P_{vrm} \text{ at } T_r = 0.7) = \sum(x_i \log(P_{vri} \text{ at } T_{ri} = 0.7))$$

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,

$$a_m = \sum(x_i a_i) \quad b_m = \sum(x_i b_i)$$

$$c_m = \sum(x_i c_i) \quad d_m = \sum(x_i d_i)$$

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 at 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, Vol21(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.

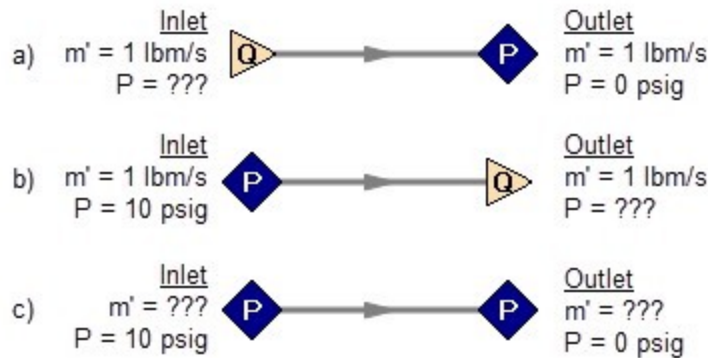


Figure 2: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.

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.

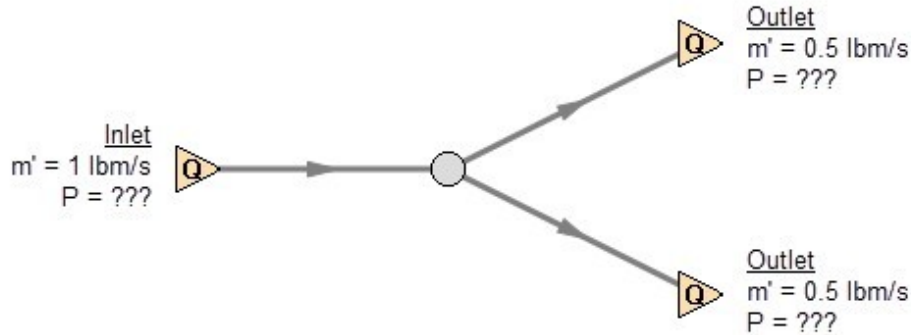


Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.

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.

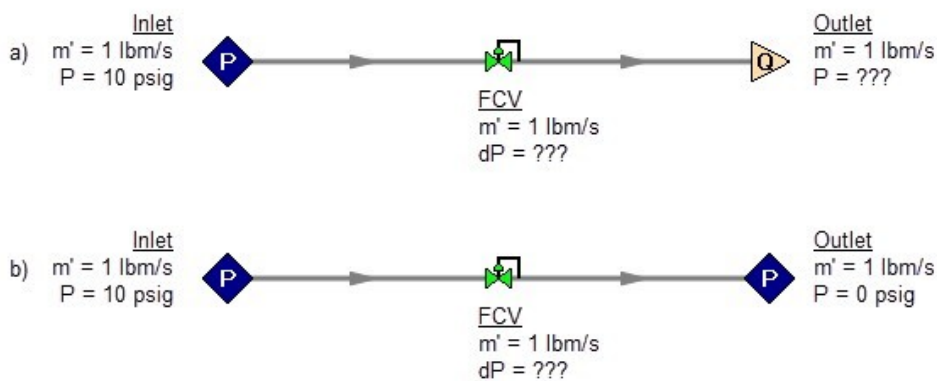


Figure 4a-b.: The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists.

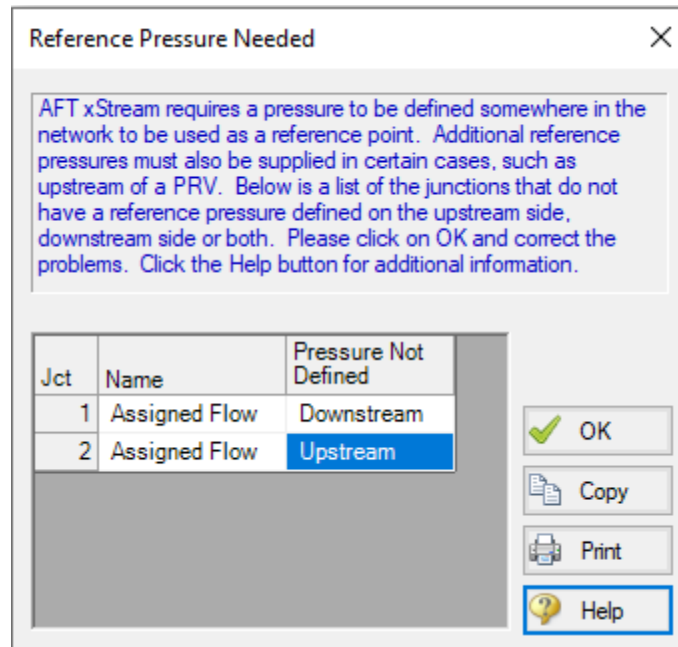


Figure 5: AFT xStream message when you try to run the model shown in Figure 4a.

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.

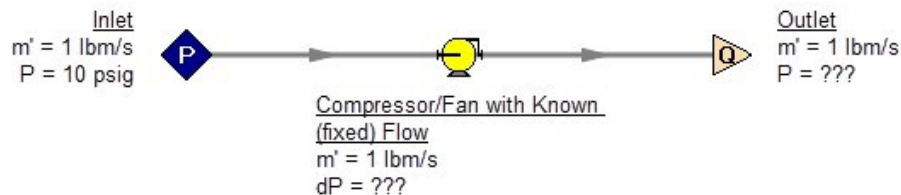


Figure 6: A compressor/fan modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists

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.

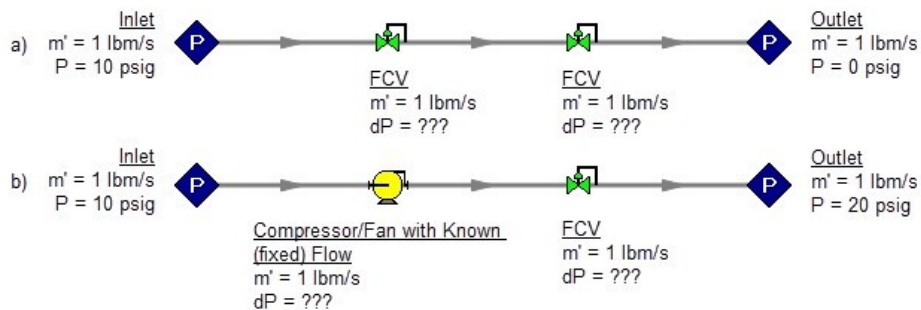


Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a compressor/fan modeled as a fixed flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions 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.

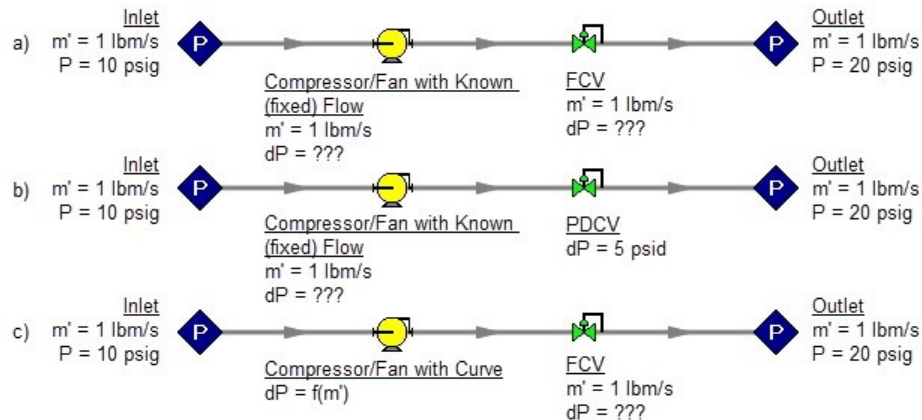


Figure 8a-c: The model at the top is the same as Figure 7b and does not have a unique solution. To size the compressor/fan, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the compressor/fan is sized, a 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.

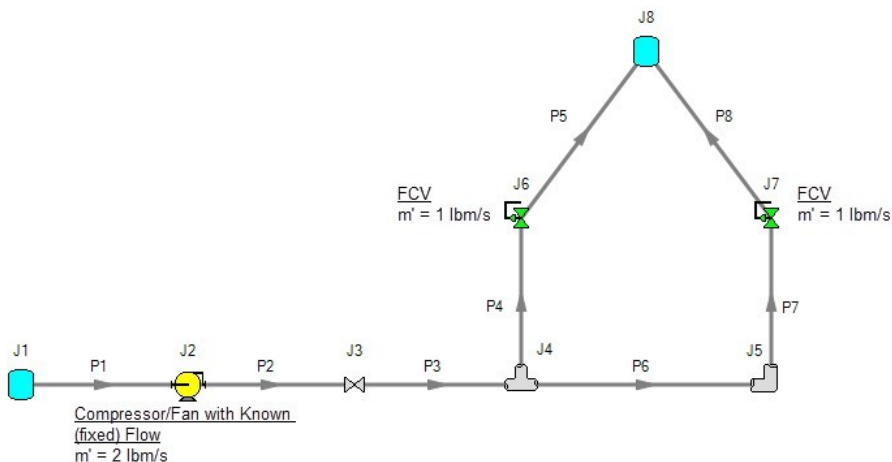


Figure 9: A compressor/fan with a fixed flow in series with two parallel FCV's. The model in its current form does not have a unique solution. To size the compressor/fan, first change the FCV at J7 to a PDCV.

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).

Jct	Name	Valve Type	Mass Flow (lbm/sec)	dP Stag. (psid)	P Static In (psia)	K	Valve State
3	Valve	REGULAR	2.000	0.000	158.7	0.00	Open
6	Control Valve	FCV	1.000	5.841	156.2	19.84	Open
7	Control Valve	PDCV	1.000	5.000	155.4	16.89	Open

Jct	Results Diagram	Mass Flow (lbm/sec)	dP Stag. (psid)	Ideal Power (hp)	Comp. Ratio Stag. (Percent)
2	Show <input type="button" value="..."/>	2.000	59.83	51.69	160.1

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.

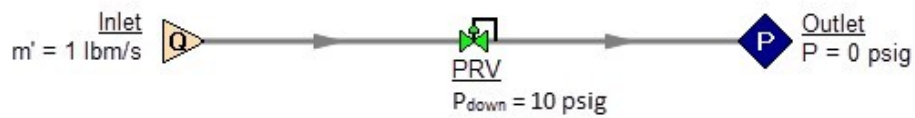


Figure 11: If a PRV is downstream of a known flow the PRV pressure drop cannot be calculated. This model does not have a unique solution.

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.

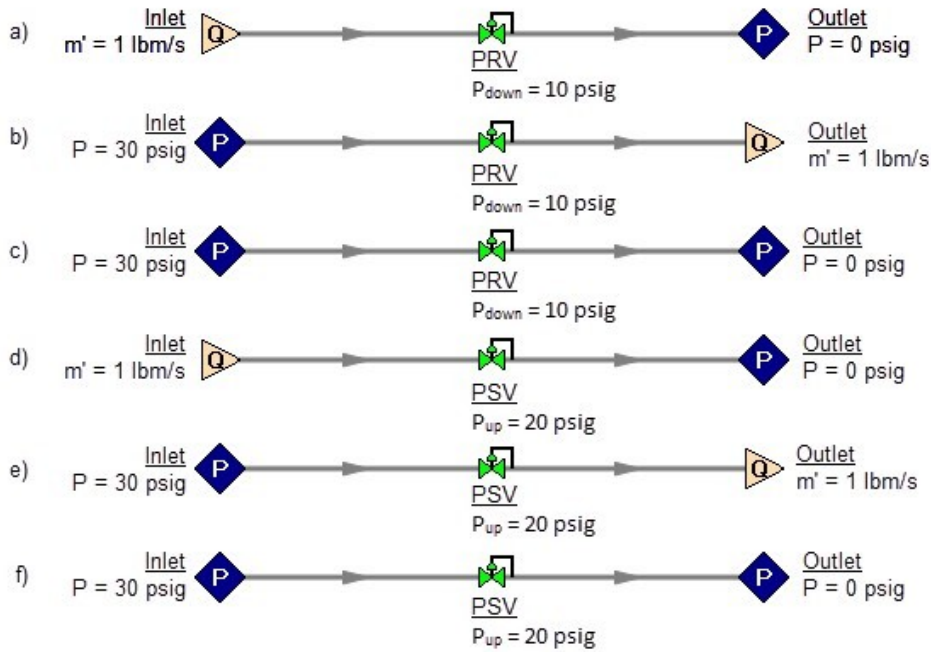


Figure 12a-f.: Six possibilities for PRV and PSV configurations. Cases A and E do not have unique solutions. See Table 2 for comments.

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.

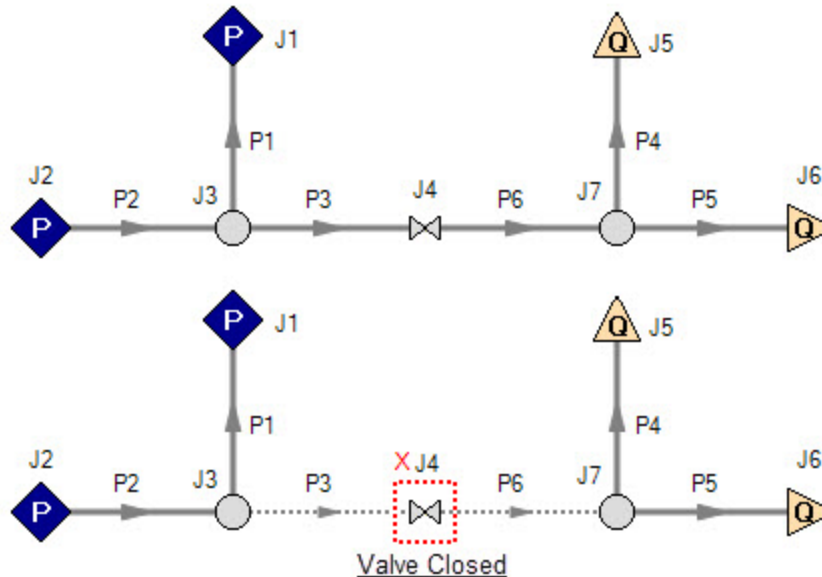


Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.

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](#)

Output Messages & Warnings

Various types of messages may be given in the [output](#) when you run a model. These messages are intended to alert you about problems with the model or solution. There are different message classifications depending on the severity, but all message types should be reviewed when they are displayed in the output.

The links below will take you to a message list for the corresponding message type with a brief explanation given for each.

1. [CRITICAL WARNING](#) - **Very High Severity** - The model is not accurately predicting behavior, and the solution may violate the [engineering assumptions](#) of the software. These issues must be resolved for the model to be trustworthy, and the output should only be used for error resolution or troubleshooting purposes and never for engineering decisions.
2. [WARNING](#) - **High Severity** - A serious issue with the model or results exists. For example, the solution has exceeded a valid data range, or potentially unrealistic behavior is occurring. A real system may encounter operational problems under the conditions.
3. [CAUTION](#) - **Moderate/Low Severity** - Something may be wrong with the model or of interest for the user to double check. Depending on the application, the results may or may not be affected. If results are affected, it would be to a lower degree compared to a Warning.
4. [INFORMATION](#) - An Information message is intended to make the user aware that an advanced feature is being used in the model, or to provide comments or suggestions relating to an advanced feature. Following suggestions may achieve higher accuracy results if the right conditions apply, but can also cause incorrect results if not applied properly. If the user understands the feature then no action is required.

Any message that directly lists a pipe/junction can be double-clicked to take you directly to the object on the Workspace. Whenever a Warning or Critical Warning message type is displayed, the General Results section will display red text.

Related Topics

Use the Model Data Window.....	581
Solution Method Panel.....	120
Simulation Mode/Duration Panel.....	114

Critical Warnings

The model is not accurately predicting behavior, and the solution may violate the [engineering assumptions](#) of the software. These issues must be resolved for the model to be trustworthy, and the output should only be used for error resolution or troubleshooting purposes and never for engineering decisions.

CRITICAL WARNINGS

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..... 490

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 Pressure/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 compressors](#) 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 on 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.

Pipe 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.

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.

These results do not represent a 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.

Warnings

A Warning is of High Severity, and should always be reviewed before considering the results to be meaningful for the system.

A serious issue with the model or results exists. For example, the solution has exceeded a valid data range, or potentially unrealistic behavior is occurring. A real system may encounter operational problems under the conditions.

WARNINGS

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.

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

The [Flow Control Valve](#) 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 Text

There are several variations of this Warning:

- Fluid is no longer in the vapor phase because the fluid **pressure is above the Saturation Pressure** for the following:
Pipe X Inlet/Outlet, Actual Pressure = Y, Saturation Pressure = Z
- Fluid is no longer in the vapor phase because the fluid **temperature is below the Saturation Temperature** for the following:
Pipe X Inlet/Outlet, Actual Temperature = Y, Saturation Temperature = Z

Explanation

The steady-state temperature and pressure conditions at the specified pipes are such that one or more gas components has reached saturation conditions. When the fluid property conditions fall below saturation the solution accuracy will be decreased since one of the fundamental [Engineering Assumptions](#) is that the fluid is single-phase and superheated. The Solver will attempt to extrapolate gas properties below the saturation line.

The variations of the Warning depend on which [fluid library](#) is used and different programming methods are required due to differences in the library sources.

- Variation for "pressure is above the Saturation Pressure" is only applicable to AFT Standard or User Library Fluids.
- Variation for "temperature is below the Saturation Temperature" is applicable to ASME Steam, Chempak, and NIST REFPROP fluids.

Required Action

Adjust the model such that the fluid properties remain superheated, single-phase, and outside the saturation conditions. If this is not possible, then a larger uncertainty in results must be accepted and the results may not be accurate depending on how far the properties extend into saturation.

If there is flexibility in adjusting the model, there are various changes that could be made depending on the goal of the analysis and what is acceptable:

- Check for typos or incorrect inputs in the Model Data (including units), especially at the boundary condition junctions or any components with significant temperature or pressure changes
- Increase or decrease the upstream boundary condition temperature (such as the temperature in a Tank junction)
- Adjust the heat transfer in pipes
- Run a scenario using a different [fluid library](#) if the fluid can be found in the other Library sources, and compare the results

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 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.

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 [C_dA](#) 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 C_dA.

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 C_dA 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 C_dA 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 the [Tolerance panel](#) of Analysis Setup, 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

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 specified limits in the [Pressure and Temperature Solver Limits panel](#) of the [AFT Arrow Steady Solution](#) in Analysis Setup. These limits are intended to aid in convergence by restricting the search space while solving, and should generally be left as the AFT default values. This warning indicates that the final solution had one or more pipes outside the limit range, and the temperature results have been artificially limited by the Solver. The results are therefore not valid. To resolve this either remove or change the temperature limit.

Note: These limits should not be used as Design Alerts, or to approximate some physical phenomenon.

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 at inlet/exit of pipe X is above/below max/min fluid temperature range

A [Warning message](#) in AFT xStream

Full Message Text

There are several variations of this Warning:

- Temperature at **inlet** of pipe X is **below min** fluid temperature range.
- Temperature at **exit** of pipe X is **below min** fluid temperature range.
- Temperature at **inlet** of pipe X is **above max** fluid temperature range.
- Temperature at **exit** of pipe X is **above max** fluid temperature range.

Explanation

The temperature used to calculate fluid properties is being predicted outside the valid range of data for the [AFT Standard fluid or User Library Fluid](#). In order to produce results the software is extrapolating data or using a curve fit beyond the defined range of acceptable values. The fluid properties may be incorrect and may affect the result accuracy. The max and min temperature for the fluid data can be seen by opening the Library menu, selecting Edit Fluids, selecting the fluid used in the model, and looking at the top of the Properties section.

Required Action

Adjust the model such that the temperatures remain within the bounds of fluid data. This could take on many forms depending on the goal of the analysis and what is acceptable:

- Check for typos or incorrect inputs in the Model Data (including units), especially at the boundary condition junctions or any components with significant temperature changes
- Increase or decrease the upstream boundary condition temperature (such as the temperature in a Tank junction)

- Adjust the heat transfer in pipes such that the temperature does not drop below or rise above the bounds of data
- Add more data to the custom fluid that extends the range to cover the temperatures seen in the model
- Run a scenario using a different [fluid library](#) if the fluid can be found in the other Library sources, and compare the results

The maximum difference between the AFT Arrow Steady solution and the MOC Steady Solution for velocity is X%

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.

The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X locations

A [Warning message](#) in AFT xStream

Full Message text is:

The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X location(s). See the Pulsation Summary.

This warning appears in AFT xStream when peak-to-peak pressure limits defined by the API 618 standard have been violated at one or more locations. This warning only appears when the PFA module is active, when the Frequency Analysis Mode is set to Steady State Pulsation, and the option Evaluate with API 618 is active in the [Pulse Setup Panel](#). The Pulsation Summary tab in the [Output Window](#), shown in Figure 1 below, provides a summary of each location where this occurred along with the limit at that location and the frequency at which the API 618 limit was violated.

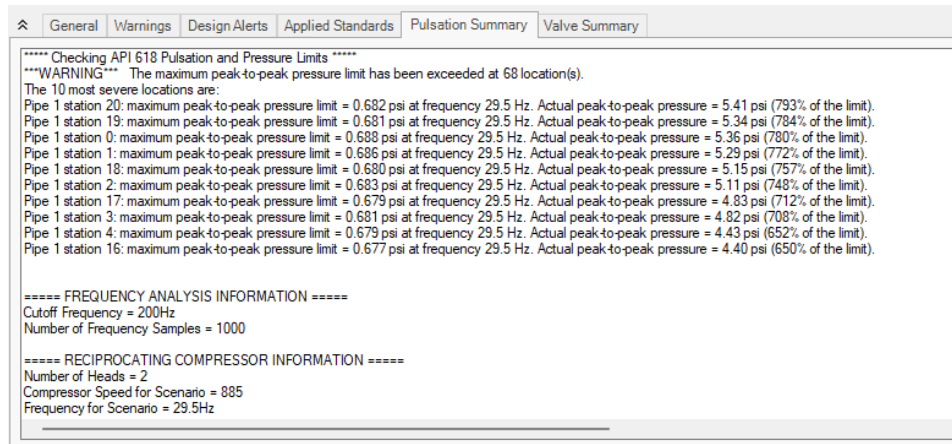


Figure 1: The Pulsation Summary tab shows frequency analysis and reciprocating compressor information along with information describing locations where API 618 was violated

Related Topics

Defining Pulsation Settings.....	434
Pulse Setup Panel.....	103
Pulsation peak-to-peak pressures are within allowable API 618 levels.....	570

The MOC Steady Solution did not reach a nominally steady state before the transient started

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.

Transient choking occurred at Junction X

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 above/below fluid maximum/minimum temperature

A [Warning message](#) in AFT xStream

Full Message Text

There are several variations of this Warning:

- Transient temperature was **below** fluid maximum temperature at X seconds. Results after this time may not be valid.
- Transient temperature was **above** fluid maximum temperature at X seconds. Results after this time may not be valid.

Explanation

The temperature used to calculate fluid properties is being predicted outside the valid range of data for the [AFT Standard fluid or User Library Fluid](#). In order to produce results the software is extrapolating data or using a curve fit beyond the defined range of acceptable values. The fluid properties may be incorrect and may affect the result accuracy. The max and min temperature for the fluid data can be seen by opening the Library menu, selecting Edit Fluids, selecting the fluid used in the model, and looking at the top of the Properties section.

xStream will extrapolate below the minimum or above the maximum temperature and assume the fluid remains in the superheated gas phase. In some cases the minimum temperature for the library fluid represents saturation conditions, in which case xStream will be extrapolating below the saturation line.

The reported time step is the first occurrence of this issue during the simulation. There may be additional occurrences of the temperature being above the max or below the min throughout the transient simulation. Results from the MOC Transient after the reported time step should be treated with caution.

Required Action

Adjust the model such that the temperatures remain within the bounds of fluid data. This could take on many forms depending on the goal of the analysis and what is acceptable:

- Check for typos or incorrect inputs in the Model Data (including units), especially at the boundary condition junctions or any components with significant temperature changes
- Increase or decrease the upstream boundary condition temperature (such as the temperature in a Tank junction)
- Adjust the heat transfer in pipes such that the temperature does not drop below or rise above the bounds of data
- Add more data to the custom fluid that extends the range to cover the temperatures seen in the model
- Run a scenario using a different [fluid library](#) if the fluid can be found in the other Library sources, and compare the results

Transient temperature was below saturation temperature

A [Warning message](#) in AFT xStream

Full Message Text

Transient temperature was below saturation temperature at X seconds. Results after this time may not be valid.

Explanation

The transient temperature and pressure conditions at one or more pipes were such that one or more gas components reached saturation conditions. When the fluid property conditions fall below saturation the solution accuracy will be decreased since one of the fundamental [Engineering Assumptions](#) is that the fluid is single-phase and superheated. The Solver will attempt to extrapolate gas properties below the saturation line, or optionally, the user can choose to limit properties to Saturated Conditions in the [Transient Solver Advanced Parameters panel](#).

The reported time step is the first occurrence of this issue during the simulation. There may be additional occurrences of the fluid properties being saturated throughout the transient simulation. Results from the MOC Transient after the reported time step should be treated with caution.

Required Action

Adjust the model such that the fluid properties remain superheated, single-phase, and outside the saturation conditions throughout the entire transient simulation. If this is not possible, then results after the reported time step should be treated with caution since there is a larger uncertainty and the results may not be accurate depending on how far the properties extend into saturation.

To determine the location of the saturated fluid properties, it is recommended to create various [Profile Graphs](#) using the "Temperature Static" and "Temperature Saturation" graph parameters with the same units and axis range. When the static temperature drops below the saturation temperature on the graph, use the Show Junction Locations option to determine which pipe has the occurrence and the approximate station. Then, a Transient Pipe Graph can be created for the related stations to show the time step that saturation occurs. Knowing the approximate location of saturation can be useful to understand which upstream junctions may be impacting the results, and the temperature/pressure drops at these junctions should be reviewed.

If there is flexibility in adjusting the model, there are various changes that could be made depending on the goal of the analysis and what is acceptable:

- Check for typos or incorrect inputs in the Model Data (including units), especially at the boundary condition junctions or any components with significant temperature or pressure changes
- Increase or decrease the upstream boundary condition temperature (such as the temperature in a Tank junction)
- Adjust the heat transfer in pipes, or consider adding insulation
- Run a scenario using a different [fluid library](#) if the fluid can be found in the other Library sources, and compare the results

Cautions

Something may be wrong with the model or of interest for the user to double check. Depending on the application, the results may or may not be affected. If results are affected, it would be to a lower degree compared to a 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

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

A [Caution message](#) in AFT xStream

There are two variations of this Caution:

- Junction X is downstream of a choking point, **static** properties cannot be calculated at junction
- Junction X is downstream of a choking point, **stagnation** properties cannot be calculated at junction

This caution only occurs at pressure type junctions (e.g., [Assigned Pressure](#) or [Tank](#)) when the model is run in the [AFT Arrow Steady mode](#). When [sonic choking](#) occurs at a junction with a fixed pressure, 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 pressure at the junction, and vice versa.

For the variation stating that static properties cannot be calculated at the junction, all other results for the model are valid except for the static pressure, static temperature, and static enthalpy at the junction.

For the variation stating that stagnation properties cannot be calculated at the junction, all other results for the model are valid except for the stagnation pressure and stagnation temperature at the junction. Stagnation enthalpy is conserved across the choke point, therefore, it is known at the junction.

Note that xStream does not model the dynamics of how the flow stream exits into the ambient conditions.

Maximum transient pipe Mach number significantly 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 does not have time data that covers the entire time frame of the simulation as defined on the [Simulation Mode/Duration panel](#) of Analysis Setup. The final data point will be used for the remaining time steps of the simulation. You can enter additional time data points at Junction X to avoid this message.

Transient temperature was within 10 deg. F of fluid minimum/maximum

A [Caution message](#) in AFT xStream

Full Message Text

There are two variations of this Caution:

- Transient temperature was within 10 deg. F of fluid **minimum** temperature at X seconds.
- Transient temperature was within 10 deg. F of fluid **maximum** temperature at X seconds.

Explanation

The temperature at some point in the system approached the minimum/maximum temperature of the fluid according to the library data during the transient simulation. For an [AFT Standard fluid or User Library Fluid](#), the max and min temperature for the fluid data can be seen by opening the Library menu, selecting Edit Fluids, selecting the fluid used in the model, and looking at the top of the Properties section.

This Caution does not necessarily mean that fluid properties were calculated outside the valid range and that results are incorrect; however, the user should investigate accordingly. Some fluid properties may be extrapolated beyond the range of the available data points that were used for the polynomial curve fits or interpolated data. Fluid property calculations at these extremes will have higher uncertainty and may result in model convergence issues. In some cases this Caution may also mean that the saturation conditions were reached, and may be displayed with an accompanying [Warning about saturation](#).

Required Action

The user should review the results to verify that saturation conditions were never reached. Flow must remain superheated at all times in the simulation (no two-phase flow or condensation) as stated in the [engineering assumptions](#). A useful method of checking for condensation is to create various [Transient Pipe graphs](#) at different locations in the system and display the "Temperature Static" and "Temperature Saturation" parameters using the same units and axis range. Prime locations to check for saturation would be at the pipe inlets/outlets next to junctions like valves that have transient events. One of the graph parameters can be added as a secondary y-axis to create a stacked graph which overlays both series. Double click the empty box in front of the parameter name, or select the parameter row and click the right facing blue arrow above the parameters to indent it and make it a secondary y-axis. Then, verify that the Static Temperature does not drop below the Saturation Temperature for any time step.

If saturation conditions are never reached at any location throughout the entire transient simulation, then this Caution can be disregarded.

Transient temperature was within 10 deg. F of saturation

A [Caution message](#) in AFT xStream

Full Message Text

Transient temperature was within 10 deg. F of saturation temperature at X seconds.

Explanation

The temperature at some point in the system approached saturation conditions during the transient simulation. It does not necessarily mean that condensation occurred or that the results cannot be trusted; however, the user should investigate accordingly. Fluid property calculations close to saturation conditions may result in model convergence issues.

Required Action

The user should review the results to verify that saturation conditions were never reached. Flow must remain superheated at all times in the simulation (no two-phase flow or condensation) as stated in the [engineering assumptions](#). A useful method of checking for condensation is to create various [Transient Pipe graphs](#) at different locations in the system and display the "Temperature Static" and "Temperature Saturation" parameters using the same units and axis range. Prime locations to check for saturation would be at the pipe inlets/outlets next to junctions like valves that have transient events. One of the graph parameters can be added as a secondary y-axis to create a stacked graph which overlays both series. Double click the empty box in front of the parameter name, or select the parameter row and click the right facing blue arrow above the

parameters to indent it and make it a secondary y-axis. Then, verify that the Static Temperature does not drop below the Saturation Temperature for any time step.

If saturation conditions are never reached at any location throughout the entire transient simulation, then this Caution can be disregarded.

Information

An Information message is intended to make the user aware that an advanced feature is being used in the model, or to provide comments or suggestions relating to an advanced feature. Following suggestions may achieve higher accuracy results if the right conditions apply, but can also cause incorrect results if not applied properly. If the user understands the feature then no action is required.

INFORMATION

Flow at the Following Junctions is Laminar

An [Information message](#) in AFT xStream

The flow at the listed Junctions is in a laminar regime. Further loss corrections may be necessary.

Pulsation peak-to-peak pressures are within allowable API 618 levels

An [Information message](#) in AFT xStream

This informational message appears in AFT xStream when peak-to-peak pulsation pressures fall below the maximum allowable limits defined by the API 618 standard at all locations and all evaluated frequencies. This message only appears when the PFA module is active, when the Frequency Analysis Mode is set to Steady State Pulsation, and the option Evaluate with API 618 is active in the [Pulse Setup Panel](#).

Related Topics

Defining Pulsation Settings.....	434
Pulse Setup Panel.....	103
The maximum peak-to-peak pressure limit defined in the API 618 standard has been exceeded at X locations.....	560

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

Reducing Run Time	572
Transient Solver Errors	577
Diagnostics Window	580
Use the Model Data Window	581
Transfer Results to Initial Guesses	582
Poor Compressor Curve Fits	583
Troubleshooting Resistance Curves	583
Use the Output Window Sort Feature	584
Lower the Flow Rate Relaxation	585
Try Absolute Tolerance	585
Make Initial Flow Rate Guesses for Pipes	585
Change Boundary Conditions	586
Turn Off Parts of the Model	586
Break a Large Model Into Submodels	586
Use an AFT Standard Fluid	586
Run the MOC Steady Solution	587
Simplify Event Transients	587

Limit Properties to Saturation Conditions	587
Enable Transient Debug Files	588
Identifying and Restoring Corrupted Model Files	588

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	41
Transient Result Accuracy	59
Transient Sensitivity Analysis Tutorial	62

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.....	59
Transient Sensitivity Analysis Tutorial.....	62

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 Pipes/Junctions](#) 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.....	59
Transient Sensitivity Analysis Tutorial.....	62

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.....	59
Transient Sensitivity Analysis Tutorial.....	62

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.....	59
Transient Sensitivity Analysis Tutorial.....	62

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.

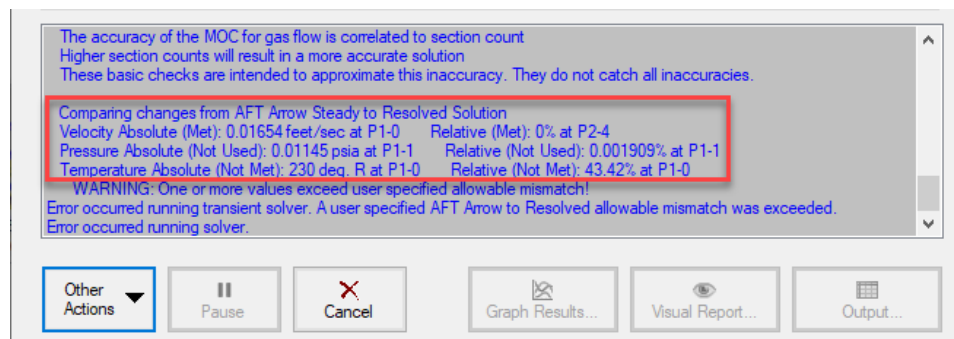


Figure 1: The bottom of the Solution Progress window provides additional information on the results of the artificial transient resolution period

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.

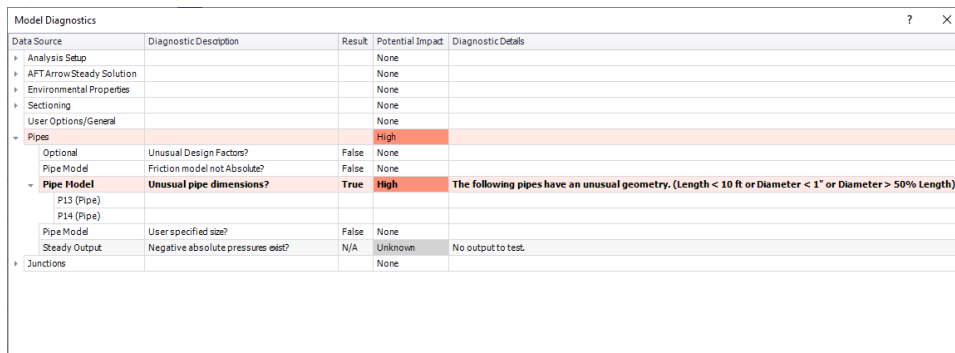
Diagnostics Window

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 exit?	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.

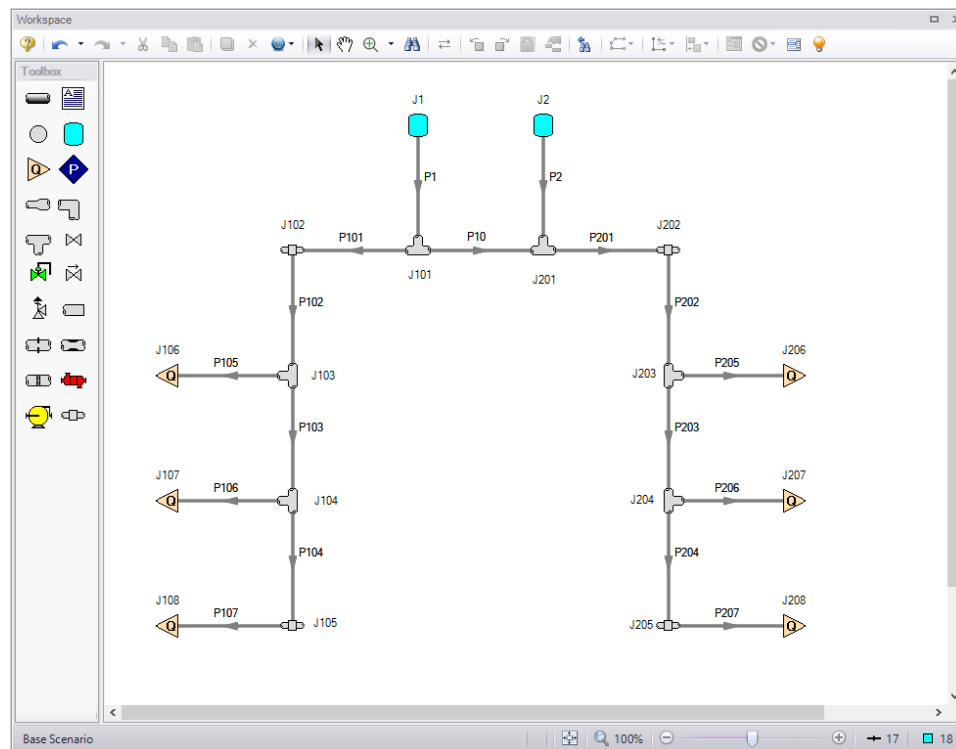


Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown).

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 [toolbar](#).

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.

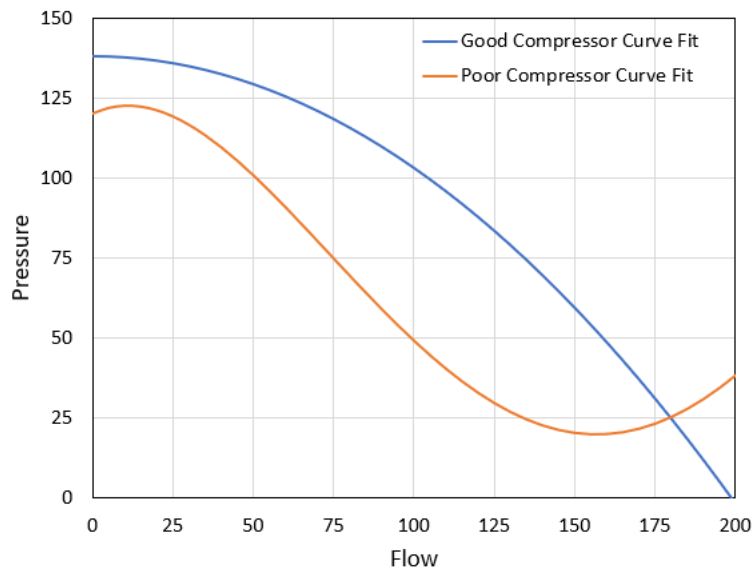


Figure 1: A good compressor/fan performance curve fit in blue and a poor compressor/fan performance curve fit in orange. All compressor/fan curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.

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.

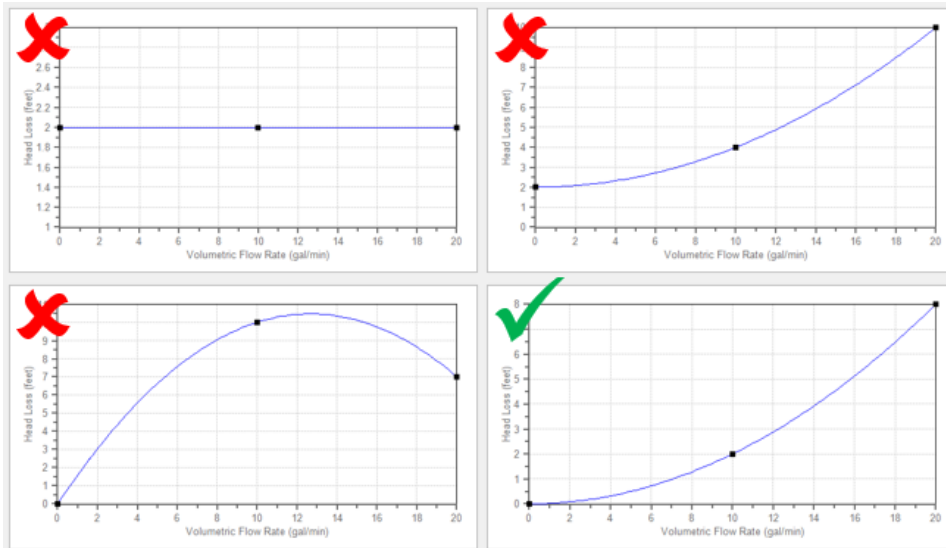


Figure 1: Example resistance curves showing correct and incorrect curve types

These common issues create two potential problems in a model:

1. If the system sees a constant pressure drop even at low flowrates, the iterative Solver can encounter difficulty converging on a solution since the system flow and pressure drop are no longer dependent on each other.
2. 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 [Resistance Curve](#) topic gives more information on defining this type of curve.

Related Topics

Resistance Curve Loss Model.....	231
General Component.....	281

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.....210

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.....	87
------------------	----

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.....	120
Transient Solver Initialization Panel.....	114

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

Transient Solver Advanced Parameters.....	157
---	-----

Fluid Panel..... 87
Engineering Assumptions..... 18

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.

Identifying and Restoring Corrupted Model Files

The most common way to tell if a model file is "corrupted" is an error message that says "An error occurred reading the file into AFT xStream. Can't find [TOOLBOX PREFERENCES] heading."

You may see multiple of these messages after pressing OK, and the part between the brackets [] may say something else each time. This is indicating that the software is trying to read different sections of the file, and it is finding that there is either no data, or corrupted data. It is then showing you an error for each section of the file that the software tries to read when loading the model, similar to Figure 1.

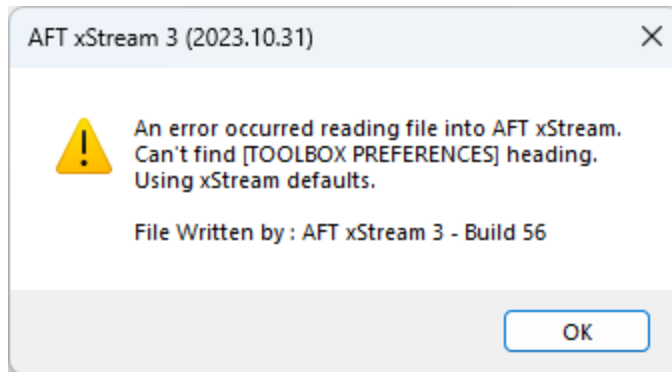


Figure 1: Error message indicating that some portion of the model file cannot be found

Another indicator is if the model's file size has been reduced to 0 or 1 KB. This indicates the data inside the file has been lost. This can be seen in the file size in the Windows File Explorer (Figure 2).

Name	Size
Corrupted Model File.xtr	1 KB
Normal Model File.xtr	105 KB

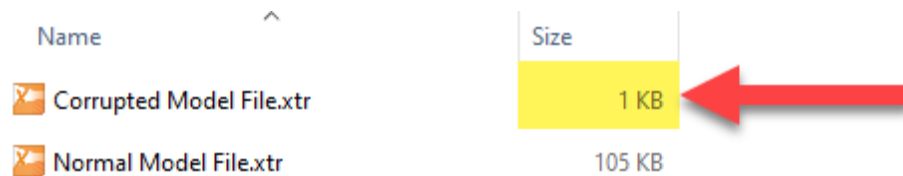


Figure 2: Very low file sizes can be an indication of model file corruption

What are the backup files (.bak extensions)?

AFT software will automatically generate backup versions of your model file each time you save. By default, these files will be located in the same directory where you have saved your model. These files have a specific naming convention, but generally the file extension will start with ".bak" and end with three numbers. Newer versions of AFT software will have a fourth letter after the .bak, and it signifies which application the backup is for.

- AFT Fathom.**bakF-001**
- AFT Arrow.**bakA-001**
- AFT Impulse.**bakI-001**
- AFT xStream.**bakX-001**

The numbers signify previous versions of the model, with the 001 extension being the most recent backup. Of course, if your current model file is corrupted and you need to restore a backup, the 001 backup won't contain the most recent updates you made to the model, but it will be better than losing everything! The backups named 002 and 003 are the second and third most recent backups, respectively. Generally, you won't need to use these files unless you want to revert to even earlier state of the model file, or there is an issue with the 001 backup.

When you save a model file, the application will copy the previous version of the file into .bak001, the previous .bak001 will be copied into .bak002, and so on, creating a cascading effect. This way .bak001 is always the most up to date backup version of the model.

How to Restore a Backup File

1. Close the AFT application if it is still open.
2. In the same location as your model file, locate the file with the extension *.bak-001.
3. Create a copy of *.bak-001. This is useful in case something goes wrong, and you need to try again.
4. Rename the copied file and change the file extension to the appropriate application file extension below. Windows will prompt you to confirm the extension change; click Yes.
 - Fathom: *.fth
 - Arrow: *.aro

Troubleshooting

- Impulse: *.imp
 - xStream: *.xtr
5. Try opening the restored file. If this didn't solve the problem, you can try the same steps with an earlier backup file, using .bak-002 or .bak-003 if they exist. However, note that doing so will revert even more changes because these are even earlier versions of the model from multiple saves prior.

Name	Date modified	Type	Size
MM_2022-12-29.bakF-001	12/29/2022 5:44 PM	BAKF-001 File	88 KB
MM_2022-12-29.bakF-002	12/29/2022 3:51 PM	BAKF-002 File	88 KB
MM_2022-12-29.bakF-003	12/29/2022 2:11 PM	BAKF-003 File	88 KB
MM_2022-12-29.fth	12/29/2022 6:33 PM	AFT Fathom 12 File	78 KB

Latest backup

Figure 3: The newest backup file, identifiable by file extension numbering and date modified

Name	Date modified	Type	Size
MM_2022-12-29 - Restored.fth	12/29/2022 5:44 PM	BAKF-001 File	78 KB
MM_2022-12-29.bakF-001	12/29/2022 5:44 PM	BAKF-001 File	78 KB
MM_2022-12-29.bakF-002	12/29/2022 3:51 PM	BAKF-002 File	88 KB
MM_2022-12-29.bakF-003	12/29/2022 2:11 PM	BAKF-003 File	88 KB
MM_2022-12-29.fth	12/29/2022 6:33 PM	AFT Fathom 12 File	78 KB

Figure 4: Changing file name extension to restore the backup file

Tips to Prevent File Corruption

The best way to prevent file corruption is to open, edit, and save the file locally on your PC. It is not recommended to edit and save files directly to a shared server, network drive, or cloud storage platform like Dropbox or OneDrive.

If the connection to the network location is interrupted or cuts out while saving the model file, it could corrupt the file or cause data loss. This can be especially common on VPNs. If you need to have the file in a shared location for colleagues to access, it is recommended that you download the file to your local PC, edit/save the file locally on your machine, then upload the file afterwards.

Customize Your User Settings Related to Backups

By default, only a maximum of three backups will be kept at a time. If desired, you can change this number in Number of Backup Files Created, which can be accessed in [User Options](#) as shown in Figure 5. Here, you can also choose to check the box that will automatically save your model in a specified time interval.

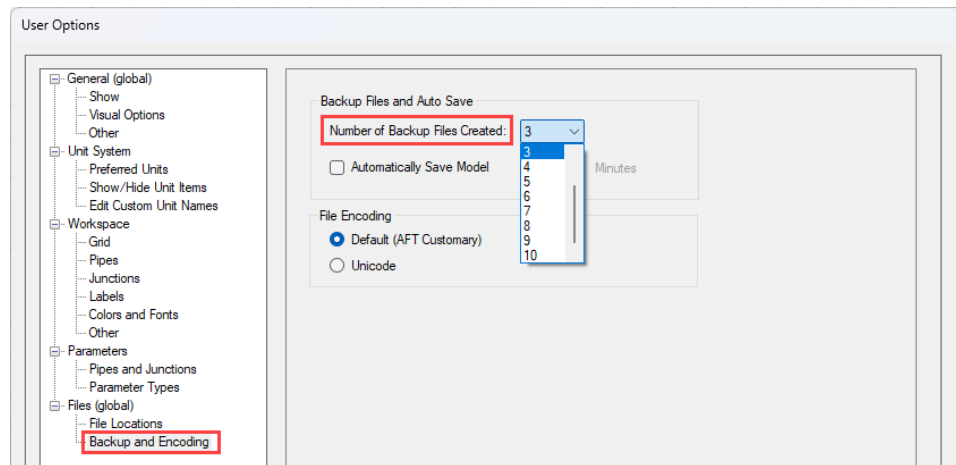


Figure 5: The number of backup files created and options to save automatically can be customized in User Options

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

Technical Support Options	593
Contacting AFT	594
Information on Software Installation	594
Chempak Database Installation	594
Output Files	596
Updated Fluids Library	596
Special File Control Features	597
Converting Models from Earlier Versions	598

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. AFT is limited in the 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.pmttechgroup.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
- **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:

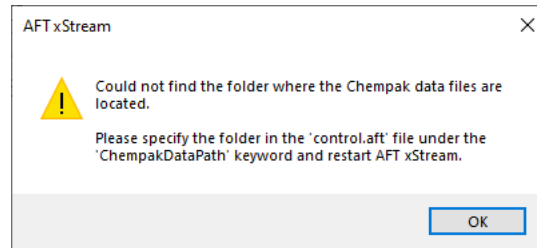


Figure 1: Error message when Chempak data cannot be found in default location

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.

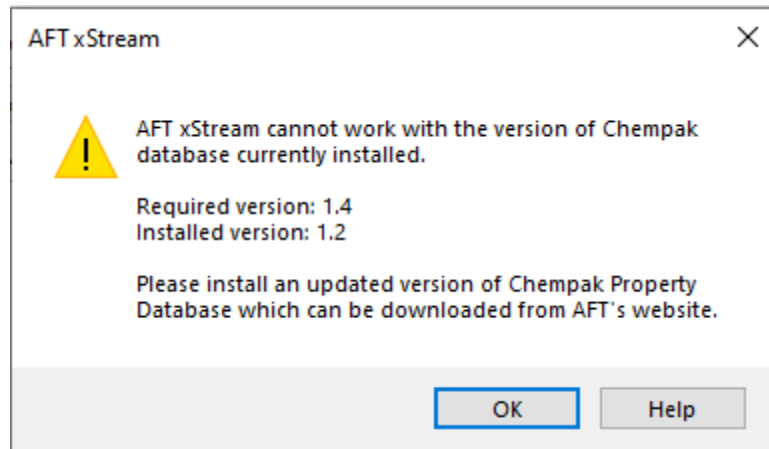


Figure 3: Outdated Chempak data

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.

Output Files

Numerical results and output from the Solver are saved in Output Files (*.out file extension). Output Files are stored in the same location as the model file, and a unique file will exist for each [scenario](#) that is run, differentiated by numbers appended to the file name. The General section of the [Output window](#) will list which Output File is used for the current scenario.

The Output File will only remain on disk if the model file is saved after it has been run.

Changing various input data parameters for the model will invalidate the previous results, and a prompt will appear stating: "This action will clear all results and delete the output file for the current scenario and any children. Do you want to continue the action?"

Renaming the model file in the Windows File Explorer will unlink the model file from any existing Output Files. Instead, open the model file and choose File > Save As which will rename and copy all Output Files. Alternatively, the Save Scenario to File options can be used from the Scenario Manager.

The file size will be heavily dictated by the simulation duration and amount of time steps saved as specified on the [Simulation Duration panel](#) in Analysis Setup. The [Sectioning Panel](#) will also have an impact and will increase the file size if the Output Pipe Stations are set to All Stations.

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:

The following fluid(s) used in the scenario have different property data than what is currently in the library:

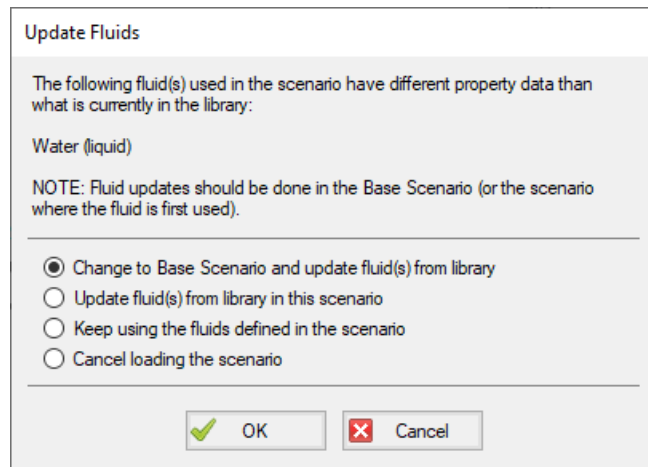


Figure 1: Update Fluids dialog window which appears if the fluid properties for the model have changed since it was last opened

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 "Keep using the fluids defined in the scenario" option 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 versions 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 (two or more versions behind), the automatic conversion is not successful. If this occurs, please contact our [support team](#) with a copy of the original model file.

Other files such as libraries (previously called "databases") or preference files can also be converted.

The [Workspace Layers Import Assistant](#) can be used for models created in versions prior to the existence of Workspace Layers. This tool allows views from Visual Report (which is now deprecated) and customized Workspace appearance to be preserved.

Index

A

Activate Modules 302, 311, 318
Add Component to Library 303
Additional (Minor) Losses 190
Additional Losses 190
AFT Standard fluid 87
Align 304
Analysis Setup 301
Animation
 Workspace Layers Animation 55, 332
Area Change 242
ASME Steam 87
Assigned Flow 244
Assigned Pressure 246

B

Backup Settings 387
Bend 248
Bounded by known flow rates 72
Branch 252

C

Cautions 547
Check Valve 254
Checklist 311
Chempak Fluid 87

Clear Initial Guesses 300
Color Map 330
Colors and Fonts 384
Compare Scenarios 302
Compressor/Fan 255
Control Valve 277
Conventions 70, 221
Converting models from earlier versions 598
Critical Warnings 547
Custom Libraries 443, 447-448, 456
Custom Units 303
Cutoff Frequency 107, 437

D

Dead End 280
Design Alerts 302
Design Factors 211
Disable Pipes Snapping to Isometric Grid 305
Duplicate Special 300

E

eLicense Information 303
Exit Valves 294
Export Model Data 298

F

Favorite Fittings & Losses 191
File Locations 387

Fittings & Losses 303
Fittings and Losses 190
Flip Vertex 305
Fluid Library 303
Fluid Property Viewers 302

G

General Component 281
Graph Results 310
Grid 382

H

Heat Exchanger 282

I

Import Graph Data 298
Information Messages 547
Initial Flow Guess 210
Inspection 182
Insulation 195
Insulation Library 303
Integration Query 302
Interpolated X-Y Data 78

J

Junction library 303
Junctions
 Area Change Properties Window 242
 Assigned Flow Properties Window 244

Assigned Pressure Properties Window 246
Bend Properties Window 248
Branch Properties Window 252
Check Valve Properties Window 254
Compressor/Fan Properties Window 255
Control Valve Properties Window 277
Dead End Properties Window 280
General Component Properties Window 281
Heat Exchanger Properties Window 282
Orifice Properties Window 282
Relief Valve Properties Window 285
Screen Properties Window 287
Tank Properties Window 289
Tee/Wye Properties Window 292
Valve Properties Window 293
Venturi Properties Window 295

K

Keep Network eLicense Checked Out 303

L

Labels 328, 383
Last Scenario 302
Libraries 443, 456
Load Background Picture 299
Lock Object 305
Loss factor 221

M

Make Selected Pipes and Jcts Orthogonal 305
Merge 298
Minimizing the Quick Access Panel 320
Minimum Number of Frequency Samples 108, 438
Model Data 310
Model Data Control 302
Model Data Print Content 302
Model Diagnostics 303
Model Status Light 319
Multi-Condition Events 302

N

Network Libraries 444
NIST REFPROP 87

O

Open Percentage Data 294
Orifice 282
Output 310
Output Control 302
Output File 596
Output Language 302
Output Messages 547
Output Print Content 302
Overview Map 317

P

Parallel Pipes 211
Pipe Drawing Mode 304
Pipe Line Size and Color 327
Pipe Material Library 303
Pipe Segments 305
Polynomial 78
Pressure Drop 190
Primary Windows 309
Properties Tab 313
Pulsation Frequency Analysis 302

Q

Quick Access Panel 324
Quick Graph Formatting 302

R

Recovery File 388
Relief Valve 285
Reset Label Locations 305
Reverse Direction 305
Rotate Icon 304
Rotate Selection 304
Run Model 302

S

Satisfying Connectivity Requirements 181
Scale/Flip Workspace 305

Scenario Comparison Tool 302
Scenario Manager 302, 312
Screen 287
Select Flow Path 300
Select Graph Parameters 302
Select Special 300
Set Pipe and Junction Locations to Parent 305
Show Grid 304
Show Object Status 182
Snap to Grid 304
Sonic Choking 294
Sort Table By 304
Special Conditions 220

T

Tank 289
Tee/Wye 292
Time Simulation 301
Transfer Results to Initial Guesses 300
Troubleshooting Models 583

U

Undefined Objects Window 181
Unique solution 72
Use Status Feature 181
User Library Fluid 87
User Options 303, 382, 386

V

Valve 293

Venturi 295

W

Warnings 547

Workspace 310, 324

Workspace Icon 220

Workspace Layers 324

 Active Layer 326

 Animation Settings 56, 330

 From Workspace Selections 326

Workspace Layers Tab 316

Workspace Symbols 332, 385