

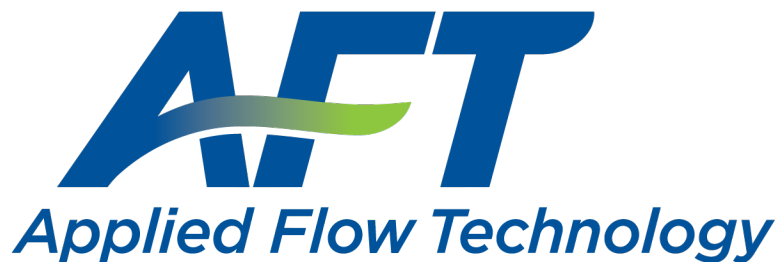
AFT ArrowTM

Product Help Guide

AFT Arrow Version 10

Compressible Pipe Flow Modeling

Published: March 08, 2024



*Dynamic solutions for a fluid world*TM

Contents

Overview of AFT Arrow	17
Engineering Assumptions	19
New in Version 10	19
Interface Improvements for Arrow 9	28
New in Version 9	29
New in Version 8	34
New in Version 7	38
New in Version 6	42
FAQ - Frequently Asked Questions	44
Arrow Learning Resources	56
Startup Window	57
Fundamental Modeling Concepts	61
Pipes and Junctions	61
Reference Positive Flow Direction	62
Features for Modeling Irrecoverable Losses	63
Philosophy of Computer Modeling	64
Role of Pressure Junctions and How They Work	65
Open vs. Closed Systems	68
Curve Fitting	71
Stagnation vs. Static Pressure Boundaries	73
Analysis Setup Overview	76
Modules Group	77
Modules Panel	78
Fluid Properties Group	81

Fluid Panel	82
Pipes and Junctions Group	97
Pipes and Junctions Panel	97
Transient Control Group	99
Simulation Duration Panel	99
Goal Seek and Control Group	100
Variables Panel	101
Goals Panel	106
Numerical Controls Panel	111
Steady Solution Control Group	111
Solution Method Panel	112
Tolerance Panel	141
Iterations Panel	144
Pressure and Temperature Limits Panel	146
Environmental Properties Group	147
Environmental Properties Panel	148
Cost Settings Group	149
Cost Settings Panel	150
Automatic Sizing Group	152
Sizing Summary Panel	153
Miscellaneous Group	153
Heat Transfer Sectioning Panel	154
Advanced Parameters Panel	155
Working with Pipe and Junction Objects	157
Selecting and Finding Objects	157
Selecting Objects	158

Finding Workspace Objects	160
Groups	161
Pipe Drawing Modes	162
Global Edit	169
Global Pipe Editing	169
Global Junction Editing	170
Toolbox	171
Creating Pipes - the Pipe Drawing Tool	173
Joining or Splitting Pipes	175
Creating Junctions	177
Connecting Objects	177
Moving Objects	178
Editing Objects	179
Morphing Junctions	180
Defining Objects	180
Highlight Required Input	182
Inspecting Objects	183
Notes	184
Status	185
Workspace Pan	185
Moving Pipe and Junction Labels	186
Special Conditions	187
Pipes	189
Pipe Properties Window	189
Common Pipe Input Parameters	191
Pipe Model	192

Fittings & Losses	194
Heat Transfer in Pipes Overview	196
Constant Heat Transfer	198
Convective Heat Transfer	199
Buried Pipe Heat Transfer	201
Heat Tracing Flux Based On Wire Turns and Length	202
Heat Transfer In Pipes - Detailed Discussion	206
Insulation for Pipes	213
Optional Input for Pipes	214
Add/Remove Pipe Segment	215
Reverse Direction	215
Pressure Drop in Pipes Detailed Discussion	216
Junctions	221
General Junction Information	222
Two Window Formats	224
Common Junction Input Parameters	226
Optional Input for Junctions	228
Specifying Losses	229
Conventions for Specifying Basis Area	230
Reference Density	230
Equal Percentage Valves	232
Junction Loss Models	233
K Factor Loss Model	234
Cv/Kv and Xt Loss Model	235
Resistance Curve Loss Model	237
Area Change	239

Area Change Loss Model	239
Assigned Flow	241
Assigned Pressure	242
Bend	243
Bend Loss Model	244
Branch	247
Check Valve	248
Control Valve	249
Control Valve Setpoint Not Achievable	252
Compressors & Fans	253
Centrifugal Compressor	255
Fans	260
Positive Displacement Compressor	261
Compressor/Fan Configuration	263
Compressor/Fan Efficiency Data	267
Compressor/Fan Sizing	269
Compression Ratio Reference Inlet Conditions	270
Multiple Configurations	271
Affinity Laws	276
Dead End	277
General Component	278
Heat Exchanger	278
Heat Exchanger Heat Transfer	279
Orifice	283
Orifice Loss Model	284
Relief Valve	287

Screen	290
Screen Loss Model	291
Separator	292
Spray Discharge	293
Tank	295
Tee/Wye	295
Tee/Wye Loss Model	296
Valve	299
Valve Modeling Closures with K Factors	302
Venturi	302
Interface and User Options	305
Menus and Toolbars	305
File Menu	306
Edit Menu	307
View Menu	309
Analysis Menu	309
Tools Menu	310
Library Menu	311
Arrange Menu	312
Window Menu	314
Help Menu	314
Toolbars	315
Printing	316
Primary Window Overview	318
Status Bar	319
Quick Access Panel	320

Model Status Light	329
Solution Progress Window	329
Workspace Window	329
Workspace Layers	331
Annotations	341
Renumbering Workspace Objects	345
Merging Models	346
Scale/Flip Workspace	347
Magnifier Tool	347
Duplicate Special	348
Undefined Objects Window	349
Keyboard Shortcuts for Workspace	350
Extended Model Check	353
Raw Data Changes	353
Model Data Window	354
Model Data Control	356
Bill of Materials	358
Output Window	359
Output Control	364
Output Parameters	370
Applied Standards	371
Graph Results Window	371
Quick Graph Formatting Window	378
Graph Types	378
Compressor Fan vs. System Curve Methods	381
Basic Graph Tutuorial	382

Multi Scenario Compressor Fan Vs. System Curves Tutorial	390
User Options Overview	395
Unit System	396
Workspace	398
Parameters	401
Files	402
Sizing Preferences	404
File Import and Export	406
Importing Piping Component Files	406
Importing CAESAR II Neutral Files	411
Importing GIS Shapefiles	415
Importing and Exporting EPANET Files	417
Importing and Exporting from Other AFT Products	421
Exporting to Excel	422
Importing from Excel	428
Design Alerts	441
Design Alerts From Output	444
Design Alerts Importing & Exporting	445
Scenario Manager	447
Creating, Organizing, and Editing Scenarios	448
Modifying Individual Scenarios	451
Scenario Inheritance	452
Running Models in Batch Mode	455
Open Model Using Last Scenario	457
Disconnected Scenario	457
Scenario Comparison Tool Overview	457

Accessing Scenario Comparison Tool	458
Modify Comparison	459
Scenario Comparison Grid	461
Cost Calculation	465
Enabling Cost Calculation	466
Energy Cost Library	467
Cost Application Manager	470
Scale Tables	472
Select Size From	474
Cost Definitions	475
Cost Libraries	478
Cost Summary Window	483
Custom Monetary Units	484
Cost Report	485
Add-on Modules	489
GSC - Goal Seek & Control	489
Goal Seek & Control in Model Data	491
GSC in Output	493
GSC in Scenarios	493
Changing Input Values to GSC Results	494
Goal Seek and Control Troubleshooting	495
ANS - Automated Network Sizing	500
Sizing Fundamentals	502
Sizing Window	516
Solution Progress with ANS	536
Sizing Results	538

Workspace Integration	539
Junction Loss Factors	542
Improving and Troubleshooting the Sizing Process	542
Sizing Components Other Than Pipes	544
Search Methods	557
Sizing Theory	567
XTS - Extended Time Simulation	586
Choosing Time Simulation	588
Junction Transient Data	588
XTS in Workspace	600
XTS in Model Data	601
XTS in Output	601
XTS in Graph Results	603
Libraries	613
Library Manager	614
Library Browser	614
Junction Library	618
Fluid Library	619
Pipe Material Library	620
Insulation Library	623
Fittings & Losses Library	624
Network Library Overview	629
Benefits of Shared Libraries	630
Creating an Enterprise-Wide Network Library System	631
Library Verification	634
Library Locations in General Preferences	636

Theory and References	639
Nomenclature	639
Review of Compressible Flow Theory	642
Compressible Flow Theory in Single Pipes	657
Extension of Single-Pipe Methods to Networks	658
Fundamental Equations	659
Continuity Equation at a Branch	660
Energy Equation at a Branch	661
Concentration Equation at a Branch	662
Network Implementation	662
Verifying Network Solutions	663
Sonic Choking	665
Subsonic vs. Sonic Pressure Losses	665
Sonic Choking Detailed Description	666
Multiple Choking Points in Single Flow Path	671
Role of Pressure Junctions - Detailed Discussion (Long)	674
Chempak Technical Information - Detailed Discussion	686
Stagnation Properties	698
Sound Power Level	699
References	701
Pipe Material Library References	703
Output Messages & Warnings	709
Critical Warnings	709
FCV/PRV/PSV Could Not Control Pressure - Pressure Was Added to Achieve Control Flow/Pressure... ..	710
Global energy balance disabled by user in Advanced Parameters panel	710
Global mass balance disabled by user in Advanced Parameters panel ...	710

Junction X Had Reverse Flow - Compressor/Fan Pressure Could Not Be Predicted	711
Junction X uses normal or standard flow units like Y in a flow path with dynamic gas mixing... ..	711
Pipe Sequences of zero friction exist, flow rate cannot be calculated	711
Required flow to supply the pressure control valve at junction X is above sonic flow and cannot be supplied	711
Setting relaxation lower than relative pressure/flow tolerance gives unreliable results	712
Specified flow through/out of junction X is above sonic flow and cannot be supplied	712
These results do not represent a converged solution	712
Warnings	713
Chempak temperature and/or pressure is out of range for the following ..	713
Compressor X is operating at a corrected flow greater/lower than the entered data	713
Compressor X is operating at a corrected speed greater/lower than the entered data	714
Compressor X is operating at a non-dimensional flow greater/lower than the entered data	714
Compressor X is operating at a non-dimensional speed greater/lower than the entered data	715
Concentration at Junction X may not be balanced	715
Control valve open percentage outside valid range of 0-100%	715
Energy may not be balanced at Junction X	715
FCV at Junction X could not control flow	716
Flow at Junction X may not be balanced	716
Fluid is no longer in the vapor phase because the fluid temperature/pressure is... ..	716
GSC Goal may not be valid	718

Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy	718
Junction X Compressor/Fan Added Negative Pressure	718
Junction X Exceeded Runout Flow Rate... ..	718
Maximum system pressure exceeded enthalpy table's maximum pressure of X	718
Physical flow area is less than the sonic area	719
Pipe/Junction X property calculation exceeded maximum local iteration limit	719
Pipe/Junction X property calculation failed to continue towards convergence	720
Pipe X temperature is equal to or greater/less than the max/min temperature limit of Y	720
PRV at Junction X could not control pressure	721
PSV at Junction X could not control pressure	721
Separator junction X is trying to remove Y but none is entering junction ..	721
Separator junction X is trying to remove Y of Z but only A is entering junction	721
Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range	722
The following junctions have a predicted Sound Power Level greater than 155 dB	723
Unique Solution Not Possible - Relief Valve Cycles	723
Cautions	723
Check Valve closed at Junction X	723
Flow through Junction X is negative. Loss factors may be in error.	724
Flow solution for Junction X is above/below the maximum/minimum value specified for the resistance curve. Flow rate: Y	724
Junction X is downstream of a choking point	724
Note No Scenarios have been set in the Multi-Scenario Output display ..	725

Pipe X length is shorter than the elevation change	725
Pipe X steady state Reynold's Number is less than 100. Minor Losses may be significantly lower than expected.	725
Spray Discharge Junction X closed because inflow occurred and Spe- cial Condition - No Inflow was set	726
The following Detailed Tee/Wye junctions have a different automatic ori- entation than defined on the Workspace	726
Transient data did not extend to Stop Time at Junction X	727
Information	727
Flow at the Following Junctions is Laminar	727
Troubleshooting	729
Diagnostics Window	730
Use the Model Data Window	731
Transfer Results to Initial Guesses	734
Poor Compressor/Fan Curve Fits	735
Troubleshooting Resistance Curves	736
Use the Output Window Sort Feature	737
Tee/Wye Junction Complexity	738
Lower the Flow Rate Relaxation	739
Try Absolute Tolerance	740
Make Initial Flow Rate Guesses for Pipes	741
Review the Iteration History	742
Change Boundary Conditions	742
Turn Off Parts of the Model	743
Break a Large Model Into Submodels	744
Change Solution Method	745
Try a Different Fluid, Equation of State or Accuracy Option	745

Using Lumped Adiabatic as a Starting Point	746
Identifying and Restoring Corrupted Model Files	747
Technical Support	751
Technical Support Options	751
Contacting AFT	752
Information on Software Installation	753
Chempak Database Installation	753
Output Files	755
Special File Control Features	756
Converting Models from Earlier Versions	757
Index	759

Overview of AFT Arrow

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

AFT Arrow brings increased productivity to the pipe flow engineer by combining traditional engineering hydraulic analysis with an intuitive graphical user interface. By implementing a drag-and-drop method of pipe flow modeling, AFT Arrow simplifies the pipe network modeling process.

AFT Arrow 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 Arrow'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 Arrow increases your productivity in the modeling process.

What Can AFT Arrow Do?

AFT Arrow 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
- Pressure driven systems
- [Compressor or fan](#) driven systems, including multiple compressor/fans in parallel or in series
- Compressor/fans with [variable speed](#), controlled pressure and controlled flow
- Systems with [pressure and/or flow control valves](#)
- Systems with [valves closed and compressor/fans turned off](#)
- Heat transfer analysis and system energy balance
- Systems with non-ideal gases
- Systems that experience [sonic choking](#), including multiple sonic choking points
- Systems with non-reacting [flow stream mixing](#) and user defined mixtures (with optional Chempak)
- Systems with [elevation changes or rotation](#) such as in turbomachinery
- Multiple design cases in a single model file
- Cost calculation of compressor and fan energy usage

AFT Arrow provides hundreds of standard [loss models](#) for pipe system components, but also allows you to enter your own loss data. Variable loss models that depend on flow are supported.

AFT Arrow's powerful solution engine is based on standard techniques used for many years in industry. The Newton-Raphson method is used to solve the fundamental equations of pipe flow that govern mass and momentum balance. Solutions are obtained by iteration, and matrix methods optimized for speed are employed to obtain convergence.

AFT Arrow provides [two independent solution](#) methods that can be run individually or in combination with each other to allow flexibility and cross-verification of results. In addition, AFT

Arrow offers two [lumped methods](#) which offer improved computation speed with a slight loss in accuracy. Modeling of low to high pressure and low to high velocity systems can be performed. Sonic capabilities allow modeling of high velocity conditions such as in relief systems.

AFT Arrow can model a large range of gas systems including steam, natural gas, air, and chemical and petrochemical process gases. Support for real gases is offered through several generalized [equation of state models](#) or the optional [Chempak](#) add-on.

Where to Start

The best place to start learning about AFT Arrow is to work through our examples and video tutorials found in the [Arrow Learning Resources](#).

Add-on Module Capabilities

AFT Arrow has two add-on modules which extend AFT Arrow's extensive modeling capabilities into new areas.

- The [Goal Seek & Control \(GSC\) module](#) allows the engineer to perform multi-variable goal seeking and simulate control system functions.
- The [Automated Network Sizing \(ANS\) module](#) automates the process of sizing pipes and ducts in a complex network based on required conditions, such as maximum pressure or minimum flow, to reduce cost or weight.
- The [Extended Time Simulation \(XTS\) module](#) allows the engineer to model time varying system behavior.

Who Can Use AFT Arrow

Effective use of AFT Arrow requires a basic understanding of compressible flow theory. Another topic offers a review of compressible flow theory and gives several references of more detailed information. AFT Arrow users should have a working knowledge of all the concepts discussed in [Review of Compressible Flow Theory](#). In some cases it may be advisable that users attend training courses offered by Applied Flow Technology or AFT Arrow recognized authorities.

In addition to familiarity with compressible flow theory, engineers who use AFT Arrow should be familiar with standard industry practice in analyzing gas piping systems.

Related Topics

New in Version 9.....	29
New in Version 8.....	34
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56

Engineering Assumptions.....	19
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

Engineering Assumptions

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

- [Compressible flow](#)
- Steady-state conditions (the [XTS Module](#) allows transient modeling through steady-state snapshots)
- One-dimensional flow
- System is fluid-full
- All gases are superheated (single phase)
- No chemical reactions
- No supersonic flow in the system

Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 8.....	34
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

New in Version 10

View the links below for key updates from past versions:

[New in Version 9](#), [New in Version 8](#), [New in Version 7](#), [New in Version 6](#)

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.

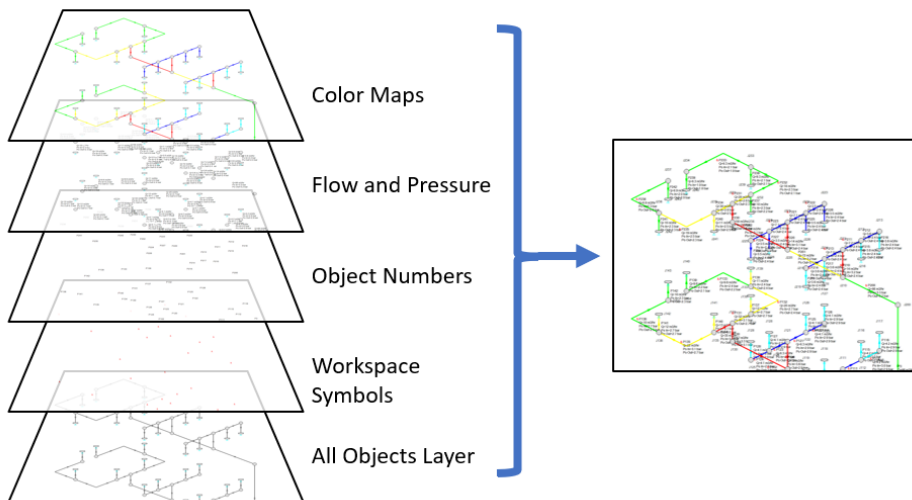


Figure 1: Workspace Layers can display input and output values superimposed over the model, show Color Maps, and control visibility of objects

Compressor Map

Model centrifugal compressors more accurately by accounting for multiple curves from a non-dimensional or corrected [Compressor Map](#). The Compressor Map feature allows the utilization of more precise flow curves obtained directly from the manufacturer. These curves provide a more accurate representation of the centrifugal compressor's performance when operating away from its design conditions. With the Compressor Map, AFT Arrow dynamically determines the closest curves to the actual operating point within the modeled system. It then performs interpolation between these curves to accurately determine the pressure ratio. This ensures that the simulation reflects the real-world behavior of the centrifugal compressor, even when it operates outside of its originally specified conditions.

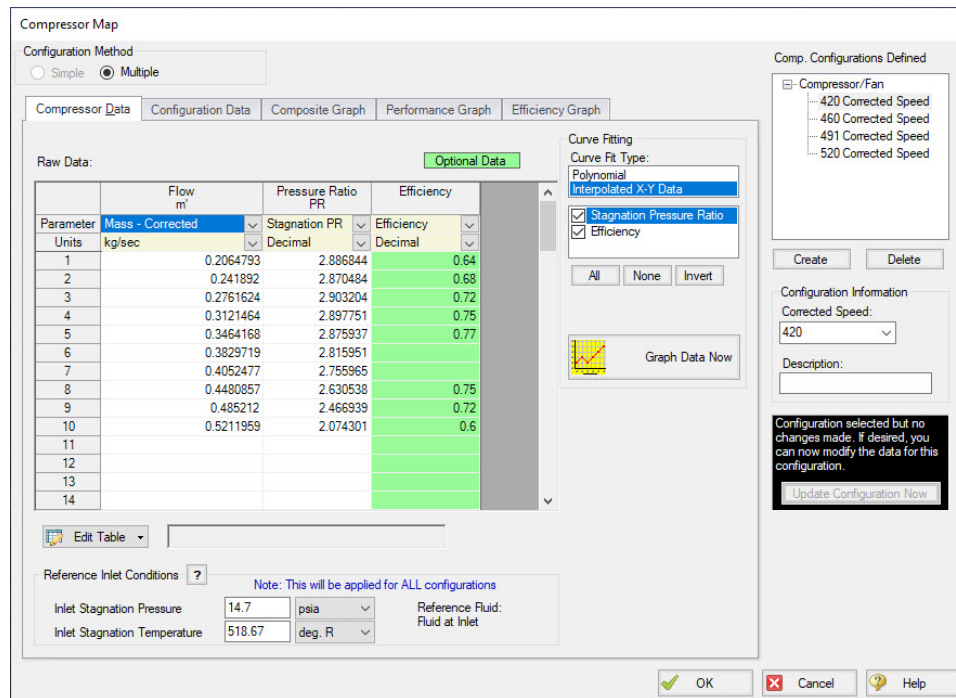


Figure 2: Compressor Map Menu Allowing for the Configuration of Multiple Speeds

Calculate Sound Power Levels

Estimate junction [Sound Power Levels](#) according to Energy Institute Guidelines. A new output parameter has been added for Sound Power Level, as calculated by the Energy Institute Guidelines for the Avoidance of Vibration Induced Fatigue Failure in Process Pipework. Sound Power Level is a function of pressure drop across inline devices, and serves as an initial calculation step to assess the presence of Acoustically Induced Vibration (AIV) issues in your system. If the calculated Sound Power Level exceeds a predefined acceptable value, a warning is issued to the user, and the Energy Institute Guideline indicates further calculation should be done to determine Likelihood of Failure (LOF). These additional calculations are possible with an AFT-supplied ancillary Excel Workbook and the Excel Export tool.

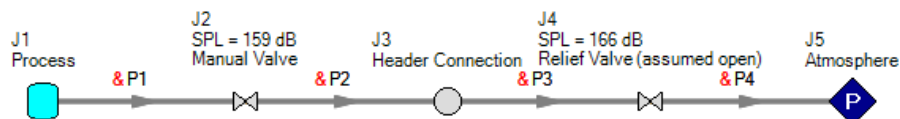


Figure 3: Calculated Sound Levels Displayed With Junction Names

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.

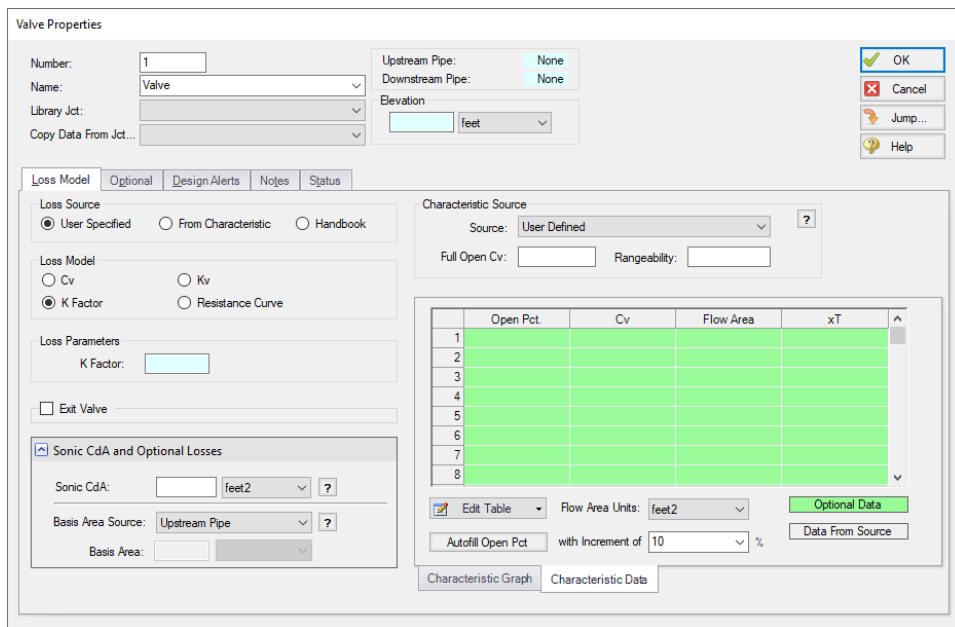


Figure 4: Redesigned Valve Properties window

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.

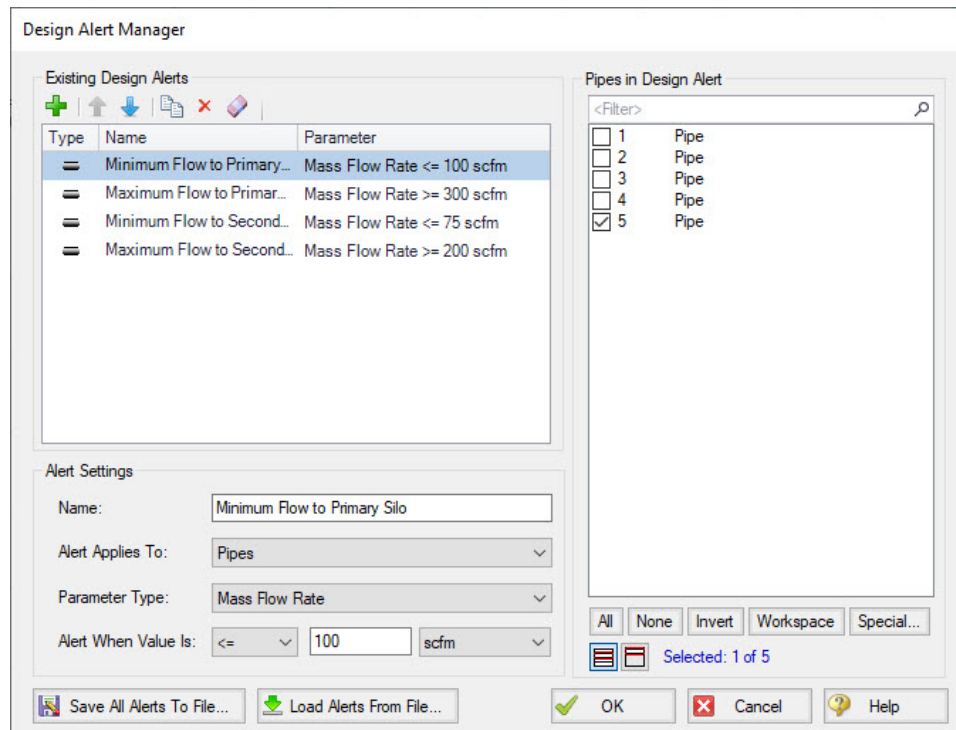


Figure 5: Steamlined Design Alerts Manager

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.

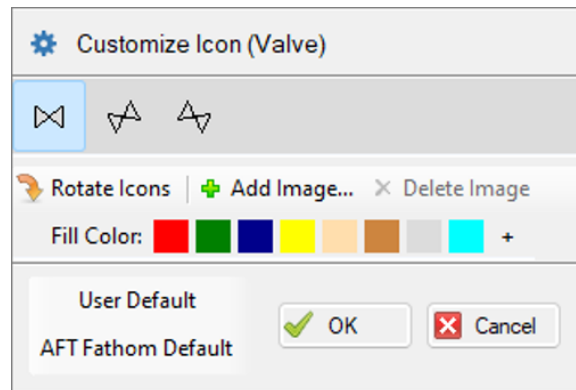


Figure 5: The Customize Icon window allows a custom image to be applied to any junction

- **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.

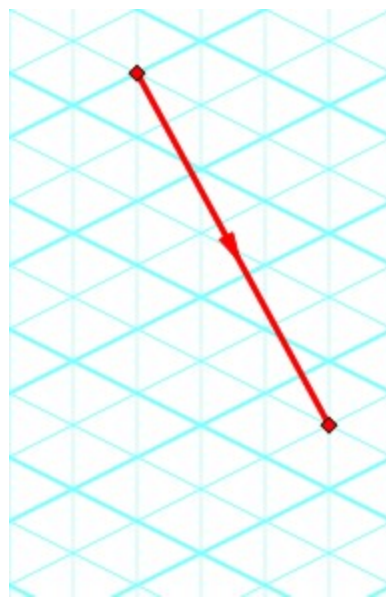


Figure 7: Isometric Freeform drawing mode does not force pipes to follow gridlines

- **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.

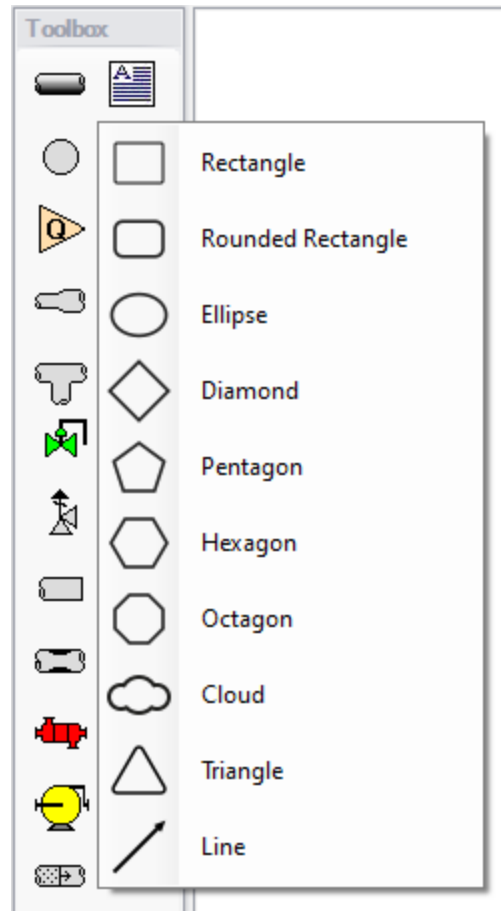


Figure 8: New Annotation options

- **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.

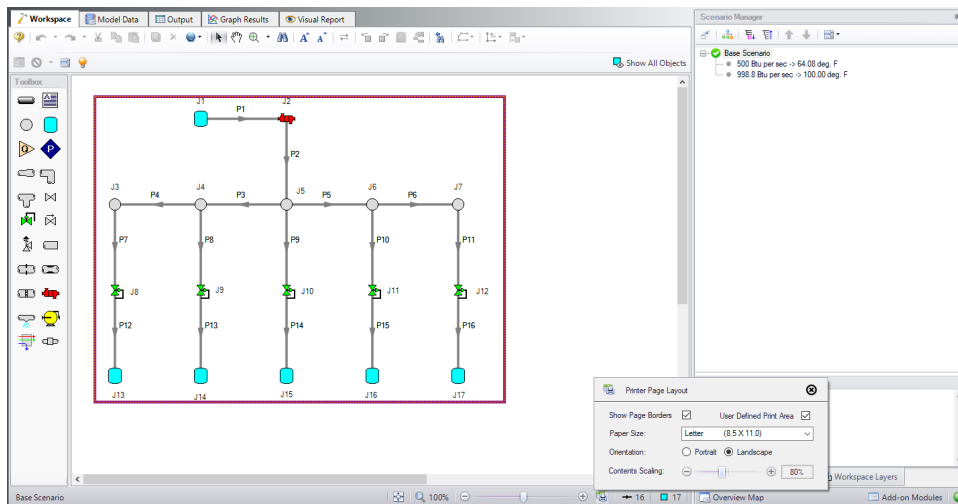


Figure 9: The Printer Page Layout feature allows the user to set the Workspace print area

- **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.

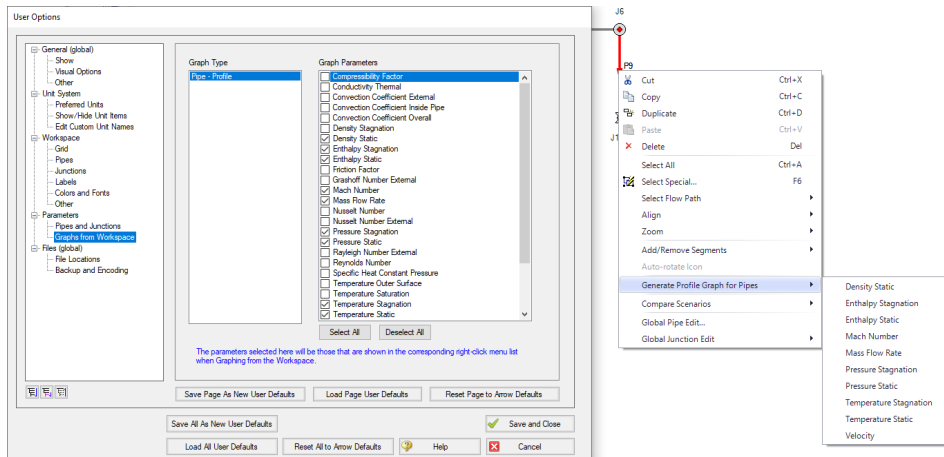


Figure 10: The right-click context menu for pipes on the Workspace can have the graphing parameter options modified

Miscellaneous Improvements

- **Modeling Additions and Improvements**
 - **Prevent Backflow at Sprays** - New "No Inflow" Special Condition for Spray junctions.
 - **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.
- **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** - 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** - Provide estimated xT value as used in the Solver for valve loss models that do not use the Cv/xT loss model.
 - **Profile Graphs for Temperature Saturation and Temperature Superheat** - Graph saturation temperature and temperature above saturation (superheat) along flow paths.
 - **Laminar Information Message** - Alerts the user if flow at junctions is in a [laminar range](#).

- **POR/AOR in Pump Summary** - Show the values for Preferred and Allowed Operating Ranges in the Compressor Summary.
- **Event Messages in New Format** - Events are now shown in a cleaner, collapsible format similar to Warnings.

Interface Improvements for Arrow 9

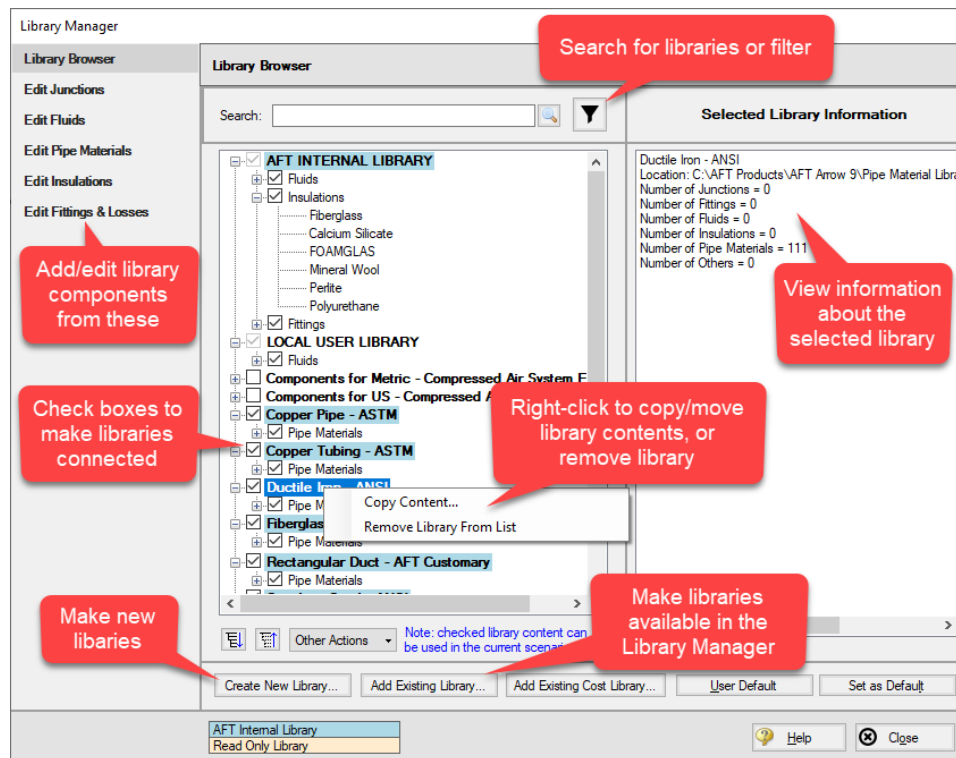
There were many improvements made to the interface in Arrow 9. These changes primarily involve consolidating inputs with the new Analysis Setup window, and updating the functionality of databases, which have been renamed as libraries.

Analysis Setup Sources in Arrow 9

- **System Properties:** The System Properties window has been split up into the [Fluid Properties group](#) and the [Environmental Properties group](#). The System Properties icon is still available from the Analysis menu and toolbar to directly open the Fluid Properties panel.
- **Solution Control:** The Solution Control parameters have primarily moved to the [Steady Solution Control group](#). The Special Controls have moved to the [Advanced Parameters panel](#) under the Miscellaneous group
- **Checklist:** The checklist which was shown in the Quick Access Panel has been replaced by the status icons in the [Analysis Setup window](#). Red exclamation marks denote which inputs are incomplete
- **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.
- **Cost Settings:** Cost calculations were previously enabled from the Analysis menu and defined in the Cost Settings window. Enabling and defining cost settings has now been moved to the [Cost Settings group](#)
- **Add-on Modules Activation:** The activation window for add-on modules has moved to the [Modules panel](#)
- **Goal Seek & Control Manager ([GSC Module Only](#)):** Settings have been moved to the [Goal Seek and Control group](#)
- **Transient Control ([XTS Module Only](#)):** Settings have been moved to the [Transient Control group](#)
- **Sizing Summary ([ANS Module Only](#)):** The Sizing Summary has been added to the [Sizing Summary panel](#) in Analysis Setup. The Sizing Summary can also still be viewed from the Sizing window. No other ANS module interface changes have been made.

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

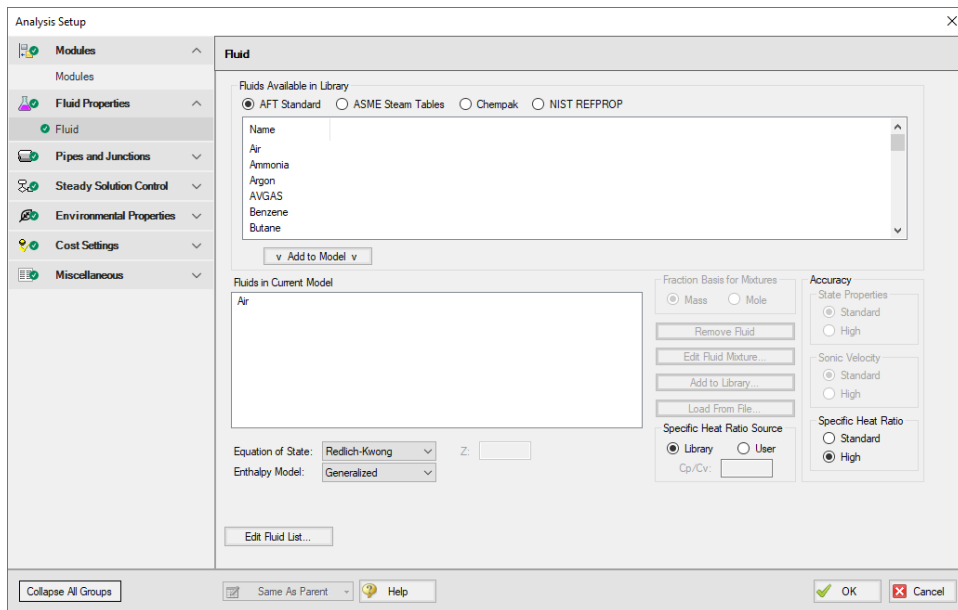


New in Version 9

Analysis Setup

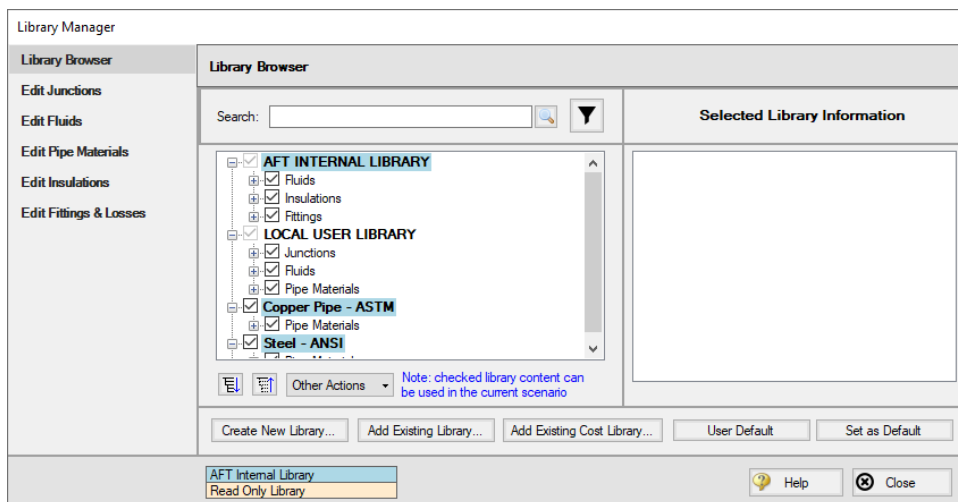
The [Analysis Setup window](#) consolidates windows for System Properties, Solution Control, Activate Modules, Extended Options, Goal Seek & Control Manager, and Transient Control all into one location. This allows users to activate and enable modules, specify fluids, change solution

tolerances or iterations, and so on, all from a single window. For a list of windows that have merged into the Analysis Setup window see the topic on [Interface Improvements for Arrow 9](#).



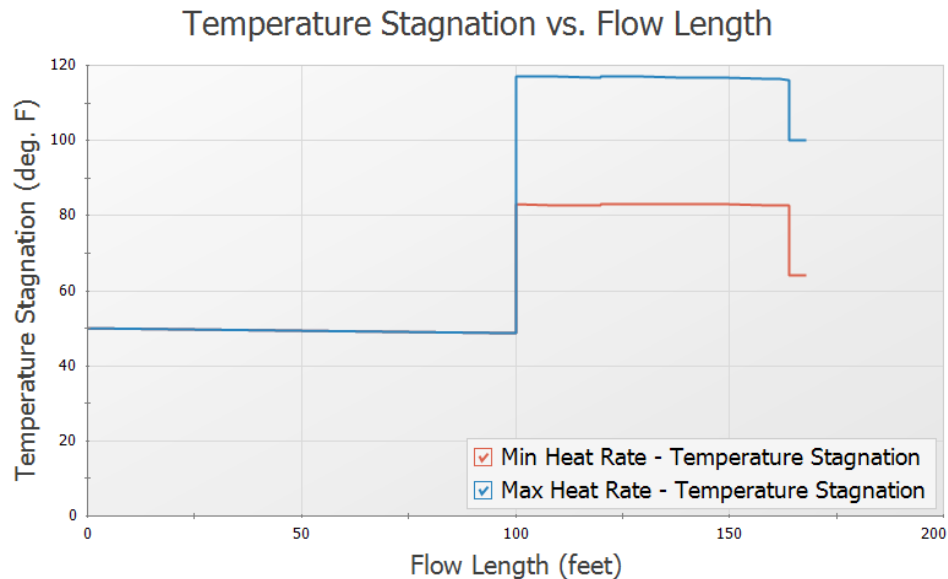
Libraries

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.



Multi-Scenario Graphing

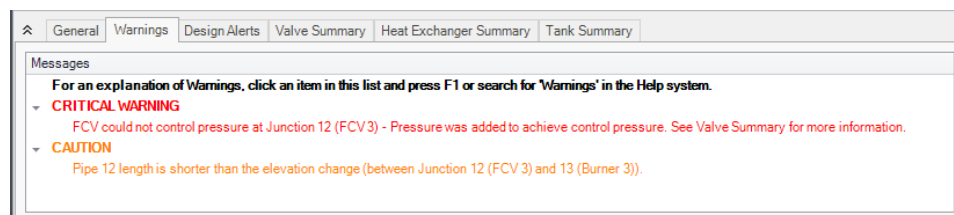
Profile graphs (and [Transient Graphs](#) in the XTS module) can now include output from multiple scenarios, making it easy to compare the results along a flow path for different scenarios.



Improved Organization of the Warnings in the Output

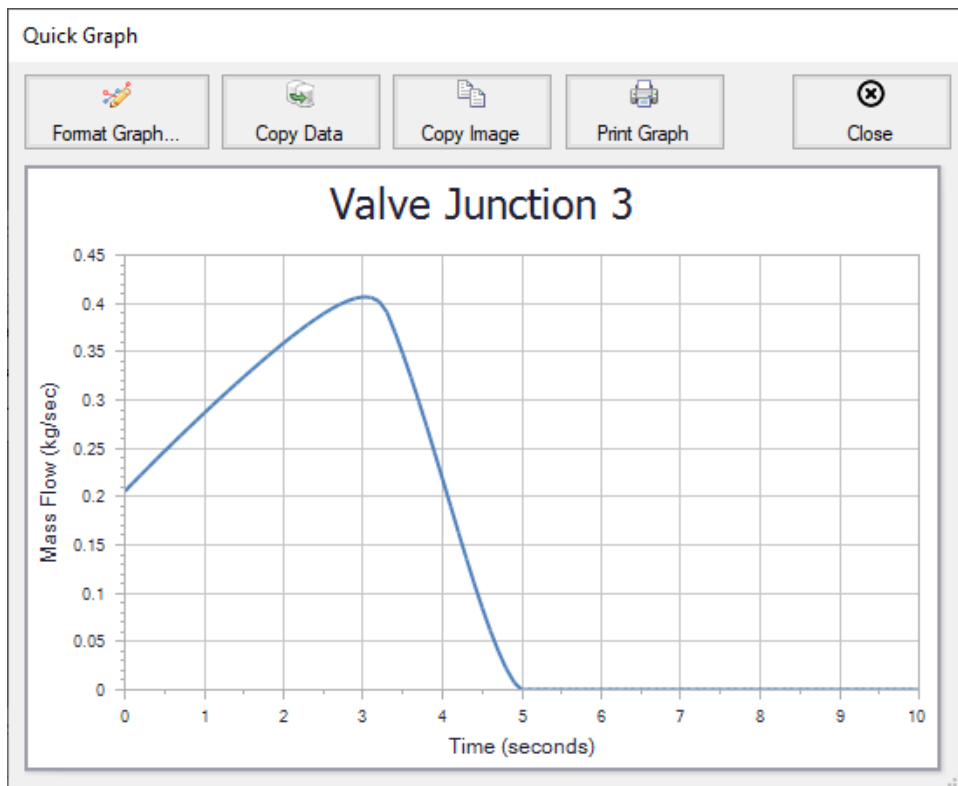
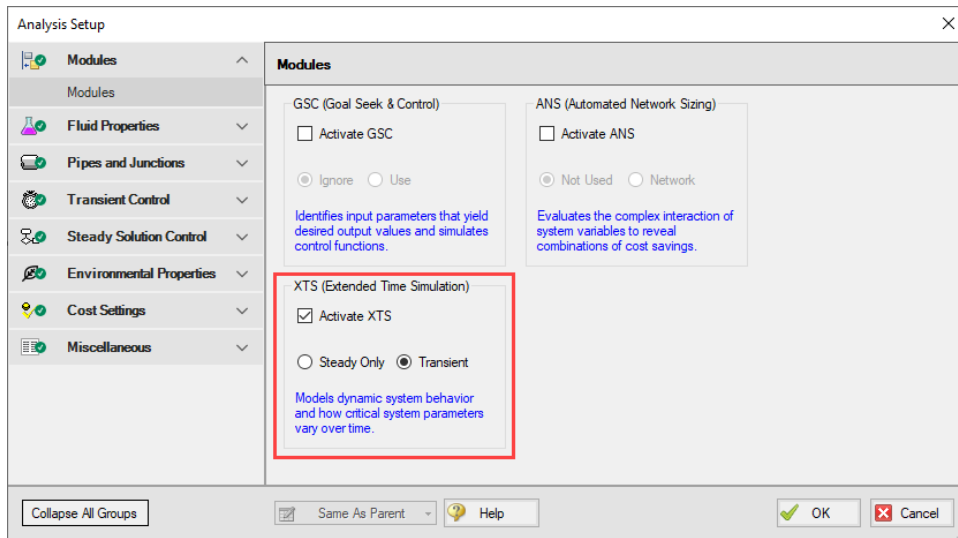
An improved display of the warnings and design alerts makes it easier to review and understand information in the following ways:

- [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
- Warnings, Cautions, and Design Alerts are now displayed for all scenarios when using the Multi-scenario Output feature



Extended Time Simulation (XTS) Add-on Module

The [XTS Module](#) is now available as an [add-on module](#) for AFT Arrow. The XTS module allows slow [transients](#) to be modeled such as pressurizing a tank, controlled valve opening/closing, and slow compressor/fan transients. Users can view transient output in the [Output window](#), create [transient graphs](#), or create transient Visual Reports.



More Improvements

- **Pipes/Junctions**
 - Pipes can be defined as zero-length connectors to connect two junctions which are directly flanged together
- **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
 - Heat Transfer output parameters added to Visual Report
 - [Design alerts](#) are grouped together in the General Output section
- **Miscellaneous**
 - SRK and Peng-Robinson [equations of state](#) are now available for 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

Related Topics

Overview of AFT Arrow.....	17
New in Version 8.....	34
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

New in Version 8

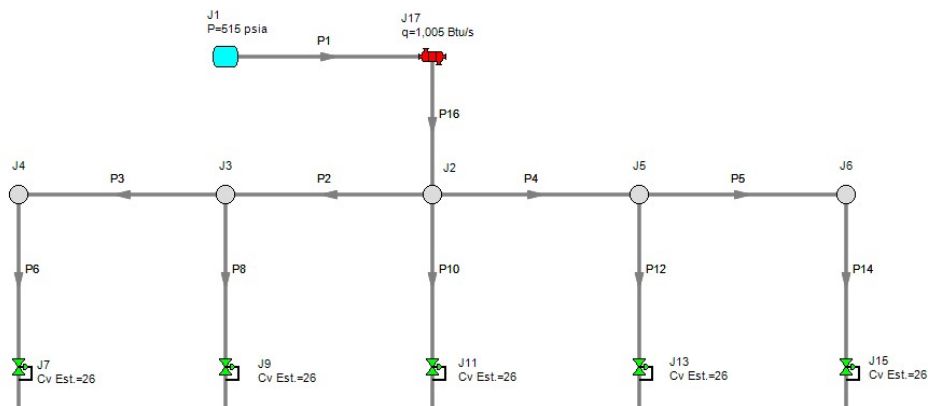
Scenario Comparison Tool

- Quickly get a comparison of different scenarios within a model file
- Useful to double-check input results for old models, or for troubleshooting purposes
- For a detailed description see [Scenario Comparison Tool Overview](#)

Item	Parameter	Scenario					
		Diffusion Only	Diffusion & Premix	Premix Only	Diffusion Only	Diffusion & Premix	Premix Only
Pipes (11 of 11 compared)							
P4	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
P7	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
P10	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Scaled Hydraulic Diameter	3.548 inches	4.026 inches	4.026 inches	4.026 inches	3.548 inches	3.548 inches
	Size	3-1/2 inch	4 inch	4 inch	4 inch	3-1/2 inch	3-1/2 inch
Junctions (12 of 12 compared)							
J5 (Diffusion Gas FCV)	Control Setting	22 lbm/sec	5 lbm/sec	5 lbm/sec	22 lbm/sec	5 lbm/sec	5 lbm/sec
J8 (Premix Gas FCV 1)	Special Condition	Closed	None	None	Closed	None	None
J11 (Premix Gas FCV 2)	Special Condition	Closed	None	None	Closed	None	None
Solution Control (4 of 4 compared) - No Differences							
System Properties (4 of 4 compared) - No Differences							

Junction- specific Parameters in Visual Report

- Output such as the compression ratio of a compressor or the C_v of a control valve can now be displayed in the Visual Report



Notes on Output

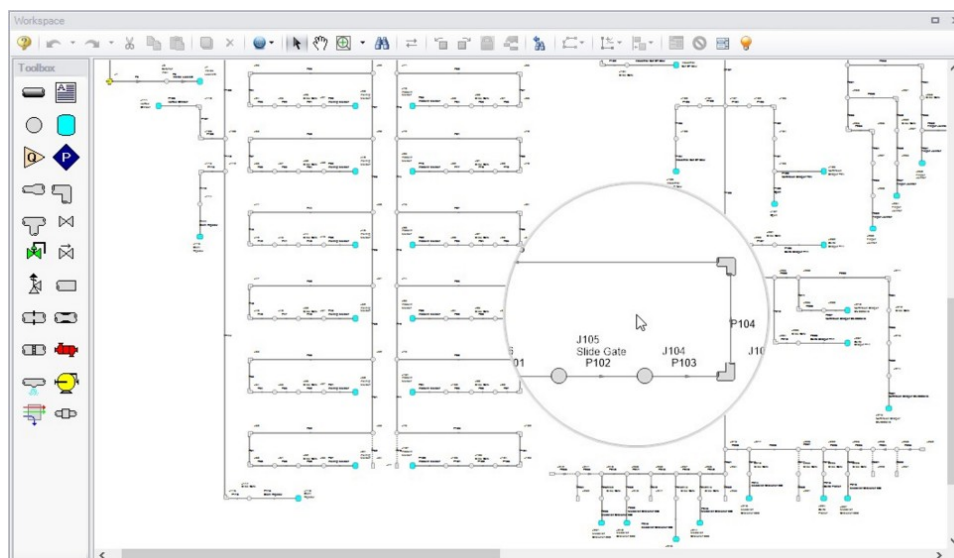
- Notes can be added to any cells in the [Output](#) to save observations and comments directly with the model file

Pipe	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)
3	Pipe	5.000	44.47	44.41	0.03054	0.03053	0.2667	469.9	469.6	469.6	469.4	0.8987
4	Pipe	10.000	88.93	89.05	0.06106	0.06117	1.0632	471.0	469.9	469.9	468.8	0.5056
5	Pipe	5.000	44.47	44.41	0.03054	0.03053	0.2667	469.9	469.6	469.6	469.4	0.8987
6	Pipe	5.000	76.54	76.56	0.05262	0.05264	0.2198	469.6	469.4	468.8	468.6	0.1490
7	Discharge Burner #1	5.000	318.70	321.41					114.7	112.3	111.3	0.1370
8	Pipe	5.000	76.64	76.65					469.7	469.1	468.9	0.1522
9	Discharge Burner #2	5.000	319.28	321.99					114.7	112.2	111.3	0.1402
10	Pipe	5.000	76.54	76.55					470.8	470.2	470.0	0.1539
11	Discharge Burner #3	5.000	319.57	322.29					114.7	112.2	111.3	0.1418
12	Pipe	5.000	76.64	76.65					469.7	469.1	468.9	0.1522
13	Discharge Burner #4	5.000	319.28	321.99					114.7	112.2	111.3	0.1402
14	Pipe	5.000	76.54	76.56					469.4	468.8	468.6	0.1490
15	Discharge Burner #5	5.000	318.70	321.41	0.21569	0.21754	0.9460	115.6	114.7	112.3	111.3	0.1370
16	Pipe	25.000	221.31	224.46	0.15215	0.15431	6.7158	477.7	471.0	470.9	464.1	0.4791

Maximum Burner Discharge velocity is 400 ft/sec

New Workspace Functionality

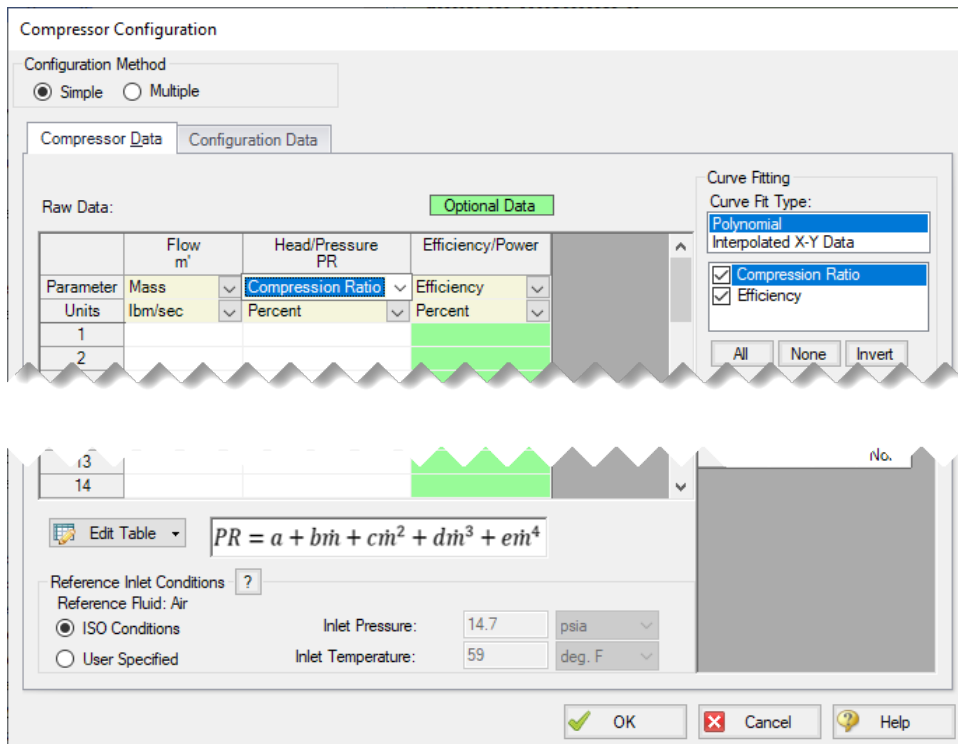
- Multi-level Undo** - Drawing, arranging and moving objects in the Workspace now allows for multiple levels of undo
- Workspace Magnifier** - The [magnifier tool](#) provides an enlarged view of sections of the model to make it easier to manage and navigate large models



- **Automatic Junction Rotation** - Arrow will automatically rotate junctions to visually align with neighboring pipes to speed-up model formatting
- **Combine Pipes** - a series of pipes and junctions can be [combined](#) to simplify a model, with any losses being imported to the Pipe Fittings and Losses tab
- **Select Shortest Flow Path** - The shortest path between two selected junctions can be found using this feature in the [Edit menu](#)

Define Compressors in terms of Compression Ratio

- The compression ratio at different flow rates can be directly specified in the [Compressor Configuration window](#)



Diagnostics Window

- Provides a list of features and settings which might cause issues as a starting point for model troubleshooting
- See the [Diagnostics Window](#) topic for more information

Multi-scenario Compressor/Fan vs. System Curves

- Compressor/Fan vs System curves from different scenarios can be plotted on the same graph within AFT Arrow
- Easily compare changes to compressor/fan operation in different cases
- For more information see [Multi-Scenario Compressor/Fan vs. System Curves Tutorial](#)

More Improvements

- **Overall Performance**
 - Improved intelligence on when a model needs to be saved and re-run to preserve the output
 - Faster model loading speed
 - Improved language refresh speed
 - New and updated themes
- **Workspace**
 - The [Undefined Objects Panel](#) (launched from clicking "Define Pipes and Junctions" in the Checklist) remains open when pipes/junctions are being edited
 - Double-clicking a junction allows more than one of that junction to be placed in the Workspace at a time
 - Snapping can be turned off for pipes when the [isometric drawing mode](#) is enabled
- **Output**
 - Output parameters can now be filtered/searched for simpler selection
 - The average thermal conductivity used for calculations can be displayed in the Output
- **Visual Report**
 - Visual Report Control has been redesigned to better filter parameters
- **Pipes**
 - [Heat tracing](#) now allows the entry of power/length and number of turns to allow Arrow to calculate the heat flux directly
- **Junctions**
 - Updated [Tee/Wye Properties window](#) which clarifies the input and automatically defines the pipe arrangement according to the direction and orientation on the Workspace
 - Built-in K-factors are now included for bends with r/D between 0.5 and 1 (see the [Bend Loss Model](#) for more information)
 - Notes can be globally edited for junctions
- **Design Alerts**

- Design Alerts can be [saved to and loaded from a file](#) to quickly apply common Design Alerts to multiple models
- Ability to [import custom Fittings & Losses](#) from a file

Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

New in Version 7

New & Improved Microsoft Excel Integration

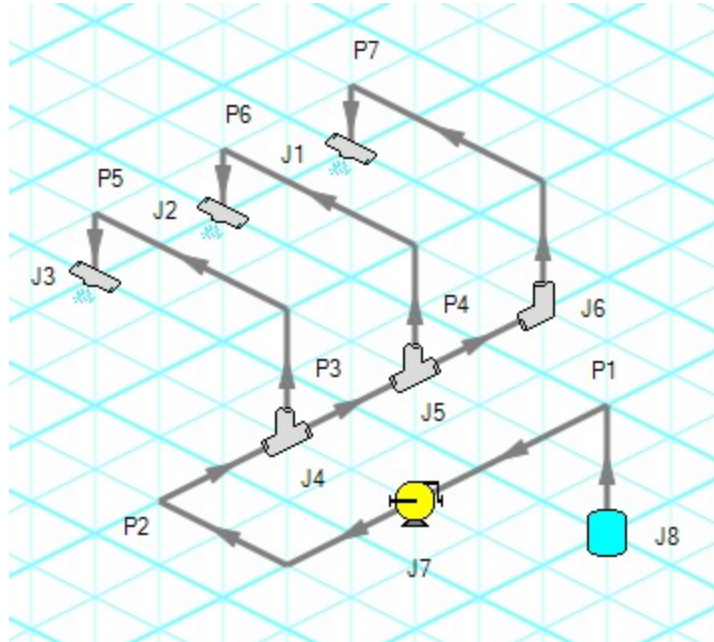
- Improved keyword based method for importing model changes from an Excel Worksheet
 - Use the included template Worksheet to quickly and easily define model changes
 - Change **multiple scenarios at once**
 - For a detailed description see [Importing From Excel](#)

	A	B	C	D	E	F	G	H
1	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	HEX heat rate set	Yes	Heat Exchanger	1	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	No	Heat Exchanger	2	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	No	Heat Exchanger	3	Heat Rate	Set equal to value	3700	Normal Heat
5	FCV setpoint set	Yes	Control Valve	15	Control Setting	Set equal to value	300	
6	FCV setpoint reduce by 20 lbm/sec	Yes	Control Valve	16	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increase by 10%	No	Control Valve	16	Control Setting	Change by percent (+/-)	10	
8	Fan Speed set	Yes	Compressor/Fan	22	Speed (%)	Set equal to value	90	

- New feature for directly exporting specified Output data to new or existing Excel Workbooks.
 - Export only the data that is desired - no need to sort through unnecessary data
 - Specify exactly where the data will end up - insert data into a pre-formatted Workbook
 - Automatically export after a model run
 - Export data from multiple scenarios with a [batch run](#)
 - For a detailed description see [Exporting To Excel](#)

New Isometric Drawing Mode

- New [Isometric Pipe Drawing Mode](#)
 - Show and Snap to an isometric grid on the Workspace
 - View models in an intuitive "3D" layout, and easily build from existing isometric drawings



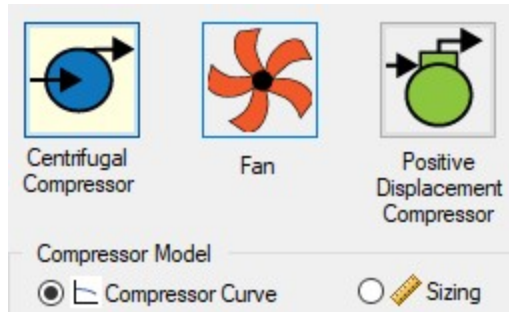
NIST REFPROP Fluid Database Now Included

- Over 150 fluids from the National Institute of Standards and Technology (NIST) Reference Fluid Thermodynamic and Transport Properties Database (REFPROP) Version 10
 - Select a [NIST REFPROP](#) fluid directly in System Properties
 - Use accurate and complete fluid information in your models without the need to locate and enter fluid properties manually

Improved Compressor/Fan Window

- New [Fan](#) Model
 - Clarifies the specification of fans
 - Includes Fan Curve and Sizing options
- New [Positive Displacement Compressor](#) Model

- Clarifies the specification of Positive Displacement Compressors
- Added Compressor Curve and Sizing Analysis Types to [Centrifugal Compressor](#) Model
 - Clarifies required data and model intention
 - For a detailed description see [Compressor/Fan Properties Window](#)



New Thermal Models

- Two new Heat Transfer Models for Pipes
 - [Convective Heat Transfer With Heat Tracing](#) - model the addition of heat between a pipe wall and an insulation layer
 - [Buried Pipe](#) - model the thermal effects of burying a pipe
- Calculate a Forced Convection Coefficient from ambient fluid velocity and temperature with the [Churchill-Bernstein correlation](#)

Improved Startup Window

- Improvements to the capability and usability of the Startup Window
 - Set modeling preferences including - [active modules](#), [working fluid](#), [units](#), [language](#), [2D or Isometric](#) drawing mode, and [default pipe material](#)
 - Displayed units can now be easily refined by selecting Common Units and Industry-Specific Units

New Integration with Common Piping Layout formats

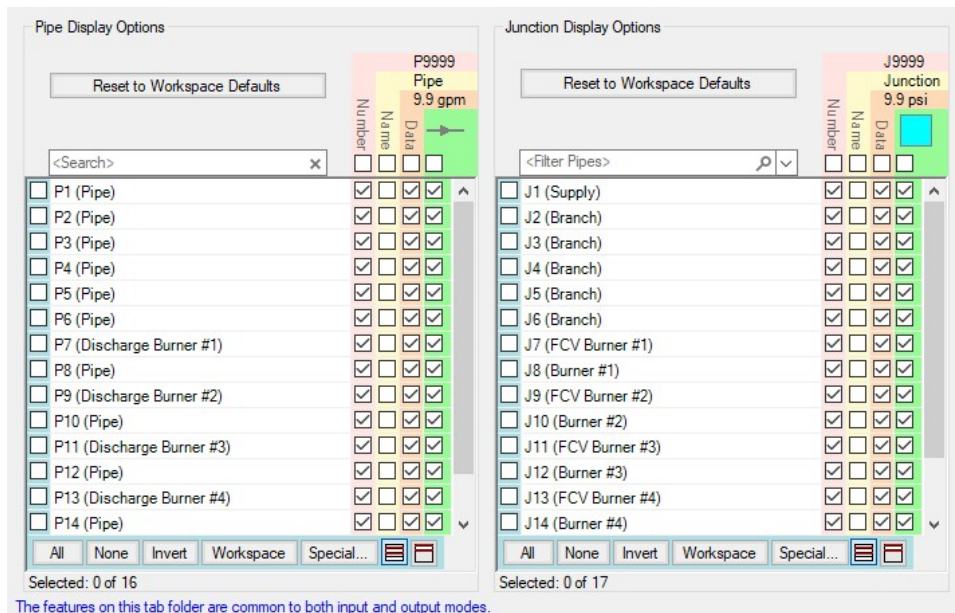
- Save modeling time by importing existing Piping Layouts from various neutral file types
 - Import [CAESAR II Neutral Files](#) (.cii)
 - Import [Piping Component Files](#) (.pcf)
 - Import and Export [EPANET Files](#) (.inp)
 - These common file formats are generated from many common engineering programs such as AutoCAD Plant 3D, SmartPlant, PDS, CadWorx, and many more.

Updated Standards

- Output displays [Applied Standards](#) - standards that were used in calculation of model results
- [Relief Valve loss](#) can be specified with Designation Size per *API 526 - Flanged Steel Pressure Relief Valves*

More Improvements

- **General**
 - Many new [default Pipe Materials](#)
 - [Control Valve](#) setting for Always Control (Never Fail) can be set to a default to off
 - Increased the loading speed for large models
- **Workspace**
 - The [Workspace background color](#) can be changed based on the level of the current scenario - additional levels have been added
 - [Annotation](#) background opacity can be changed
 - New [keyboard shortcuts](#) to zoom (CTRL + +/-) and pan (CTRL + arrow keys) the Workspace
 - [Full Scenario Path Name](#) can be copied to clipboard by right-clicking a scenario
 - [Selection box visually indicates](#) the behavior of the selection - select everything intersecting (right to left) or only objects completely inside (left to right)
 - Pipes with Fittings & Losses defined now display a [special character](#) next to their number (default of &)
- **Output**
 - [Design Alerts can be created directly from the Output](#) by right-clicking on a result - immediately select the appropriate object and parameter
- **Model Data**
 - [Quickly locate a specific object on the Workspace](#) by right-clicking the item in Model Data
 - Added additional parameters such as insulation thickness
- **Visual Report**
 - Reworked Display Option controls for Pipes and Junctions in Visual Report Control



Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 8.....	34
New in Version 6.....	42
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

New in Version 6

General Interface

- Improved tab interface and enhance Quick Access Panel functionality
- Output reports in German and Chinese languages
- Main menus are translated into available output languages

Graph Results

- Multiple Graphs can be added to Graph Folders for easy generation and display
- Multiple graphs tabs on main Graph tab

- Multiple Graphs tiled on same Graph Folder tab
- Easy formatting of titles, axis labels, and plot lines
- Plot series lines can be turned on/off for quick inspection
- Graph controls now located on the Quick Access Panel for easy graph definition and changes

Scenario Manager

- Ability to create child scenarios without results saving duplication and load time

Workspace

- GIS Shapefiles can be imported to create a model
- Text Alignment options in annotations
- Annotations can be layered on the Workspace

Output

- Results Diagrams which graphically show data for pipe heat transfer and compressors
- Intermediate Axial Points calculations are shown for all pipes in a collapsible table
- Default colors for multi-scenario output

Visual Report

- Automatic Color Map generation

Pipes

- Design Alerts are specified and named globally and applied to pipes
- New Design Alerts Available

Junctions

- Volumetric flow rate set points can be used for Control Valves
- Compressor/Fans compression ratio can be set
- When the speed is set for a Compressor/Fan, both the original and modified curves are shown on the Compressor/Fan property window

- Design Alerts can be applied to junctions
- Forward velocity to close check valves

Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 8.....	34
New in Version 7.....	38
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
Startup Window.....	57
FAQ - Frequently Asked Questions.....	44

FAQ - Frequently Asked Questions

- [How do AFT Arrow's solution methods compare to traditional handbook methods for gas flow?](#)
- [Does AFT Arrow handle sonic choking?](#)
- [How has AFT Arrow been verified?](#)
- [Can AFT Arrow be used to model liquid systems?](#)
- [How does AFT Arrow account for real gas effects?](#)
- [How does AFT Arrow account for heat transfer in pipe systems?](#)
- [Does AFT Arrow account for the heat of compression in a compressor?](#)
- [Does AFT Arrow perform an energy balance across each junction in the model?](#)
- [What is the difference between using a junction to input pressure loss data and the fittings & losses in pipes?](#)
- [What is the difference between a Tank junction and an Assigned Pressure junction?](#)
- [How does AFT Arrow account for pressure losses at tees and wyes?](#)
- [How do I size a fan or compressor when there are not any flow control valves?](#)
- [How do I size a fan or compressor when there are flow control valves?](#)
- [How do I model a variable speed fan?](#)
- [How do I account for changes to a fan curve based on different operating pressures?](#)
- [How do I find the fan speed to obtain a desired flow or discharge pressure?](#)
- [How do I close a valve, fan or pipe?](#)
- [How do I model the pressure drop across a component when there is only one data point?](#)
- [How do I model a relief valve in a system?](#)
- [How do I model a valve loss given the manufacturer Cv?](#)
- [How do I model choked flow through a valve, orifice or similar element?](#)

- [How do I model a desuperheater in AFT Arrow?](#)
- [How do I show the pipe or junction names on the Workspace?](#)
- [How do I change the display of pipe or junction names on the Workspace after the model has been built?](#)
- [How do I set my preferred engineering units as defaults?](#)
- [How do I change the input data for multiple pipes or junctions all at one time?](#)
- [How do I create a gas mixture?](#)
- [How do I model a dynamic mixing process?](#)
- [How do I merge two models together?](#)
- [How do I automatically run multiple models right after each other?](#)
- [How do I enable or disable the Highlight feature in the Pipe and Junction Properties windows?](#)
- [How do I automatically save my results after the model is run?](#)
- [How do I quickly find a particular pipe or junction on the Workspace?](#)
- [How do I change the reference positive flow direction for one or more pipes?](#)
- [How do I access pipe and junction Properties windows from the Model Data window?](#)
- [How do I display only selected pipes or junctions in the output?](#)
- [How do I quickly change the units for an output parameter?](#)
- [How do I model elevation changes in my pipe system?](#)
- [How do I model a rotating pipe system such as might exist in turbomachinery?](#)
- [How do I show data for only selected pipes or junctions in Workspace Layers?](#)
- [How do I make the Pipe Drawing Tool stay active so that I can continue to draw pipes without having to click it each time?](#)
- [How do I add graphical segments to a pipe so it is not constrained to a single straight line?](#)
- [How do I set up custom libraries on my local PC?](#)
- [How do I set up custom libraries on our local or wide area network?](#)
- [How do I change the icon for a junction in AFT Arrow?](#)
- [Can I customize the junction icons in AFT Arrow?](#)
- [What is the limit to the size of model I can create with AFT Arrow?](#)
- [Can I use special friction equations such as Panhandle and Weymouth?](#)
- [Can AFT Arrow model a turbine in the system?](#)
- [Can AFT Arrow model reacting flows?](#)
- [Can I install AFT Arrow onto a local or wide area network?](#)

How do AFT Arrow's Solution Methods Compare to Traditional Handbook Methods for Gas Flow?

AFT Arrow solves the governing equations of compressible without simplifying assumptions, and is thus more accurate than traditional handbook methods.

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.

Does AFT Arrow Handle Sonic Choking?

AFT Arrow accurately models sonic choking at all three possible geometric conditions. See [Sonic Choking](#) for more information.

How Has AFT Arrow Been Verified?

There are several aspects to AFT Arrow verification.

First, AFT Arrow has been compared to published data where it exists. Available published data is usually for a single pipe, which is not very challenging for a network solver. Models, comparisons and explanations for such published predictions are given the Verification folder within AFT Arrow.

Second, it can be verified that the AFT Arrow predictions agree with the fundamental equations for [single pipes](#) and [networks](#).

Third, AFT Arrow offers two independent solution methods, which can be compared against each other.

Fourth, incompressible gas flow predictions for networks show good agreement with incompressible calculation methods such as AFT Arrow.

Finally, AFT Arrow predictions have been compared to test data and other analytical methods on numerous occasions and good agreement has been shown. For example, see [Winters, B.A., and T.W. Walters](#), "X-34 High Pressure Nitrogen Reaction Control System Design and Analysis".

Can AFT Arrow Be Used to Model Liquid Systems?

No, AFT Arrow cannot model liquid systems. AFT Arrow's solution methodology is built on a [gas equation of state](#), which is not compatible with liquid flow calculations. AFT Arrow can, however, model incompressible gas flow such as occurs in venting systems and many HVAC systems.

How Does AFT Arrow Account for Real Gas Effects?

AFT Arrow's solution methodology is based on the ideal gas equation modified with a compressibility factor. This compressibility factor can be obtained from an equation of state model or property library. The effect of the compressibility factor is carried through the fundamental equation derivations, and thus accounted for directly.

How Does AFT Arrow Account for Heat Transfer in Pipe Systems?

Heat transfer effects show up in several places in AFT Arrow. First, in pipes users may assign a convective coefficient and pipe wall resistance and/or allow AFT Arrow to calculate convective heat transfer based on standard methods. These heat transfer calculations and the associated physical properties are determined over each solution section. Thus properties and convective coefficients can and do vary along the length of the pipe. Second, heat transfer can occur in Heat Exchanger junctions. Here the user assigns a heat load or chooses a heat transfer model for the heat exchanger. AFT Arrow performs the appropriate energy balance calculation across the heat exchanger. Third, heat transfer occurs in compressor/fans due to heat of compression.

Finally, at each branching location in the network an energy balance is performed so that energy is balanced from one pipe to the next.

Does AFT Arrow Account for the Heat of Compression in a Compressor?

Yes, AFT Arrow accounts for heat of compression based on the polytropic constant specified by the user or the compressor/fan efficiency.

Does AFT Arrow Perform an Energy Balance Across Each Junction in the Model?

Yes, every junction except for heat exchanger and compressor/fans is assumed to have a constant stagnation enthalpy (no heat or work transferred to/from the fluid). The static (thermodynamic) enthalpy can and does change based on the energy equation.

What Is the Difference Between Using a Junction to Input Pressure Loss Data and the Fittings & Losses in Pipes?

A junction has several advantages. First, solutions are given at all junctions, so the user can check the results at the junction. In contrast, [Fittings & Losses](#) are lumped into the pipe and it is not possible to give results at the loss. Second, many junctions (such as valves) have the ability to specify a CdA for sonic choking calculations. No such ability exists for Fittings & Losses; thus, sonic choking is always ignored for Fittings & Losses. Third, when using a junction the location in the pipe system of the pressure loss is specified. In other words, the upstream and downstream pipe lengths are specified. Since the location of the junction loss along a pipe can affect how much pressure is lost, the junction loss calculation is more accurate than the pipe fitting loss. A short explanation of this is that the pressure loss due to a K factor depends on the velocity squared. Typically, the velocity of the gas will change along the pipe, so the pressure drop of the K factor loss will depend on the local velocity. In the case of pipe fitting loss, it assumed to act like a friction factor and be evenly distributed along the pipe.

The pipe fitting approach has the advantage of being able to specify multiple losses quickly and easily, and not cluttering up the model Workspace with numerous junctions.

What Is the Difference Between a Tank Junction and an Assigned Pressure Junction?

There is an overlap in capability between the Tank junction and an Assigned Pressure junction and frequently they are interchangeable. Here are the differences. The Tank junction input pressure and temperature always corresponds to stagnation properties. In the Assigned Pressure junction, they can correspond to either static (with one connecting pipe) or stagnation properties (with one or more pipes). The ability to represent static conditions is the reason the Assigned Pressure can connect only to one pipe when using the static option. Static conditions have a flow velocity associated with them and multiple pipes would have multiple velocities. If the stagnation option is used in the Assigned Pressure junction, it will behave identically to a Tank junction. Finally, the Tank junction can act as the reference pressure for a closed system, while the Assigned Pressure cannot.

How Does AFT Arrow Account for Pressure Losses at Tees and Wyes?

AFT Arrow uses models developed under incompressible flow conditions to calculate [losses at tees and wyes](#). The methods, which come from [Idelchik](#), take into account losses that depend on the flow split, area change and angle of the branch pipe. These loss models may not be accurate for some systems.

How Do I Size a Fan or Compressor When There Are Not Any Flow Control Valves?

See [Size a Compressor or Fan](#).

How Do I Size a Fan or Compressor When There Are Flow Control Valves?

See [Size a Compressor or Fan with a Flow Control Valve](#).

How Do I Model a Variable Speed Fan?

Enter the fan data as a polynomial in the Compressor/Fan Properties window and then set the fan speed directly. No entry is assumed to be 100% speed. If you enter something other than 100%, AFT Arrow uses the [affinity laws](#) to adjust the curve.

How Do I Account for Changes to a Fan Curve Based on Different Operating Pressures?

If performance data is entered as head rise, no density correction is necessary. If entered in terms of pressure, enter the fan data as a polynomial in the Compressor/Fan Properties window, select the option to modify the curve for [density difference](#), and then set the basis density for the curve (usually atmospheric density for the gas). AFT Arrow will modify the fan curve based on the operating density.

How Do I Find the Fan Speed to Obtain a Desired Flow or Discharge Pressure?

Enter the fan data as a polynomial in the Compressor/Fan Properties window and set the desired flow or discharge pressure on the "Variable Speed" tab. AFT Arrow will calculate the fan speed required to produce the flow or pressure conditions specified and display the speed in the Compressor/Fan Summary report.

How Do I Close a Valve, Fan or Pipe?

Select the pipe or junction you want to close and choose [Special Condition](#) from the Edit menu. By default, AFT Arrow will display a red "X" next to the pipe or junction label on the Workspace. It will also redraw your Workspace and show the closed sections of the model with dashed lines for the pipes and dashed outlines for each junction.

Some special condition settings do not close the junction but have different purposes. For example, the normal condition for a relief valve is to be closed, so its special condition causes it to be open.

How Do I Model the Pressure Drop Across a Component When There Is Only One Data Point?

Use the [Resistance Curve Loss Model](#).

How Do I Model a Relief Valve in a System?

You can model a relief valve using either a Relief Valve junction or a regular Valve junction. The Relief Valve junction is always closed when you run the model (unless you set its Special Condition), and AFT Arrow will run the Solver to determine if sufficient pressure exists to crack it. If so, it will run the Solver again with the relief valve open.

If you know the condition you are modeling will crack the relief valve, it is more efficient to just use a regular valve junction. In this case, AFT Arrow assumes the valve is open from the start, and does not have to go through the extra step of solving the network to check for sufficient system pressure to crack it.

How Do I Model a Valve Loss Given the Manufacturer Cv?

AFT Arrow supports the [ANSI/ISA-75.01.01-2012](#) definition of valve Cv for gas flow. This can be modeled in valve junctions, relief valves, check valves, or control valves.

How Do I Model Choked Flow Through a Valve, Orifice or Similar Element?

You must specify the CdA for Choking on the junction loss model tab. AFT Arrow uses this as the base area for sonic calculations. If modeling the subsonic loss as Cv, you can account for choking by specifying Xt.

How Do I Model a Desuperheater in AFT Arrow?

You can use a Branch junction to model a desuperheater. Enter an inflow (source flow) at the branch and specify the enthalpy of the source. AFT Arrow will perform an energy balance using this enthalpy, and assume the discharge is still superheated, but at the new lower energy level.

How Do I Show the Pipe or Junction Names on the Workspace?

Open the pipe or junction Properties window, click the Optional tab, and check or clear the check boxes for showing the number or name. This will affect the current pipe or junction. You can set the default behavior in the Workspace Preferences area available on the [User Options](#) window.

How Do I Change the Display of Pipe or Junction Names on the Workspace After the Model Has Been Built?

You can use the Global Edit windows to change the current model settings for all pipes or junctions or only selected ones. See the [Global Pipe Edit](#) or [Global Junction Edit](#) window for more information.

How Do I Set My Preferred Engineering Units as Defaults?

From the Tools menu, select User Options. Then select Preferred Units under [Unit System](#). Drop down the list of units for each Unit Family to select the Preferred Unit. At this point, the preferred unit applies only to the current model. To make this the default for all models, click the Save as New User Defaults button at the bottom.

How Do I Change the Input Data for Multiple Pipes or Junctions All at One Time?

The [Global Pipe Edit](#) and [Global Junction Edit](#) windows offer tremendous power and flexibility in changing all or parts of your model input all at once.

How Do I Create a Gas Mixture?

You can [create gas mixtures](#) with the NIST REFPROP fluid library or the optional Chempak fluid library add-on. The AFT Standard gas library does not support mixing. To create a mixture, open the Fluid Properties panel in the Analysis Setup window, choose the NIST REFPROP or Chempak library, and then click the “Create New Mixture and Add...” button. Here you can specify the mixture components and percentages.

How Do I Model a Dynamic Mixing Process?

You can model dynamic gas mixtures with the included NIST REFPROP library or the optional Chempak add-on. The AFT Standard gas library does not support mixing. To model dynamic mixing, open the Fluid Properties panel in the Analysis Setup window, choose the NIST

REFPROP or Chempak library, and then choose the gases in the list and add them to the model. You can add any number of gases, and also create gas mixtures and add them.

Once the desired gases are added to the model, they can be assigned to different source junctions such as Tanks, Assigned Pressures and Assigned Flows. AFT Arrow applies the gas or gas mixture to the junction's connecting pipes. It then solves the network, carrying the gas composition to each section of the model in accordance with species mass conservation. As the solution progresses, the mass flows will adjust as they approach convergence and AFT Arrow updates the concentration balance for each iteration based on the current global mass balance.

How Do I Merge Two Models Together?

Use the Merge feature on the [File menu](#) to merge models together.

How Do I Automatically Run Multiple Models Right After Each Other?

Multiple models can be run sequentially using the [Batch Run](#) feature.

How Do I Enable or Disable the Highlight Feature in the Pipe and Junction Properties Windows?

There are three ways to enable or disable the Highlight feature. The first is toggling the option on the view menu. The second is pressing the F2 function key while in a Pipe and Junction Properties window. The third is double-clicking the anywhere in the Pipe and Junction Properties window.

How Do I Automatically Save My Results After The Model Is Run?

Open the [Output Control](#) window from the Analysis menu, change to the Format & Action folder, and choose to select the Transfer Results to Initial When Done, Transfer Valve States When Done, and Save Model When Results are Transferred options.

How Do I Quickly Find a Particular Pipe or Junction on the Workspace?

Use the [Find](#) feature to quickly find a pipe or junction.

How Do I Change The Reference Positive Flow Direction for One or More Pipes?

Select the pipe or pipes and choose the Reverse Direction feature on the Arrange menu.

How Do I Access Pipe and Junction Properties Windows from the Model Data Window?

In the Model Data tabular displays, double-click the far-left column where the pipe or junction number is located and the appropriate Properties window will be opened.

How Do I Display Only Selected Pipes or Junctions in the Output?

Open the Output Control window from the Analysis menu, change to the Show Selected Pipes/Junctions folder, and select the pipes and or junctions you want to display.

How Do I Quickly Change the Units for an Output Parameter?

Whereas the Output Control window allows you specify units for all parameters, when in the Output window viewing results you can quickly change the units for parameters in the tabular displays by double-clicking the column header.

How Do I Model Elevation Changes in my pipe system?

Set the elevations in the elevation fields for each junction. Pipes are assumed to be linear between junctions. If you need to model a system other than a stationary earth-based system, the gravitational acceleration can be changed in the Environmental Properties panel.

How Do I Model a Rotating Pipe System Such as Might Exist in Turbomachinery?

Specify the rotational speed in the Environmental Properties panel. The zero datum is the rotational centerline. Elevation input in Junction windows take on the alternate meaning of distance to rotational centerline. See [Rotating Systems](#) for more information.

How Do I Show Data for Only Selected Pipes or Junctions in Workspace Layers?

This is controlled via the [Label Control](#) group of the Layer Settings window.

How Do I Make the Pipe Drawing Tool Stay Active so That I Can Continue to Draw Pipes Without Having to Click It Each Time?

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.

If you double-click the Pipe Drawing tool it remains active until you click it again a single time. This allows you to draw a series of pipes without returning to the Toolbox each time.

How Do I Add Graphical Segments to a Pipe so It Is Not Constrained to a Single Straight Line?

Use the [Segment Pipe](#) tools found on the Arrange Menu.

How Do I Set Up Custom Libraries on My Local PC?

See [Library Browser](#).

How Do I Set Up Custom Libraries on Our Local or Wide Area Network?

See [Network Library Overview](#).

How Do I Change the Icon for a Junction in AFT Arrow?

Open the Junction Properties window, click the Optional tab, and then click the Change Icon button.

Can I Customize the Junction Icons in AFT Arrow?

No, AFT Arrow icons are in a resource file that cannot be edited by the user. This capability is planned for a future release.

What Is the Limit to the Size of Model I Can Create with AFT Arrow?

There are no theoretical limits to model size, but there are a few practical limits. First, AFT Arrow accepts pipe and junction ID numbers up to 99,999. This limits the model size to 99,999 pipes and 99,999 junctions. Before you reach that limit, however, you will likely encounter a limitation

of your available RAM to hold all of the solver parameters. To determine how much RAM you need, add up the number of branches and tees in the model. Take the square of this number. Then multiply it by 32 to get the amount of RAM that must be available. For example, with 1,000 branches and tees, the square is 1 million, and after multiplying by 32 you need 32 million bytes of RAM (i.e., 32 MB).

Can I Use Special Friction Equations such as Panhandle and Weymouth?

Yes, either Panhandle or Weymouth can be specified by selecting them in the Pipe Properties window.

Can AFT Arrow Model a Turbine in the System?

AFT Arrow can approximate a turbine using a heat exchanger junction. By assigning pressure drop and heat rate data, the discharge conditions of a turbine can be modeled.

Can AFT Arrow Model Reacting Flows?

No, AFT Arrow models only non-reacting flows.

Can I Install AFT Arrow onto a Local or Wide Area Network?

Yes, AFT Arrow can run off the network or local PC. When installed on a network, the number of concurrent users is limited to the number of licenses purchased.

Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 8.....	34
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
Startup Window.....	57

Arrow 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 Series

The Quick Start video tutorial series provides a short introduction to the basic features of AFT Arrow and shows how to build, run, and view output for the model.

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

Beginner: Air Heating System

This is a simple example model which is recommended as a starting point for brand new AFT product users.

- Beginner: Air Heating System

Walk Through Examples

This links to a variety of example problems for beginning to advanced users which showcase different features and applications of AFT Arrow.

- Walk Through Examples

Module Walk Through Examples

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

- Module Walk Through Examples

Startup Window

The Startup Window is displayed when AFT Arrow 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 Arrow 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](#).
 - **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 Arrow release, or sign up for notifications of new releases via email.
 - **Learning Center** - Provides links to various [Learning Resources](#) that are designed to help get a new user up and running with AFT Arrow as quickly as possible. The resources include the Quick Start Guide PDF, Examples, and video tutorials.

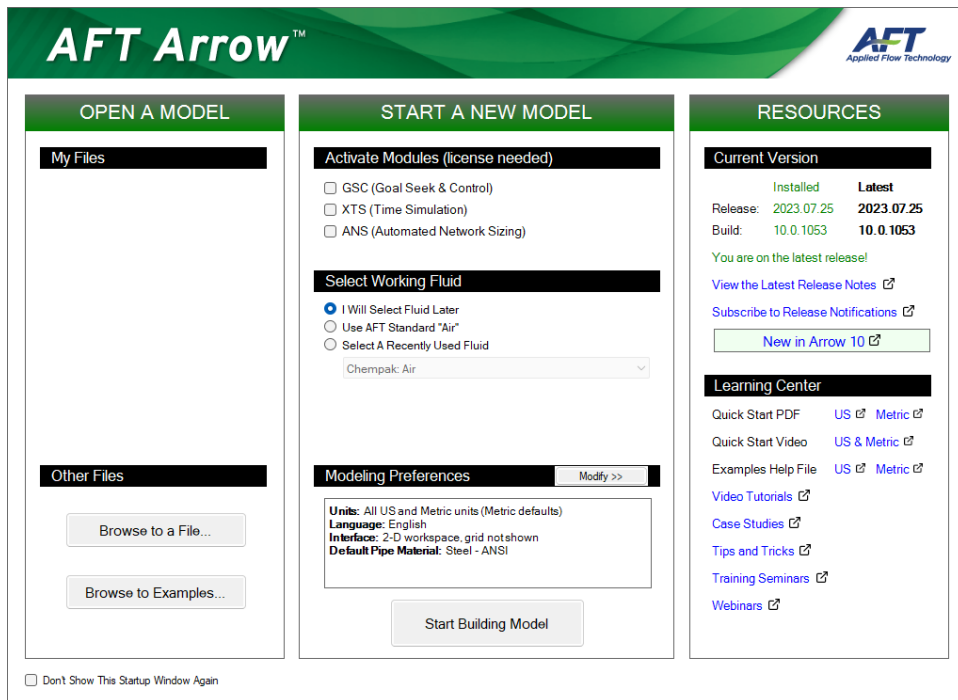


Figure 1: Startup Window in AFT Arrow

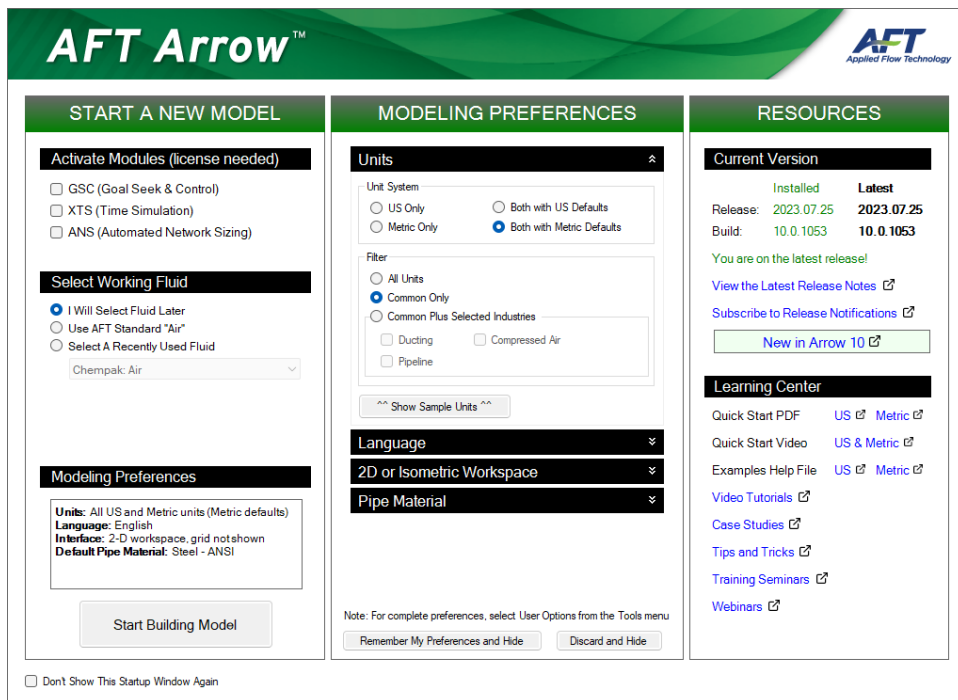


Figure 2: Startup Window showing Modeling Preferences

Related Topics

Overview of AFT Arrow.....	17
New in Version 9.....	29
New in Version 8.....	34
New in Version 7.....	38
New in Version 6.....	42
Arrow Learning Resources.....	56
Engineering Assumptions.....	19
FAQ - Frequently Asked Questions.....	44

Fundamental Modeling Concepts

This section focuses on the critical concepts and theories that comprise the model building process in AFT Arrow. 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	61
Reference Positive Flow Direction	62
Features for Modeling Irrecoverable Losses	63
Philosophy of Computer Modeling	64
Role of Pressure Junctions and How They Work	65
Open vs. Closed Systems	68
Curve Fitting	71
Stagnation vs. Static Pressure Boundaries	73

Pipes and Junctions

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

[Pipes](#) are conduits for steady-state, compressible, one-dimensional gas flow. The flow rate through the entire length of the pipe is always constant. [Pressure drop due to friction](#) occurs in pipes. [Heat transfer](#) may or may not occur depending on the type of model you select.

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

In addition to balancing flow and energy, junctions also influence the flow or pressure behavior of the system. For example, a [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 [irrecoverable pressure losses](#) for gas behavior. In some cases you can specify special kinds of thermal 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 Arrow 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, it is important to specify the proper flow direction for pipes which connect to junctions such as [compressor/fans](#) or [valves](#). AFT Arrow uses the flow direction of the pipes to determine in which direction to increase or reduce pressure.

Related Topics

Fundamental Modeling Concepts.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

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 Arrow 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 Arrow 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 Arrow 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).

Related Topics

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

Features for Modeling Irrecoverable Losses

AFT Arrow 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 C_d (discharge coefficients) for orifices.

No provision has been made for the equivalent length approach to solving pressure drop problems.

Related Topics

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

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 Arrow 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 Arrow, you should follow these guidelines:

- Be aware of what AFT Arrow can and cannot do and the [assumptions](#) it makes.
- Learn and use the tools in AFT Arrow 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 Arrow 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 Arrow 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 analyses and reduce modeling errors.
- Check your input data. Use the features available in AFT Arrow to review your results for input errors. Pay attention to the information in the output. 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

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

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 Arrow 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 Arrow and if you try and run this model, AFT Arrow will inform you that it cannot run it because of the lack of a reference pressure (Figure 2). In AFT Arrow there are several junctions that can act as a reference pressure: the tank, assigned pressure, relief valve junctions which are open to external pressure, pressure control valves, orifice (with the exit orifice option) and valves (with the exit valve option).

When you try and run a model that has one or more sections completely bounded by known flows (or Flow Control Valves or [Compressor/Fans](#) operating in fixed 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](#) for more information.

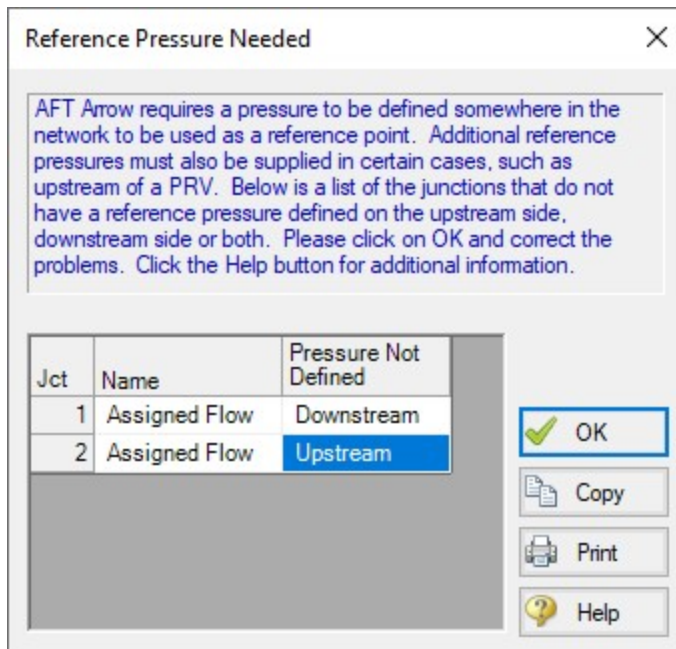


Figure 2: AFT Arrow message when you try to run a model without sufficient reference pressures

How Pressure Junctions Work

Pressure junctions (e.g., [Tanks](#) or [Assigned Pressures](#)) in AFT Arrow 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 Arrow. 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 3. This system compresses air from a low pressure supply tank to a high pressure discharge tank.

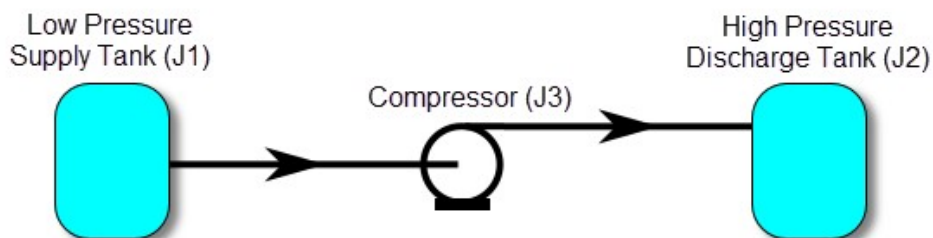


Figure 3: Physical compressor system to transfer air from low pressure tank (J1) to high pressure discharge tank (J2)

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 so builds an AFT Arrow model as shown

in Figure 4. The Figure 4 model is attempting to represent the physical system in Figure 3, but it in fact represents the physical system in Figure 5.

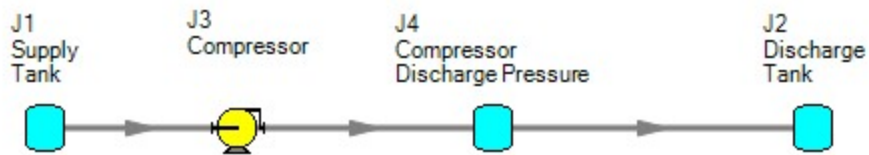


Figure 4: Model to size compressor attempting to represent system in Figure 3, but actually represents the system in Figure 5

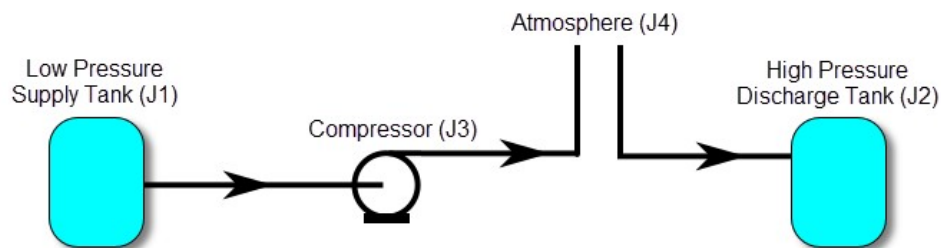


Figure 5: Physical system represented by model in Figure 4

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 3 be modeled? It should be modeled as shown in Figure 6.

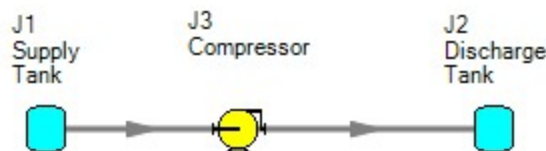


Figure 6: Proper model of physical system shown in Figure 3

If the goal is to size the compressor at J3, the model in Figure 6 can be used with the compressor modeled as a fixed flow compressor.

Related Topics

Role of Pressure Junctions - Detailed Discussion (Long).....	674
Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Open vs. Closed Systems.....	68
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

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 Arrow 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.

When using a Tank junction to model a closed system, you should select the Balance Energy in Tank option, which will allow the tank temperature to vary and be calculated in a manner consistent with the overall energy balance.

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 Arrow views a closed system model. AFT Arrow 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 AFT Arrow's solution engine 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.

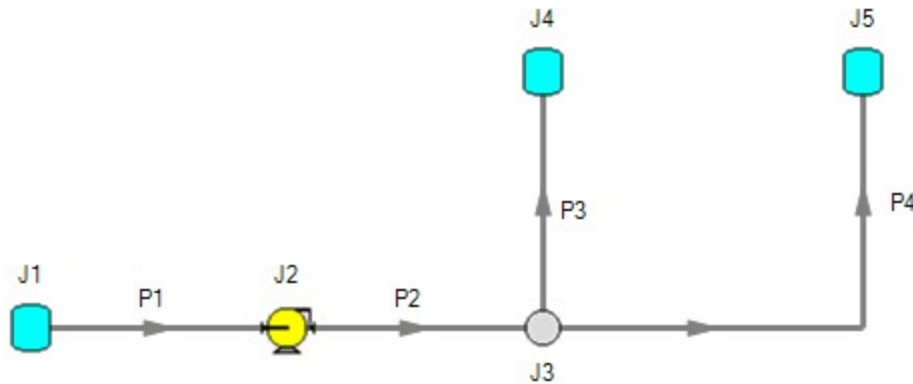


Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5

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 Arrow it will appear as an identical system to the first system. The reason is that AFT Arrow 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 Arrow.

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 Arrow is not applying a mass balance to J1 directly. It is merely the result of an overall system mass balance.

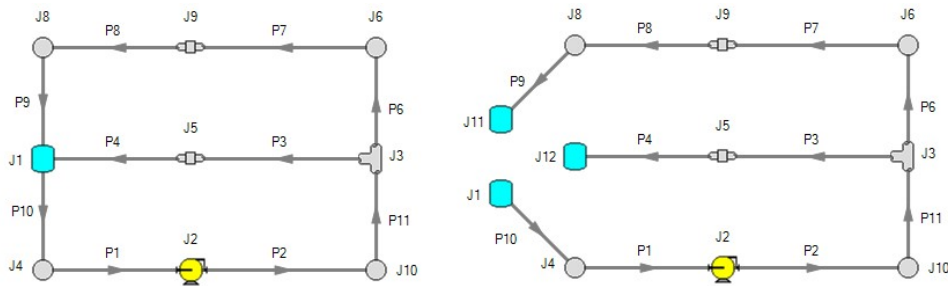


Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two systems will appear identical to AFT Arrow.

Balancing Energy

Here the thermal aspects of open and closed systems are discussed. If you have not read and grasped the discussion on mass balance in closed systems, please read that section before this one.

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 a temperature of 100° F at J1. This temperature will be the inlet pipe stagnation temperature for all pipes that flow out of J1. 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 is the function of the “Balance Energy At Tank” feature in the [Tank junction](#). The junction temperature (input as maybe 100° F) is allowed to “float” and find its own equilibrium. Each iteration the floating tank temperature is passed into pipe P10 until convergence. When passed into P10, this will affect the return temperatures of P4 and P9. 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 in the [Network Implementation](#) topic.

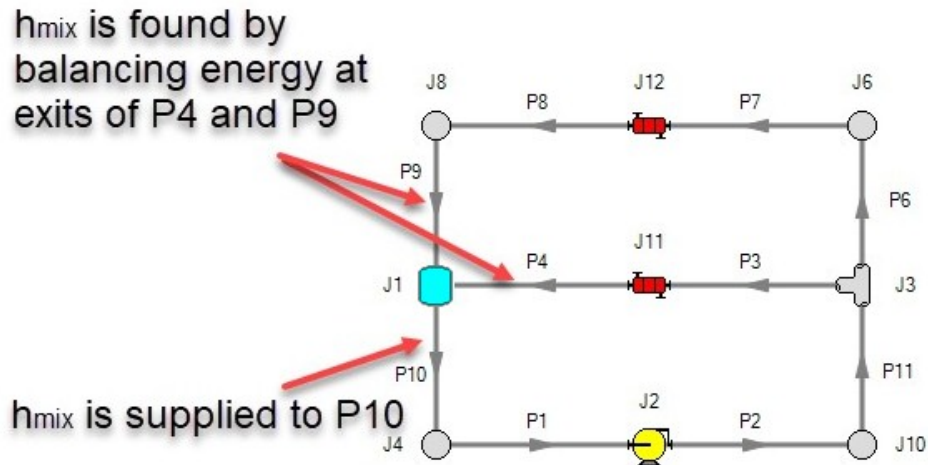


Figure 3: Example of balancing energy in a closed system

Related Topics

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Curve Fitting.....	71
Stagnation vs. Static Pressure Boundaries.....	73

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 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:

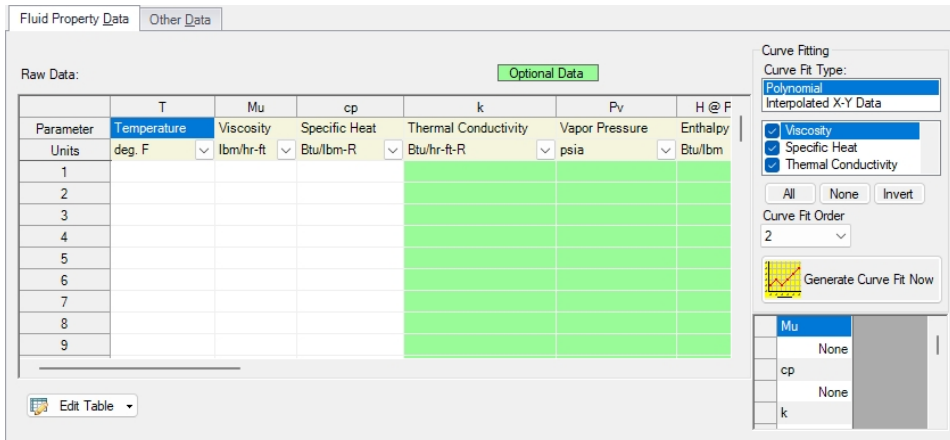


Figure 1: Custom Fluid Definition

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:

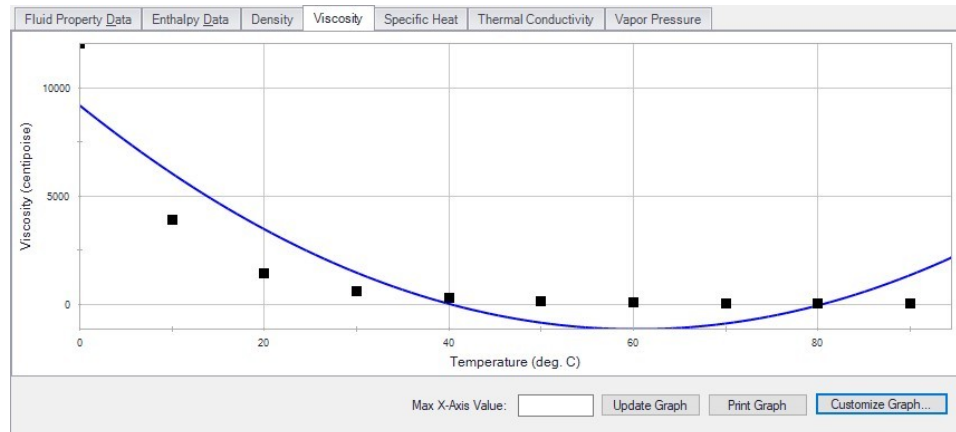


Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data

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

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Stagnation vs. Static Pressure Boundaries.....	73

Stagnation vs. Static Pressure Boundaries

With two exceptions (to be discussed), all pressure-type boundary conditions in AFT Arrow 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 Arrow by use of a [Tank junction](#).

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

Fundamental Modeling Concepts

It is tempting to say that because there is no velocity in the reservoir, then it does not matter whether the reservoir 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 Arrow model is applied at the exit of the junction, not within the junction itself. Inside the reservoir, 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.

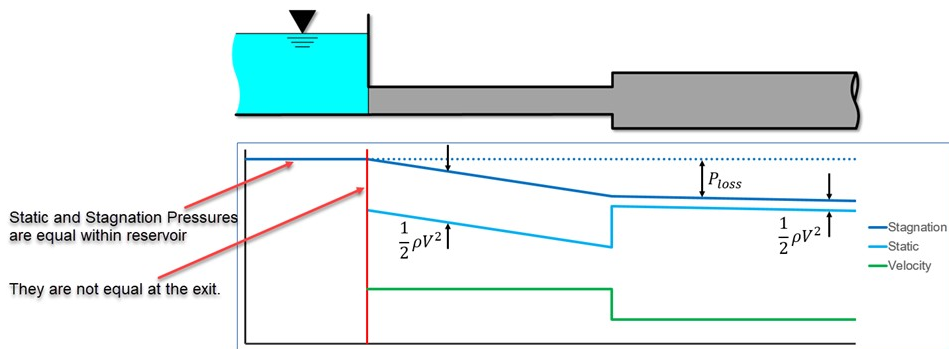


Figure 1: Reservoir with one connecting pipe (incompressible flow)

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 Arrow, 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 Arrow 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

Fundamental Modeling Concepts.....	61
Pipes and Junctions.....	61
Reference Positive Flow Direction.....	62
Features for Modeling Irrecoverable Losses.....	63
Philosophy of Computer Modeling.....	64
Role of Pressure Junctions and How They Work.....	65
Open vs. Closed Systems.....	68
Curve Fitting.....	71

Analysis Setup Overview

Analysis Setup contains all of the settings which the user must define to run the model, including fluid properties, pipe and junction status, environmental properties, and other related options. Launch Analysis Setup 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](#)
- [Goal Seek and Control Group](#) - ([GSC Module](#) only)
- [Steady Solution Control Group](#)
- [Environmental Properties Group](#)
- [Cost Settings Group](#)
- [Automatic Sizing Group](#) - ([ANS Module](#) only)
- [Miscellaneous Group](#)

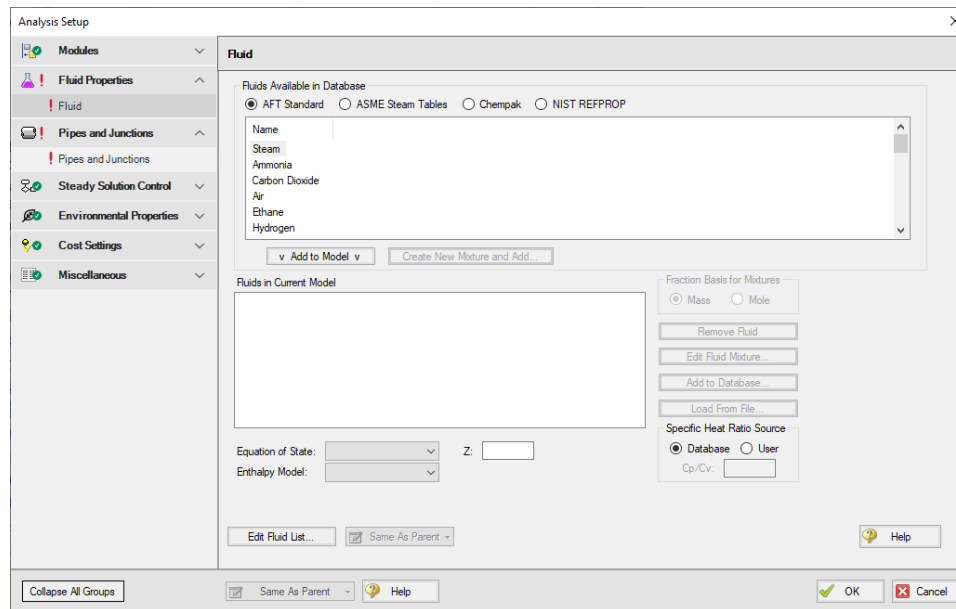


Figure 1: Default state of Analysis Setup in a new blank model

Modules Group

AFT Arrow has several optional add-on modules which extend its modeling capabilities into other areas. The modules allow multivariable goal seeking and control simulation, time varying simulation, and full system sizing. The [Modules panel](#) within the Modules Group in [Analysis Setup](#) allows you to activate and enable any of the add-on modules. An additional license to the base AFT Arrow license is required to use a module. These modules include:

- [GSC \(Goal Seek & Control\)](#) - Identifies input parameters that yield desired output values and simulates control functions.
- [XTS \(Extended Time Simulation\)](#) - Models dynamic system behavior and how critical system parameters vary over time.
- [ANS \(Automated Network Sizing\)](#) - Evaluates the complex interaction of system variables to reveal combinations of cost savings.

Related Topics

Modules Panel.....	78
Add-on Modules.....	489
Analysis Setup Overview.....	76
Fluid Properties Group.....	81
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111

Environmental Properties Group.....	147
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Miscellaneous Group.....	153

Modules Panel

The Modules panel allows users to activate and enable the optional add-on modules.

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 GSC, XTS, or ANS add-on modules.

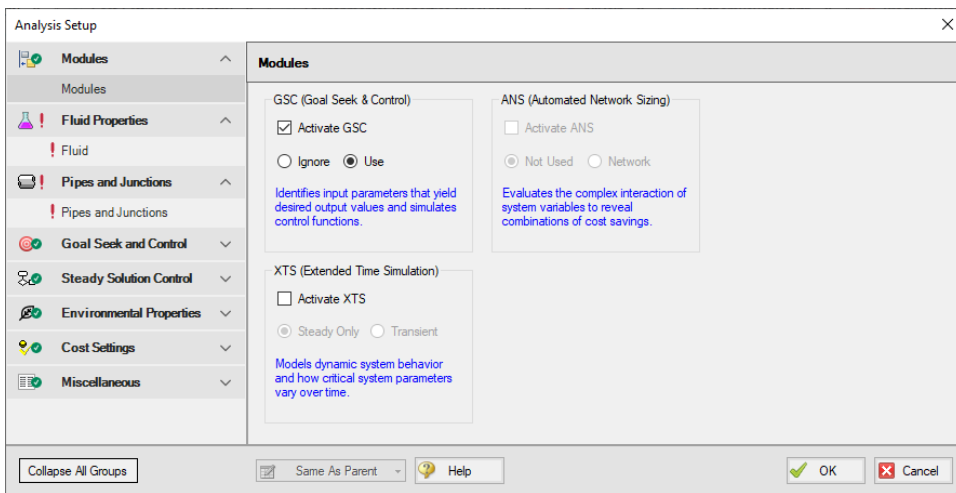


Figure 1: The default state of the Modules panel

GSC (Goal Seek & Control)

Activate the Goal Seek & Control module by checking the box next to Activate GSC. Once activated, enable it by selecting Use or disable it by selecting Ignore. When the GSC module is activated, the [Goal Seek and Control Group](#) appears in the Analysis Setup window. The Goal Seek and Control group has three items: [Variables](#), [Goals](#), and [Numerical Controls](#).

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

Setting the GSC module to Ignore causes the GSC module to function like the base AFT Arrow application. One difference is that users are still able to enter goal seeking data with this data

retained in the model. If the model is opened in the base AFT Arrow application, this data will be lost. Table 1 relates the differences between using AFT Arrow without the GSC module activated, using it with the GSC module activated but disabled (Ignore mode), and using it with the GSC module activated and enabled.

The GSC module can also be toggled between Use and Ignore from the Analysis menu in the Goal Seek & Control option as shown in Figure 2. Use this feature to quickly toggle the GSC module between Use and Ignore without opening the Modules panel in Analysis Setup.

Table 1: AFT Arrow feature accessibility based on the GSC module activation and enablement

Features	GSC Not Activated	GSC Activated	
		GSC Disabled	GSC Enabled
Goal Seek and Control Group in Analysis Setup	Not Visible	Visible	Visible
Goal Seek and Control in Analysis menu	Not Visible	Visible	Visible
Goal Seek and Control tab in General Section of Model Data window	Not Visible	Visible	Visible
GSC Variables and GSC Goals tabs in General Section of Output window	Not Visible	Not Visible	Visible

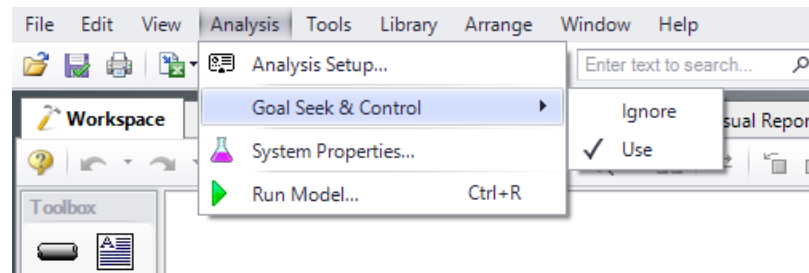


Figure 2: Select Use from the Goal Seek & Control option in the Analysis menu to perform goal seeking when running the model

XTS (Extended Time Simulation)

Activate the [Extended Time Simulation module](#) by checking the box next to Activate XTS. Once activated, enable it by selecting Transient or disable it by selecting Steady Only. When the XTS module is activated, the [Transient Control group](#) appears in the Analysis Setup. The Transient Control group has one item: [Simulation Duration](#).

Activating the XTS module can be done before or after a model is built. If activated after, then additional input will be required for transient junctions. Pre-existing models built with the base AFT Arrow application can have the XTS module activated.

Transient modeling can be turned off at any time by selecting Steady Only. The Steady Only mode causes the XTS module to function like standard steady-state AFT Arrow. One difference is that users can still enter transient data and this data is retained in the model. If the model is opened in standard AFT Arrow, this data will be lost. Additionally, the Transient Control group and Simulation Duration panel are still accessible. Table 4 relates the differences between not using XTS and using it in Steady Only mode.

The XTS module can be toggled between Steady Only and Transient from the Analysis menu in the Time Simulation option as shown in Figure 4. Use this feature to quickly toggle the XTS module between Steady Only and Transient without opening the Modules panel in Analysis Setup.

Table 4: AFT Arrow feature accessibility based on the XTS module activation and enablement

Features	XTS Not Activated	XTS Activated	
		XTS Disabled	XTS Enabled
Transient Control Group in Analysis Setup	Not Visible	Visible	Visible
Time Simulation in Analysis Menu	Not Visible	Visible	Visible
Transient tab in junction properties windows	Not Visible	Visible	Visible
Transient Pipe and Jct tabs in Output	Not Visible	Not Visible	Visible
Transient tab in junction section of Model Data	Not Visible	Visible	Visible
Time control slider and buttons in Output	Not Visible	Not Visible	Visible
Time control slider and buttons in Workspace	Not Visible	Not Visible	Visible
Transient tab on Select Graph Data	Not Visible	Not Visible	Visible
Animation controls in Graph Results	Not Visible	Not Visible	Visible

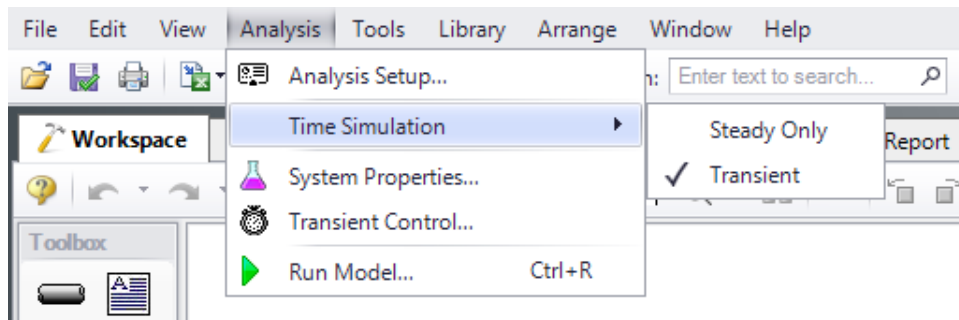


Figure 4: Transient mode is enabled from the Analysis menu

ANS (Automated Network Sizing)

Activate the Automated Network Sizing module by checking the box next to Activate ANS. Once activated, enable it by selecting Network or disable it by selecting Not Used. When the ANS module is activated, the [Automatic Sizing group](#) appears in the Analysis Setup. The Automatic Sizing group has one item: [Sizing Summary](#). Additionally, the [Sizing Window](#) becomes available when ANS is activated.

Activating the ANS module can be done before or after a model is built. If activated after, then additional sizing data will be required to run the model. Pre-existing models built with the base AFT Arrow application can have the ANS module activated.

ANS can be disabled at any time by selecting Not Used from the same menu. The Not Used mode causes the ANS module to function like the base AFT Arrow application. However, all ANS data and settings in the model will still be retained. If the model is opened in the base AFT Arrow application, this data will be lost. The purpose of Not Used mode is that it allows ANS to be quickly turned off and on without having to change data in the [Sizing Objective](#).

The ANS module can be toggled between Network and Not Used from the Analysis menu in the Automated Sizing option as shown in Figure 5. Use this feature to quickly toggle the ANS module between Network and Not Used without opening the Modules panel in Analysis Setup.

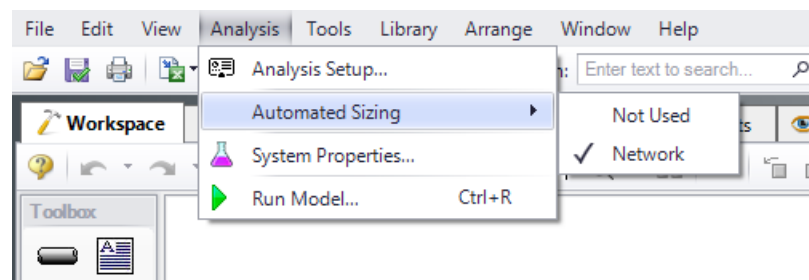


Figure 5: Enabling Automated Sizing from the Analysis menu

Related Topics

Modules Group.....	77
Add-on Modules.....	489

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 only one item in the Fluid Properties Group, the [Fluid panel](#).

Related Topics

Fluid Panel.....	82
Creating Gas Mixtures.....	86
Equation of State Models.....	88
Enthalpy Models.....	93
Fluid Accuracy Settings.....	95
Analysis Setup Overview.....	76
Modules Group.....	77
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111
Environmental Properties Group.....	147
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Miscellaneous Group.....	153

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 Fluid 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 is included for Arrow releases after May 11, 2022.
- **[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, cus-

tom 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 Arrow are treated as compressible and superheated as stated in the [Engineering Assumptions](#).

AFT Standard 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.

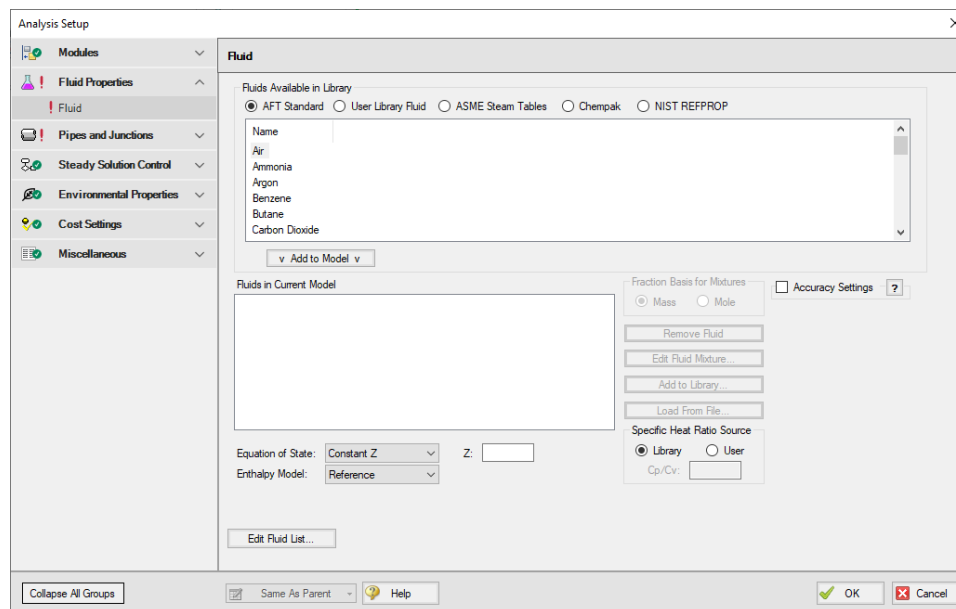


Figure 1: Fluid panel in Analysis Setup with the AFT Standard fluid library selected

ASME Steam Tables Library

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](#).

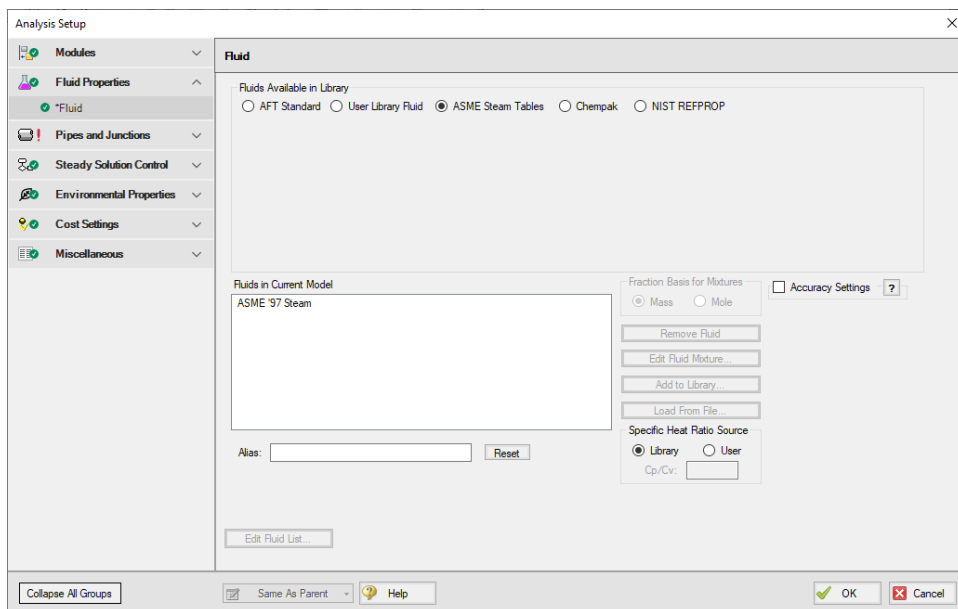


Figure 2: Fluid panel in Analysis Setup with the ASME Steam fluid library selected

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.

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.

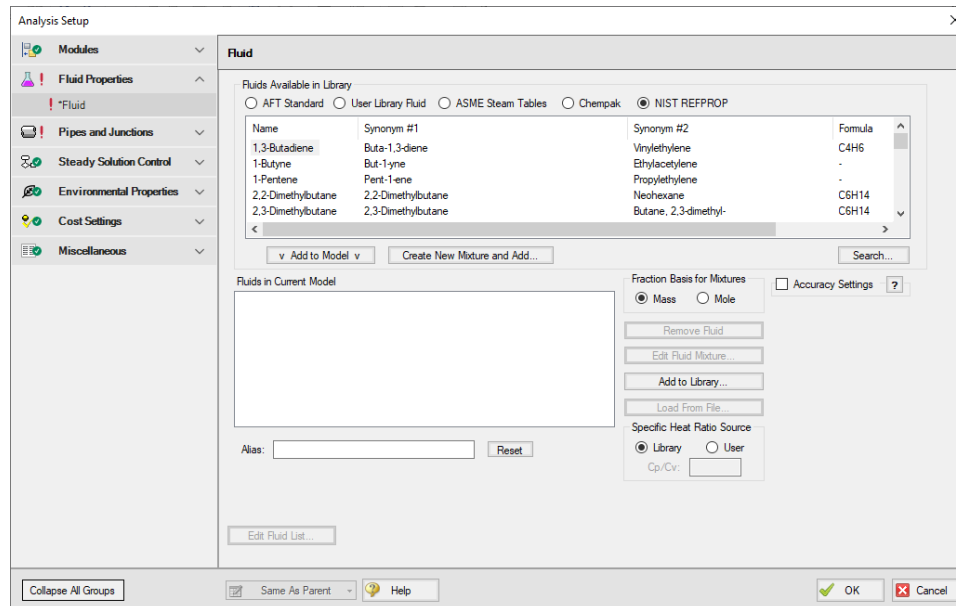


Figure 3: Fluid panel in Analysis Setup with the NIST REFPROP fluid library selected

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](#).

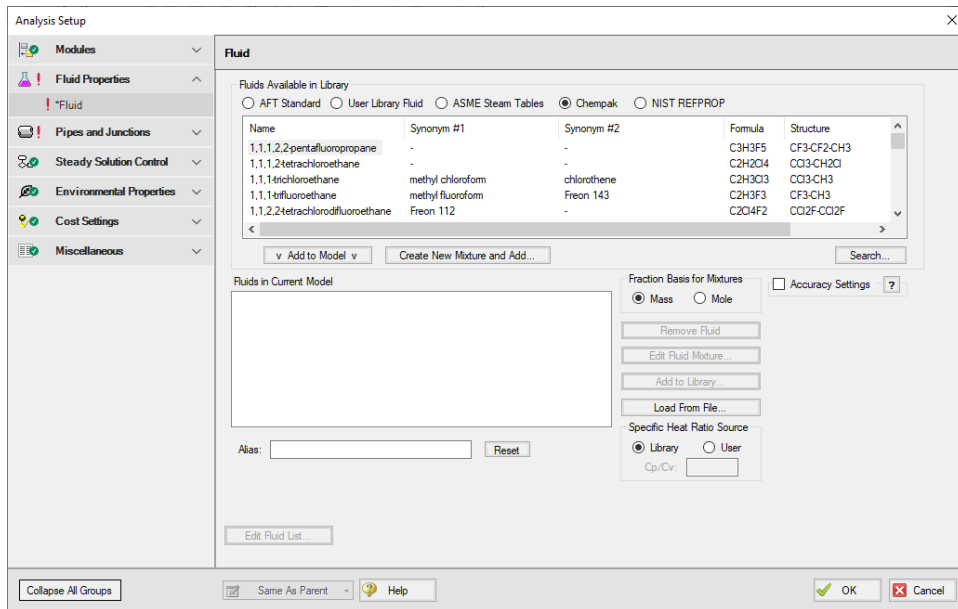


Figure 3: Fluid panel in Analysis Setup with the Chempak fluid library selected

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

Fluid Properties Group.....	81
Creating Gas Mixtures.....	86
Equation of State Models.....	88
Enthalpy Models.....	93
Fluid Accuracy Settings.....	95
Chempak temperature and/or pressure is out of range for the following.....	713
Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range.....	722
Fluid is no longer in the vapor phase because the fluid temperature/pressure is....	716

Creating Gas 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.

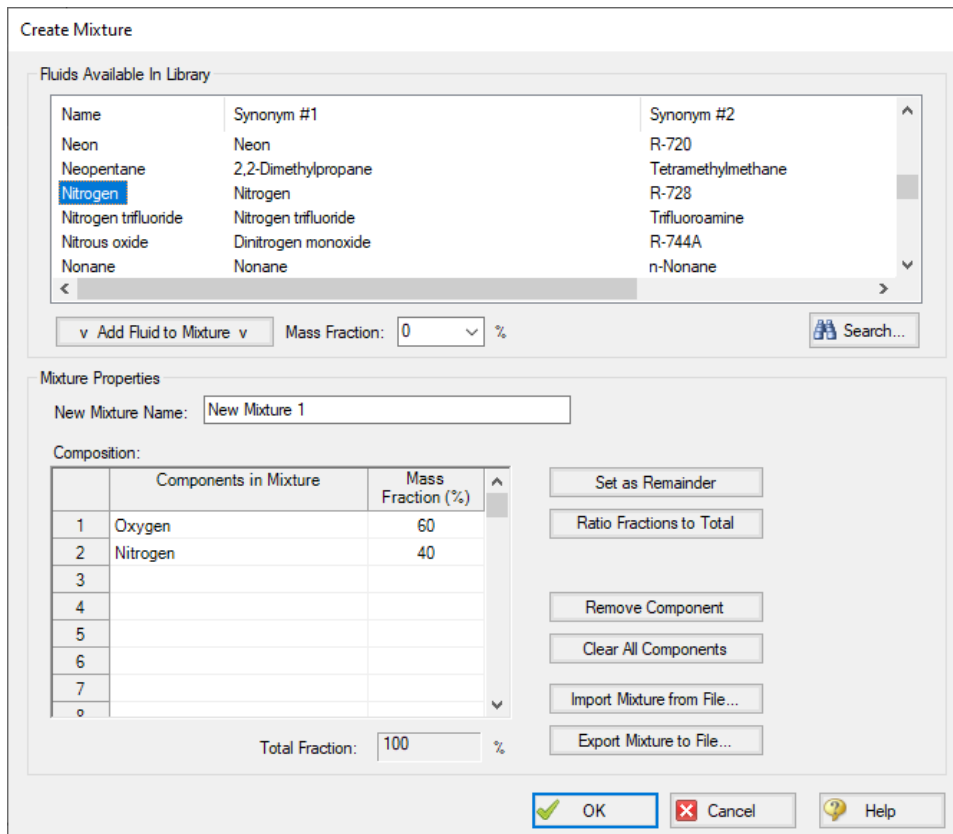


Figure 1: The Create Mixture window (opened from the Fluid panel) allows you to create predefined mixtures for the model

Related Topics

Fluid Properties Group.....	81
Fluid Panel.....	82
Equation of State Models.....	88
Enthalpy Models.....	93
Fluid Accuracy Settings.....	95

Equation of State Models

AFT Arrow 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. Arrow 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 Arrow 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

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

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

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

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

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

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

Three-parameter

Arrow 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 Arrow 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 P_r and T_r are the [reduced pressure and temperature](#), respectively.

The constants B_0 and B_1 are given by the following:

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

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

and ω is the [acentric factor](#).

Related Topics

Fluid Properties Group.....	81
Fluid Panel.....	82
Creating Gas Mixtures.....	86
Enthalpy Models.....	93
Fluid Accuracy Settings.....	95

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 Arrow offers three ways to calculate the enthalpy. 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:

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

B_0 and B_1 are the constants defined for the [Three-parameter equation of state](#), 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 Arrow'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

Fluid Properties Group.....	81
Fluid Panel.....	82
Creating Gas Mixtures.....	86
Equation of State Models.....	88
Fluid Accuracy Settings.....	95

Fluid Accuracy Settings

Several advanced settings are available to fine-tune the accuracy of the fluid properties for the Arrow 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.

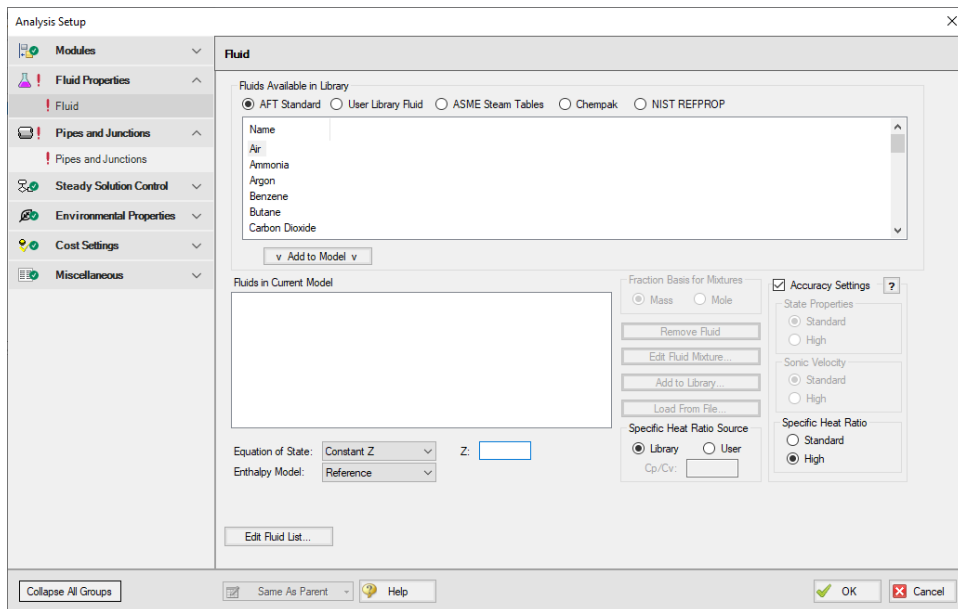


Figure 1: Fluid panel in Analysis Setup with the Fluid Accuracy Settings enabled

State Properties

This option is only applicable 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 Arrow 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 only applicable 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.

Related Topics

Fluid Properties Group.....	81
Fluid Panel.....	82
Creating Gas Mixtures.....	86
Equation of State Models.....	88
Enthalpy Models.....	93

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.

Related Topics

Analysis Setup Overview.....	76
Modules Group.....	77
Fluid Properties Group.....	81
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111
Environmental Properties Group.....	147
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Miscellaneous Group.....	153
Pipes and Junctions Panel.....	97

Pipes and Junctions Panel

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

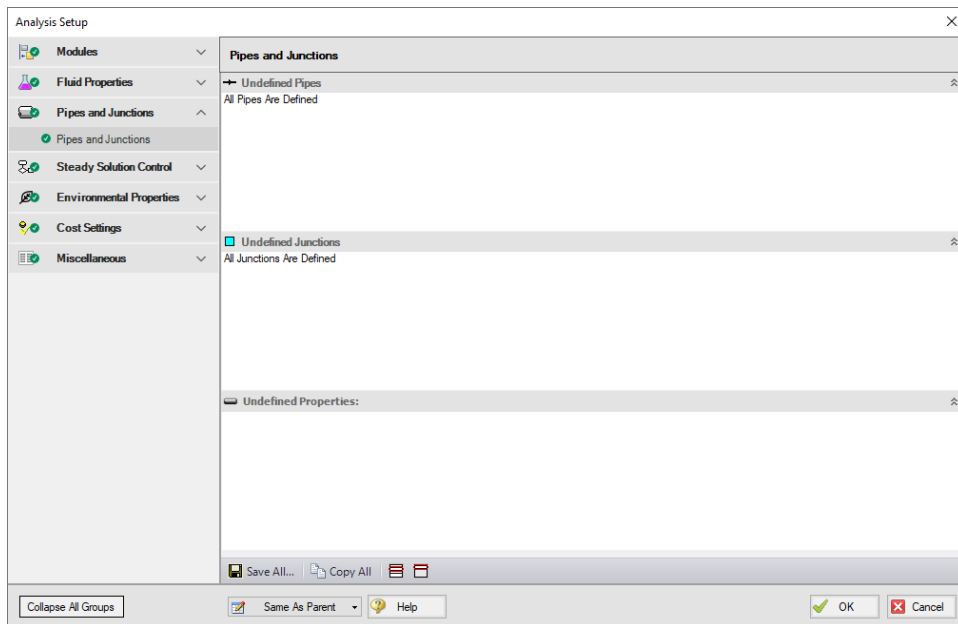


Figure 1: Default state of the Pipes and Junctions panel

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.

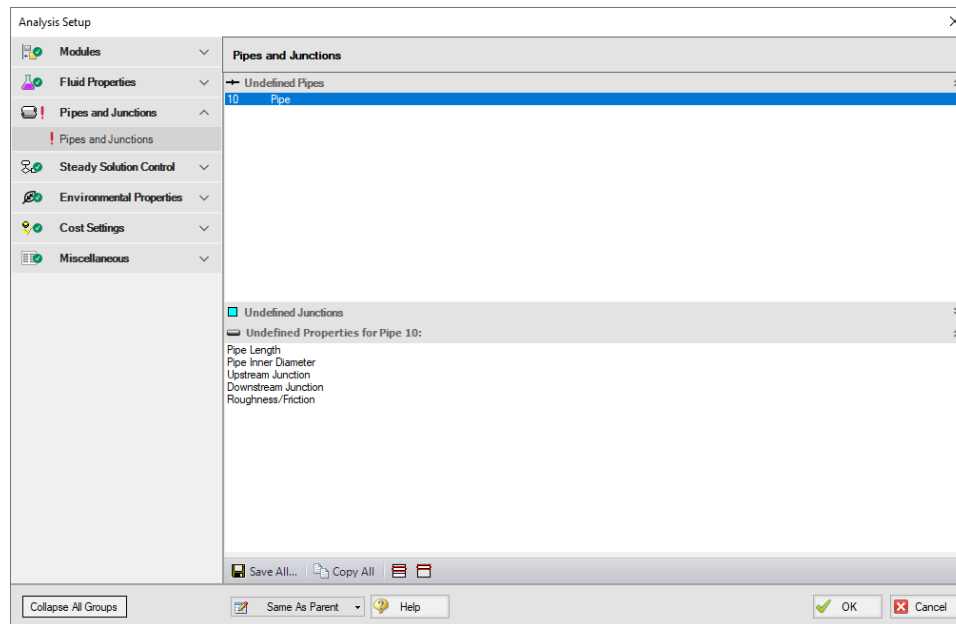


Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed

Transient Control Group

([XTS Module Only](#)) The Transient Control Group in [Analysis Setup](#) has one item: the Simulation Duration panel. The Transient Control Group will have a green checkmark next to its name when the XTS module is disabled (Steady Only mode) or when all required fields in the Simulation Duration panel are defined. The model cannot run until the Transient Control Group has a green checkmark.

Related Topics

XTS - Extended Time Simulation 586

Simulation Duration Panel

([XTS Module Only](#)) When the XTS module is activated, the Transient Control Group appears in Analysis Setup. Whether the XTS module is enabled (Transient mode) or disabled (Steady Only mode), the Simulation Duration panel can be opened and modified.

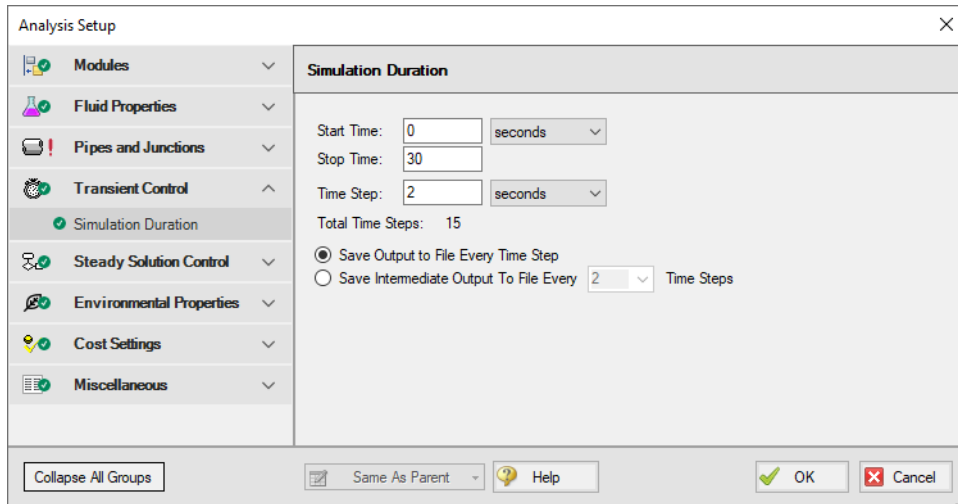


Figure 1: Simulation Duration panel fully defined with example values

The transient simulation is controlled by information the user enters in the Simulation Duration panel of the Transient Control group of the Analysis Setup window. This panel is used to specify the start and stop time of the transient simulation, the size of the time step, and the frequency of the data saved to the transient output file.

Related Topics

XTS - Extended Time Simulation 586

Goal Seek and Control Group

[\(GSC Module Only\)](#) The GSC module is controlled through the Goal Seek and Control items in the [Analysis Setup window](#). In these panels, the user specifies all of the system variables, as well as the desired goals. As the user adds variables and goals to the model, AFT Arrow will display a symbol beside the pipes and junctions that have variables or goals applied to them. The default is a V for variables, and a G for goals. The goal symbol is not displayed next to objects that are part of a group goal.

The Goal Seek and Control Group has three panels:

[Variables panel](#) - Allows users to define and apply variables to be used for the GSC run.

[Goals panel](#) - Allows users to define and apply goals to be used for the GSC run.

[Numerical Controls panel](#) - Allows GSC search engine parameters to be adjusted for troubleshooting purposes.

Related Topics

Analysis Setup Overview..... 76
 Modules Group..... 77
 Fluid Properties Group..... 81
 Pipes and Junctions Group..... 97
 Steady Solution Control Group..... 111
 Environmental Properties Group..... 147
 Cost Settings Group..... 149
 Automatic Sizing Group..... 152
 Miscellaneous Group..... 153
 GSC - Goal Seek & Control..... 489
 Variables Panel..... 101
 Goals Panel..... 106
 Numerical Controls Panel..... 111

Variables Panel

(GSC Module Only) The Variables panel is where variables are defined and applied for use with the GSC module (Figure 1). Variables are input parameters that the GSC module will change to achieve the user's desired goals. Generally, there must be one variable applied for each goal that is applied.

Types of Variables

All variables are junction parameters. The variables that are available for each junction type are shown in Table 1. There are over 30 types of variables.

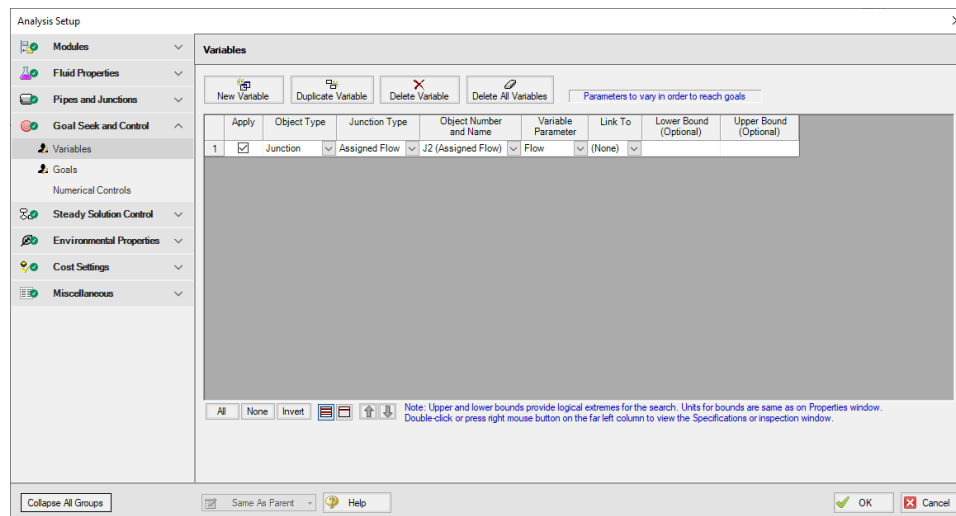


Figure 1: GSC Variables are input parameters to be automatically varied

Types of Variables

All variables are junction or pipe parameters. The variables that are available for each object type are shown in Table 1. There are over 30 types of variables.

Table 1: List of object types with available variables

Object Type	Variables	Restrictions
Pipes	Design Factor Friction ID Reduction (Scaling) Insulation Thickness	
Assigned Flow	Flow Temperature	
Assigned Pressure	Pressure Temperature	
Branch	Imposed Flow	
Check Valve	Loss Value CdA	
Compressor/Fan	Speed Flow Head Rise Impeller Size Pressure Rise	
Control Valve	Control Setpoint	
General Component	Loss Factor CdA	
Heat Exchanger	Loss Factor Heat Rate Discharge Temperature Temperature Rise Enthalpy Rise Heat Transfer Area Overall Heat Transfer Coefficient Secondary Flow Secondary Inlet Temperature	

Orifice	Diameter/Area Loss Factor CdA CdA Sonic/Subsonic	
Relief Valve	Loss Value CdA	
Screen	Open Area Loss Factor CdA	
Spray Discharge	Spray Area Number Sparger Holes CdA	Only for Spar- ger
Tank	Pressure Temperature	
Valve	Loss Value CdA Open Percent	

Creating and Deleting Variables

To create a variable, click the New Variable button on the Variables panel (Figure 1). This will insert a new item into the variable list. After the variable has been added, the variable data is defined by entering the necessary data in each of the data columns.

Variables may also be added by duplicating an existing variable, then modifying the data for the new variable. To duplicate a variable, select the variable to be duplicated from the variable list, and click the Duplicate Variable button.

To delete a variable, select the variable by clicking on the appropriate row in the variable list. After selecting the variable to be deleted, click the Delete Variable button. If you want to delete all of the variables in the list, click the Delete All Variables button.

Defining Variables

For each variable the following settings can be defined:

- **Object type** - The Object Type defines whether the variable applies to a pipe or a junction.
- **Junction type** - The Junction Type column is used to define the type of junction to which the variable is being applied.
- **Object number and name** - The specific junction to vary is selected in the Junction Number and Name column. The list will only display junctions of the type selected in the Junction Type column.
- **Variable parameter** - Each junction type has a specific set of parameters that are available as variables (Table 1). The list will only display parameters that are available for the junction type selected in the Junction Type column.
- **Linking variables** - The Link To column is used to force a variable on multiple junctions of the same type to solve to the same value.

An example where this would be desirable would be when using flow setpoint as a variable for several control valves in parallel. It is often desirable to solve for a condition where the setpoint is the same for all of the valves. This can be accomplished by using variable linking. A setpoint variable would be defined for each of the control valves. Then, one of the control valves would be selected as the basis object, which just means no linking is specified for this control valve. The other control valves would be linked to the basis object by selecting the basis control valve from the list in the Link To column.

- **Variable bounds** - During a goal search, the GSC module will modify the values for the defined variables. Sometimes it is helpful to specify upper and lower bounds for variables to provide logical extremes during the goal search. Some examples of logical bounds that can be applied would be lower and upper bounds on valve open percentage of 0% and 100%, respectively.

Note that bounds can have engineering units, but these units are not displayed. It is assumed the units are the same as the junction input parameter Properties Window.

Organizing Content

The lower left corner of the panel features options for rearranging and selecting variables and goals. All will select all available variable or goals. None will deselect all variables or goals. Invert will invert which variables and goals are selected. There are also options to show all variables or goals and to show only the applied variables or goals. The goals and variables can also be reordered using the turquoise arrows to change their position.

The Same As Parent button allows the variables, goals, numerical control, or all of the parameters (in regards to the Goal Seek and Control Group) of the parent scenario to be copied down to the child scenario.

Reviewing Pipe/Junction Data

Data for the selected objects in the Variables panel can be reviewed and edited without having to close the Analysis Setup window.

- **Reviewing object input data** - If you would like to see the input data for the junction selected as a variable, press the right mouse button on the far left column in the Variables table. The [inspection window](#) will display, showing you the input data.
- **Editing object input data** - Similar to the inspection feature just described, you can open the [Pipe](#) or [Junction Properties window](#) by double-clicking the far left column in the Variables table. From there you can change any data desired.

Applying Variables

Once a variable has been created, the user must specify if a variable is to be used when the GSC module is run. To apply a variable, select the checkbox in the Apply column.

Using the Apply feature allows the user to define multiple variables that can be used in alternate cases or analyses. Any of the variables that are not being used for a particular analysis can remain in the list for later use. Click the Apply checkbox for the variables that are to be used in the current analysis.

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek and Control Group.....	100
Goals Panel.....	106
Numerical Controls Panel.....	111

Goals Panel

[\(GSC Module Only\)](#) The Goals panel is where goals are defined and applied for use with the GSC module (Figure 1). Goals are output values you would like to achieve. The GSC module adjusts the applied variables until the applied goals are met. Generally there should be one goal applied for each applied variable.

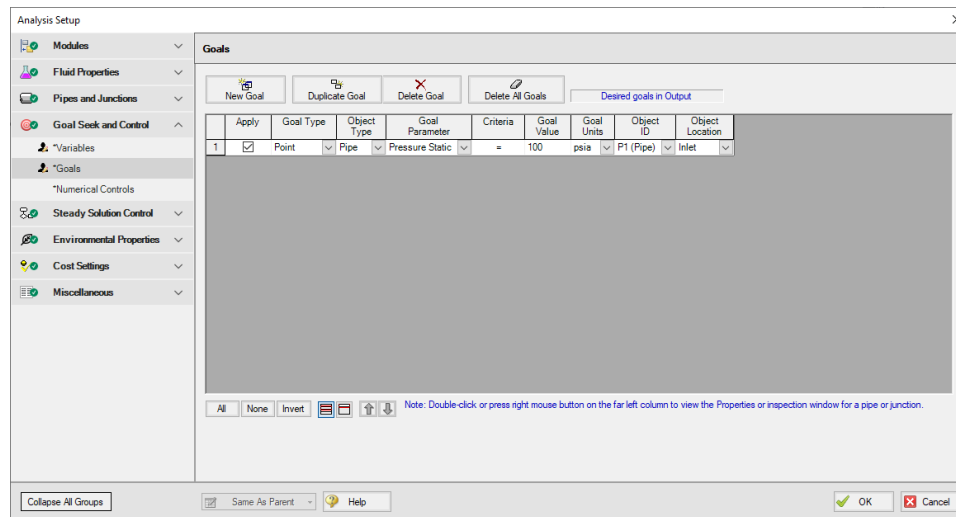


Figure 1: GSC Goals are output parameters to be achieved by changing the variables

Types of Goals

The GSC module offers over 150 goals applied to pipes, junctions or groups of pipes and/or junctions. Table 1 lists all of the types of goals.

Table 1: List of object types with available goals

Object Type	Goals	Restrictions
Pipe	Enthalpy Stagnation Enthalpy Static Heat Rate Into Pipe Insulation #1 Surface Temperature Insulation #2 Surface Temperature Insulation #3 Surface Temperature Mach Number Mass Flow Rate Pressure Drop Friction Pressure Gradient Pressure Stagnation Pressure Static Pressure Static Maximum Pressure Static Minimum Reynolds Number Temperature Loss Stagnation Temperature Loss Static Temperature Stagnation Temperature Static Temperature Superheat Velocity Volumetric Flow Rate Wall Temperature	Only wall friction Only wall friction
Area Change	Mass Flow Rate Pressure Stagnation Pressure Static Temperature	
Assigned Flow	Pressure Stagnation Pressure Static	
Assigned Pressure	Mass Flow Rate Net Into Jct	Net flowrate
Bend, Branch, Check Valve, General Component, Orifice, Relief Valve, Screen, Valve, and Venturi	Mass Flow Rate Pressure Stagnation Pressure Static Temperature	
Compressor/Fan	Head Rise Mass Flow Rate Power Overall Pressure Rise Pressure Stagnation Pressure Static Temperature	Only for non-fixed head rise Only for non-fixed flow Only for non-fixed pressure rise
Control Valve	Cv Kv Mass Flow Rate Open Percentage Pressure Loss Pressure Stagnation Pressure Static Temperature	If Open Pct Data Exists If Open Pct Data Exists Only for PRV, PSV If Open Pct Data Exists Not PDCV Only for FCV Only for FCV
Dead End	Pressure	
Heat Exchanger	Heat Rate In Mass Flow Rate Pressure Stagnation Pressure Static Temperature	
Spray Discharge	Mass Flow Rate Out	Net flowrate
Tank	Mass Flow Rate Net Into Jct	Net flowrate
Tee or Wye	Pressure Stagnation Pressure Static Temperature	
Group Sum	Heat Rate In Mass Flow Rate Pressure Drop Friction	
Group Max/Min	Uses same goals as group type	Group must be all pipes or same junction type

Creating and Deleting Goals

To create a goal, click the New Goal button on the Goals tab. This will insert a new item into the goals list. After the goal has been added, the goal data is defined by entering the necessary data

in each of the data columns.

Goals may also be added by duplicating an existing goal, then modifying the data for the new goal. To duplicate a goal, select the goal to be duplicated from the goal list, and click the Duplicate Goal button.

To delete a goal, select the goal by clicking on the appropriate row in the goal list. After selecting the goal to be deleted, click the Delete Goal button. If you want to delete all of the goals in the list, click the Delete All Goals button.

Defining Goals

For each goal the following settings can be defined:

- **Goal Type** - Select the type of goal to be used from the following:
 - **Point Goal** - A point goal is specified when the desired goal can be specified at a specific object location in the model.
 - **Differential Goal** - A differential goal is used when the goal value is actually determined by the difference in values between two specific object locations. An example of a differential goal might be the pressure difference between two locations in the model.
 - **Group Goal** - A Group goal is used when a goal is to be applied to a group of objects at the same time. In order to use a Group goal, all of the objects must be added to a [Group](#) by using the Groups command on the Edit menu.
- **Object Type** - Select the type of object to which the goal is to be applied. See the section below for further description of the object types.
- **Goal Parameter** - Select the type of goal to be applied to the selected object, such as pressure, flow rate, or temperature.
- **Criteria** - Specify the criteria which is to be met for the specified goal parameter, such as equals, greater than or equal, or less than or equal.
- **Goal Value/Units** - Specify the value and units for the goal parameter.
- **Object ID** - Specify the pipe, junction, or group name to which the goal applies.
- **Object Location** - Specify, as appropriate, whether the goal applies to the inlet or outlet of the selected object or group.

Organizing Content

The lower left corner of the panel features options for rearranging and selecting variables and goals. All will select all available variable or goals. None will deselect all variables or goals. Invert will invert which variables and goals are selected. There are also options to show all variables or goals and to show only the applied variables or goals. The goals and variables can also be reordered using the turquoise arrows to change their position.

The Same As Parent button allows the variables, goals, numerical control, or all of the parameters (in regards to the Goal Seek and Control Group) of the parent scenario to be copied down to the child scenario.

Object Types

For **Point and Differential goal types**, the object type indicates whether the goal is to be applied to a pipe or a junction object. If the goal is to be applied to a junction, the type of junction is selected from the list.

For the Group goal type, the user must select between a Group Max/Min or a Group Sum object type.

A **Group Max/Min goal** allows a single goal to be applied to a group of objects. The GSC module applies a Group Max/Min goal by ensuring the final goal value is either greater than or equal to (a Min goal) or less than or equal to (a Max goal) the specified value for each object in the group. For example, a Group Max/Min goal can be applied to ensure a minimum flow rate out of each spray discharge nozzle in a group of nozzles. Group Max/Min goals only work for groups where all group members are of the same type (i.e., all pipes or all valve junctions).

A **Group Sum goal** is used to set the sum of the goal parameter for all the members in a group to a specified value. For example, the Group Sum goal can be applied to set the sum of the volumetric flow rates through all of the nozzles in a group of spray discharge nozzles to a particular value. Groups used in Group Sum goals can consist of pipes and/or junctions, with junctions of different type.

Applying Goals

Once a goal has been created, the user must specify if a goal is to be used when GSC is run. To apply a goal, select the checkbox in the Apply column.

Using the Apply feature allows the user to define multiple goals that can be used in alternate cases or analyses. Any of the goals that are not being used for a particular analysis can remain in the list for later use. Click the Apply checkbox for the goals that are to be used in the current analysis.

Reviewing Pipe/Junction Data

Data for the selected objects in the Goals panel can be reviewed and edited without having to close the Analysis Setup window.

- **Reviewing object input data** - If you would like to see the input data for the junction selected as a goal, press the right mouse button on the far left column in the Goals table. The [inspection window](#) will display, showing you the input data. This feature does not work for group goals.
- **Editing object input data** - Similar to the inspection feature just described, you can open the [Pipe](#) or [Junction Properties Window](#) by double-clicking the far left column in the Goals table. From there you can change any data desired. This feature does not work for group goals.

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek and Control Group.....	100
Variables Panel.....	101
Numerical Controls Panel.....	111
GSC Goal may not be valid.....	718
When Goals Cannot Be Achieved.....	495

Numerical Controls Panel

[\(GSC Module Only\)](#)

Numerical control of the GSC search engine parameters can be adjusted on the Numerical Control tab. It is recommended the user **avoid** changing these parameters unless the user understands their purpose or it has been recommended by AFT technical support.

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek and Control Group.....	100
Variables Panel.....	101
Goals Panel.....	106

Steady Solution Control Group

The user has control over a number of parameters that influence numerical convergence.

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

Analysis Setup Overview.....	76
Modules Group.....	77
Fluid Properties Group.....	81
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Environmental Properties Group.....	147
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Miscellaneous Group.....	153
Tolerance Panel.....	141
Iterations Panel.....	144
Pressure and Temperature Limits Panel.....	146
Solution Method Panel.....	112

Related Blogs

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

Solution Method Panel

The Solution Method panel defines the calculation method and relevant parameters for the calculation (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.

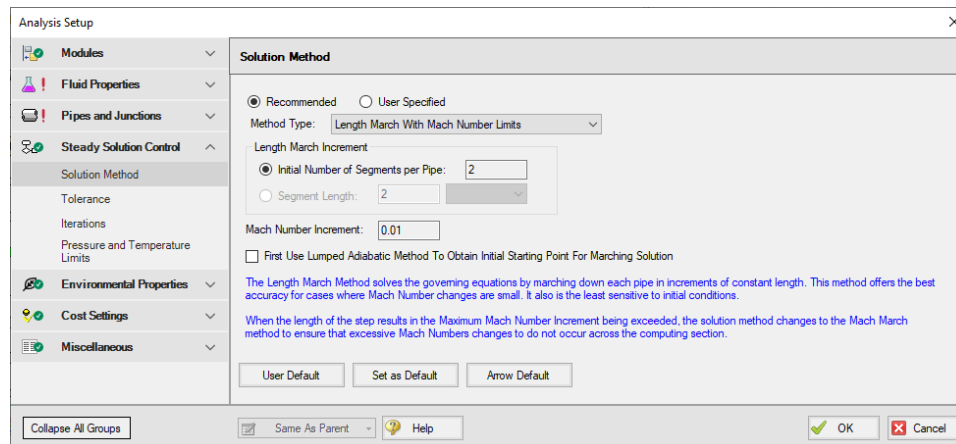


Figure 1: Default settings on the Solution Method panel

Solution Method Overview

AFT Arrow provides several solution methods for compressible pipe flow. 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. For some applications these effects are minimal, 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	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

Steady Solution Control Group.....	111
Tolerance Panel.....	141
Iterations Panel.....	144
Pressure and Temperature Limits Panel.....	146

Related Blogs

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

Length March Method - Detailed Discussion

The Length March [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.

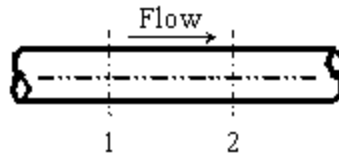


Figure 1: Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations.

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:

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

Because we know the parameters above, we also know:

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

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

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

If there is no elevation change, then:

$$z_2 = z_1$$

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

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

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

If the pipe is isothermal, then:

$$T_2 = T_1$$

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

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

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

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

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

1. 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.
2. Notice how the velocity and Mach number increase along the pipe length.
3. Notice how the static temperature drops. This is due to cooling of the gas as it expands with pressure drop.
4. Notice that the stagnation enthalpy is constant in accordance with adiabatic flow.
5. Notice how the static enthalpy drops along the pipe length. This shows that the flow is not isenthalpic.
6. 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.

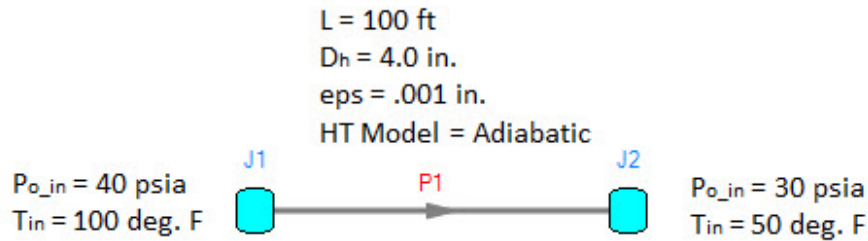


Figure 2: Model of an air system dropping from 40 to 30 psia

Pipe -- Point	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	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	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 Arrow model Verify_Governing_Equations.aro using the Length March method

The model, Verify_Governing_Equations.aro, 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:

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

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

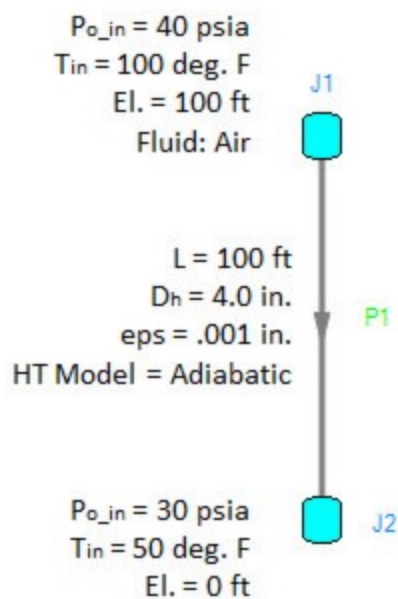


Figure 5: AFT Arrow model solved in Figure 1 and verified in Table 1. Fluid is air. File is Verify_Governing_Equations.aro.

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):

$$\rho g dz = \rho_{avg} g (z_1 - z_2) \tag{53}$$

Table 1: Verification showing how AFT Arrow 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/ft)	f	dp (lbm/ft3)	V (ft/sec)	dV (ft/sec)	rho (lbm/ft3)	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:

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

The four terms in Equations 53 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 Arrow does not solve Equation 49. Rather, when using the Length March method, AFT Arrow 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 Arrow 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 Arrow. 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.aro, 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.

Pipes		Axial Points		Heat Transfer							
Pipe -- Point	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.aro 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.

Pipe	P Static In (psia)	Rho Static In (lbm/ft3)	Vel. In (feet/sec)	dH Stag. (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

Jct	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

Solution Method Panel	112
Nomenclature	639

Mach Number March Method - Detailed Discussion

The Mach Number March [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

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

Four useful parameters that simplify notation are:

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

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

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

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

Using the four new parameters in Equation 18 simplifies to

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

Integrating Equation 19 yields

$$\begin{aligned} \ln \frac{M_2^2}{M_1^2} - (x_2 - x_1) \left(\bar{F}_f \frac{f}{D} + \bar{F}_g \frac{g \sin \theta}{R \bar{Z} \bar{T}_o} \right) - \bar{F}_{T_o} \left(\ln \frac{T_{o2}}{T_{o1}} + \ln \frac{Z_2}{Z_1} \right) \\ - \bar{F}_\gamma \left(\ln \frac{\gamma_2}{\gamma_1} \right) = 0 \end{aligned} \quad (20)$$

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:

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

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

P_h = Heated perimeter

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

Equating Equation 32 and 33 yields

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

Rearranging Equation 34 yields:

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

Integrating Equation 35 yields

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

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

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

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

Constant Heat Flux

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

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

The stagnation temperature change is thus:

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

Integrating Equation 39 yields:

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

Substituting this into Equation 20 yields:

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

Example

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

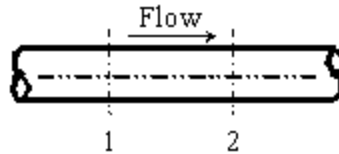


Figure 1: *Marching equations are applied at successive computing sections along the pipe. Conditions at location 1 are known, and at location 2 are being solved using one of the marching equations.*

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

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

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

Because we know the parameters above, we also know:

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

Let's turn our attention to the Mach Number March method. Depending on the heat transfer model, AFT Arrow 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 Dx 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 Arrow 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:

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

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

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

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 DM , 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 Arrow, 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.

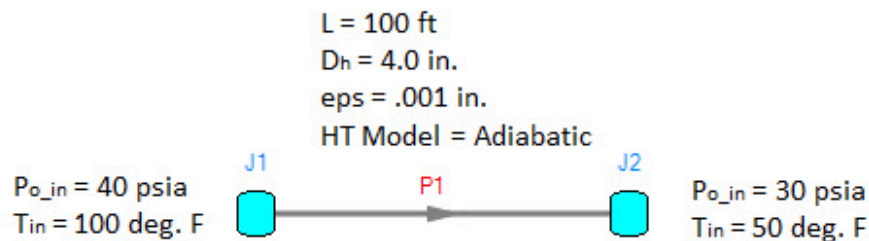


Figure 2: Model of an air system dropping from 40 to 30 psia

Axial Points											
Pipe -- Point	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

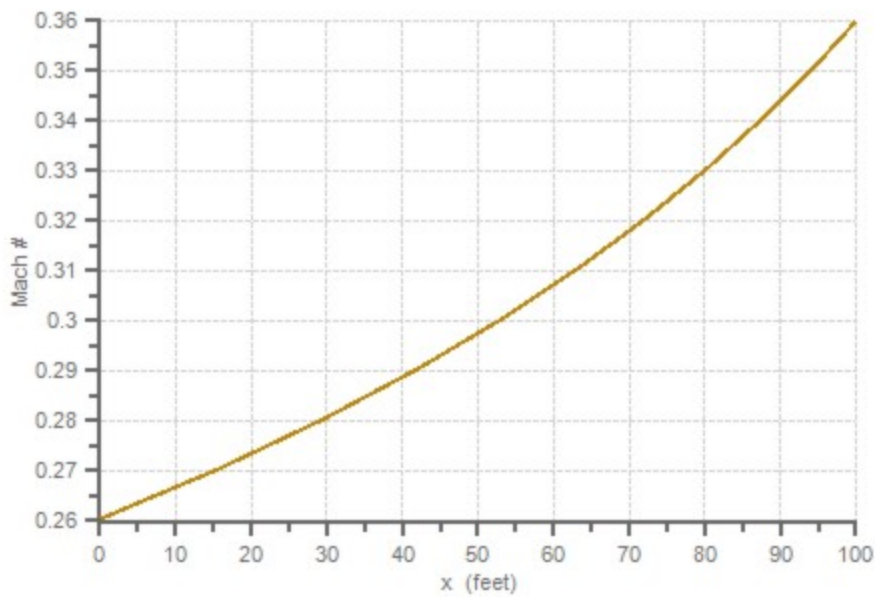


Figure 4: The Mach number increases with length in this plot of the Figure 3 results

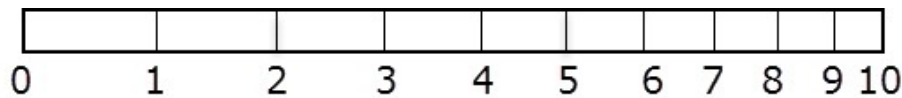


Figure 5: Relative pipe section locations for Figure 3 results. Notice how computing sections get closer as a result of gas acceleration as the end of the pipe is approached.

Mach Number March Method Verification

By changing the solution method for Verify_Governing_Equations.aro 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	x (feet)	P Static (psia)	Rho Static (lbm/ft3)	Velocity (feet/sec)	H Stag. (Btu/lbm)	f
1 -- 0	0.000000	38.138699	0.18667813	301.36948	202.17825	0.014931721
1 -- 1	15.597554	36.717522	0.17988963	312.74225	202.19830	0.014931537
1 -- 2	29.455023	35.396301	0.17358769	324.09607	202.21611	0.014931159
1 -- 3	41.812508	34.164791	0.16772209	335.43042	202.23198	0.014930761
1 -- 4	52.870377	33.014107	0.16224930	346.74472	202.24620	0.014930346
1 -- 5	62.796864	31.936487	0.15713137	358.03854	202.25896	0.014929913
1 -- 6	71.734070	30.925154	0.15233517	369.31125	202.27043	0.014929463
1 -- 7	79.802666	29.974119	0.14783145	380.56238	202.28081	0.014928997
1 -- 8	87.105637	29.078117	0.14359452	391.79132	202.29019	0.014928515
1 -- 9	93.731346	28.232458	0.13960154	402.99759	202.29871	0.014928017
1 -- 10	99.755882	27.432985	0.13583225	414.18063	202.30644	0.014927504
1 -- 11	100.000000	27.399925	0.13567653	414.65601	202.30676	0.014927232

Figure 6: Axial Points for AFT Arrow model Verify_Governing_Equations.aro using the Mach Number March method

Table 1: Verification showing how AFT Arrow Mach Number March method predictions satisfy momentum, mass and energy balance equations. Taken from spreadsheet Verify_Governing_Equations_Mach.xls.

AFT ARROW OUTPUT						AVERAGED VALUES (CALCULATED)						MOMENTUM BALANCE					MASS BALANCE	ENERGY BALANCE	
P Static	Rho Static	Velocity	H Stag.			dp	V	d/	dho			dp Total	dp Fric.	dp Accel.	dp Grav.	Balance	Balance	Balance	
x (feet)	(psia)	(lbm/ft3)	(feet/sec)	(Btu/lbm)	f	p (lbm/ft3)	(lbm/ft3)	(feet/sec)	(feet/sec)	(Btu/lbm)	dx (feet)	dx (feet)	(psi)	(psi)	(psi)	(psi)	(psi)	(lbm/sec)	(lbm/sec)
0.00	38.139	0.1867	301.369	202.178	0.0149	-	-	-	-	-	-	-	-	-	-	-	-	-	-
15.60	36.718	0.1799	312.742	202.198	0.0149	0.1833	0.00679	307.066	-11.373	-0.0200	-15.60	15.60	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	0.01287	-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	380.562	202.281	0.0149	0.1501	0.00450	374.937	-11.251	-0.0104	-8.07	8.07	0.95104	-0.82281	-0.13665	0.00841	-0.00002	-5.64E-09	-0.010
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	0.00016	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

Solution Method Panel	112
Nomenclature	639

Lumped Adiabatic Method - Detailed Discussion

When this [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. One application where this is definitely not the case is in [Rotating Systems](#).

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

Solution Method Panel	112
Nomenclature	639

Lumped Isothermal Method - Detailed Discussion

When this [Solution Method](#) is used, all heat transfer data entered for individual pipes is neglected unless the user has specified in the Solution Method panel that individual pipe temperatures should be used. 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. One application where this is definitely not the case is in [Rotating Systems](#).

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 [Solution Method panel](#). This temperature is used for all pipes unless directed otherwise by the user.

Related Topics

Solution Method Panel	112
Nomenclature	639

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, Temperature Tolerance, and Concentration Tolerance (if [dynamic mixing](#) occurs).

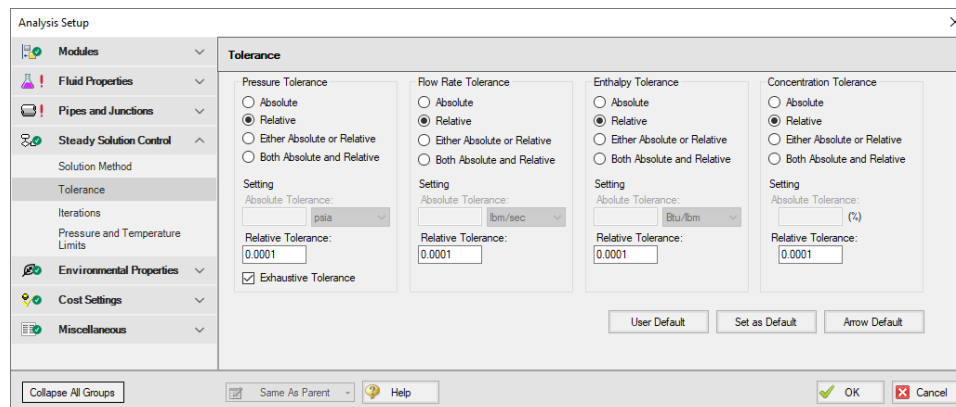


Figure 1: Tolerance panel in Analysis Setup

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*.

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	0.66666-7	8.00000-0	NO	NO	NO	NO
3	14.0000	0.14285-7	2.00000-0	NO	NO	NO	NO
4	13.5000	0.03703-7	0.50000-0	NO	NO	NO	NO
5	13.6000	0.00735-3	0.10000-0	NO	NO	NO	NO
6	13.5900	0.00073-6	0.01000-0	NO	NO	NO	NO
7	13.5913	0.00009-6	0.00013-0	NO	YES	YES	NO
8	13.5907	0.00004-4	0.00060-0	NO	YES	YES	NO
9	13.5909	0.00001-5	0.00020-0	YES	YES	YES	YES

Exhaustive Tolerance

AFT Arrow 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 Topics

Steady Solution Control Group.....	111
Iterations Panel.....	144
Pressure and Temperature Limits Panel.....	146

Solution Method Panel 112
 Try Absolute Tolerance 740
 Energy may not be balanced at Junction X 715
 Flow at Junction X may not be balanced 716

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.

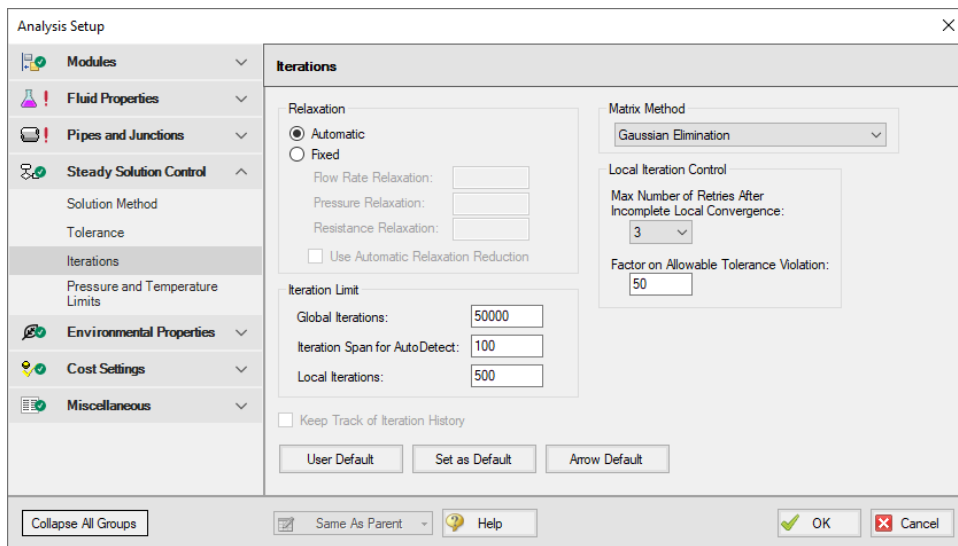


Figure 1: Iterations panel in Analysis Setup

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

- **Global 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 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 AFT Arrow, 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 Topics

Steady Solution Control Group.....	111
Tolerance Panel.....	141
Pressure and Temperature Limits Panel.....	146
Solution Method Panel.....	112
Lower the Flow Rate Relaxation.....	739
Review the Iteration History.....	742
I.....	329

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.

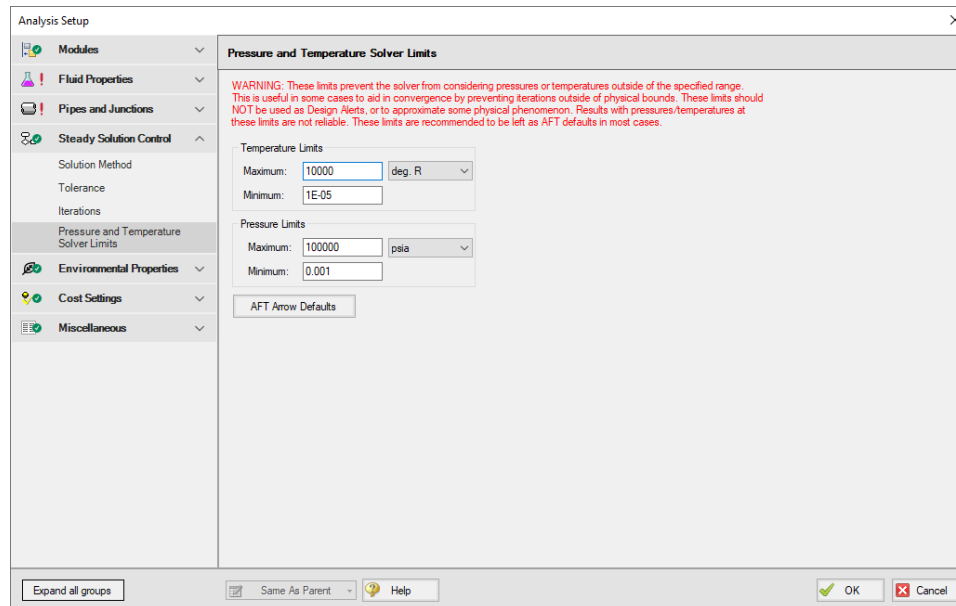


Figure 1: Pressure and Temperature Solver Limits panel within Analysis Setup

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

Related Topics

Steady Solution Control Group.....	111
Tolerance Panel.....	141
Iterations Panel.....	144
Solution Method Panel.....	112

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.

Related Topics

Analysis Setup Overview.....	76
------------------------------	----

Modules Group.....	77
Fluid Properties Group.....	81
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Miscellaneous Group.....	153
Environmental Properties Panel.....	148

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.

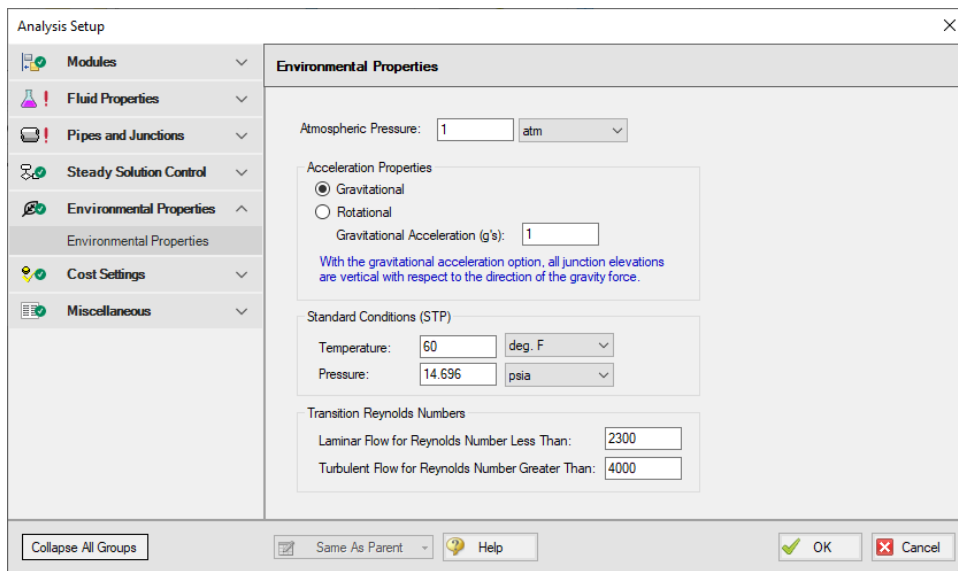


Figure 1: Environmental Properties panel in the Analysis Setup window

Atmospheric Pressure

AFT Arrow 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.

Acceleration Properties

The default for gravitational acceleration is 1 standard earth gravitational acceleration, or 32.174 feet/sec². You can change this to a multiple of standard earth accelerations (that is, number of g's) according to the design environment for your system. In principle, the body force on a fluid system does not have to be due to gravity.

You also can specify a rotational acceleration such as might be experienced in turbomachinery. You are prompted to enter a rotation rate in rpm (revolutions per minute). In this case, AFT Arrow interprets the junction elevation data as distance from the rotation centerline, and uses a varying acceleration body force on each individual pipe as determined by the laws of rotational dynamics.

Each individual pipe computing section will use the acceleration based on the arithmetic average over its length. You cannot specify both gravitational and rotational accelerations at the same time.

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 Arrow 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 Arrow uses a default standard temperature of 60 deg. 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 deg. F (0 deg. C) for temperature. The normal pressure and temperature can not be changed by the user.

Transition Reynolds Numbers

AFT Arrow 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 and convection heat transfer.

Related Topics

Environmental Properties Group..... 147

Cost Settings Group

The Cost Settings Group in [Analysis Setup](#) has one item: the [Cost Settings panel](#). This group is always present regardless of whether or not cost calculation is enabled. When all fields are defined, the Cost Settings Group will have a green checkmark next to its name.

Related Topics

Analysis Setup Overview.....	76
Modules Group.....	77
Fluid Properties Group.....	81
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111
Environmental Properties Group.....	147
Automatic Sizing Group.....	152
Miscellaneous Group.....	153
Cost Settings Panel.....	150
Cost Calculation.....	465

Cost Settings Panel

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The Cost Settings panel must be defined in order to run a model. By default, cost calculation is not enabled and no further input is required to define the panel.

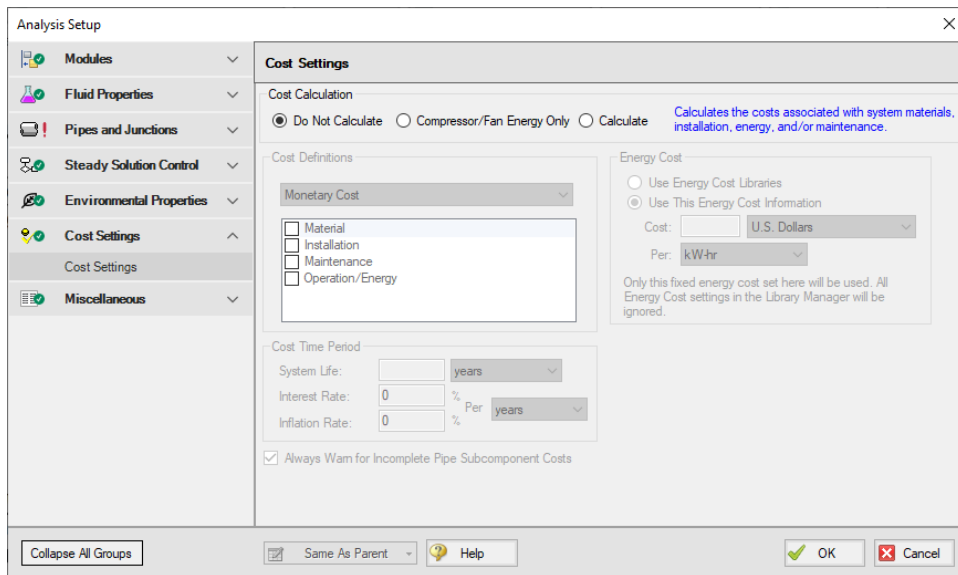


Figure 1: The Cost Settings panel

Enable Cost Calculation

Enable Cost Calculations from the [Cost Settings panel](#) in Analysis Setup. The following options are available:

- **Do Not Calculate** - Selected by default.
- **Compressor/Fan Energy Only** - Only calculate the costs associated with the power consumption of compressors. The energy cost and system life must be entered in the Cost Settings panel.
- **Calculate** - Enable full calculation of costs. In addition to defining Cost Settings, [Cost Libraries](#) must be defined to use this feature.

Including Objects in Cost Report

Workspace objects must be included in the Cost Report to have their cost calculated. This allows only analyzing the cost of certain parts of the model - useful if only one compressor is being replaced, or if an expansion to an existing system is being considered. To include the objects in the Cost Report simply select Include Cost in Report on the object's Cost tab, as shown in Figure 2. Finer detail in how the Cost is included can be set by clicking the [Cost Application](#) button.

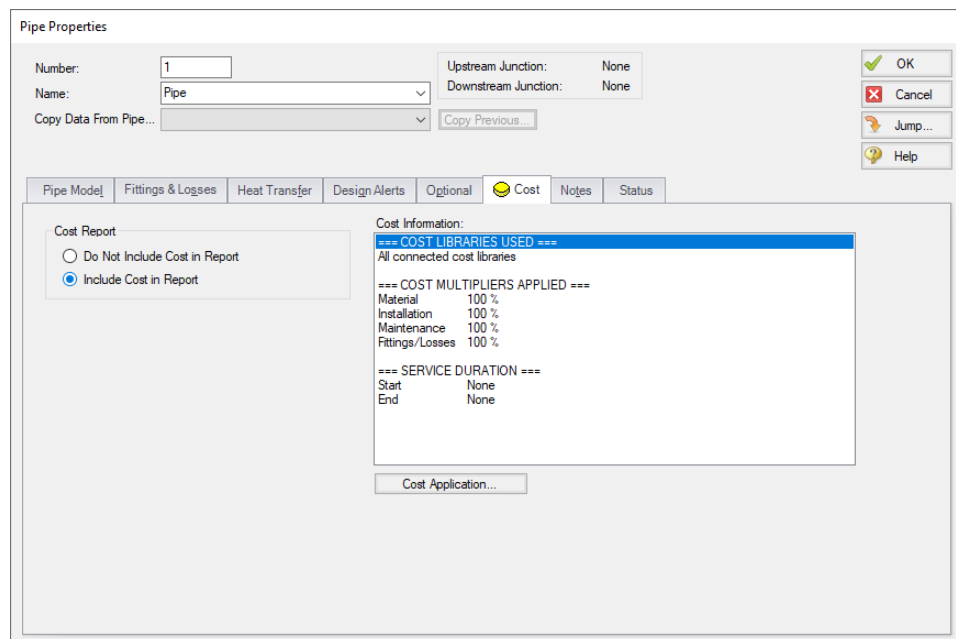


Figure 2: Include Cost in Report

Energy Cost

Energy cost data can be specified through an [energy cost library](#), or by directly entering an energy cost on the Cost Settings window. Using an energy cost library allows you to specify multiple energy costs, such as energy costs at both peak and off-peak rates.

Cost Definitions

The Cost Definitions section allows you to specify the type of costs to be determined. Check the boxes next to the items that are of interest. The options are:

- Material
- Installation
- Maintenance
- Operation/Energy

Cost Time Period

The system lifetime to be applied for the cost calculations is specified in the Cost Time Period section. You can also specify interest and inflation rates to be applied to the cost calculations.

The System Life affects all recurring costs. It can also affect non-recurring costs if the Service Duration is used.

See [Cost Libraries](#) for detailed discussions regarding recurring and non-recurring costs.

Related Topics

Cost Settings Group.....	149
Cost Calculation.....	465

Automatic Sizing Group

[\(ANS Module Only\)](#) The Automatic Sizing group in [Analysis Setup](#) only appears when the ANS Module is activated from the [Modules panel](#). The Automatic Sizing group has one panel: the [Sizing Summary panel](#).

Related Topics

Analysis Setup Overview.....	76
Modules Group.....	77
Fluid Properties Group.....	81

Pipes and Junctions Group..... 97
 Goal Seek and Control Group..... 100
 Steady Solution Control Group..... 111
 Environmental Properties Group..... 147
 Cost Settings Group..... 149
 Miscellaneous Group..... 153
 ANS - Automated Network Sizing..... 500
 Sizing Summary Panel..... 153

Sizing Summary Panel

The Sizing Summary panel in [Analysis Setup](#) displays the [Sizing Summary](#) when the [ANS module](#) is active. The Sizing Summary can also be viewed in the [Sizing Window](#).

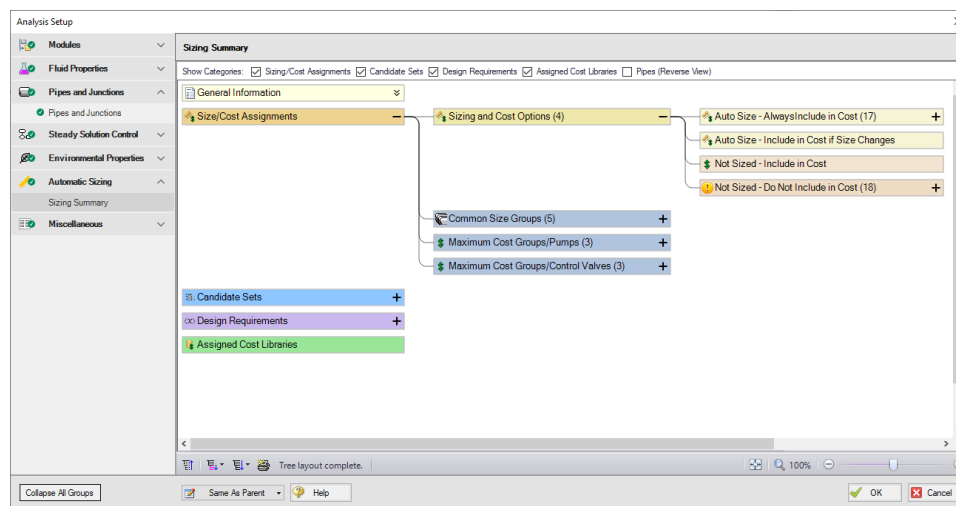


Figure 1: Sizing Summary panel with all tree nodes expanded

Related Topics

ANS - Automated Network Sizing..... 500
 Automatic Sizing Group..... 152

Miscellaneous Group

The Miscellaneous group contains settings that are used infrequently. By default, the Miscellaneous Group is fully defined, with a green checkmark next to its name, without any additional input from the user.

The Miscellaneous group has two panels:

- [Heat Transfer Sectioning panel](#) - Contains advanced settings for heat transfer calculations for troubleshooting purposes

- [Advanced Parameters panel](#) - Contains advanced parameters that can be used for troubleshooting purposes

Related Topics

Analysis Setup Overview.....	76
Modules Group.....	77
Fluid Properties Group.....	81
Pipes and Junctions Group.....	97
Goal Seek and Control Group.....	100
Steady Solution Control Group.....	111
Environmental Properties Group.....	147
Cost Settings Group.....	149
Automatic Sizing Group.....	152
Heat Transfer Sectioning Panel.....	154
Advanced Parameters Panel.....	155

Heat Transfer Sectioning Panel

The number of thermal sections that are used to calculate heat transfer in Heat Exchangers can be modified here.

Because the amount of heat transfer is dependent on the temperature difference at any given point, an overall average would not give an accurate estimate. Sectioning heat transfer components increases the accuracy of the calculation due to the changing thermal profile. Very large thermal changes may require additional sections.

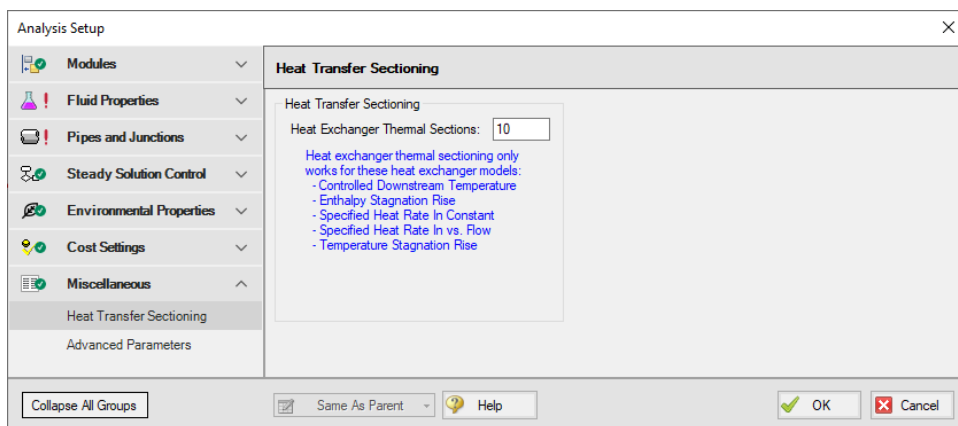


Figure 1: Default state of the Heat Transfer Sectioning panel

Related Topics

Miscellaneous Group.....	153
--------------------------	-----

Advanced Parameters Panel..... 155

Advanced Parameters Panel

The Advanced Parameters panel, shown in Figure 1, of [Analysis Setup](#) offers various settings intended only for troubleshooting purposes.

These parameters should not be changed under normal circumstances and will significantly impact results.

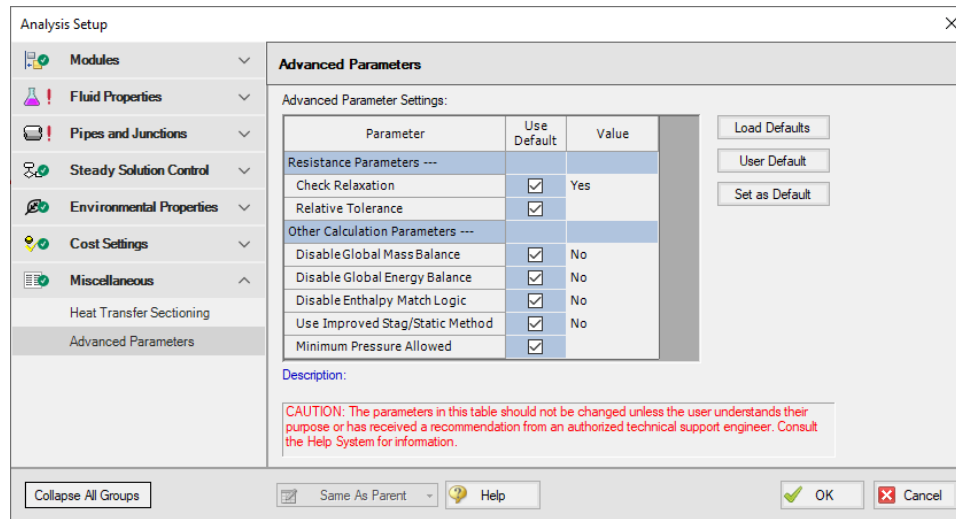


Figure 1: Default state of the Advanced Parameters panel

Related Topics

Miscellaneous Group..... 153
 Heat Transfer Sectioning Panel..... 154
 Global energy balance disabled by user in Advanced Parameters panel..... 710
 Global mass balance disabled by user in Advanced Parameters panel..... 710

Working with Pipe and Junction Objects

This section focuses on interactions with the Pipe and Junction objects in the AFT Arrow 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	157
Pipe Drawing Modes	162

Selecting and Finding Objects

AFT Arrow 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

Working with Pipe and Junction Objects.....	157
Pipe Drawing Modes.....	162
Global Edit.....	169
Selecting Objects.....	158
Finding Workspace Objects.....	160
Groups.....	161

Selecting Objects

A single left click on any object will select that object. Selecting multiple objects can be accomplished by:

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

Select Flow Path

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

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

If a junction is selected which has multiple flow paths attached to it (e.g., a branch with 3 or more pipes) then objects in each of the connected flow paths will be selected. 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 Parameters
- Zero-Length Connectors
- Notes
- Undefined Objects
- User-Colored Pipes
- Transient Data Exists (XTS only)
- Groups
- Used as Goal or Variable
- 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.....	157
Finding Workspace Objects.....	160
Groups.....	161

Finding Workspace Objects

Workspace Find

The Find window utility helps you find a particular pipe or junction by its ID number and name. The Find utility becomes more useful for large models.

You can access the Find utility by selecting Find on the [View menu](#), by clicking the binocular button on the [Toolbar](#), or by pressing the CTRL+ F keys on the keyboard. 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 the 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.

Related Topics

Selecting and Finding Objects	157
Selecting Objects	158
Groups	161

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 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

Selecting and Finding Objects	157
Selecting Objects	158
Finding Workspace Objects	160

Pipe Drawing Modes

AFT Arrow 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.

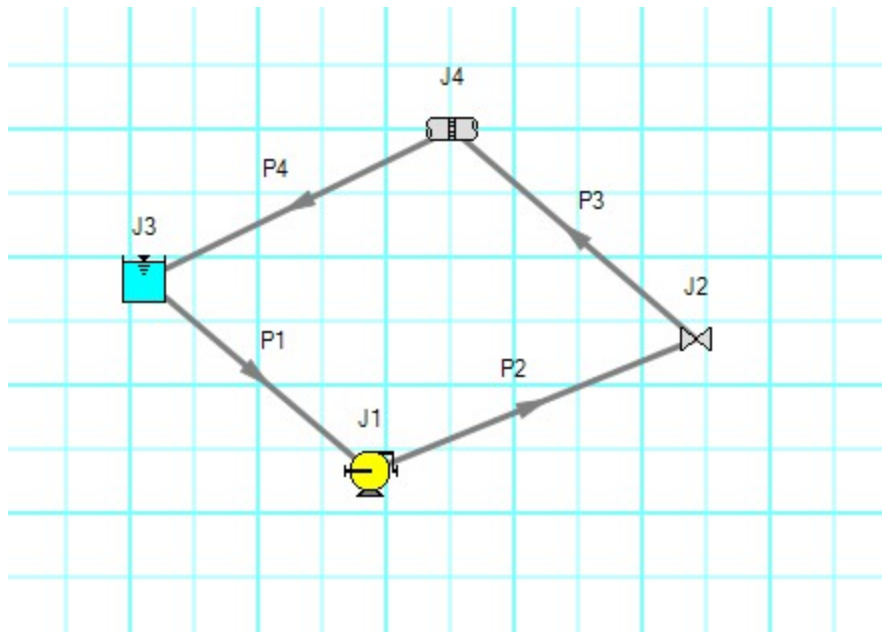


Figure 1: 2D Freeform with Snap to Grid off

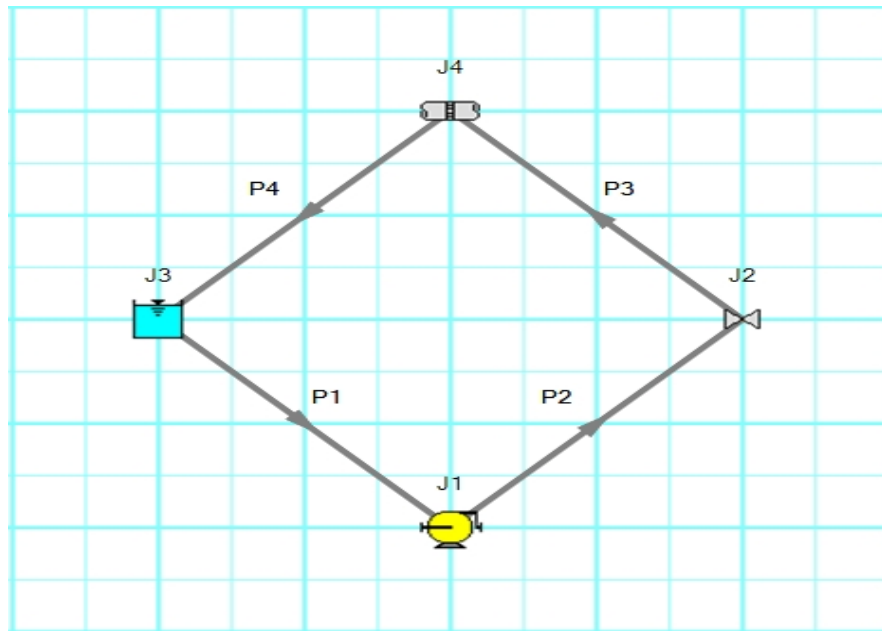


Figure 2: 2D Freeform with Snap to Grid on

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.

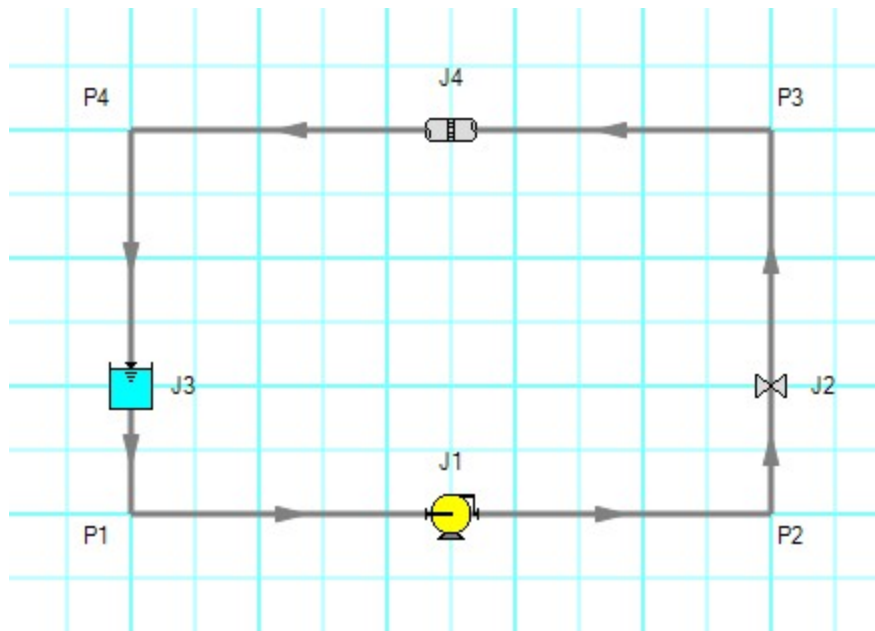


Figure 3: Model in 2D Orthogonal mode with Snap to Grid on

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.

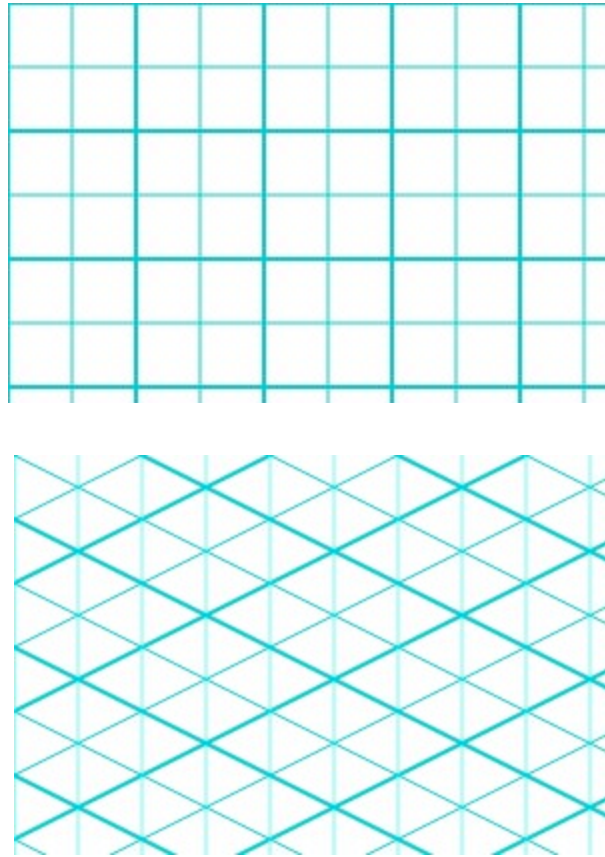


Figure 4: 2D (left) and isometric (right) grids

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.

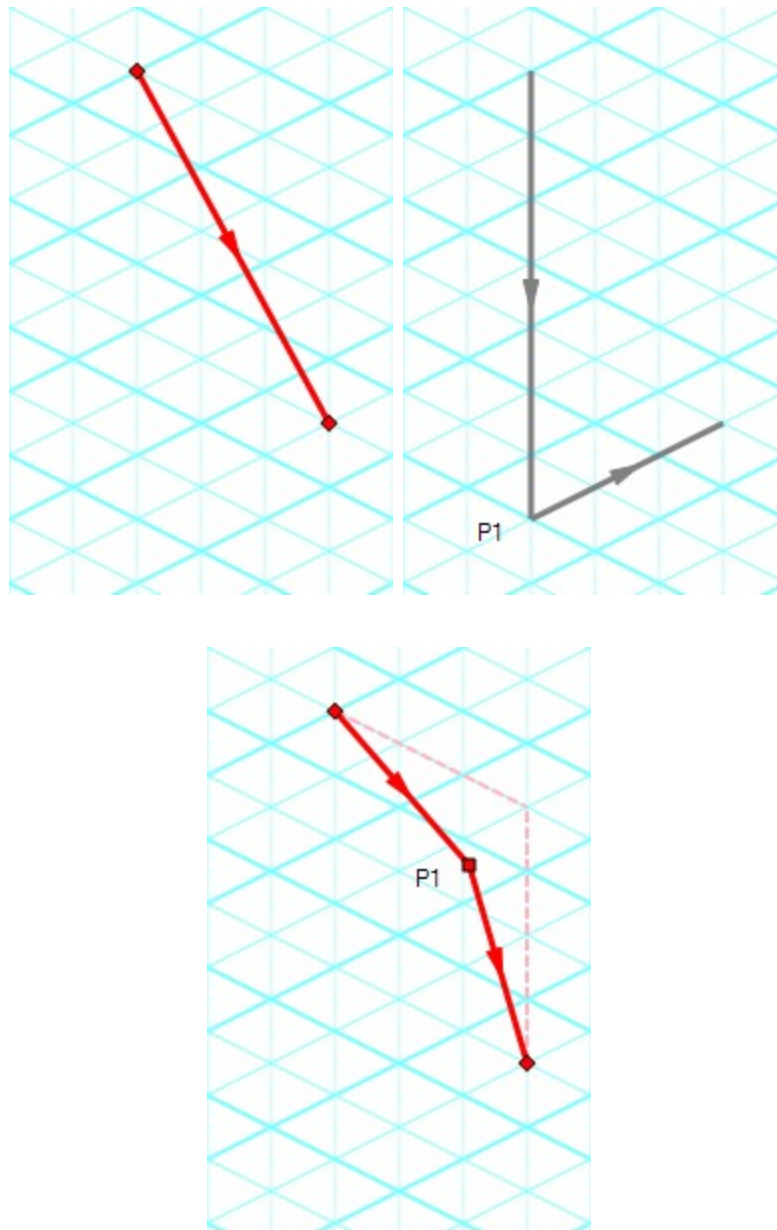


Figure 5: Automatic layout of isometric pipes

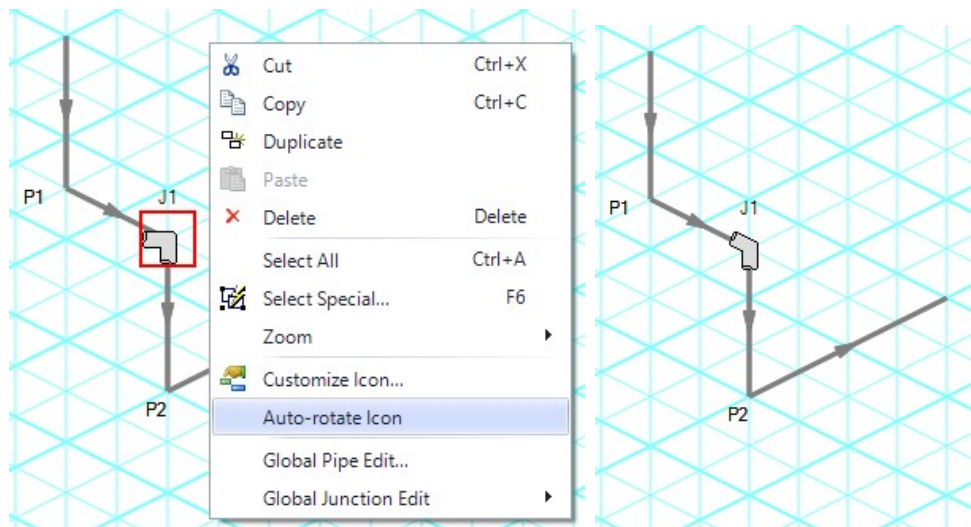
The Snap to Grid Points option will force the end points of the pipes to snap to the points where the grid lines intersect without adding segments to it. This is akin to the 2D Freeform mode while Snap to Grid is on.

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 Arrow 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.



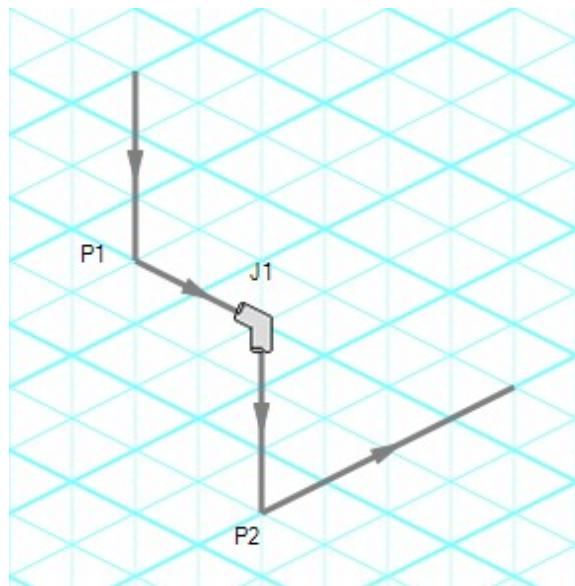
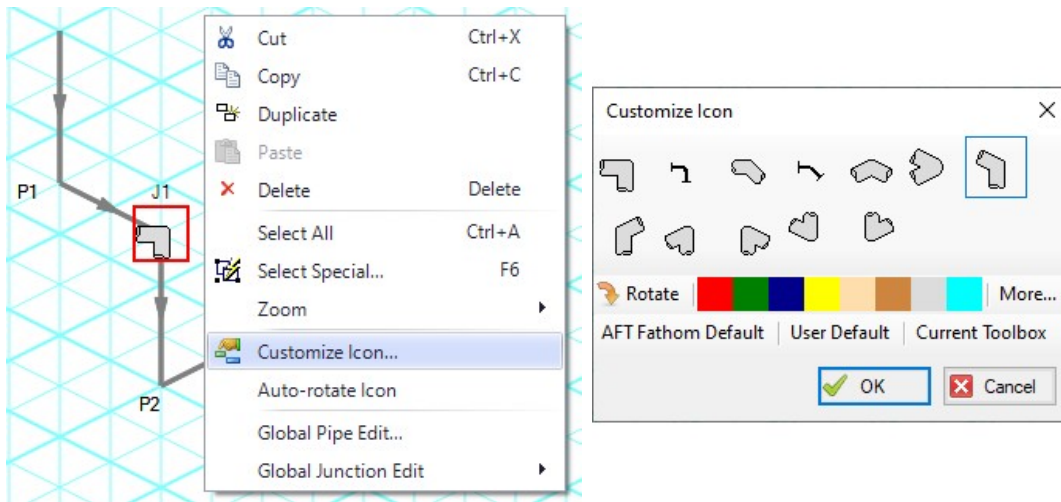


Figure 6: Icon rotations in isometric space

Related Topics

Working with Pipe and Junction Objects.....	157
Selecting and Finding Objects.....	157
Global Edit.....	169

Global Edit

AFT Arrow 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.
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 the [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 cancel 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 Topics

Global Edit	169
Global Junction Editing	170

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 pump curves and number of heat exchanger tubes.

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. Here Spray Discharge is selected.
4. Click the Select Spray Discharge 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 Spray Discharge Properties Window.
5. Enter the junction data you want to change and click the OK button in the Spray Discharge 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 (Spray Discharge junctions in this case).
8. Click the Apply Selections button.

Related Topics

Global Edit	169
Global Pipe Editing	169

Related Blogs

[Mass Modify your Model with Global Edit](#)

Toolbox

The bundle of tools at the left of the [Workspace](#) window that allow you to create or select 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.

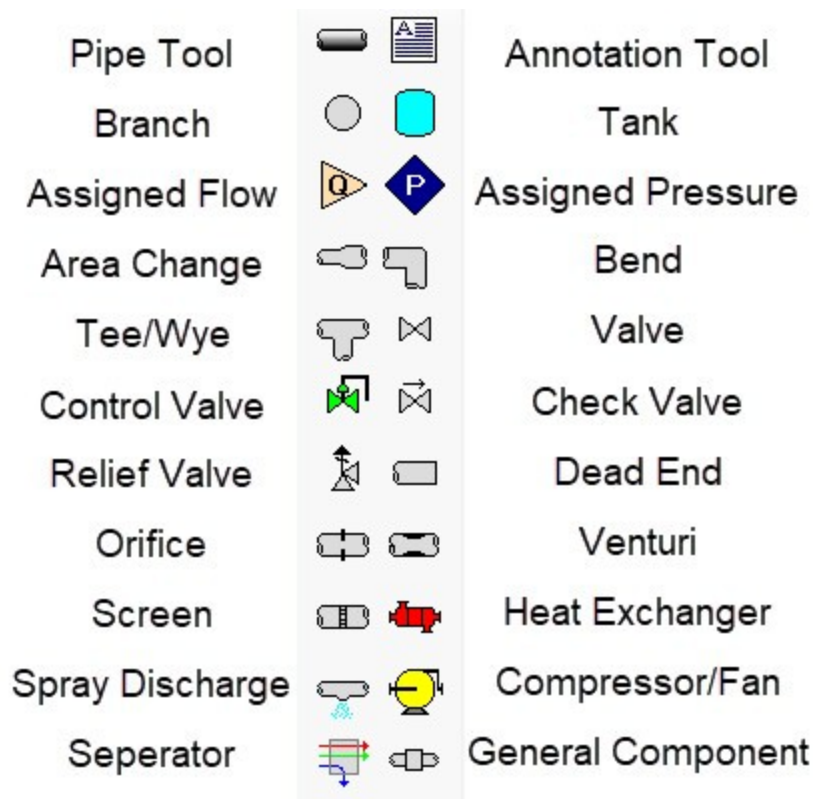


Figure 1: Toolbox items

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.

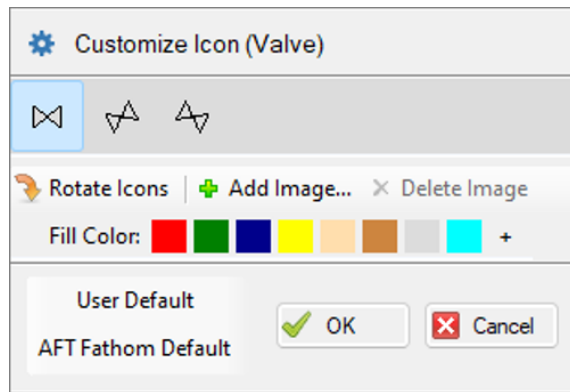


Figure 2: The Customize Icon window allows a custom image to be applied to any junction

Related Topics

Creating Pipes - the Pipe Drawing Tool.....	173
Creating Junctions.....	177
Keyboard Shortcuts for Workspace.....	350
Annotations.....	341

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 Arrow 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. An arrow near the ID number indicates the [reference flow direction](#) for the 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 does remain with the pipe in child scenarios in the [Scenario Manager](#). 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 +/- image 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

Toolbox.....	171
Creating Junctions.....	177
Keyboard Shortcuts for Workspace.....	350
Annotations.....	341
Pipes.....	189
Joining or Splitting Pipes.....	175
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185

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 ([XTS module](#)).

To join pipes and junctions in the model, select the "Join Pipes/Junctions" option from the Edit menu. The Join Pipes window will be shown with more information on pipes that can be combined in the model. The behavior of the Join Pipes/Junctions window will vary depending on if junctions are selected on the Workspace when the Join Pipes/Junctions option is selected.

Note: For maintenance releases prior to August 2022 the Join Pipes/Junctions window did not exist. Instead pipes and junctions selected in the Workspace would be joined automatically when the "Join Selected Pipes/Junctions" option was selected in the Edit menu.

- If junctions are selected on the Workspace, the Join Pipes/Junctions window will open with only the selected junctions shown in the list.
- If no junctions are selected on the Workspace, the Join Pipes/Junctions window will open with all junctions that can be combined shown in the list. Any junctions that can be combined and won't cause a loss in data at that junction will be selected automatically.

Once the Join Pipe/Junctions window is opened the selections can be modified before joining the pipes. The options below the table can be enabled/disabled in order to filter the shown junctions based on if they are selected on the workspace, if they will loose data when combined, or if they can't be combined.

If any junctions are selected that would loose data when combined, an additional button will appear near the bottom of the window that allows junctions that would have data loss to be deselected. It is recommended to select this button before joining the pipes/junctions unless you are certain the lost data is no longer required.

To join the pipes at the junctions selected in the Join Pipes/Junctions window, click the Join Selected Junction(s) button at the bottom of the window.

Splitting Pipes

An existing pipe in the workspace can be split into multiple pipes by [holding the Shift](#) key while placing a junction on the pipe. This will split the pipe in two and insert the selected junction at the specified length. Any Fittings & Losses defined in the pipe can be placed in either of the split pipe segments.

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185

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.

To add multiple of the same junctions to a model, double click the desired icon and click the workspace to add the icon to the desired location. The selected junction icon will remain engaged until you click the junction icon again or press the escape key.

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

Toolbox	171
Creating Pipes - the Pipe Drawing Tool	173
Keyboard Shortcuts for Workspace	350
Annotations	341

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 to submit to the Solver, 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.

However it is possible to represent two junctions that are directly connected, such as two components that are welded together by using a pipe with the [Model As Zero-Length Connector](#) option.

AFT Arrow will attempt to automatically connect pipes and junctions when they are moved. This eliminates unnecessary opening of the pipe and junction properties windows. If Arrow 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 Arrow. 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. 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 Arrow'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 Arrow disconnects the two objects. However, if you move a group of connected objects, their model connectivity is maintained.

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

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 the object, 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 of objects 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.

Related Topics

Pipes	189
Creating Pipes - the Pipe Drawing Tool	173
Joining or Splitting Pipes	175
Connecting Objects	177
Editing Objects	179
Defining Objects	180
Highlight Required Input	182
Inspecting Objects	183
Notes	184
Status	185
Junctions	221
Morphing Junctions	180
Moving Pipe and Junction Labels	186

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 multiple levels of undo for each editing operation. 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

Pipes	189
-------------	-----

Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175
Connecting Objects.....	177
Moving Objects.....	178
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

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 the 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

Junctions.....	221
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185
Moving Pipe and Junction Labels.....	186

Defining Objects

To specify the required property data for an object, open its Properties window. This can be done in several ways:

- Double-click the object on the Workspace
- Select the object (by clicking on it once, or using the [Selection Box](#)) and press Enter
- Select the object and click the Open Pipe/Jct Window icon on the [Workspace Toolbar](#)

-
- Double-click the left column of the appropriate table in the Model Data window (where the object ID number is shown)
 - [Jump](#) from another Properties window
 - 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, a venturi must have exactly two connecting pipes, whereas a reservoir 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 compressor/fan on the Workspace. A compressor/fan requires inlet and outlet piping. AFT Arrow reflects this by requiring each compressor/fan to be assigned an upstream pipe and a downstream pipe. Entering only the compressor/fan constants and setting an elevation is **necessary but not sufficient information to define the compressor/fan** according to AFT Arrow's requirements. Once you have entered the compressor/fan data and properly connected it, the compressor/fan 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.

Related Topics

Pipes	189
Creating Pipes - the Pipe Drawing Tool	173
Joining or Splitting Pipes	175
Connecting Objects	177
Moving Objects	178
Editing Objects	179
Highlight Required Input	182
Inspecting Objects	183
Notes	184
Status	185
Junctions	221
Morphing Junctions	180
Moving Pipe and Junction Labels	186

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 Arrow or when you are having difficulty obtaining a defined object status.

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Inspecting Objects.....	183
Notes.....	184
Status.....	185
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

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.

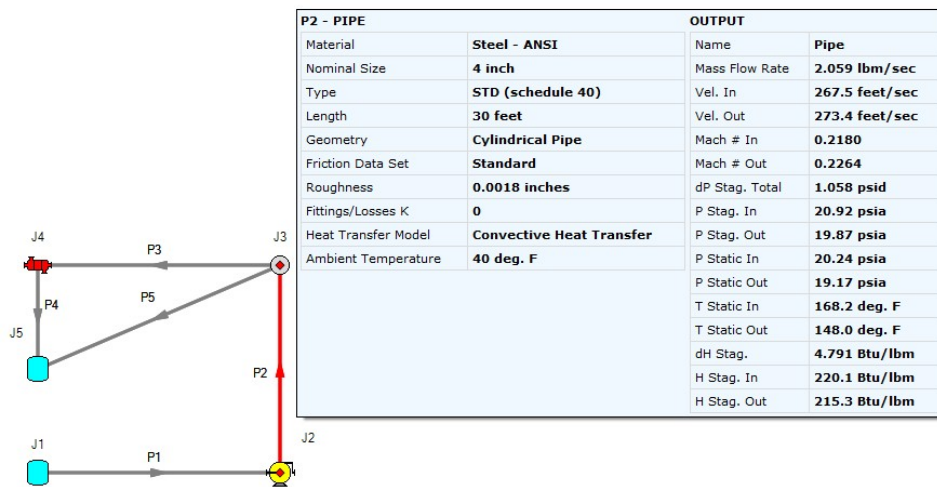


Figure 1: Holding the right mouse button on a Workspace object displays the associated data

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Notes.....	184
Status.....	185
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

Notes

Each object can have notes associated with it. This is useful for listing assumptions, reference documents, drawings, measurements, etc.

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175

Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Status.....	185
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

Status

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

Related Topics

Pipes.....	189
Creating Pipes - the Pipe Drawing Tool.....	173
Joining or Splitting Pipes.....	175
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Junctions.....	221
Morphing Junctions.....	180
Moving Pipe and Junction Labels.....	186

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 (represented with a hand icon) 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

Moving Pipe and Junction Labels.....	186
Workspace Layers.....	331

Related Blogs

[AFT Responds to 2020 Customer Survey Feedback](#)

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

Related Topics

Junctions.....	221
Connecting Objects.....	177
Moving Objects.....	178
Editing Objects.....	179

Morphing Junctions.....	180
Defining Objects.....	180
Highlight Required Input.....	182
Inspecting Objects.....	183
Notes.....	184
Status.....	185
Workspace Pan.....	185
Workspace Layers.....	331

Special Conditions

Some junctions have Special Conditions that alter the normal state of the junction. For example, a valve can be closed, a compressor/fan turned off, or a relief valve opened. Pipes can also be set closed which will effectively close the pipe to flow at both ends.

The special conditions are set in one of three ways:

- By selecting the junction(s) or pipe(s) on the [Workspace](#) and choosing Special Conditions from the [Edit menu](#)
- By selecting the junction(s) or pipe(s) on the Workspace and clicking the Special Condition icon on the [Toolbar](#)
- By opening the junction's or pipe's Properties window and clicking the appropriate condition on the Optional tab

The pipes or junctions that 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 displayed 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 [compressor/fans](#) can be turned off and either allow flow through or stop the flow.

Related Topics

Optional Input for Junctions.....	228
Optional Input for Pipes.....	214
Turn Off Parts of the Model.....	743

Pipes

This section focuses on specifying and modifying the Pipe object in the AFT Arrow 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 for AFT Arrow, and how to insert fittings and losses within the pipes.

Contents

Pipe Properties Window	189
Common Pipe Input Parameters	191
Pipe Model	192
Fittings & Losses	194
Heat Transfer in Pipes Overview	196
Insulation for Pipes	213
Optional Input for Pipes	214
Add/Remove Pipe Segment	215
Reverse Direction	215
Pressure Drop in Pipes Detailed Discussion	216

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, heat transfer conditions, and two connecting junctions. If the pipe is not purely cylindrical, additional geometry information is also required.

Related Topics

Pipes	189
Common Pipe Input Parameters	191
Pipe Model	192
Fittings & Losses	194

Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216
Pipe X length is shorter than the elevation change.....	725

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 dropdown 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 child [scenarios](#), 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.

Related Topics

Pipes	189
Pipe Properties Window.....	189
Pipe Model.....	192
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196

Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216

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.
- **Pipe Geometry** - Several pipe geometries can be modeled. The critical value for flow calculations is hydraulic diameter, based on cross-sectional area A_c and wetted perimeter P_w :

$$D_H = \frac{4 A_c}{P_w}$$

- **Cylindrical Annulus** - Flow in an annulus or "ring" pattern. This requires both an outer and inner diameter to calculate hydraulic diameter.
- **Cylindrical Pipe (default)** - A typical pipe. The hydraulic diameter and inner diameter are identical
- **Helical Tube** - Requires hydraulic diameter and coil diameter. Specification of this geometry affects the friction model pressure loss.
- **Noncylindrical Pipe** - Requires flow area (cross-sectional area) and wetted perimeter to calculate hydraulic diameter.
- **Rectangular Duct** - Requires width and height to calculate hydraulic diameter.
- **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.
- **Geometric Specification** - Several required fields that change depending on the selected Pipe Geometry. Only editable for User Specified Pipe Material.

- **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](#).

Model As Zero-length Connector can be checked to turn the pipe into a lossless connection between two junctions, such as for two components that are directly welded together. Zero-Length Connectors (ZLCs) are denoted by a red C icon next to the pipe name on the Workspace. Intermediate branching flow is not allowed between two ZLCs.

ZLCs are hidden by default in the Model Data and Output. Generally the output for ZLCs is not of interest because they will have the same inlet and outlet properties.

You can enable ZLCs to be shown in the Model Data by opening the Model Data Control, going to the Show Selected Pipes/Junctions tab, and selecting the Show Zero-Length Connectors checkbox under the list of pipes. The same checkbox is present in the Output Control on the Show Pipes/Jcts tab. You will then need to click on the pipe numbers in the list to make them visible in the Model Data or Output, or you can use the All button to select all pipes in the model.

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.
- **Weymouth equation** - A model for natural gas pipelines under fully turbulent flow, using pipe diameter to characterize pressure loss.

- **Panhandle equation** - A model for natural gas pipelines, using Reynold's Number to characterize pressure loss.

For detailed information on how these Friction Models are defined, see [Pressure Drop in Pipes - Detailed Discussion](#).

Related Topics

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216

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.

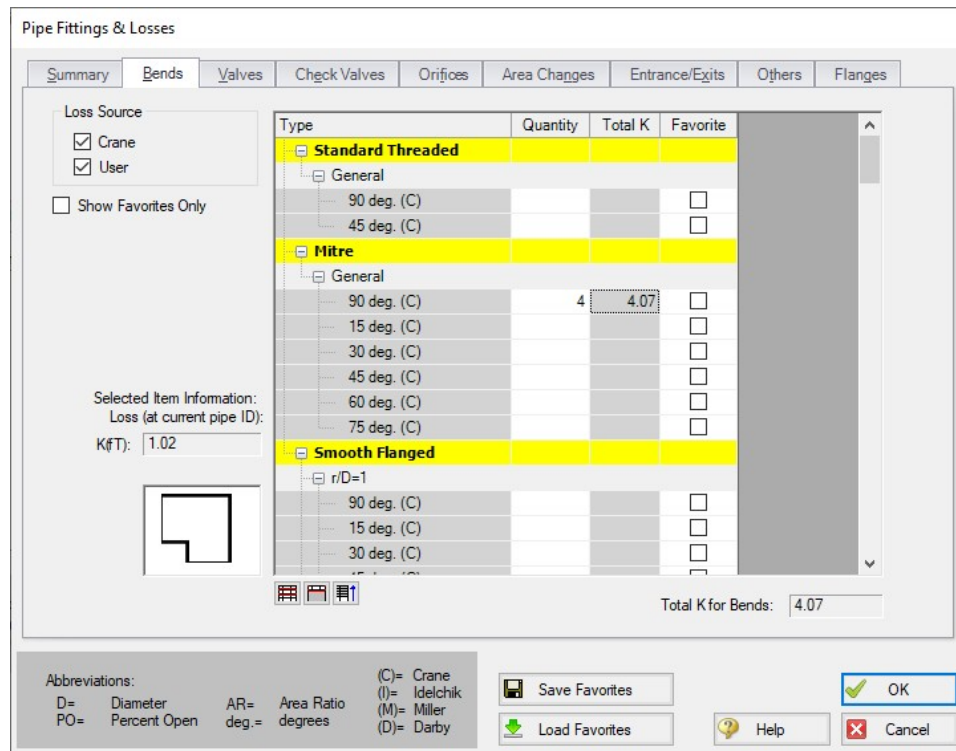


Figure 1: Pipe Fittings & Losses window

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

Pipes	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192

Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216
Fittings & Losses Library.....	624

Heat Transfer in Pipes Overview

Each pipe responds thermally to the environment outside of the pipe. AFT Arrow offers several models that allow you to model the heat transfer characteristics of a pipe:

- [Adiabatic \(Perfectly Insulated\)](#) - An adiabatic pipe is one that has perfect insulation and thus no heat transfer takes place. This model differs from Isothermal in that the temperature into the pipe is taken from the upstream junction. This is the default heat transfer model. [Insulation data](#) can be entered but is not used for thermal purposes.
- [Isothermal](#) - The Isothermal model maintains the pipe at the specified temperature. [Insulation data](#) can be entered but is not used for thermal purposes.
- [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).
- [Constant Heat Flux](#) - The constant heat flux model is useful in cases where the temperature difference between the fluid and the ambient environment is extremely large or in cases where a large radiant heat flux exists. Insulation data can be entered but is not used for thermal purposes.
- [Constant Heat Rate](#) - This model allows you to specify a constant heat rate into or out of the pipe.
- [Buried Pipe](#) - This model allows you to specify that the pipe is buried in a thermally conductive medium.

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.

Pipe	Results Diagram	Name	T Fluid Inlet (deg. F)	T Fluid Outlet (deg. F)	T Inlet Wall Inside (deg. F)	T Outlet Wall Inside (deg. F)	T Inlet Outer Wall (deg. F)	T Outlet Wall Outside (deg. F)	T Ambient (deg. F)
1	Show	Pipe	299.8	99.92	224.5	89.98	223.4	89.84	70.00

Figure 1: Thermal Data table in Output window Pipe area shows temperature distribution in pipe

Clicking the button next to Show brings up a Results Diagram for that particular pipe:

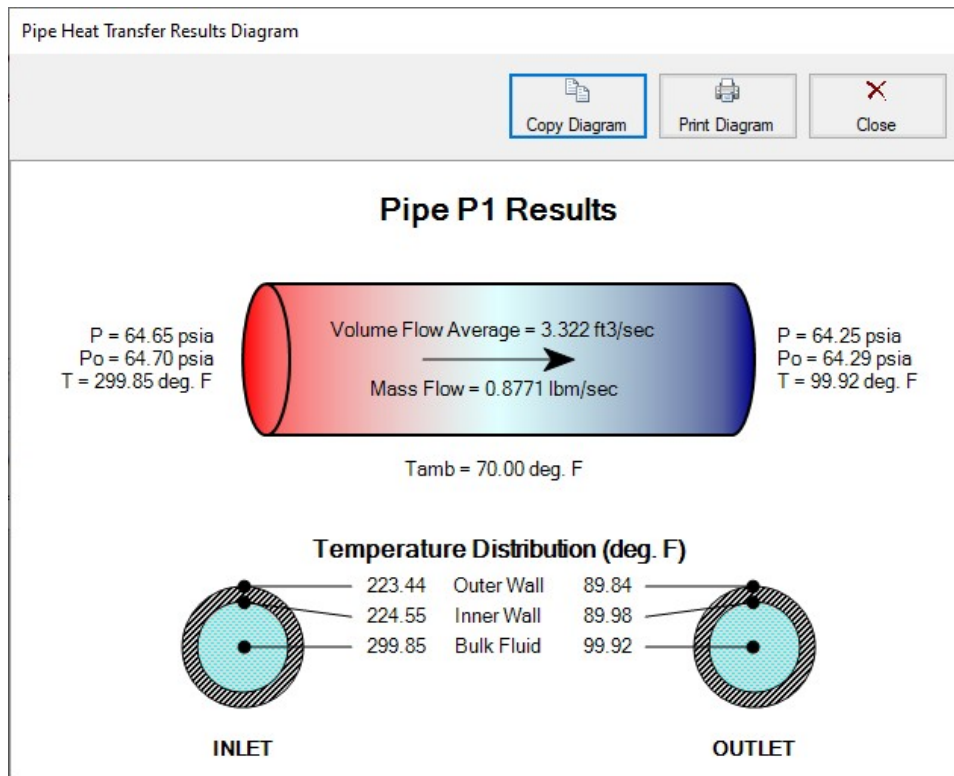


Figure 2: Pipe Heat Transfer Results Diagram

Related Topics

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192
Fittings & Losses.....	194
Insulation for Pipes.....	213
Optional Input for Pipes.....	214

Add/Remove Pipe Segment215
 Reverse Direction215
 Pressure Drop in Pipes Detailed Discussion216

Constant Heat Transfer

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

- **Adiabatic (Perfectly Insulated)** - The heat transfer between the pipe and the environment is fixed to zero. 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.
- **Isothermal** - The fluid temperature is set to a specified value. Heat transfer is still modeled in this case - instead of varying the fluid temperature, the heat rate into or out of the pipe is found that satisfies the isothermal constraint.
- **Constant Heat Flux** - A constant heat flux (heat rate per area) is applied to the pipe. Positive values indicate heat entering the pipe. Flux is often a more readily available and flexible value for some conditions, such as radiation heat transfer. For this model, the heat flux is applied to the area defined by the ID of the pipe. To model a heat flux underneath insulation, use the Convective Heat Transfer with Heat Tracing model.
- **Constant Heat Rate** - It may also be known that a certain pipe transfers (or needs to transfer) a specific heat rate, regardless of the size of the pipe. This model allows the specification of a constant energy per time into or out of the pipe.

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.

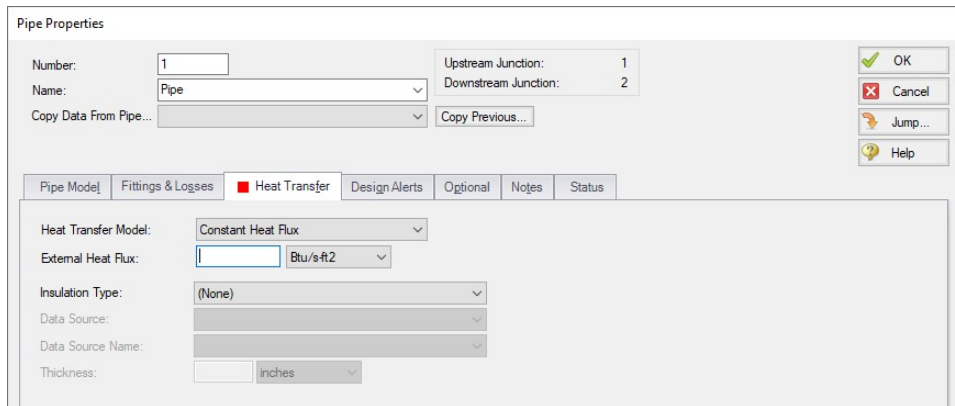


Figure 1: Constant Heat Flux

Related Topics

Heat Transfer in Pipes Overview.....	196
Convective Heat Transfer.....	199
Heat Tracing Flux Based On Wire Turns and Length.....	202
Buried Pipe Heat Transfer.....	201
Heat Transfer In Pipes - Detailed Discussion.....	206

Related Blogs

[Let the Heat Flow: Modeling Heat Transfer in Pipes in AFT Fathom and AFT Arrow](#)

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 is easy to determine [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 Arrow 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.

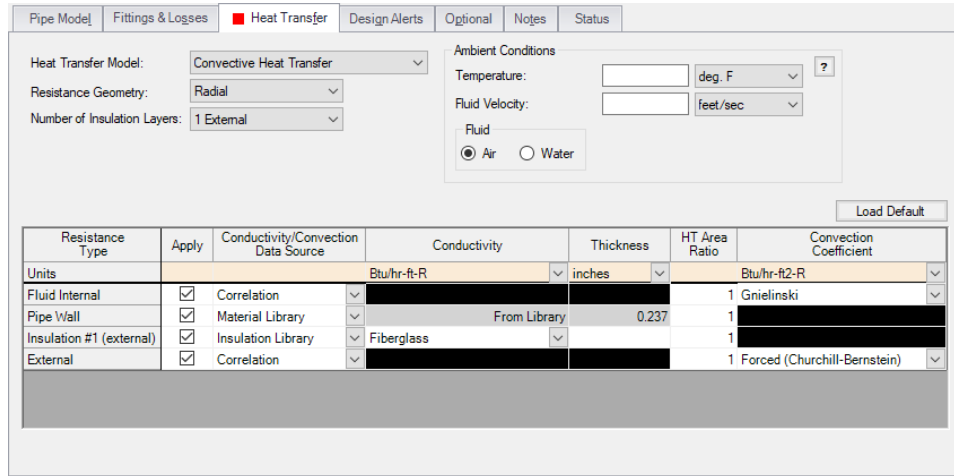


Figure 1: Convective Heat Transfer Layout

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.....	196
Constant Heat Transfer.....	198
Heat Tracing Flux Based On Wire Turns and Length.....	202
Buried Pipe Heat Transfer.....	201
Heat Transfer In Pipes - Detailed Discussion.....	206

Buried Pipe Heat Transfer

Buried Pipe Heat Transfer follows a form very similar to the [Convective Heat Transfer](#) models.

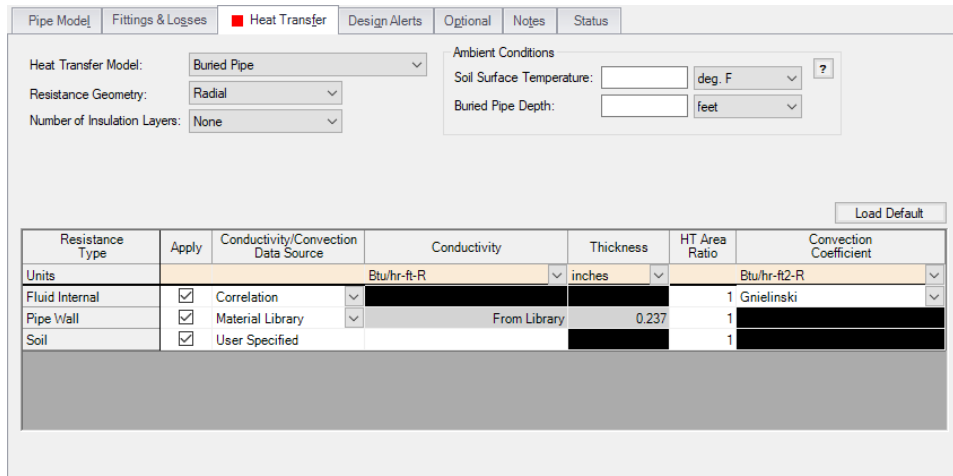


Figure 1: Buried Pipe Window

The differences between convective heat transfer and buried pipe heat transfer are:

- The Ambient Conditions are now the Soil Surface Temperature and Buried Pipe Depth. Specifically, the Soil Surface Temperature asks for the temperature of the soil at the ground surface (as opposed to the ambient air temperature), while the Buried Pipe Depth asks for the distance between the ground surface and pipe centerline.
- The External Resistance Type is replaced with Soil. This requires a User Specified value of conductivity for the soil

Related Topics

Heat Transfer in Pipes Overview..... 196
 Constant Heat Transfer..... 198
 Convective Heat Transfer.....199
 Heat Tracing Flux Based On Wire Turns and Length.....202
 Heat Transfer In Pipes - Detailed Discussion.....206

Heat Tracing Flux Based On Wire Turns and Length

The Convective Heat Transfer with Heat Tracing model requires the user enter additional information to represent the thermal effect of a heat tracing.

Arrow 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 Arrow will use to determine the heat flux as described below.

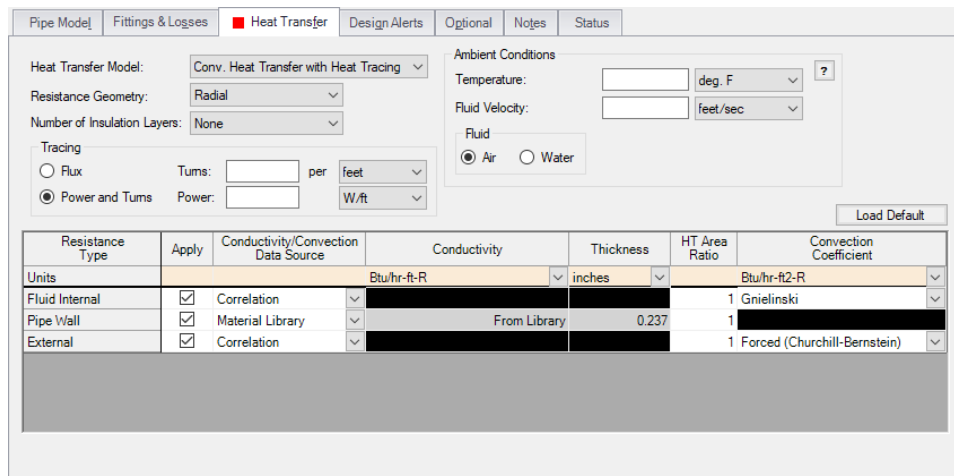


Figure 1: Pipe Properties window with the Conv. Heat Transfer with Heat Tracing model enabled

Power and Turns - Detailed Discussion

When you select the option to model Heat Tracing based off a heated wire, Arrow models that heat flux based on the theory described below.

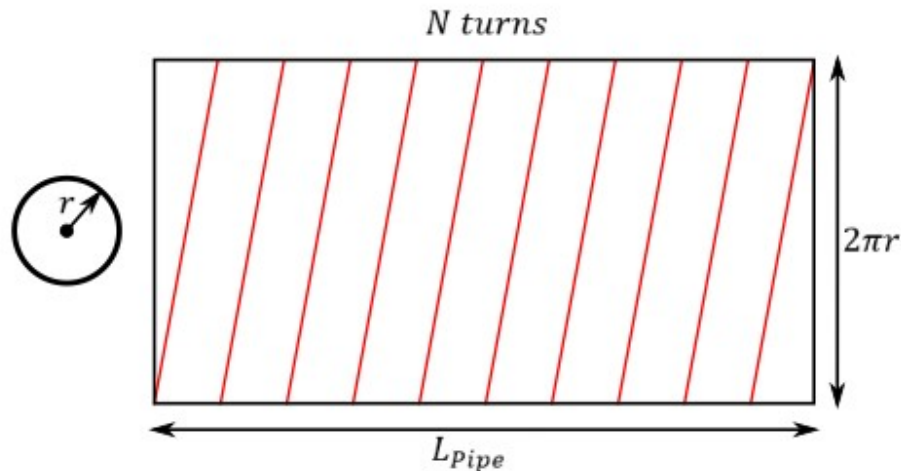


Figure 2: Wire tracing geometry

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..... 196

Constant Heat Transfer..... 198
 Convective Heat Transfer.....199
 Buried Pipe Heat Transfer.....201
 Heat Transfer In Pipes - Detailed Discussion.....206

Heat Transfer In Pipes - Detailed Discussion

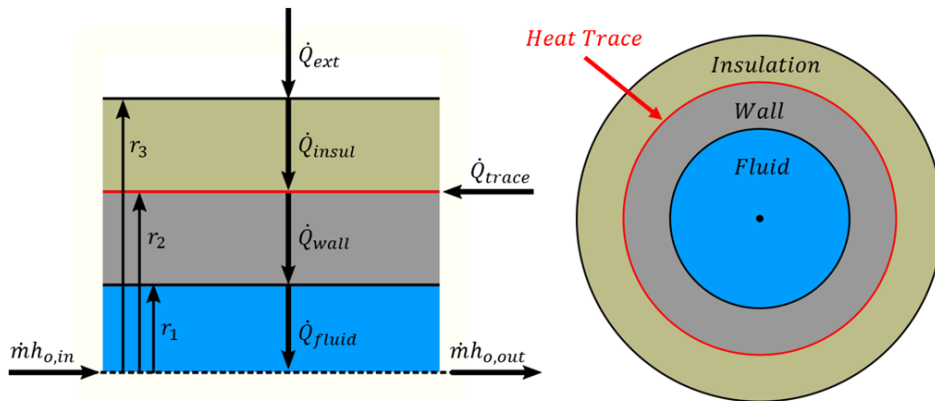


Figure 1 - Diagram of a pipe with Heat Tracing and Insulation

There are many possible combinations of heat transfer that can be defined in AFT Arrow. 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. Because this is a steady-state problem, any control volume that is drawn around sub-sections or the entire section must have entering heat equal to leaving heat.

Balancing these heat rates and temperatures via iterative methods allows AFT Arrow to come to a unique solution.

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 this 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 Arrow 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:

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

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

$$R_{convection} = \frac{1}{h A}$$

$$R_{conduction} = \frac{\Delta r}{k A}$$

$$R_{soil} = \frac{1}{2 \pi k L} \ln \left[\frac{z_{depth}}{r_{pipe}} + \sqrt{\left(\frac{z_{depth}}{r_{pipe}} \right)^2 - 1} \right]$$

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 Arrow 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 Arrow 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.

$$\overline{Nu} = 0.3 + \frac{0.62 Re^{1/2} Pr^{1/3}}{\left[1 + \left(\frac{0.4}{Pr}\right)^{2/3}\right]^{1/4}} \left[1 + \left(\frac{Re}{282,000}\right)^{5/8}\right]^{4/5} \quad \text{valid for } (Pr * Re \geq 0.2)$$

Buried Pipe Heat Transfer

[Buried Pipe heat transfer](#) is modeled by solving directly for the thermal resistance of the soil layer. AFT Arrow uses a model developed by [Eckert and Drake](#) to directly calculate the thermal resistance of soil on top of a pipe for use in the overall thermal resistance network:

$$R_{soil} = \frac{1}{2\pi k L} \ln \left[\frac{z_{depth}}{r_{pipe}} + \sqrt{\left(\frac{z_{depth}}{r_{pipe}}\right)^2 - 1} \right]$$

In this model, z is the distance between the ground surface and the pipe center-line, while r is the outside radius of the pipe.

Related Topics

Heat Transfer in Pipes Overview.....	196
Constant Heat Transfer.....	198
Convective Heat Transfer.....	199
Heat Tracing Flux Based On Wire Turns and Length.....	202
Buried Pipe Heat Transfer.....	201

Related Blogs

[Let the Heat Flow: Modeling Heat Transfer in Pipes in AFT Fathom and AFT Arrow](#)

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 Arrow 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 Topics

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216

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.

The Solver generates initial guesses on its own, which is usually sufficient to produce a converged solution. If this is not the case, user defined guesses can help obtain a solution.

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.

Design Factors

A design factor can be used to add safety margins into the hydraulic analysis. For pipes, three 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) is multiplied by the design factor assigned to the pipe.
- **Pipe Heat Transfer:** the thermal resistance for each [heat transfer layer](#) is divided by the design factor assigned to the pipe.

Special Condition

The [Special Condition](#) for the pipe can be set to None or Closed. Setting this to closed prevents flow through the pipe, like closing a valve. A symbol (default of "X") will be placed before the pipe label and the pipe will appear dashed, indicating that it is closed.

Related Topics

Special Conditions.....	187
Optional Input for Junctions.....	228
Turn Off Parts of the Model.....	743
Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192

Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Add/Remove Pipe Segment.....	215
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216
Pipe X length is shorter than the elevation change.....	725
Transfer Results to Initial Guesses.....	734
Make Initial Flow Rate Guesses for Pipes.....	741

Add/Remove Pipe Segment

Visual segments can be added to pipes.

To add or remove segments, use the Arrange > Pipe Segments submenu. These options are also available 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 [Bend](#) junction. See discussion in [Creating Pipes - the Pipe Drawing Tool](#) for more information.

Related Topics

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Reverse Direction.....	215
Pressure Drop in Pipes Detailed Discussion.....	216

Reverse 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 Arrow 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.

Related Topics

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214
Add/Remove Pipe Segment.....	215
Pressure Drop in Pipes Detailed Discussion.....	216

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 Arrow 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.

$$\Delta P = f \frac{L}{D} \frac{1}{2} \rho V^2 \quad (\textit{incompressible})$$

This model requires the calculation of a friction factor, accomplished by various methods 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.

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

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

- **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.
- - A model for natural gas pipelines under fully turbulent flow, using pipe diameter to characterize pressure loss. The full form of the Weymouth equation relates flow to pressure drop, specific gravity, pipe length, and temperature. [Crane](#) (page 1-8) presents an equivalent form using friction factor:

$$f = \frac{0.032}{D^{1/3}}$$

Note: Crane indicates that this friction factor is "identical to the Moody friction factor in the fully turbulent range for 10-inch I.D. pipe only. Weymouth friction factors are greater than Moody factors for sizes less than 20-inch, and smaller for sizes larger than 20-inch."

- **Panhandle Equation** - A model for natural gas pipelines, using Reynolds Number to characterize pressure loss. Valid for pipes 6 to 24 inch diameter, Reynolds Numbers 5E6 to 14E6 and specific gravity of 0.6. The full form relates flow to a flow efficiency factor, diameter, pressure drop, and pipe length. Similar to the Weymouth model, Crane (page 1-8) presents an equivalent form using friction factor:

$$f = 0.1225 \left(\frac{D}{Q * S.G.} \right)^{0.1461}$$

Note: Crane indicates: "In the flow range to which the Panhandle formula is limited, this results in friction factors that are lower than those obtained from either the Moody data or the Weymouth friction formula."

Frictionless

The pipe will not have any pressure drop across it. This is inherently *unrealistic* behavior, but can be useful for troubleshooting purposes.

Helical Tubes

For helical tubes the pressure drop is modified from the equivalent "straight pipe" pressure drop via the following relationship ([Ito 1959](#)):

$$\Delta P_{helical} = \Delta P_{straight} \left[Re \left(\frac{D_{pipe}}{2 r_{coil}} \right)^2 \right]^{0.05}$$

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

Pipes.....	189
Pipe Properties Window.....	189
Common Pipe Input Parameters.....	191
Pipe Model.....	192
Fittings & Losses.....	194
Heat Transfer in Pipes Overview.....	196
Insulation for Pipes.....	213
Optional Input for Pipes.....	214

Add/Remove Pipe Segment215
Reverse Direction215

Junctions

This section focuses on the Junctions in the AFT Arrow workspace as well as the theories that govern the junctions. Topics included discuss the types of junctions that AFT Arrow provides, the required inputs for each junction, and theory topics that indicate how the inputs to the junction are used.

Contents

General Junction Information	222
Junction Loss Models	233
Area Change	239
Assigned Flow	241
Assigned Pressure	242
Bend	243
Branch	247
Check Valve	248
Control Valve	249
Compressors & Fans	253
Dead End	277
General Component	278
Heat Exchanger	278
Orifice	283
Relief Valve	287
Screen	290
Separator	292

Spray Discharge	293
Tank	295
Tee/Wye	295
Valve	299
Venturi	302

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 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 Arrow does not prevent negative flow through junctions. Should this occur, AFT Arrow 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. Should negative flow occur in any junction, AFT Arrow will give a caution in the output.

Table 1: Comparison of junction capabilities

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		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
Separator	2	Yes		Yes	1
Spray Discharge	1-4	Yes	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

Two Window Formats	224
Common Junction Input Parameters	226
Optional Input for Junctions	228
Specifying Losses	229
Conventions for Specifying Basis Area	230
Reference Density	230

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 Arrow 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/Fan junction](#) which will add pressure to the system in the direction of positive flow through the connecting pipes. The [Compressor/Fan junction](#) interprets where to add the pressure based on the directions of the connecting pipes.

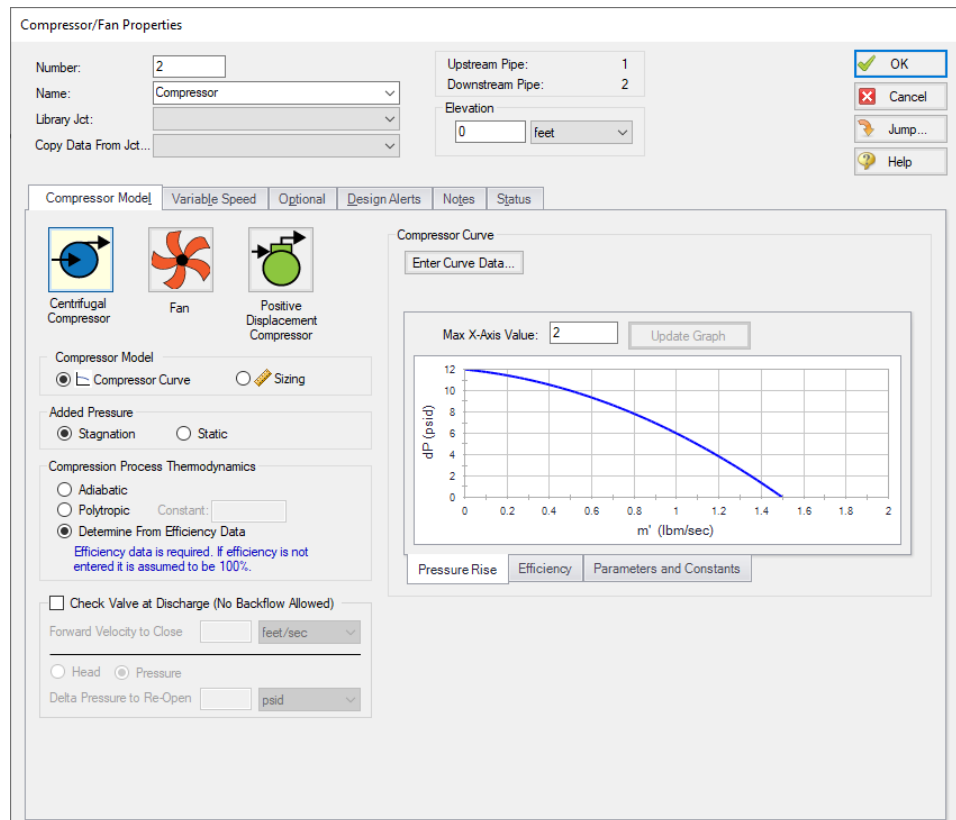


Figure 1: Properties window format #1 (fixed connecting pipe display)

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 twenty-five connecting pipes; except for the [Tee/Wye junction](#), which allows only three.

Because the number of pipes connected to a junction may vary, the second basic format uses a table sized according to the number of connecting pipes.

An example of this second window format is the [Branch Properties](#) window, shown in Figure 2, which has three connecting pipes. To determine AFT Arrow's interpretation of the model connectivity for this second window format, you can review the contents of the table areas.

K-factors in the first column will be applied for flow into the pipe and out of the junction, while K-factors in the second column will be applied for flow out of the pipe and into the junction.

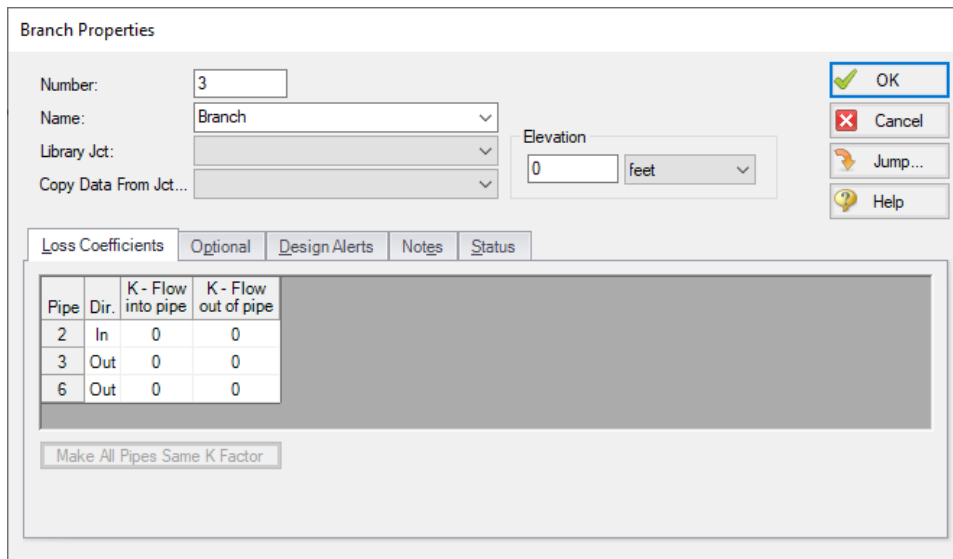


Figure 2: Properties window format #2 (pipes displayed in a table)

Related Topics

General Junction Information.....	222
Common Junction Input Parameters.....	226
Optional Input for Junctions.....	228
Specifying Losses.....	229
Conventions for Specifying Basis Area.....	230
Reference Density.....	230

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 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 Junction

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 User Options](#), automatically setting new junctions to the specified elevation.

Note: Junction elevation takes on a modified meaning for rotating systems. In rotating systems, the elevation is the distance from the centerline of rotation. This distance is used with the rotational velocity specified in the [Environmental Properties panel](#) to determine rotational acceleration for the pipe.

Jump Feature

Quickly move to the next junction. See the [Finding Objects](#) topic.

Related Topics

General Junction Information.....	222
Two Window Formats.....	224
Optional Input for Junctions.....	228
Specifying Losses.....	229
Conventions for Specifying Basis Area.....	230
Reference Density.....	230
Pipe X length is shorter than the elevation change.....	725

Optional Input for Junctions

Initial Guesses

A user defined initial guess can be entered for pressure (or head) and temperature for most junctions. This is only an initial guess for the Solver - the solution **will** vary from this value, and the value has **no effect** on the end solution.

A good initial guess can be helpful to decrease the number of Solver iterations required, and potentially help with convergence issues.

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.

Design Factor

A design factor can be used to add safety margins into the analysis. For junctions, three types of design factors exist:

- **Compressor/Fan Junctions:** head rise at the compressor/fan is multiplied by the design factor assigned to the compressor/fan.
- **Heat Exchanger Junctions:** total rate of heat transfer is multiplied by the Heat Transfer design factor assigned to the heat exchanger. The total loss factor is multiplied by the Friction Loss design factor assigned to the heat exchanger.
- **All other junctions:** total loss factor is multiplied by the design factor assigned to the junction.

Note: The design factor for compressors/fans has the opposite effect on a system that the design factor for other junction types has. A design factor of 1.1 for junctions with a pressure loss will give an extra 10% pressure drop at that junction, adding resistance to the system. A design factor of 1.1 for the compressor/fan junction will give an extra 10% pressure rise, effectively modeling the compressor/fan as having extra capacity than expected. To add a 10% safety margin to the compressor junction, a design factor of 0.9 should be used.

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

Special Conditions.....	187
Optional Input for Pipes.....	214
Turn Off Parts of the Model.....	743
General Junction Information.....	222
Two Window Formats.....	224
Common Junction Input Parameters.....	226
Specifying Losses.....	229
Conventions for Specifying Basis Area.....	230
Reference Density.....	230
Transfer Results to Initial Guesses.....	734
Make Initial Flow Rate Guesses for Pipes.....	741

Specifying Losses

Local losses can be specified in two ways. The most flexible way of including a loss factor in an AFT Arrow 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

General Junction Information.....	222
Two Window Formats.....	224
Common Junction Input Parameters.....	226
Optional Input for Junctions.....	228
Conventions for Specifying Basis Area.....	230
Reference Density.....	230

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 Arrow 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

General Junction Information.....	222
Two Window Formats.....	224
Common Junction Input Parameters.....	226
Optional Input for Junctions.....	228
Specifying Losses.....	229
Reference Density.....	230

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.

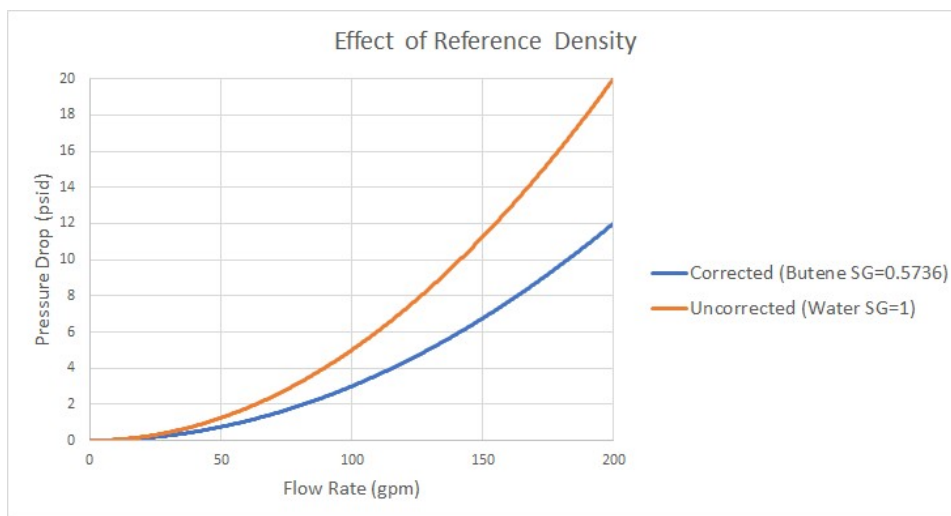


Figure 1: Visual comparison of Pressure Loss Curves for differing densities. Head curves would be identical for this component.

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.

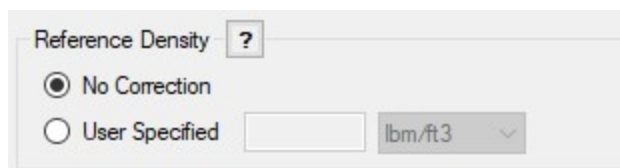


Figure 2: Reference Density Selection

There are two options:

- **No Correction (default)** - This selection assumes that the component was tested with the fluid defined in the [Fluid panel](#).
- **User Specified** - This directly specifies the reference density. If the fluid in the Fluid panel differs, the entered curve will be corrected.

Related Topics

General Junction Information.....	222
Two Window Formats.....	224
Common Junction Input Parameters.....	226
Optional Input for Junctions.....	228
Specifying Losses.....	229
Conventions for Specifying Basis Area.....	230

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:

$$C_v = \frac{e^x}{T} C_{v,max}$$

$$x = \ln(\tau)H$$

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

Junction Loss Models

There are several common loss models offered to represent the frictional pressure loss at junctions in Arrow. See the topics below for information on common subsonic loss models. Loss models that are unique to individual junctions are described in separate topics, refer to the specific junction properties window topics for more information.

If sonic choking occurs a different sonic loss calculation method is used. See below for a summary of the differences.

Subsonic vs Sonic 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, discharge coefficient for subsonic flow, or one of the other loss models discussed in the [Junctions](#) section.
- **Sonic** losses result when [sonic choking](#) occurs. Sonic losses are dependent on the minimum effective flow area and are unrelated to the K-factor. This is the C_dA for sonic choking.

The two loss mechanisms cannot occur simultaneously in nature or in AFT Arrow. If one mechanism occurs during the solution, the input data for the other mechanism is ignored by AFT Arrow.

It is also important to note that sonic choking at a restriction cannot be calculated if the C_dA for Sonic Choking is not entered. Thus, Arrow will always use subsonic loss calculations if no C_dA for Sonic Choking has been entered, even if the actual system would experience choked flow at that restriction.

Note: The C_dA for sonic choking may be different from the subsonic C_dA loss model option in Arrow. 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

K Factor Loss Model.....	234
Cv/Kv and Xt Loss Model.....	235
Resistance Curve Loss Model.....	237

K Factor Loss Model

AFT Arrow models component losses according to the following equation:

$$\Delta P_{loss} = K \frac{1}{2} \rho V^2$$

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 loss models used in AFT Arrow. 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
Bend	Crane 1988, A-29 Miller 1990, 207-209
Area Change	Crane 1988, A-26; Idelchick 2007, 246 & 254
Orifice	Idelchick 2007, 258-259 & 262
Screen	Idelchick 2007, 598
Tee/Wye	Idelchick 2007, Chapter 7; Miller 1990, 315-316, 323
Valve	Ingersoll-Rand 1970; Crane 1988, A-29

[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 all 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 hydraulic 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 ([Miller 1990](#)) is another reference offering good general purpose hydraulic data.

K Factor Data 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.

Arrow 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.

Related Topics

Junction Loss Models.....	233
Cv/Kv and Xt Loss Model.....	235
Resistance Curve Loss Model.....	237

Cv/Kv and Xt Loss Model

AFT Arrow allows the user to enter valve 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 sub-sonic 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:

$$Y = 1 - \frac{X_{sizing}}{3X_T}$$

Where X_T is the pressure ratio, X , when the valve has just begun to choke.

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 or Kv in Output

The Output window can display an estimated C_v or K_v when it was not explicitly defined by the user so long as the appropriate parameter is added in the [Output Control](#). The estimated C_v or K_v value is calculated using the mass flow rate equation shown above, with X_{sizing} estimated as follows:

$$X_{sizing} = \frac{\gamma_1}{1.4} \frac{\Delta P}{P_1}$$

Related Topics

Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Valve.....	299
Valve Modeling Closures with K Factors.....	302
Sonic Choking.....	665

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 with the [XTS Module](#).

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

Junction Loss Models.....	233
K Factor Loss Model.....	234
Cv/Kv and Xt Loss Model.....	235
Troubleshooting Resistance Curves.....	736

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.

If the standard loss factors provided are inadequate for your application, you may specify your own custom loss factor referenced to the upstream pipe flow area.

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. That is the purpose of the optional C_dA input parameter. The minimum possible sonic flow area is entered as the C_dA and if sonic conditions are reached AFT Arrow will solve the system on a [choked flow](#) basis.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Area Change Loss Model.....	239
Subsonic vs. Sonic Pressure Losses.....	665

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](#), A-26) 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, A-26) 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

K Factor Loss Model.....	234
Area Change.....	239
Subsonic vs. Sonic Pressure Losses.....	665

Assigned Flow

The Assigned Flow junction type allows you to connect one pipe and to input a known flow rate entering or leaving at a particular location in your system. An irrecoverable loss can also be included.

The Assigned Flow Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipe information in a fixed format. Depending on whether you specify the junction as an inflow or outflow type, the required connecting pipe (upstream/inlet or downstream/outlet) will be enabled while the other pipe is disabled. Note that if the flow is specified as Inflow (to the system) then the reference positive flow direction of the connected pipe must be away from the junction (i.e., the junction is upstream of the pipe).

The Assigned Flow junction will automatically switch the inflow/outflow parameter based on the reference flow direction of the connected pipe.

The Assigned Flow junction type allows you to specify positive flow rates as mass flow rates. Negative flow rates are not accepted. 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 flow direction to be consistent with the actual direction and change from an outflow type to an inflow type.

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](#).

Loss Factor

On the Optional tab, a loss factor can be entered to account for losses due to fittings or components at the entry/exit to the connected pipe.

Transient

[\(XTS Module Only\)](#) On the Transient tab, a flowrate transient can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233

Assigned Pressure

The Assigned Pressure junction allows for the specification of a fixed boundary pressure and temperature.

This junction is very similar to the [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.

There is also the option to Balance Energy in the junction - this adjusts the temperature in order to make incoming and outgoing energy balance. This option is only appropriate for closed loop systems, read more here: [Open vs Closed Systems - Balancing Energy](#).

CdA for Sonic Choking

An optional input in the Assigned Pressure window is the CdA. This parameter describes the effective area restriction for the purpose of calculating [sonic choking](#). This is useful if you are using the assigned pressure junction as a vent to atmosphere and a restriction exists at the pipe exit. If sonic choking occurs at the vent exit, the CdA will be the limiting area for flow. The CdA will only apply when there is one connecting pipe.

Loss Factor

On the Loss Coefficients tab, loss factors can be entered to account for losses due to fittings or components at the entry/exit to each of the connected pipes.

Transient

[\(XTS Module Only\)](#) On the Transient tab, a pressure transient can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
Subsonic vs. Sonic Pressure Losses.....	665

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. That is the purpose of the optional C_dA input parameter. The minimum possible flow area is entered as the C_dA and if sonic conditions are reached AFT Arrow will solve the system on a choked flow basis.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Bend Loss Model.....	244
Subsonic vs. Sonic Pressure Losses.....	665

Bend Loss Model

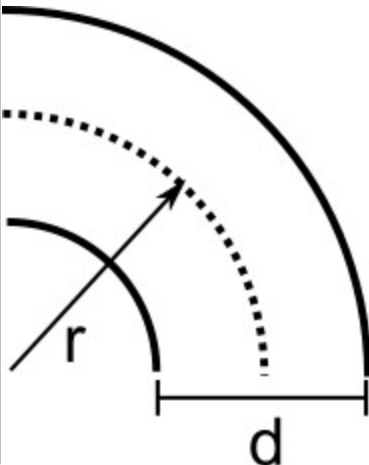
90 degree bends

The three [Bend](#) loss correlations are all for turbulent Reynolds numbers ([Crane 1988](#), A-29).

The K-factors for a **smooth flanged bend** with $r/D \geq 1$ are provided in Table 1.

Table 1: K-factors for 90 degree smooth flanged bends

r/d	K
1	$20 f_T$
1.5	$14 f_T$
2	$12 f_T$
3	$12 f_T$
4	$14 f_T$
6	$17 f_T$
8	$24 f_T$
10	$30 f_T$
12	$34 f_T$
14	$38 f_T$
16	$42 f_T$
20	$50 f_T$



The diagram shows a 90-degree arc of a pipe with a smooth flange. The radius of the bend is labeled 'r' and the diameter of the pipe is labeled 'd'. A solid line represents the outer edge of the bend, and a dotted line represents the inner edge. An arrow points from the center of the bend to the outer edge, indicating the radius 'r'.

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:

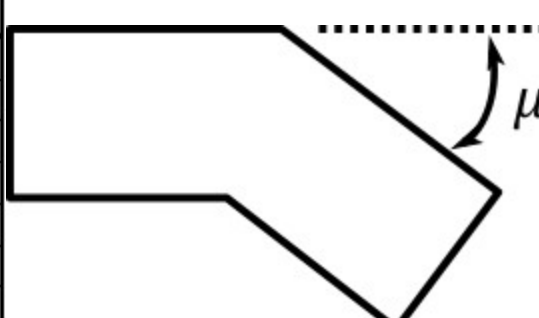
$$K_B = (n - 1) \left(0.25 \pi f_T \frac{r}{D} + 0.5 K \right) + K$$

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

K Factor Loss Model.....	234
Bend.....	243
Subsonic vs. Sonic Pressure Losses.....	665

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. Other significant features are the ability to impose a flow source or sink at the junction and the ability to specify separate loss factors for each connecting pipe. The Branch junction must have at least two connecting pipes.

The Branch Properties Window follows the second of the two basic [Properties window formats](#).

A table on the Loss Coefficients folder tab displays the connecting pipe information. This table grows in size to accommodate up to twenty-five connecting pipes. For each connecting pipe, the pipe table lists the reference flow direction and up to two loss factors. The first loss factor is for physical flow out of the branch and into the pipe. The second is for physical flow into the branch and out of the pipe. The loss factors can be specified independently or left as zero.

The Branch junction also offers you the option of specifying a flow source or sink at the junction in the Optional tab. According to [AFT Arrow's convention](#), a source of flow into the system is defined as positive, while a consumption of flow with flow out of the system (a sink) is defined as negative. Along with the flow rate you must specify the fluid and temperature or enthalpy. A branch junction can be used to model a desuperheater using the enthalpy input.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234

Check Valve

The Check Valve junction type requires two connecting pipes. This junction type allows you to model valves that prevent reverse 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.

AFT Arrow assumes that the check valve is initially open. If the flow solution indicates that forward flow will not occur, AFT Arrow closes the valve and solves the flow distribution again.

Note: If the check valve closes during the simulation, it will not re-open. If there is an unsteady situation in which the check valve would continuously open and close, there may be unexpected results. It is good practice to close check valves with a Special Condition if it is known that they will be closed. This event can be seen in the Solution Progress window as **Re-running Solver because one or more special junction states changed...**

A check valve [Special Condition](#) can be either open or closed.

Specifying Losses

You specify the losses for the Check Valve on the Loss Model tab in the Properties window. The Check Valve junction allows the C_v or K_v to be specified for pressure loss calculation. The C_v/K_v is defined according to the ANSI/ISA standard as described in the [Kv and Cv Loss Model](#) topic. The valve X_t data is also required when using C_v/K_v . When C_v/K_v is selected, data for FL and FD can optionally be entered for valves with low Reynolds numbers to calculate correction factors. If the valve diameter is entered, AFT Arrow will automatically calculate correction factors for cases where pipe diameter differs from the valve.

When you select Resistance Curve, the Check Valve Properties window makes additional features available. Using these new features you can input Check Valve 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.

CdA for Sonic Choking

An optional input in the Check Valve Properties window is the C_dA . 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 from test data. Note that the C_dA input field is disabled if C_v or K_v loss is used, as the X_t value is used for sonic choking calculations in that case.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Check Valve closed at Junction X.....	723
Cv/Kv and Xt Loss Model.....	235
Subsonic vs. Sonic Pressure Losses.....	665

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

You can model four types of control valves: Pressure Reducing Valves (PRV's), Pressure Sustaining Valves (PSV's), Flow Control Valves (FCV's), and Pressure Drop Control Valves (PDCV's). 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. You can, however, specify the full open loss. This 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 never allowed to not 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.

Action If Setpoint Not Achievable

- **Always Control (Never Fail)** - It is possible that a control valve cannot achieve a given set point. By default this option is on - to change this, uncheck the box and click Set as Default. For additional information, see [Control Valve Setpoint Not Achievable](#).
- **Use Default Actions** - The behavior of the control valve when it cannot meet its setpoint can be modified by unchecking this option. The default behaviors represent the normal behaviors for the standard valve types.

Special Conditions

Control Valves have two [Special Conditions](#). To open fully and not control, or to close.

Open Percentage Table

The Optional tab allows data to be entered for special control valve characteristics. Specifically, the valve C_v/K_v , x_t , and Flow Area can be specified vs. the Open Percentage of the valve. This data does not affect AFT Arrow's flow solution, but will affect the reported C_v/K_v . 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.

If open percentage data is not specified, then a C_v/K_v value will be estimated for the control valve.

Transient

[\(XTS Module Only\)](#) On the Transient tab, a control setpoint transient can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234

Control Valve Setpoint Not Achievable.....	252
FCV at Junction X could not control flow.....	716
FCV/PRV/PSV Could Not Control Pressure - Pressure Was Added to Achieve Control Flow/Pressure.....	710
PRV at Junction X could not control pressure.....	721
PSV at Junction X could not control pressure.....	721
Cv/Kv and Xt Loss Model.....	235
Control valve open percentage outside valid range of 0-100%.....	715
Subsonic vs. Sonic Pressure Losses.....	665

Control Valve Setpoint Not Achievable

It is possible that the control valve cannot achieve the specified control setpoint. There are several actions the valve can take in such cases. By default, the Always Control (Never Fail) feature is selected. This option is physically unrealistic but useful for modeling purposes. The reason this option is physically unrealistic is that the valve can add pressure to maintain control, thus acting like a compressor. If this occurs, the user is clearly warned in the output. The Always Control option is useful because it gives a clear indication to the user of how far away the valve is from being able to control.

Flow Control Valves

For flow control valves, other options for valves which cannot achieve control are to have the valve Open Fully or Close. By default the valve fully opens. Once the control valve loses control, whether open or closed, AFT Arrow will solve the model again.

Pressure Control Valves

Pressure control valves are more complicated. These valves have two potential actions when control cannot be achieved. The first action is when the control cannot be achieved due to insufficient upstream pressure. The second is due to excessive downstream pressure. The default behavior for pressure reducing valves is to Open Fully due to insufficient upstream pressure, and close due to excessive downstream pressure. The defaults for pressure sustaining valves are the opposite. You can override the default behavior by clearing the Use Default Actions selection and specifying the desired actions.

AFT Arrow will determine if the valve can maintain control based on the Loss When Fully Open value. If this fully-open pressure drop is greater than the pressure drop which balances the system, the control valve can't achieve its setpoint. In other words, the control valve can't open wide enough (becoming less restrictive) to achieve control, and will revert to using the Loss When Fully Open specified by the user. If you enter data in the Open Percentage table on the Optional tab, you can select to have the data taken directly from the table for the 100% open case. You can also choose to not have any losses associated with the valve in the full-open state by selecting None.

Related Topics

Control Valve.....	249
FCV at Junction X could not control flow.....	716
FCV/PRV/PSV Could Not Control Pressure - Pressure Was Added to Achieve Control Flow/Pressure.....	710
PRV at Junction X could not control pressure.....	721
PSV at Junction X could not control pressure.....	721
Cv/Kv and Xt Loss Model.....	235

Compressors & Fans

The Compressor/Fan junction allows you to model the pressure added to a system by a compressor or fan. Because all turbomachines are different, AFT Arrow does not provide any standard types. Pressure and flow data that describes the turbomachine must be obtained from the manufacturer or from test data. To reduce repeated data entry, frequently-used compressors/fans can be added to the [Component 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, they can have Design Alerts and Notes applied to them. They require two connecting pipes. Note that the direction of the connected pipes defines what side of the junction is suction and which is discharge.

Compressor/Fan Model

This tab requires the selection and definition of what the junction represents. There are three models available, which are shown in Figure 1.

- **Centrifugal** - Represents a typical centrifugal compressor in which a pressure rise is achieved by adding kinetic energy to the superheated gas flow. Arrow can help to size a compressor using the [Sizing](#) option. Once a compressor curve is attained, the compressor can be modeled by entering the curve into the [Compressor Curve](#) under the Compressor Model section.
- **Positive Displacement** - Represents a steady state approximation of a positive displacement device. The flow is fixed to a constant value, and whatever pressure is necessary to obtain that flow will be supplied.
- **Fan** - Represents a typical fan in which a large amount of gas is generally moved with a lower increase in pressure than a compressor. As with the centrifugal compressor, Arrow can help to size a fan using the [Sizing](#) option. Once a fan curve is attained, the fan can be modeled by entering the curve into the [Fan Curve](#) under the Fan Model section. The Fan model will always be treated as adiabatic. For non-adiabatic behavior the centrifugal compressor option should be chosen.



Figure 1: Compressor/Fan junction available models

Variable Speed

This tab allows the speed of the turbomachine to vary either by a fixed amount, or dynamically. These options are only available when a compressor or fan is specified with a curve.

- **Fixed Speed (%)** - Directly modifies the fan or compressor curve according to the [Affinity Laws](#). The entered curve is always assumed to be the 100% speed curve.
- **Controlled Compressor/Fan (Variable Speed)** - Dynamically change the speed of the turbomachine to meet a specified control setpoint. Instead of allowing the flow or pressure to vary with a fixed speed, the speed of the compressor/fan (and thus Compressor/Fan Curve, according to the Affinity Laws) varies to meet a fixed flow or pressure. If modeling a suction or discharge pressure, control can optionally be enforced only if the value is above/below the setpoint.

Compression Process Thermodynamics

Depending on the compression process, significant thermal effects can occur. Fans create a pressure rise that is typically so low that the temperature is not noticeably affected, and this is why the fan will always be treated as adiabatic in Arrow. The large pressure rise in compressors, however, leads to significant temperature changes. See the [Centrifugal Compressor](#) topic for details on the available methods for calculating this.

Optional

There are several optional parameters that can be defined for a compressor/fan. Some of these are available to most junctions such as Initial Guesses, Display on Workspace, Design Factor, and Workspace Icon.

Also present are several compressor/fan junction specific options:

- **Special Condition**
 - **None** - The compressor/fan operates normally.
 - **Compressor/Fan Off No Flow** - The compressor/fan is shut off and blocked. This is identical to a closed valve.
 - **Compressor/Fan Off With Flow Through** - The compressor/fan is shut off but allows flow to continue through. This is identical to a lossless connection.
- **Number of Compressor/Fans at This Location** - Several compressor/fans can be represented by one Compressor/Fan junction. This simplifies the model by reducing the number of pipes and junctions present, as well as automatically calculating the resulting composite Compressor/Fan curve. When this option is selected the compressor summary will show the total flow/pressure added from the combined compressors/fans.
- **Additional [Efficiency Data](#)** - Power and efficiency information for each stage in a VFD-Motor-Compressor/Fan assembly.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276
Junction X Had Reverse Flow - Compressor/Fan Pressure Could Not Be Predicted.....	711
Junction X Exceeded Runout Flow Rate.....	718
Junction X Compressor/Fan Added Negative Pressure.....	718
Poor Compressor/Fan Curve Fits.....	735

Related Blogs

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

Centrifugal Compressor

Analysis Type

This topic pertains to the Centrifugal Compressor Curve Analysis option which represents the real behavior of a centrifugal compressor or fan. For Sizing Analyses, see [Compressor/Fan Sizing](#).

Centrifugal Compressor Curve

A single compressor curve is useful if the compressor will operate at a fixed speed. The right side of the Compressor Model tab displays the defined curve(s) for the compressor.

To enter curves, click [Enter Curve Data](#). Once this data has been defined, previews of the applied curves can be seen on the right side of the Compressor Model tab. Note that only one curve can be entered unless different configurations under the [Multiple Configurations](#) tab have been specified.

Compressor Map

A Compressor Map is a family of compressor curves categorized by different operating speeds and is typically provided by a manufacturer in the form of a graph or data sheet. These curves can be input into AFT Arrow which uses an extension of the [Multiple Configurations](#) feature. The Solver will use the Rated Speed entered by the user to interpolate between the available curves and determine the actual operating point.

When to Use a Compressor Map

- The working fluid of the AFT Arrow model is the same as the fluid used by the manufacturer when developing the Compressor Map.
- Manufacturer data is provided in units of non-dimensional or corrected mass flowrate vs pressure ratio.
- The AFT Arrow model needs to account for inlet temperatures and pressures that differ from the manufacturer test conditions.
- Various scenarios that use different operating conditions or speeds need to be analyzed. The Compressor Map will characterize a range of operating conditions rather than being constrained to a single curve.

Entering or Modifying Compressor Map Curves

1. Select the Compressor Map radio button under the Centrifugal Compressor model.
2. Click [Enter Curve Data](#) under the Compressor Curve area which will open the Compressor Map window.
3. Select the appropriate flow parameter and units in the Raw Data table header rows (Non-dimensional or Corrected Mass Flow).
4. Enter the data for one of the curves into the table.
5. Specify the [Curve Fit type](#) (Polynomial or Interpolated X-Y Data) and click Graph Data Now / Generate Curve Fit Now.
6. Specify the Reference Inlet Conditions which are the pressure and temperature of the fluid used by the manufacturer when generating the Compressor Map curves.
7. Enter a Non-dimensional or Corrected Speed associated with the curve under the Configuration Information area on the right side of the Compressor Map window. An optional description can also be entered.
8. Click Update Configuration Now which will add the curve and associated speed to the list at the top-right of the Compressor Map window.
9. Click Create and repeat the above steps to enter the next curve for a different speed. Repeat this process until all desired curves are entered and categorized in the list.
10. Once all the curves are entered, click OK to accept and close the Compressor Map window.
11. Enter a Rated Speed on the Compressor/Fan Properties window. The Solver will use this speed to interpolate between the available curves.

Compressor Maps can be modified by clicking Enter Curve Data, selecting the speed from the list in the top-right, modifying the data of interest, clicking Graph Data Now / Generate Curve Fit Now, and clicking Update Configuration Now.

Viewing Compressor Map Curves - Individual curves can be viewed by changing the Speed dropdown menu on the Compressor/Fan Properties window, or by using the Composite Graph, Performance Graph, or Efficiency Graph tabs on the Compressor Map window.

Surge and Stall Lines - The Surge Line is described by connecting the data points with the lowest flowrate for each speed curve. The Stall Line is described by connecting the data points with the highest flowrate for each speed curve. These two lines bound the preferred range of operation inside the Compressor Map, and a Warning will be given if the [corrected](#) or [non-dimensional flow](#) is outside this range. Furthermore, a Warning will be given if the [corrected](#) or [non-dimensional speed](#) is outside the range of the data.

Variable Speed tab - The Fixed Speed (%) field will directly scale the Rated Speed that was entered on the Compressor/Fan Properties window. Note that the [Affinity Laws](#) are NOT used unlike the normal compressor curve.

Output - Corrected Mass Flow, Stagnation Pressure Ratio, and other useful output parameters are available within the [Output Control](#).

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 Arrow as pressure rises instead of head rises because the usefulness of the head term in variable-density systems is diminished.

Compression Process Thermodynamics

Depending on the compression process, significant thermal effects can occur. Fans and blowers create a pressure rise that is typically so low that the temperature is not noticeably affected, which is why Arrow treats the [Fan](#) model as adiabatic, but the large pressure rise in compressors leads to significant temperature changes. This is also known as the heat of compression, and in Arrow it can be accounted for in a compressor in multiple ways:

- **Isentropic** - The compressor is treated as 100% efficient using an isentropic efficiency. If the compression process is isentropic, the temperature rise is given by the following relationship:

$$\frac{T_2}{T_1} = \left(\frac{P_2}{P_1} \right)^{\frac{\gamma-1}{\gamma}}$$

where the pressures and temperatures are *static* and not stagnation. [Gamma \(\$\gamma\$ \)](#) is the specific heat ratio.

- **Polytropic** - If there is incidental heat loss to ambient, or if the compressor is actively cooled, the process will not be adiabatic. In this case the process is referred to as polytropic, and the temperature rise is given by the following:

$$\frac{T_2}{T_1} = \left(\frac{P_2}{P_1} \right)^{\frac{n-1}{n}}$$

where n is the polytropic exponent. If n is equal to γ , it is isentropic. Typically, n is greater than γ for compressors.

- **Determine From Efficiency Data** - If efficiency data is entered as optional data in the Compressor curve, this option can be selected and the thermal effects of the compressor due to compression heating will be calculated. The efficiency data will be applied as isentropic efficiency, which is defined as the ideal work for an isentropic process over the actual work.

Check Valve at Discharge (No Backflow Allowed)

If this option is checked when modeling a centrifugal compressor with a compressor curve, a [check valve](#) that can be considered part of the compressor itself is assumed to exist. This will never allow backflow at the compressor junction. If a check valve closes as part of the steady-state simulation, Arrow will perform recalculations under the assumption that the compressor's check valve is closed. Note that a steady check valve position must be attained in order for Arrow to apply its steady-state calculations.

- **Forward Velocity to Close** - The flow velocity, in the forward direction, that causes the check valve to close, preventing any further flow. A negative value in this field indicates that a reverse flow below the specified value will be permitted through the compressor before it closes.
- **Delta Head to Re-Open** - The differential head or pressure (upstream minus downstream) required to re-open the check valve.

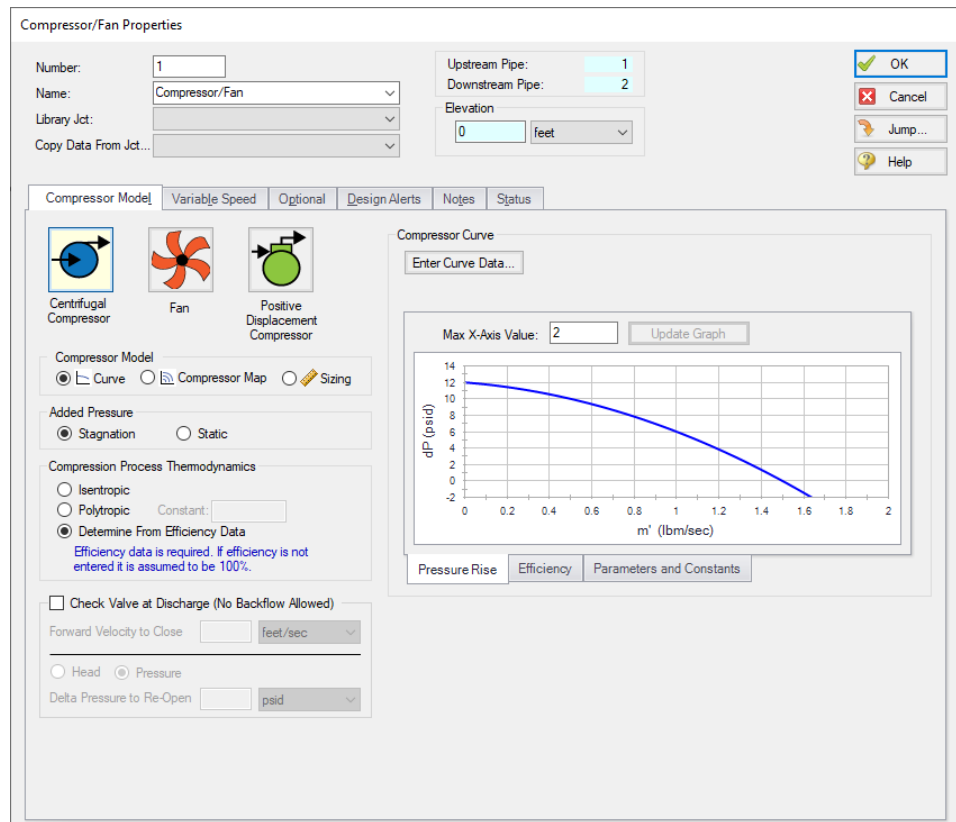


Figure 1: Compressor model with a centrifugal compressor curve defined

Related Topics

Compressors & Fans.....	253
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

Fans

Analysis Type

This topic pertains to the Fan Curve Analysis option which represents the real behavior of a fan. For Sizing Analyses, see [Compressor/Fan Sizing](#).

Fan Curve

To enter curves, click [Enter Curve Data](#). Once this data has been defined, previews of the applied curves can be seen on the right hand side of the Fan Model tab. Note that only one fan curve can be entered unless different configurations under the [Multiple Configurations](#) tab have been specified.

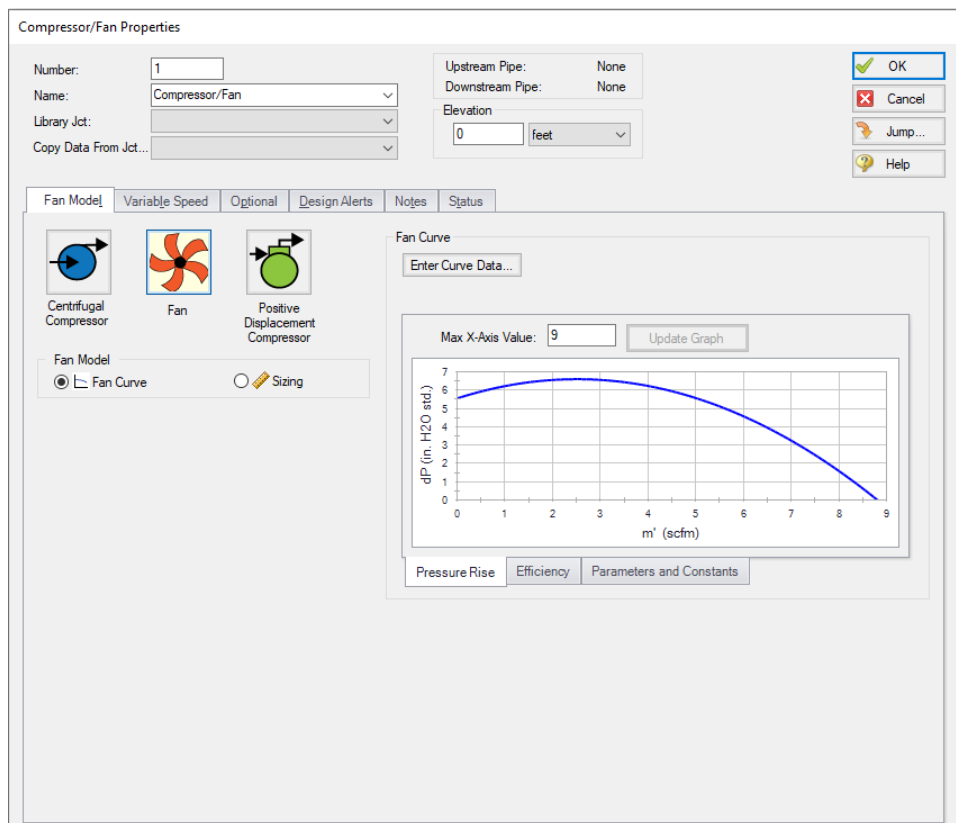


Figure 1: Fan model with a fan curve defined

The fan specification differs from the compressor specification in that all pressure values entered in the curve are assumed to be a static pressure, and the ability to change this to a stagnation pressure is not available. This simplification exists because fans generally add significantly less pressure to the fluids than compressors, and the overwhelming majority of fan vendor curves are

created using static pressures. If it is necessary to define a stagnation pressure in a fan curve, users should define this junction as a compressor and specify Stagnation pressure under the Added Pressure section.

In addition, the heat added by the fan is considered negligible, thus the fan will always be treated as adiabatic with 100% isentropic efficiency.

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

Related Blogs

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

Positive Displacement Compressor

Positive Displacement compressors/blowers can be considered to provide a fixed flow for steady state analysis. The positive displacement compressor/blower allows for a fixed flow to be defined in a straightforward manner.

To model a positive displacement compressor/blower, select the Positive Displacement option on the Compressor Model tab of the [Compressor/Fan Properties window](#).

Parameter

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

Fixed Condition

- **Fixed Flow Rate** - The value and unit of the fixed flow amount.
- **Nominal Efficiency** - A fixed efficiency, useful to calculate [power requirements](#).

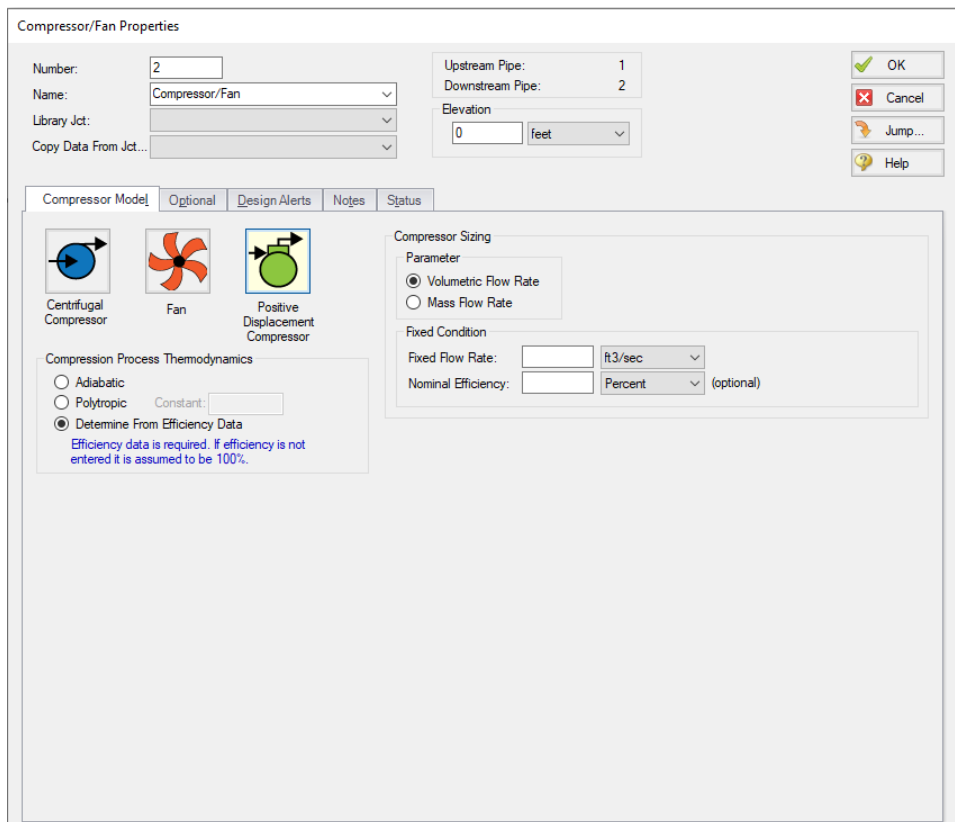


Figure 1: Positive Displacement Compressor/Blower Model

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

Related Blogs

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

Compressor/Fan Configuration

The compressor/fan configuration window is used to define the specific operation of a centrifugal compressor or fan modeled with a curve. It is opened by selecting the Enter Curve Data button from the [Compressor/Fan Properties window](#) when a centrifugal compressor or fan is specified.

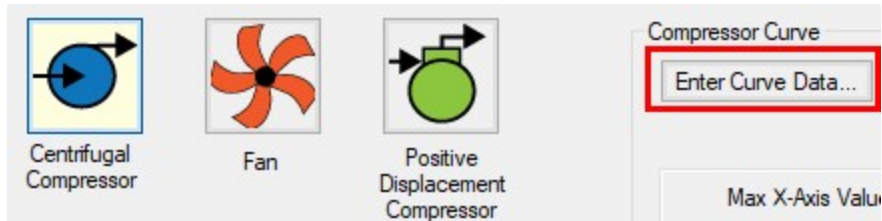


Figure 1: Enter Curve Data

Configuration Method

By default, a compressor/fan curve is defined using the Simple configuration. This means that the user inputs only one compressor/fan curve for the junction.

If the Configuration Method is changed to Multiple, additional input becomes available on the right of the configuration window. This allows defining different curves to represent the behavior of the compressor/fan at different speeds or impeller diameters. See a detailed description in Compressor/Fan [Multiple Configuration](#). This feature is especially useful when multiple curves for a given compressor/fan junction must be evaluated as it prevents the user from having to delete and enter curve data.

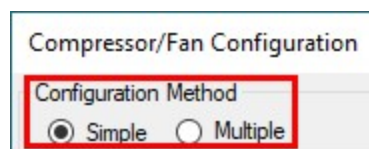


Figure 2: Compressor/Fan Configuration Method

Compressor/Fan Data

The first tab of the compressor/fan configuration controls the turbomachine's most basic behavior - the compressor/fan curves.

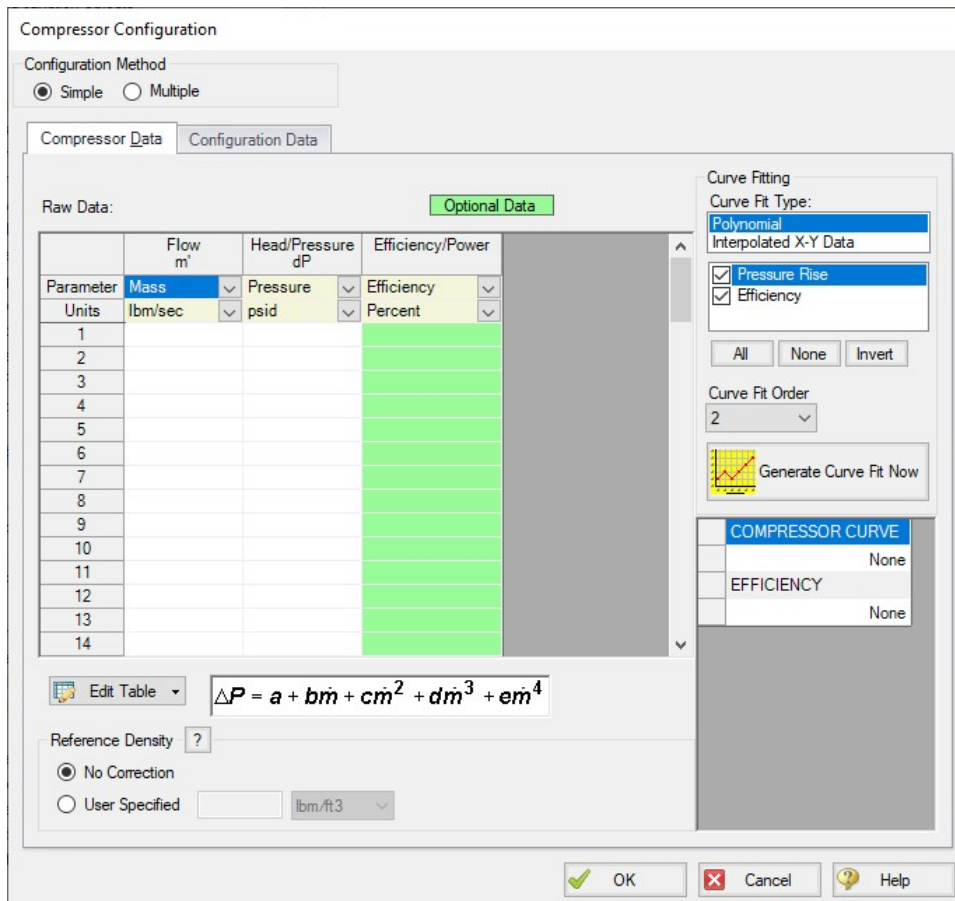


Figure 3: Compressor/Fan Data

A compressor/fan can have two types of 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). This data can also be used to determine the compression heating effects for a centrifugal compressor.

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.

If the Compression ratio is used for the curve definition, the [Reference Inlet Conditions](#) are an additional required input.

Configuration Data

This tab controls some additional behavior or input for a compressor/fan junction modeled with a curve. Entering or changing values on this tab is optional.

The screenshot shows the 'Compressor Configuration' dialog box with the 'Configuration Data' tab selected. The 'Configuration Method' is set to 'Simple'. The 'Compressor Data' tab is also visible. The 'Best Efficiency Point (BEP)' section has 'Calculate from Efficiency/Power Data' selected. The 'Preferred Operating Region (POR)' section has 'None' selected. The 'Allowable Operating Region (AOR)' section has 'None' selected. There are input fields for 'End Of Curve Flow Rate (optional)' in lbm/sec, and 'Low End' and 'High End' values for both POR and AOR, expressed as a percentage of BEPQ.

Figure 4: Configuration Data for Compressor

- **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 regardless of how this value is defined.
- **Preferred Operating Region (POR - Compressor Only)** - % 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 - Compressor Only)** - 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 these graphs to confirm the curve fit is reasonable for any potential operating region.

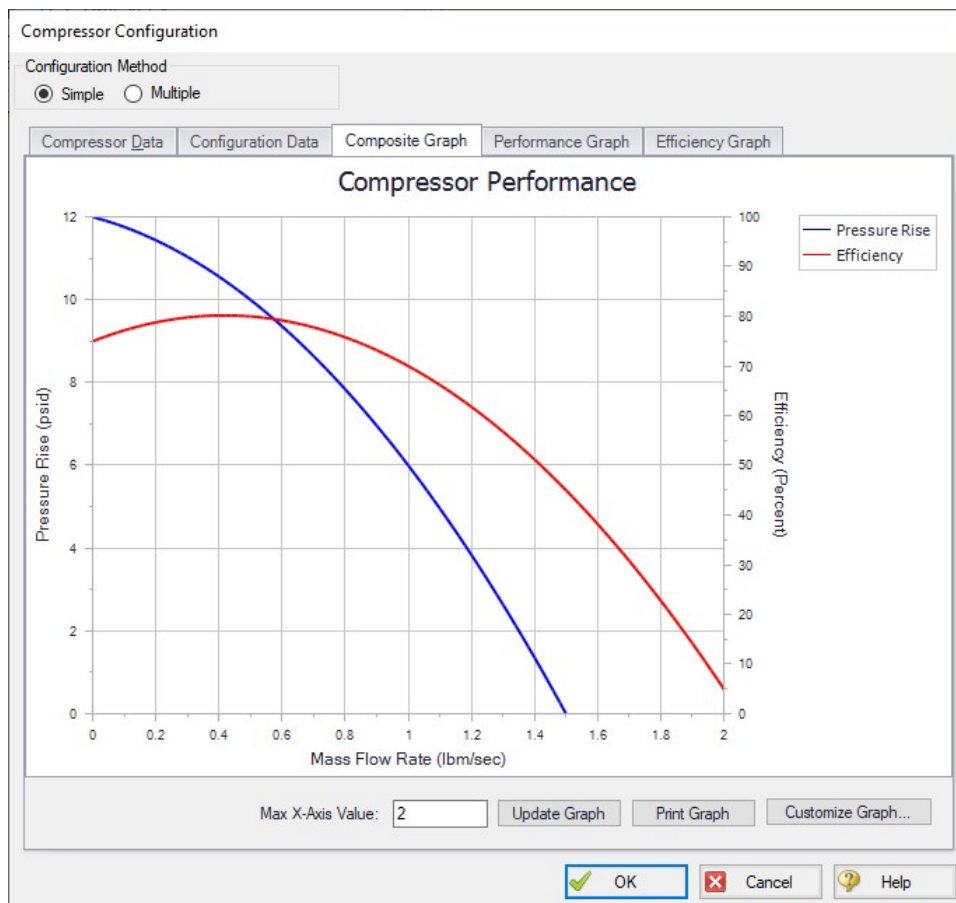


Figure 5: Composite Graph

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267

Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Affinity Laws.....	276
Poor Compressor/Fan Curve Fits.....	735

Related Blogs

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

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 in 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 the BEP, and the % of BEP which indicates how close the operating point is to the BEP.

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.

Power and Efficiency Calculations

Various stages of power or efficiency data relating to a compressor assembly can be added to a compressor junction.

The most basic value that can be accounted for is the compressor efficiency or power curve. This curve, entered in the Compressor Configuration window, is used to determine the power/efficiency being delivered to the compressor itself - the Shaft Power.

The Solver will determine the total pressure rise ($\Delta p = p_2 - p_1$) and volumetric flow (Q) through the compressor via iteration for the unique system. The isentropic power used at the compressor/fan, or the **Ideal Power**, can be calculated using the stagnation pressure values, volumetric flow, and isentropic expansion coefficient (k) as given by the following equation from [Perry, 10-39 \(1997\)](#):

$$P_{ideal} = 0.00436 \frac{k}{k-1} Q p_1 \left[\left(\frac{p_2}{p_1} \right)^{(k-1)/k} - 1 \right]$$

where power is in hp, flow is in ft³/min, and pressure is in psia. Alternate forms of the above equation can be found in Perry.

Note: In Arrow's interface, gamma (γ) is used to represent the isentropic expansion coefficient instead of k.

With a compressor power or efficiency curve, the power required by the compressor/fan, or **Shaft Power**, can be calculated:

$$P_{Shaft} = \frac{P_{Ideal}}{\eta_{Compressor}}$$

Note: The compressor efficiency value used here is the **isentropic efficiency**.

On a compressor's optional tab, further efficiency information can be defined:

	Motor Power	Motor Eff. (%)	VFD Power	VFD Eff. (%)
1				
2				
3				

Power Units: hp

Edit Table

Figure 1: Additional Efficiency Data

Entries in this table must come in pairs - both Power and Efficiency values must be present. For example, if Motor Power is entered, the Motor Efficiency column must also be entered. You can enter only motor data, only VFD data, or both.

Motor Power in this instance refers to the *output* power of the motor, which is equal to the Shaft Power calculated previously. This is because the motor driver is mechanically coupled to the shaft. Because the Shaft Power is already known, it can be looked up inside this table using linear interpolation, and the corresponding Motor Efficiency is used to calculate the **Motor Input Power**:

$$P_{Motor\ Input} = \frac{P_{Shaft}}{\eta_{Motor}}$$

VFD Power is used in a similar fashion. The VFD Power in Figure 1 refers to the *output* power of the VFD, which is identical to the Motor Input Power (but instead of being mechanically coupled, this is represented by an electrical connection). Because Motor Input Power was calculated, VFD Efficiency can be found from the table using linear interpolation, and **VFD Input Power** can be calculated:

$$P_{VFD\ Input} = \frac{P_{Motor\ Input}}{\eta_{VFD}}$$

VFD Input Power is equivalent to the **Overall Power**, viewable in the Compressor/Fan Summary along with all of the other above values. Whenever an efficiency value is left undefined, it is assumed to be 1 (perfectly efficient).

$$\eta_{overall} = \eta_{VFD}\eta_{Motor}\eta_{Compressor} \quad P_{Overall} = \frac{P_{Ideal}}{\eta_{overall}}$$

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276
Poor Compressor/Fan Curve Fits.....	735

Compressor/Fan Sizing

The [Compressor/Fan Properties window](#) offers the option of Sizing a compressor/fan. By entering either a volumetric or mass flow rate, the total developed pressure is calculated and displayed in the Compressor/Fan section of the [Output window](#). By assigning a flow to a compressor/fan, determining the size requirements is straightforward.

To size a compressor/fan, first specify if the junction is a centrifugal compressor or fan and then select the Sizing option for the Compressor/Fan Model.

Specify the parameter that should be used for the sizing. The parameters include:

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

- **Head Rise** - Fix the head rise through the compressor/fan. Note that the density used when fixing the head rise through a compressor/fan is that of the fluid at the suction of the turbomachine.
- **Pressure Rise** - Fix the pressure rise through the compressor/fan.
- **Compression Ratio** - This option is only available for compressors, and it allows you to fix the compression ratio of the compressor. This ratio is important because it directly affects the efficiency of the compressor, and it is calculated using the equation:

$$\textit{Compression Ratio} = \frac{P_{discharge}}{P_{suction}}$$

Problems with Compressor/Fan Sizing and Using a Flow Control Valve

Using the Sizing feature on a [Compressor/Fan Properties Window](#) with a Flow Control Valve (which also fixes flow rate) downstream of the compressor/fan junction results in a Reference Pressure error - the system no longer has a unique solution. Read more about this in [Role of Pressure Junctions - Detailed Discussion](#).

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

Compression Ratio Reference Inlet Conditions

When a Compression Ratio curve is entered, the Reference Inlet Conditions are needed for Arrow Solver to convert the Compression Ratio curve from the curve created at the reference conditions (by the vendor) to a curve applicable at the piping system's operating conditions. The Reference Inlet Conditions are the conditions measured at the inlet of the compressor when the Compression Ratio curve is developed by the curve vendor.

The Reference Inlet Conditions are air at ISO conditions (14.7 psia & 59 deg. F) by default. However, the Reference Inlet Pressure & Temperature can be changed. Currently, air is the only option for the Reference Fluid.

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

Multiple Configurations

A single [compressor/fan junction](#) can contain information for multiple compressor/fan configurations - impeller trims and operating speeds. This is useful to compare different variations of a compressor/fan without needing to redefine a new compressor/fan junction for every case. The configuration of the compressor/fan can easily be changed without losing any information.

Defining Multiple Configurations

When opening the [Compressor/Fan 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.

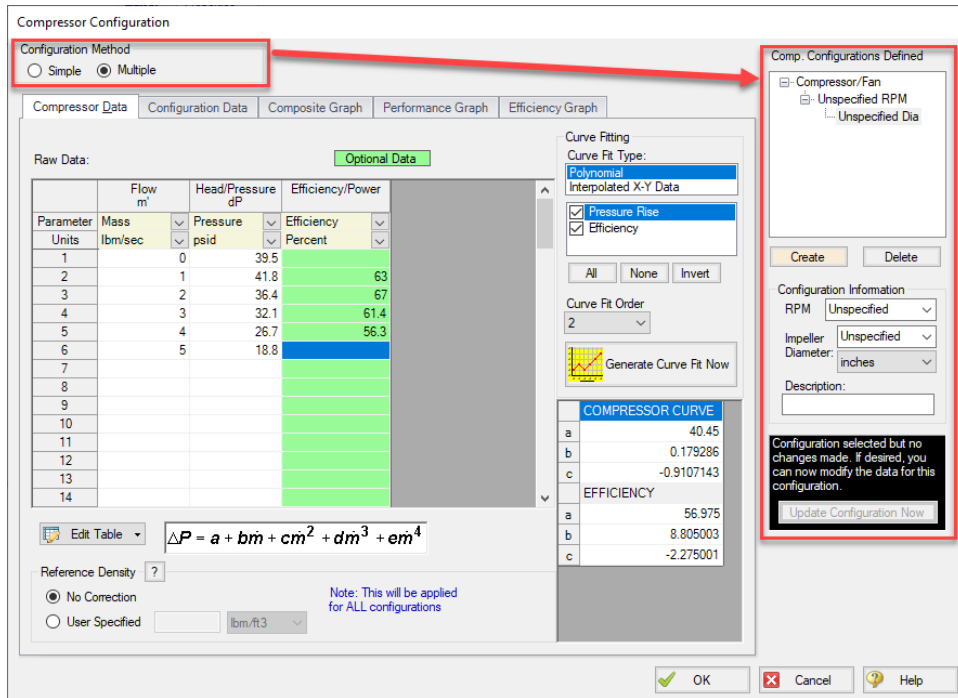


Figure 1: Multiple Configuration Method Selected

In order to make use of the Multiple Configurations 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 Now box turns yellow:

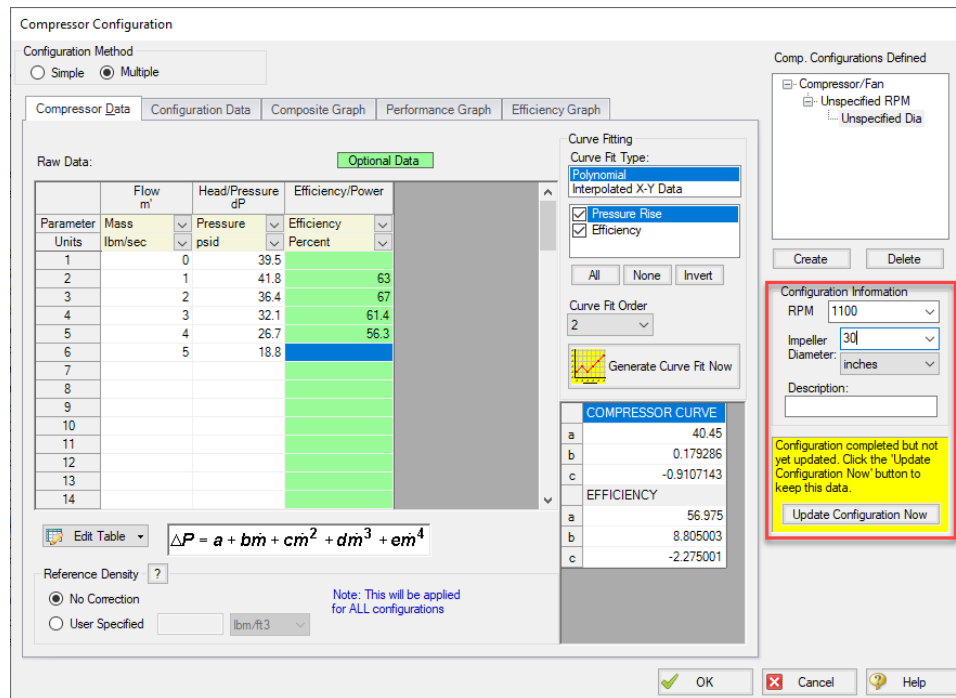


Figure 2: Update Configuration

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

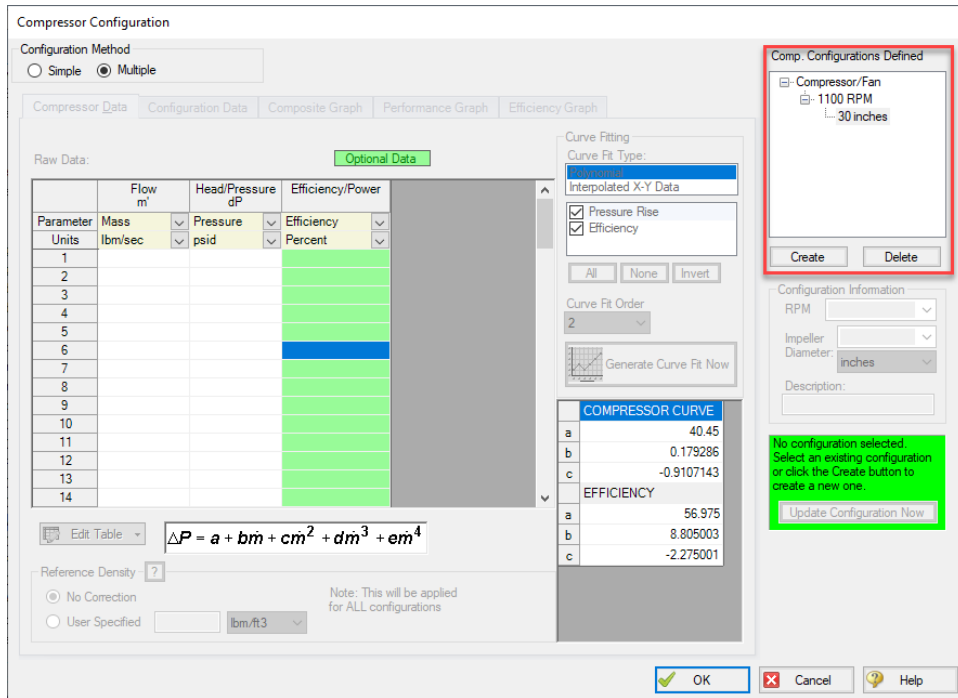


Figure 3: Current Compressor Configurations

To create an additional configuration, click Create.

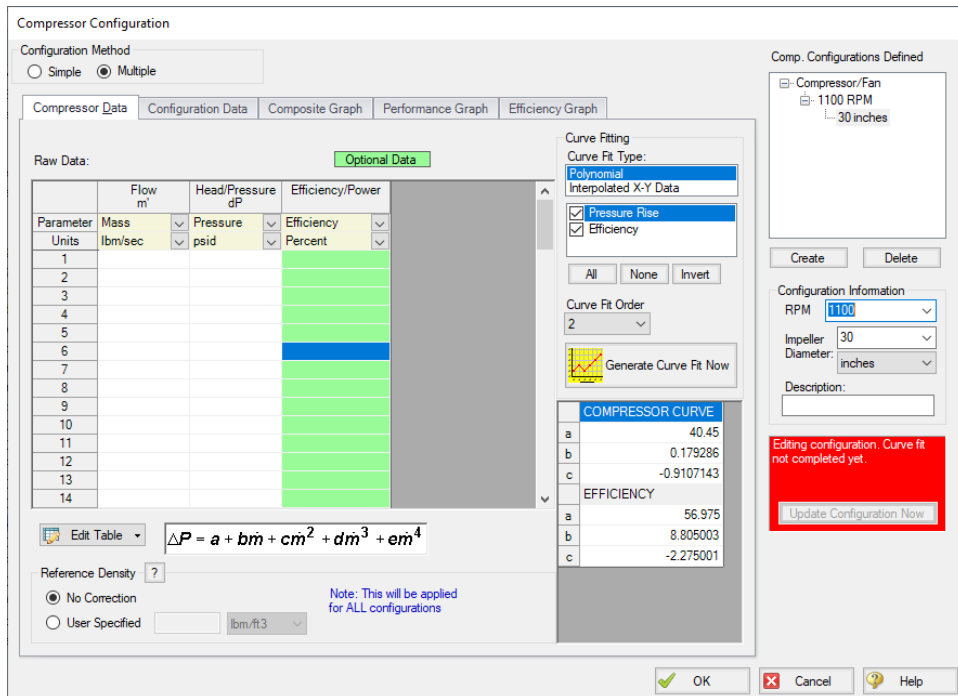


Figure 4: Creating a new Configuration

The compressor/fan curves must be created and curve fit just as is done with a compressor/fan in which only a single configuration has been specified. After completing this and clicking Update Configuration Now, the new configuration will be created and shown in the Compressor/Fan Configurations area. Configurations can be edited by selecting them in this area.

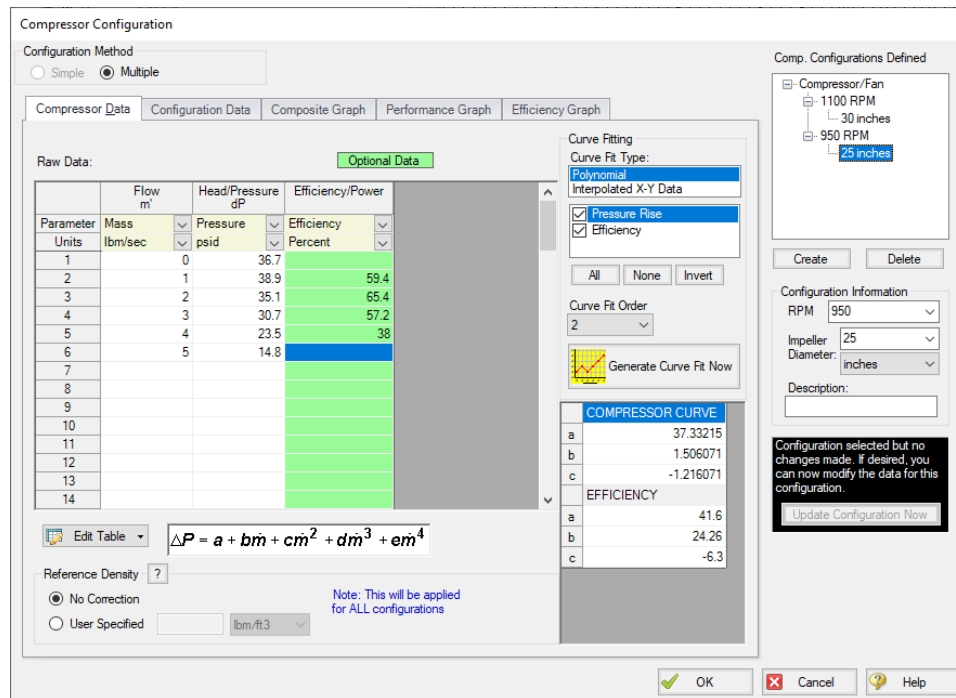


Figure 5: Editing a Configuration

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

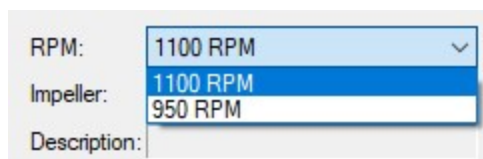


Figure 6: Selection of a Configuration

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270

Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263
Affinity Laws.....	276

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 appropriate affinity law is then applied to the compressor/fan curve, and a speed ratio, denoted with the symbol s , that meets the user's specification on the Variable Speed tab on the [Compressor/Fan Properties window](#) is determined. The affinity laws are shown below.

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 -**

$$\frac{\Delta H_1}{\Delta H_2} = s^2$$

- Compressor/Fan Power consumption -

$$\frac{P_1}{P_2} = s^3$$

- Efficiency -

$$\frac{\eta_1}{\eta_2} = 1$$

Related Topics

Compressors & Fans.....	253
Centrifugal Compressor.....	255
Fans.....	260
Compressor/Fan Efficiency Data.....	267
Compression Ratio Reference Inlet Conditions.....	270
Multiple Configurations.....	271
Compressor/Fan Sizing.....	269
Positive Displacement Compressor.....	261
Compressor/Fan Configuration.....	263

Related Blogs

[To Affinity and Beyond!](#)

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. Use of this junction allows AFT Arrow the liberty of skipping the flow solution, since the flow is known to be zero.

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

Junctions.....	221
----------------	-----

General Junction Information.....	222
Junction Loss Models.....	233

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.

Subsonic losses in the General Component junction can be represented using a K factor or by entering a resistance curve. 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.

CdA for Sonic Choking

An optional input is the C_dA for Sonic Choking. This parameter describes the effective area restriction for the purpose of calculating [sonic choking](#). In most cases, information on the C_dA must be obtained from manufacturer or test data.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Subsonic vs. Sonic Pressure Losses.....	665

Heat Exchanger

The Heat Exchanger 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 heat exchanger equipment. You also have the ability to specify loss factors as a function of a flow parameter.

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.

Subsonic losses in the Heat Exchanger junction can be represented using a K factor or by entering a resistance curve. When you select [Resistance Curve](#), the Heat Exchanger Properties window makes additional features available. Using these new features you can input heat

exchanger loss factors or pressure drops that vary with flow. To enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

Heat Transfer

On the Thermal Data tab you can select from twelve different [heat exchanger](#) models.

Note that similar to flow rate definition, heat flow is positive when flowing into the system, and negative when flowing out.

When modeling convective heat exchangers (such as parallel or counterflow) two heat exchanger junctions can be linked thermally. This [thermal link](#) allows the two heat exchanger junctions to represent two sides of the same heat exchanger. This capability can also be used to model heat transfer between loops with different gases.

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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Heat Exchanger Heat Transfer.....	279

Heat Exchanger Heat Transfer

There are twelve different heat exchanger models available in AFT Arrow.

- NTU-Effectiveness Models
 - Counter-Flow
 - Cross Flow, Both Fluids Unmixed
 - Cross Flow, Secondary Fluid Mixed
 - Cross Flow, System Fluid Mixed
 - Parallel
 - Shell & Tube, 1 Shell Pass, Mult. of 2 Tube Passes
- Directly specified thermal property

- Controlled Downstream Temperature
- Controlled Downstream Temperature (Stagnation)
- Enthalpy Stagnation Rise
- Specified Heat Rate In Constant
- Specified Heat Rate In vs. Flow
- Temperature Stagnation Rise

The screenshot shows the 'Heat Exchanger Properties' dialog box with the 'Thermal Data' tab selected. The 'Thermal Model' is set to 'Counter-Flow'. The 'Link to Heat Exchanger' checkbox is unchecked. The 'Heat Flow Into System' field is empty, with units set to 'Btu/s'. The 'Controlled Outlet Temperature' field is empty, with units set to 'deg. F'. The 'Heat Transfer Area' field is empty, with units set to 'feet2'. The 'Overall Heat Transfer Coefficient' field is empty, with units set to 'Btu/hr-ft2-R'. Under 'Secondary Fluid Data', the 'Flow Rate' field is empty (units: 'lbm/sec'), 'Specific Heat' is empty (units: 'Btu/lbm-R'), and 'Inlet Temperature' is empty (units: 'deg. F'). The top of the dialog shows 'Number: 11', 'Name: Heat Exchanger', 'Upstream Pipe: None', 'Downstream Pipe: None', and 'Elevation' set to 'feet'. Buttons for 'OK', 'Cancel', 'Jump...', and 'Help' are on the right.

Figure 1: Heat Exchanger Thermal Data

All of the NTU-Effectiveness models require the same 5 parameters as input (shown in Figure 1), but calculate the resulting heat transfer slightly differently. The direct specification models only require that the associated property be entered.

Thermal results are displayed along with pressure results in the Heat Exchanger Summary in the output, as well as information on secondary fluid conditions and log mean temperature difference.

NTU-Effectiveness Method

The NTU-Effectiveness method is used when the fluid inlet and outlet temperatures for the heat exchanger are not known.

To start, the NTU-Effectiveness method requires that the maximum possible heat transfer be found. Theoretically, the maximum heat transfer would occur in a counterflow heat exchanger of an infinite length. In this scenario, the maximum possible temperature difference is the difference between the inlet temperature of the hot fluid and the inlet temperature of the cold fluid:

$$\Delta T_{max} = T_{h1} - T_{c1}$$

The minimum heat capacity rate can then be found by multiplying the mass flow rate by the specific heat for the hot fluid and the cold fluid:

$$C_h = (\dot{m} C_p)_h$$

$$C_c = (\dot{m} C_p)_c$$

$$C_{min} = \text{smaller of } C_h, C_c$$

The fluid with the higher heat capacity rate changes temperature more quickly along the length of the heat exchanger, but the fluid with the lower heat capacity rate actually sees the maximum possible temperature change. Based on this, the maximum heat transfer between the two fluids in the theoretical heat exchanger is the minimum heat capacity rate multiplied by the maximum temperature difference:

$$q_{max} = C_{min} (\Delta T_{max})$$

The effectiveness of a heat exchanger is the ratio between the actual heat transfer rate and the maximum possible heat transfer rate:

$$\epsilon = \frac{q}{q_{max}} \quad \text{where} \quad q = C_h (T_{h1} - T_{h2}) = C_c (T_{c2} - T_{c1})$$

If the effectiveness and the inlet conditions are known, the amount of heat transferred between the two fluids can be calculated without knowing anything about either fluid's outlet temperature:

$$q = \epsilon C_{min} (T_{h1} - T_{c1})$$

The key to the NTU method becomes calculating the effectiveness, which is a function of the heat capacity ratio, C , and the number of transfer units, NTU:

$$C = \frac{C_{min}}{C_{max}}$$

$$NTU = N = \frac{U A}{C_{min}}$$

Each heat exchanger configuration has a unique relation used to calculate the effectiveness. These become the fundamental equations for the six thermal models that use the NTU method:

$$\textit{Parallel Flow} \quad \epsilon = \frac{1 - e^{-N(1+C)}}{1 + C}$$

$$\textit{Counterflow} \quad \epsilon = \frac{1 - e^{-N(1-C)}}{1 - Ce^{-N(1-C)}}$$

$$\textit{Crossflow, Both Fluids Unmixed} \quad \epsilon = 1 - e^{\left(\frac{N-0.22}{C}\right)(e^{-CN^{0.78}} - 1)}$$

$$\textit{Crossflow, Fluid with } C_{max} \textit{ mixed} \quad \epsilon = \frac{1}{C} \left(1 - e^{-C(1-e^{-N})}\right)$$

$$\textit{Crossflow, Fluid with } C_{min} \textit{ mixed} \quad \epsilon = 1 - e^{\frac{1}{C}(1-e^{-CN})}$$

$$\textit{Shell \& Tube} \quad \epsilon = 2 \left\{ 1 + C + \sqrt{1 + C^2} \left(\frac{1 + e^{-N\sqrt{1+C^2}}}{1 - e^{-N\sqrt{1+C^2}}} \right) \right\}^{-1}$$

Thermal Link

When modeling convective heat exchangers (such as parallel or counterflow) two heat exchanger junctions can be linked thermally.

A thermal link is when two heat exchanger junctions, which are hydraulically isolated, can exchange heat with each other. Typical use is when the two heat exchangers represent two sides of the same physical heat exchanger. When thermal linking is specified in a heat exchanger Specifications window, the user must specify the heat transfer area and overall

convection coefficient. AFT Arrow will force the two heat exchanger junctions to use the same value.

Related Topics

Heat Exchanger.....278

Thermal Link

Orifice

The Orifice junction type requires two connecting pipes, unless you specify it as an exit orifice, in which case only one connecting pipe is allowed. This junction type allows you to model the irrecoverable loss that occurs through a plate-type flow restriction. It also allows you to specify 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 Arrow provides two standard orifice types: sharp-edged and rounded. These loss factors are calculated based on [correlations](#). In addition to the standard orifices that AFT Arrow provides, you can enter your own custom orifice loss factor that is consistent with the application.

For convenience, you can specify constant loss characteristics of an orifice as a discharge coefficient (C_d) 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.

When you select [Resistance Curve](#), the Orifice Properties window makes additional features available. Using these new features you can input Orifice 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.

CdA for Sonic Choking

An optional input in the Orifice Properties window is the C_dA 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 C_dA 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 Arrow. 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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Orifice Loss Model.....	284
Subsonic vs. Sonic Pressure Losses.....	665

Orifice Loss Model

Sharp-Edged Orifice with Different Upstream and Downstream Diameters

The orifice loss factors are all for turbulent Reynolds numbers. The sharp-edged orifice shown in Figure 1, is given by the following ([Idelchik 2007](#), 259):

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

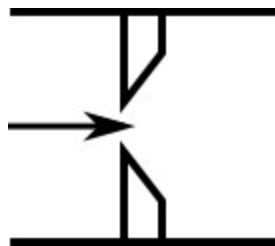


Figure 1: Sharp-edged orifice (in this case $A_{up} = A_{down}$)

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

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

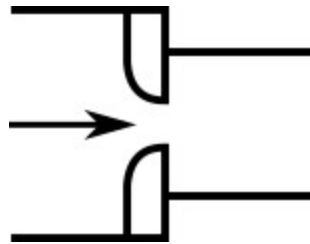


Figure 2: Round-edged orifice with different upstream and downstream pipe areas

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

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

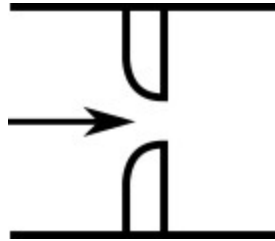


Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

For other orifice configurations, see Chapter 4 of [Idelchik's Handbook of Hydraulic Resistance](#) (2007).

Orifice Discharge Coefficient

An alternate method of modeling subsonic pressure loss uses the discharge coefficient. To model this, choose User Specified loss option and the C_d option for Loss Model.

When solving the system, AFT Arrow creates a subsonic $C_d A$ for the orifice and solves the pressure drop by iterating the following equation for pressure drop for a given mass flow rate ([Keenan 1941](#), 317):

$$\dot{m} = C_D A P_{down} \sqrt{\frac{2}{RT} \left(\frac{\gamma}{\gamma - 1} \right) \left(\frac{P_{up}}{P_{down}} \right)^{(\gamma-1)/\gamma} \left[\left(\frac{P_{up}}{P_{down}} \right)^{(\gamma-1)/\gamma} - 1 \right]}$$

where *up* and *down* subscripts refer to conditions upstream and downstream of the orifice. In this implementation, the pressures and temperature are static, but the velocity (i.e., Mach number) in the pipe is low compared to the orifice. Therefore, stagnation pressures and temperatures are used. Note that *R* in this equation is the specific gas constant, which is equivalent to the universal gas constant divided by the molecular weight of the gas in the system.

Related Topics

K Factor Loss Model.....	234
Orifice.....	283
Subsonic vs. Sonic Pressure Losses.....	665

Relief Valve

The Relief Valve junction allows users to model a valve that opens at a specified pressure in the system. Depending on how the relief valve operates, that pressure could either be the pressure at the relief valve inlet or the pressure at some remote point in the system. The differences between those valves are discussed below.

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.

There are three types of relief valves available:

- **Internal Relief Valve:** contains two connecting pipes and relieves into the downstream pipe. This type of relief valve acts like a regular valve that is closed until the specified pressure is reached.
- **Exit Valve:** contains one connecting pipe and relieves to an external ambient pressure when the specified pressure is reached.
- **Inline Exit:** contains two connecting pipes and relieves to an external ambient pressure when a specified pressure is reached. This type of relief valve acts as a lossless connection when closed and allows flow between the connecting pipes.

Opening and Closing Profiles

The Relief Valve junction has several methods by which the valve can open and close. These options are:

- **Instant:** the relief valve opens fully or closes fully from one time step to the next as the opening or closing conditions are met.
- **Pressure:** the valve passively opens or closes according to a C_v vs. Pressure profile as the opening or closing conditions are met. The C_v vs. Pressure profiles are based on the Valve Setpoints and the data entered in the Loss Model tab. Maximum C_v vs. Time rate limits can be applied.
- **Never:** the valve never closes. This method cannot be applied to the valve opening.

Balanced vs Unbalanced Valves

Relief valves can either be hydraulically balanced (constant backpressure) or non-hydraulically balanced. This characteristic of the valve is determined by how the dome of the relief valve is pressurized.

For an unbalanced valve, the dome of the valve is pressurized via the fluid downstream of the valve. Since the downstream pressure can change during the transient, the valve position is based on a pressure differential between the upstream and downstream pipes rather than based solely on the upstream pressure.

For the balanced valve, the dome of the valve is pressurized to a constant pressure, often by attaching some type of bellows device. With this system, the valve position is based only on the actual upstream pressure.

Valve Setpoints

The valve setpoints determine when the relief valve will open and when it will be closed. These setpoints can be defined using either pressure or head.

There are four possible valve setpoints the user needs to define:

- **Exit Pressure:** the ambient pressure downstream of the relief valve for the Exit or Inline Exit valve types. This pressure will only be used to determine when the valve opens and closes when the relief valve is unbalanced.
- **Overpressure:** the upstream pressure at which the valve is fully open. This input is only needed when the valve opens or closes based on pressure.
- **Set Pressure:** the upstream pressure at which the valve opens. This input must be specified for all valve profiles and is replaced by the High Pressure Setpoints when Remote Sensing is enabled.
- **Blowdown Pressure:** the upstream pressure at which the valve is fully closed. All options for valve opening and closing require this input, with the exception of the Never profile. If the relief valve closes at the same pressure where it opens, the Set Pressure and the Blowdown Pressure should be set to the same value.

Remote Sensing

The Relief Valve junction gives users the ability to model Remote Sensing Relief valves. This option is available anytime the valve operates as a pressure/pressure valve.

A Remote Sensing Relief Valve is a relief valve that opens based on the pressure at a specified location in the system other than the upstream side of the valve. In a transient system, this design allows the relief valve to be more responsive to surge events by opening the valve before the pressure surge arrives at the valve itself. In a steady state system, this design allows the valve to open at some remote pressure while the degree to which the relief valve is open is determined by the pressure at the valve.

Loss Model

The Relief Valve loss information is entered on the Loss Model tab. The available options change depending on how the valve opening and closing are defined. For all options, the Base Area for Loss Model can be specified, with the upstream pipe being used by default.

The following loss model options are available depending on which profiles are selected:

- **Instant Profiles**
 - **C_v (ANSI/ISA)**: the relief valve instantly changes from closed to a fixed C_v .
 - **K_v (Constant)**: the relief valve instantly changes from closed to a fixed K_v .
 - **C_dA (Constant)**: the relief valve instantly changes from closed to a fixed C_dA . This C_dA is defined as the product of an Orifice Effective Area (A) and a Discharge Coefficient (C_d).
 - 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.
 - **Resistance curve**: the relief valve instantly changes from closed to a resistance curve.
- **Pressure Profiles**
 - **C_v (Variable)**: the relief valve moves between the closed and open states with the C_v varying between 0 and the Fully Open C_v value.
 - **K_v (Variable)**: the relief valve moves between the closed and open states with the K_v varying between 0 and the Fully Open K_v value.

The Variable Data option for the Relief Valve Loss Model allows users to model the valve losses based on where the valve is between fully open and fully closed. The Linear based on setpoints option has Impulse determine those losses solely based on the setpoints the user has specified. If the user knows enough about the valve to describe a detailed loss profile, they can select the Non-linear option. The first data point must have a loss value of 0 at the Blowdown pressure. The final data point must have a loss value equal to the Fully Open loss value, at the Overpressure. The entered data is interpolated during the transient simulation.

C_dA for Sonic Choking

An optional input in the Relief Valve Properties Window is C_dA (for sonic choking - optional). This parameter describes the effective area restriction in the orifice for the purpose of calculating sonic choking. In most cases, information on the C_dA 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 Arrow. 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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Unique Solution Not Possible - Relief Valve Cycles.....	723
Cv/Kv and Xt Loss Model.....	235
Subsonic vs. Sonic Pressure Losses.....	665

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 Arrow 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 Arrow 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 Arrow 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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Screen Loss Model.....	291
Subsonic vs. Sonic Pressure Losses.....	665

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):

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

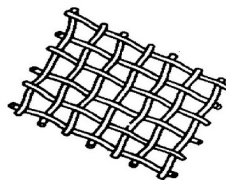


Figure 1: Circular metal wire screen

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 \(2007\)](#).

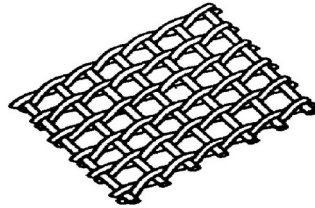


Figure 2: Silk thread screen

Related Topics

K Factor Loss Model.....	234
Screen.....	290
Subsonic vs. Sonic Pressure Losses.....	665

Separator

The Separator junction type requires two connecting pipes. This junction type allows you to separate fluids out of a [mixture](#).

The Separator 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. Separator junctions are internal to the system, with two connecting pipes.

Either the Chempak or NIST REFPROP fluid library must be used to create a mixture for the Separator junction to be used. These fluid libraries can be specified on the [Fluid panel](#). Units for the amount of gasses separated are listed below the table.

Special Conditions

The Special Condition for a Separator stops the separation, making it act like a [branch junction](#).

Related Topics

Junctions.....	221
----------------	-----

General Junction Information.....	222
Junction Loss Models.....	233
Separator junction X is trying to remove Y of Z but only A is entering junction.	721
Separator junction X is trying to remove Y but none is entering junction.....	721

Spray Discharge

The Spray Discharge junction must have at least one connecting pipe, and it allows up to four. This junction offers a flexible way of modeling a flow exiting the system through a nozzle or spray device.

The Spray Discharge Properties window follows the second of the two basic [Properties window formats](#). A table displays the connecting pipe information. The pipe table grows in size to accommodate up to four connecting pipes.

The exit pressure to which the Spray Discharge junction discharges must be entered, as well as the temperature in case of inflow.

Two geometries can be modeled. The first is a normal, one hole spray, where the discharge flow area of the hole is entered. The second is a sparger which, in principle, works in the same way as a spray. For the sparger, the flow area of a single hole and the number of holes are entered. This assumes that the holes are hydraulically close in proximity and are the same area.

For each connecting pipe, the pipe table on the Pipe Losses tab lists the reference flow direction and up to two loss factors. The first loss factor is for physical flow out of the pipe and into the junction, the second is for physical flow into the pipe and out of the junction. The loss factors can be specified independently or left as zero. To edit within the pipe table, click in the appropriate column and row.

Loss Model

The discharge coefficient for the junction is entered in the appropriate box. The physical area of the exit flow can also be entered. These two parameters are used to calculate the discharge flow rate, which depends on the difference between the internal pressure in the system and the specified exit pressure.

If you select a [Resistance Curve](#), the Spray Discharge Properties window makes additional features available. Using these new features you can input the spray pressure drop that varies with flow. To enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Arrow 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 Spray Discharge Properties window is the C_dA . This parameter describes the effective area restriction in the spray for the purpose of calculating [sonic choking](#).

Note: The C_dA for sonic choking may be different from the subsonic C_dA loss model option in Arrow. 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

Spray Discharge junctions have the following [Special Conditions](#) on the Optional tab:

- **None** - Flow is allowed to enter/exit the system at the junction as determined by the system hydraulics and external pressure.
- **Closed** - Flow is not allowed to enter/exit the system at the junction, but flow can still go in/out of the connected pipes as determined by the system hydraulics.
- **No Inflow** - Flow is not allowed to enter the system from the surroundings, but is allowed to exit the system as determined by the system hydraulics and external pressure.

Flow Net vs Through

The [Output Control](#) allows reporting of Mass/Volumetric "Flow Rate Net At Junction" or "Flow Rate Through Junction". Depending on the number of connected pipes and flow directions determined by the Solver, the Flow Rate Through Junction may report values of N/A. This is because there may not be a unique flow path "through" the junction.

- **Net At Junction** - Generally this parameter is measuring the flow entering/exiting the system. It is calculated by summing the flow into the junction and subtracting the sum of flow out of the junction. A positive value indicates flow exiting the system, and a negative value indicates flow entering the system.
- **Through Junction** - Generally this parameter is measuring the flow through a unique path in the junction if one is available. A unique flow path could include a scenario where the Special Condition is set to Closed, but flow enters from one upstream pipe, passes through the junction (lossless), and exits into one downstream pipe. Another example of a unique flow path is if the Spray has only one connected pipe.

Note: Pipes with inverted [Reference Positive Flow Directions](#) and negative flowrates can cause confusion when interpreting results with the Flow Rate Net At Junction and Flow Rate Through Junction parameters. Care should be taken when observing the signs of the parameters.

Transient

([XTS Module](#) Only) On the Transient tab the Subsonic and Sonic CdA data can be varied with time to simulate the opening or closing of the junction. For more information on transient data,

including event transients, see the [Junction Transient Data](#) topic.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Subsonic vs. Sonic Pressure Losses.....	665

Tank

The Tank junction type allows you to connect up to twenty-five pipes. One connecting pipe is required. The Tank junction is convenient for specifying a fixed pressure in your system.

A Tank junction applies a defined pressure and temperature at the junction location in the model. When solving a pipe flow system, a Tank causes the rest of the system to distribute the flow in a manner consistent with the defined pressure.

The Tank Properties window follows the second of the two basic [Properties window formats](#). A table on the Loss Coefficients folder tab displays the connecting pipe information. This table grows to accommodate up to twenty-five pipes. After you add a fifth connecting pipe, a scroll bar appears, allowing you to review and enter loss factors for all pipes in the table.

The pipe table displays the reference positive flow direction of each connecting pipe. To enter loss factors, select the cell in the table and edit the value. Each pipe can use a different loss model or custom value.

Energy Balance

For an open system, most users desire that the temperature remains fixed, like the pressure does. However, for closed, recirculating systems, you will usually want the temperature to vary. See [Open vs. Closed Systems - Balancing Energy](#) for detailed discussion.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233

Tee/Wye

The Tee/Wye junction type requires three connecting pipes.

Loss Model

There are two Loss Models available for Tee/Wye junctions: Simple and Detailed. If you choose the Simple type of Tee/Wye, no loss factors will be calculated. This is equivalent to a branch with no loss factors entered for the connecting pipes. In some systems, the losses associated with a Tee/Wye are small compared to the overall loss in the system. To avoid unnecessary calculations, you should choose the Simple type of Tee/Wye unless you need the actual losses.

If the Detailed loss model is chosen, additional features will become available to define the arrangement of the pipes connected to the tee/wye, as well as information on the tee/wye geometry. By default, Arrow will automatically define the straight and branch run pipes based off of the arrangement in the Workspace. If Arrow cannot determine an arrangement, the user will be required to define the pipe arrangement in the Tee/Wye Properties window. The arrangement defined in this window will affect the calculated loss factors.

The Detailed type of Tee/Wye will calculate the loss factors as they depend on the flow split as well as the geometry. Because of the dependence on the flow split, loss factors usually cannot be calculated before performing an analysis. AFT Arrow incorporates four models to calculate tee and wye loss factors during the actual solution. These calculations account for the developed flow pattern, requiring additional iteration. See [Tee/Wye Loss Model](#) and [Tee/Wye Junction Complexity](#) for a detailed discussion of Detailed Tees.

If you have specific loss factors that you want to use for a Tee/Wye junction, you should use a [Branch](#) junction instead and enter the appropriate K factors on the Optional tab.

Related Topics

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Tee/Wye Junction Complexity.....	738
Tee/Wye Loss Model.....	296

Related Blogs

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

[Give me some details about Detailed Tees!](#)

Tee/Wye Loss Model

Note: These correlations were developed for incompressible flow. K-Factors generally give good results when used in compressible flow, but there is uncertainty in this application. See the [K Factor Loss Model](#) topic for more discussion.

The loss factors calculated for tee and wye junctions involve complicated correlations that depend on the flow split, the ratio of flow areas, and the angle of the connecting pipes. The models used by AFT Arrow are taken mostly from Chapter 7 of Idelchik's Handbook of Hydraulic Resistance ([Idelchik 2007](#)). The equations will not be shown in this topic, but the references are given in the table below.

All tees and wyes have either diverging or converging flow streams. Two sets of equations are required for each configuration.

In the following discussion, the parameter $K_{c,s}$ represents the loss factor based on the combined flow, stream flow area, and velocity for the side branch. Similarly, $K_{c,st}$ is for the straight-through pipe (the run).

Figure 1 depicts the nomenclature. K-factor subscripts refer to base area reference and location in the Tee. For example, $K_{c,s}$ represents the K-factor for the side branch (s) referenced to the combined (c) pipe flow area.

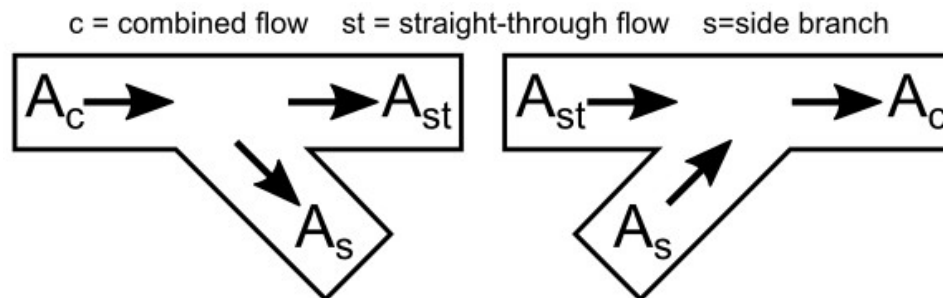


Figure 1: Idelchik's nomenclature for diverging and converging flow through Tee/Wye junction

In the AFT Arrow Tee/Wye window, a simpler and more general nomenclature is used. AFT Arrow refers to the a, b and c pipes. This allows AFT Arrow to cover all the possibilities in Table 3 without specifying exactly which pipe is combined flow pipe. In some cases, you may not know all the flow directions in a system, and might not be able to specify which is the combined flow pipe.

AFT Arrow's input method allows you to specify which pipe is the branch pipe. Idelchik calls this the "s" pipe for "side branch". AFT Arrow calls it the c pipe. If the tee angle is 90 degrees, it is not important which pipe you call the a pipe and which the b pipe. The distinction between a and b takes on more importance for non-90 degree pipes. In this case, AFT Arrow allows you to specify the angle of the branch pipe.

For tees/wyes the correlations used to calculate loss assume that the angle between the side branch and straight through flow pipes (labeled as A_s and A_{st} in Figure 2 above) can be at maximum 90 degrees. If the angle entered in the tee/wye junction exceeds 90 degrees between the side branch and straight through flow pipe, then the loss factor will be assumed to be equivalent to the loss with a 90 degree angle.

Figure 2 shows the Tee/Wye Properties window, with the Arrangement information at the bottom of the window. The a angle itself is specified on the Tee/Wye Model tab, where there are other options for how to model the tee.

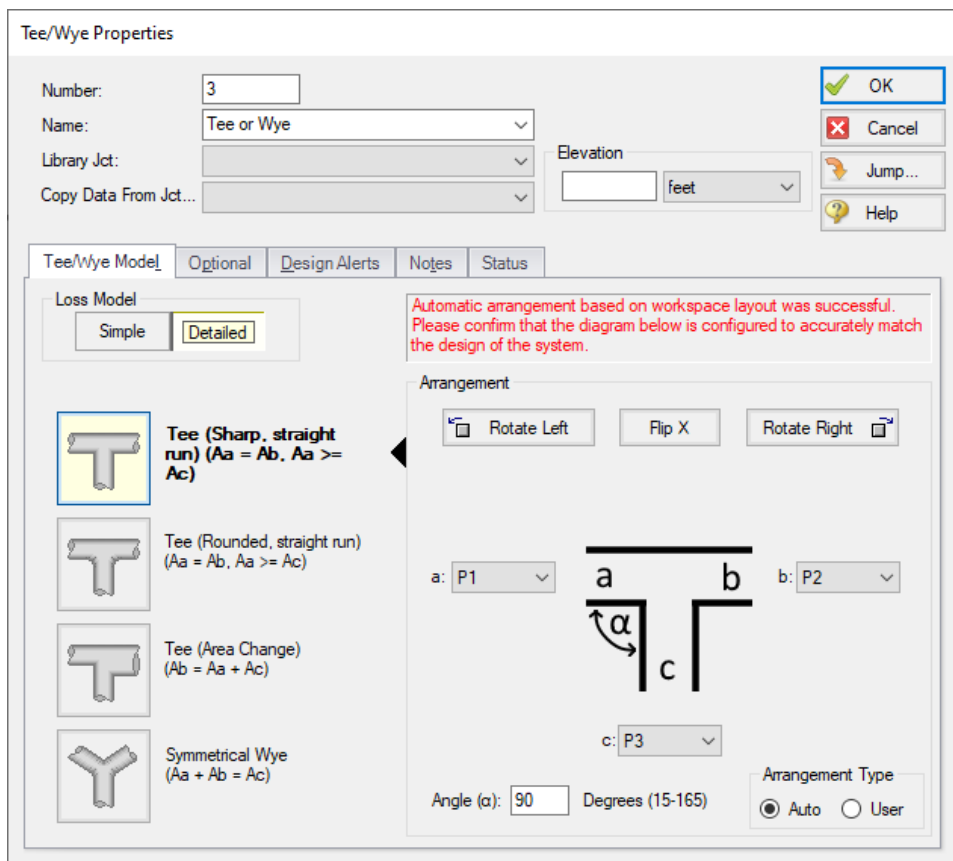


Figure 2: Tee/Wye Properties window allows you to specify the a, b and c pipes. The c pipe is always the branch pipe. This general method means you do not need to know the flow direction ahead of time.

Table 3: References for Tee/Wye junction correlations

Geometry	Type	Direction	Reference	Page
Constant Area, Sharp	Diverging Tee	Branch	Idelchik 2007	523
	Diverging Tee	Run	Idelchik 2007	525
	Converging Tee	Branch	Idelchik 2007	501-504
	Converging Tee	Run	Idelchik 2007	501-504
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	316*
Constant Area, $r/D = 0.1$	Diverging Tee	Branch	Idelchik 2007	531-533
	Diverging Tee	Run	Idelchik 2007	526
	Converging Tee	Branch	Idelchik 2007	512-514
	Converging Tee	Run	Idelchik 2007	512-514
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	315*
Area Sum, Sharp	Diverging Tee	Branch	Idelchik 2007	524
	Diverging Tee	Run	Idelchik 2007	526-527
	Converging Tee	Branch	Idelchik 2007	505-509
	Converging Tee	Run	Idelchik 2007	512-514
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	315*
Wye	Diverging Wye	Branch	Idelchik 2007	523
	Congering Wye	Run	Idelchik 2007	545
*Estimates have been made on available data				

Related Topics

K Factor Loss Model.....	234
Tee/Wye.....	295
Tee/Wye Junction Complexity.....	738

Related Blogs

[Give me some details about Detailed Tees!](#)

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.

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:

- **FL, FD, and Valve ID** - These inputs are available for the C_v or K_v loss models and allow for pressure loss correction factors to be calculated according to [ANSI/ISA-75.01.01-2012](#). FL and FD pertain to non-turbulent flow and have no impact on results for turbulent flow. Valve ID helps account for area changes between the connecting pipes.
- **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 (XTS Module Only)

The Transient tab is only visible if the [XTS Module](#) is active, and 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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233

K Factor Loss Model.....	234
Valve Modeling Closures with K Factors.....	302
Cv/Kv and Xt Loss Model.....	235
Sonic Choking.....	665
Subsonic vs. Sonic Pressure Losses.....	665

Valve Modeling Closures with K Factors

A closed valve would have an infinite K-factor. Since it is impractical to enter infinity in such cases, AFT Arrow uses the convention of $K = -1$ (negative one) to represent a closed valve.

Related Topics

K Factor Loss Model.....	234
Valve.....	299
Cv/Kv and Xt Loss Model.....	235
Sonic Choking.....	665

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 Arrow 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 Arrow 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 Arrow. 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

Junctions.....	221
General Junction Information.....	222
Junction Loss Models.....	233
K Factor Loss Model.....	234
Resistance Curve Loss Model.....	237
Troubleshooting Resistance Curves.....	736
Subsonic vs. Sonic Pressure Losses.....	665

Interface and User Options

This section focuses on the how the items within AFT Arrow's various menus and tabs can be used to customize the user's experience within the software. Topics discussed include the options present in each of the AFT Arrow 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	305
Printing	316
Primary Window Overview	318
Status Bar	319
Quick Access Panel	320
Model Status Light	329
Solution Progress Window	329
Workspace Window	329
Model Data Window	354
Output Window	359
Graph Results Window	371
User Options Overview	395

Menus and Toolbars

Many of the actions required to set up an AFT Arrow model can be completed from the various menus located at the top of the AFT Arrow 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 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.
- **Import Excel Change Data** - (Workspace and Model Data Only) - Imports data from Excel file which modifies certain input data.
- **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 Arrow runs the multiple scenarios/models automatically.
- **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 II Neutral File (.cii)** - [Import a CAESAR II Neutral File](#)
 - **GIS Shapefile (.shp)** - Allows for [Importing GIS Shapefile](#) (geospatial maps) as .shp file.
 - **EPANET File (.inp)** - [Import an EPANET File](#)
- **Export Piping Layout To**
 - **EPANET File (.inp)** - [Export an EPANET File](#)
- **Exit** - Exits AFT Arrow.

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Edit Menu

- **Undo** - (Workspace Only) - Undoes the previous workspace manipulation action. Arrow 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. Arrow 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 contents to the clipboard.
- **Copy X-Y Data** - (Graph Results Only) - Copies numerical data from graph to the Windows clipboard. Available in the [Graph Results window](#).
- **Copy Graphics** - (Workspace Only) - Copies selected [Workspace](#) content to the Windows clipboard. Formats supported include bitmap, windows metafile, and enhanced metafile.
- **Paste** - (Workspace Only) - Pastes clipboard pipes and junction contents (from previous cut or copy operation) into the Workspace model. New 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 Arrow. To enable this option, two junctions must be selected in the Workspace which Arrow 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.....	329
Model Data Window.....	354
Output Window.....	359

Graph Results Window.....	371
Menus and Toolbars.....	305

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.
- **Color Common Size Groups** - (Workspace with [ANS Module](#) Only) - Colors the pipes assigned to the selected Common Size Group(s) in the Workspace.
- **Color Design Requirements** - (Workspace with [ANS Module](#) Only) - Colors the pipes assigned to the selected Design Requirement in the Workspace.
- **Bookmarks** - (Workspace Only) - Allows creation and management of [bookmarks](#).
- **Find** - (Workspace Only) - Opens window whereby specific [pipes and junctions](#) can be found.

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Analysis Menu

Analysis Setup - Opens the [Analysis Setup](#) window.

Time Simulation ([XTS Module Only](#)) - Allows selection of either a steady-state run or a transient run.

Goal Seek & Control ([GSC Module Only](#)) - Allows selection of whether to use the Goal Seek & Control parameters in the current run.

Automated Sizing ([ANS Module Only](#)) - Allows selection of whether to use Network Sizing calculations in the current run.

Fluid Properties - Opens the [Fluid panel](#) from the Analysis Setup window.

Transient Control ([XTS Module Only](#)) - Opens the [Simulation Duration panel](#) from the Analysis Setup window.

Run Model - When the model is complete, the Run Model selection sends the model to the Solver for solution.

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Tools Menu

- **Scenario Manager** - Opens the [Scenario Manager](#) window.
- **Last Scenario** - Switches back to the previously loaded [scenario](#).
- **Generate Disconnected Scenario** - (Output Only) - Generates a scenario which is no longer connected into the [Scenario Manager](#) hierarchy and thus does not inherit changes from parents.
- **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.
- **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.

- **Cost Application Manager** - Opens the [Cost Application Manager](#) window. Cost data can be applied to all pipes and junctions.
- **Cost Summary** - Opens the [Cost Summary](#) window.
- **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 Arrow 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.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

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 insulations.
- **Edit Fittings & Losses** - Opens [Edit Fittings & Losses window](#) where you can add, delete, modify or review custom pipe fittings and losses.

- **Cost Library** - Opens the [Edit Cost Library](#) window where you can add, edit, or review cost libraries.
- **Energy Cost Library** - Opens the [Energy Cost Library](#) window to input time varying energy costs.
- **Library Manager** - Opens the [Library Manager](#) window.

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

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).

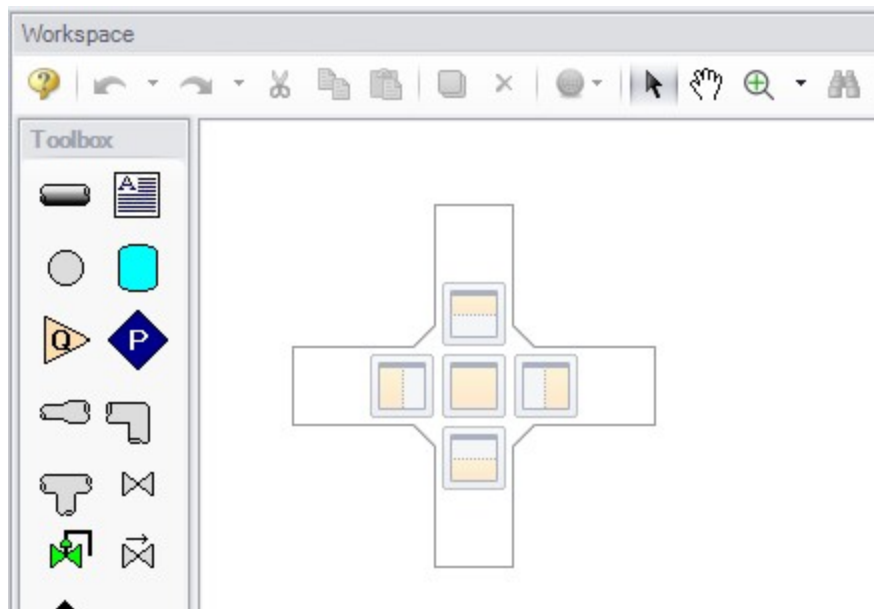


Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time

- **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](#).
- **Auto-rotate Icon** - (Workspace Only) - Rotates the selected icon(s) to align with the connected pipes if possible. 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 the [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 the [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 the [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](#).
- **Add/Remove 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.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Related Blogs

[AFT Responds to 2020 Customer Survey Feedback](#)

Window Menu

- **Workspace** - Makes the [Workspace window](#) active.
- **Sizing** - ([ANS Module Only](#)) - Makes the [Sizing 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.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Help Menu

- **Help File** - 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 on [aft.com](#)

- **Quick Start Video** - Opens ~15 minute quick start video on aft.com in English or metric units
- **Optimization Assistant** - ([ANS Module Only](#)) - Opens the Optimization Assistant to help identify input that will potentially effect the sizing results. Also appears if problems are identified when the model is run.
- **Keyboard Shortcuts** - Opens keyboard shortcut help page
- **Show Examples** - Opens help system with links to various example problems
- **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 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 Arrow release
- **AFT Notifications** - Opens page on aft.com where you can sign up for email notifications for new software updates and releases
- **About AFT Arrow** - Opens window in AFT Arrow with information on current AFT Arrow version, license number, etc.

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

Toolbars

The Toolbars offer quick access to the features used most frequently in AFT Arrow. There are actually several Toolbars in all. One is the Common Toolbar, and it is always shown. Its features are common to all Primary Windows. In addition, there is one Toolbar for each [Primary Window](#), offering features appropriate for work in that window. You only see two Toolbars at a time. The function of each icon in a given Toolbar appears when you pass the mouse over the icon button without clicking on the button. Each of the functions on the Toolbar is also available through the menu system.

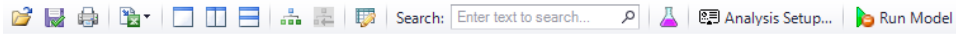


Figure 1: Common Toolbar



Figure 2: Workspace Toolbar



Figure 3: Model Data Toolbar



Figure 4: Output Toolbar



Figure 5: Graph Results Toolbar

Related Topics

Workspace Window.....	329
Model Data Window.....	354
Output Window.....	359
Graph Results Window.....	371
Menus and Toolbars.....	305

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.

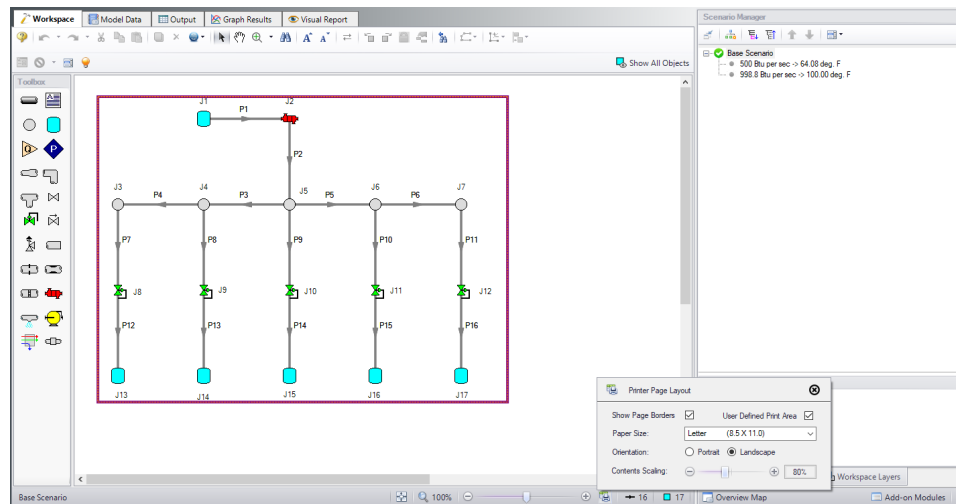


Figure 1: The Printer Page Layout options control how the model is split between pages on the Print Preview

- 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.

Related Topics

Interface and User Options.....	305
---------------------------------	-----

Primary Window Overview

AFT Arrow 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 Arrow 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.

If the [ANS Module](#) is activated, an additional Primary Window called the [Sizing window](#) will be available, which allows sizing information to be configured.

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

Related Topics

Interface and User Options.....305

Status Bar

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

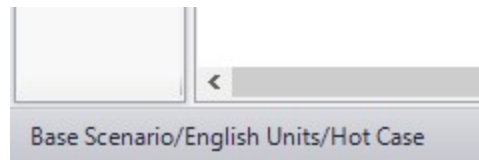


Figure 1: Current scenario displayed in the Status Bar

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.

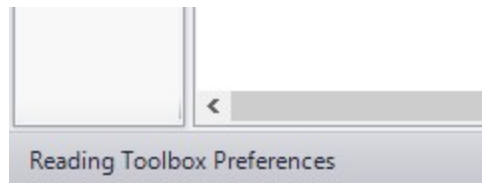


Figure 2: Status Bar displaying progress when opening a model

Related Topics

Interface and User Options.....	305
---------------------------------	-----

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

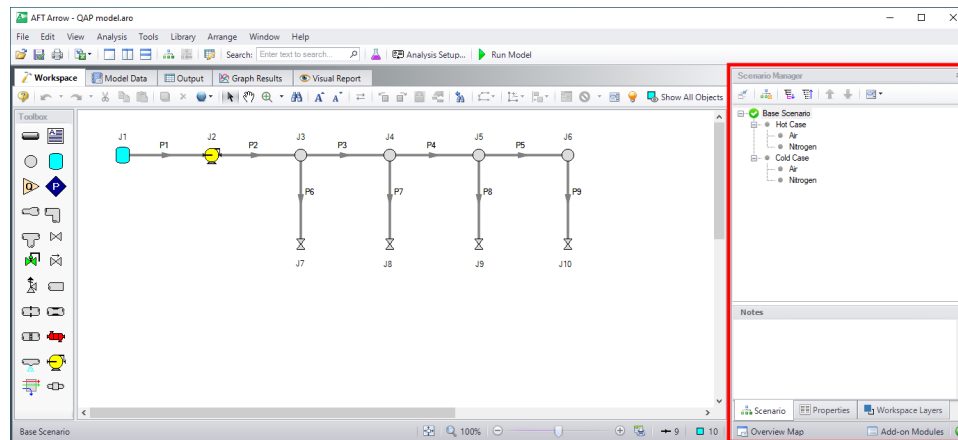


Figure 1: The Quick Access Panel

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.

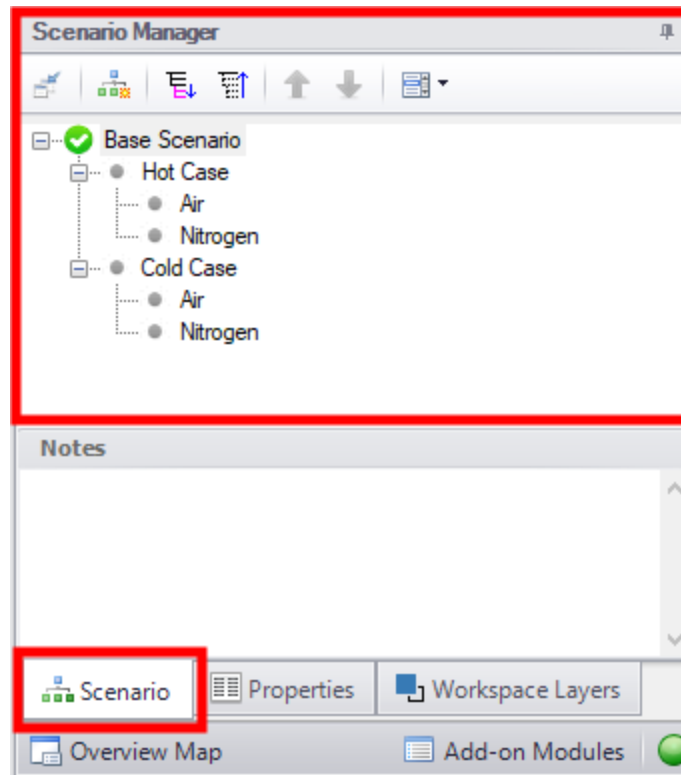


Figure 2: The Scenario Manager

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.

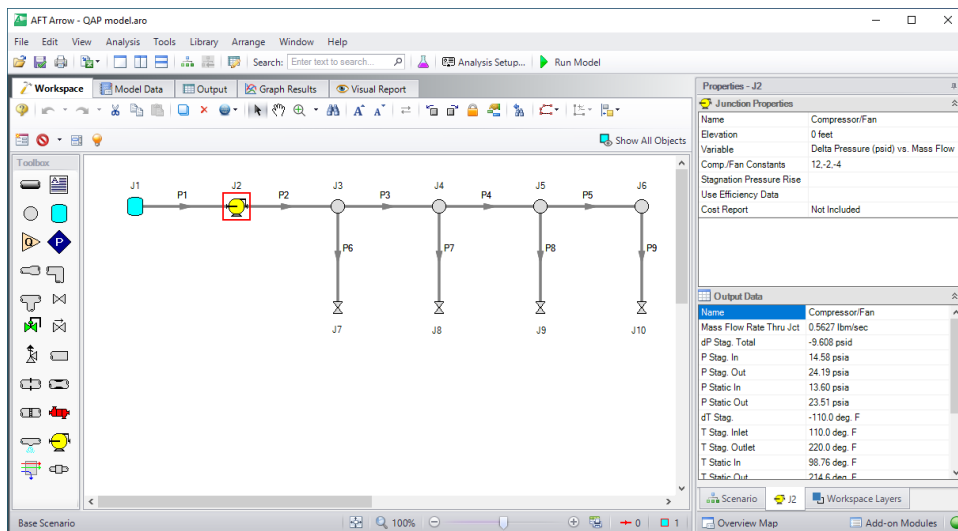


Figure 3: Viewing the Pipe/Junction Properties and Output Data from the Quick Access Panel

[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.

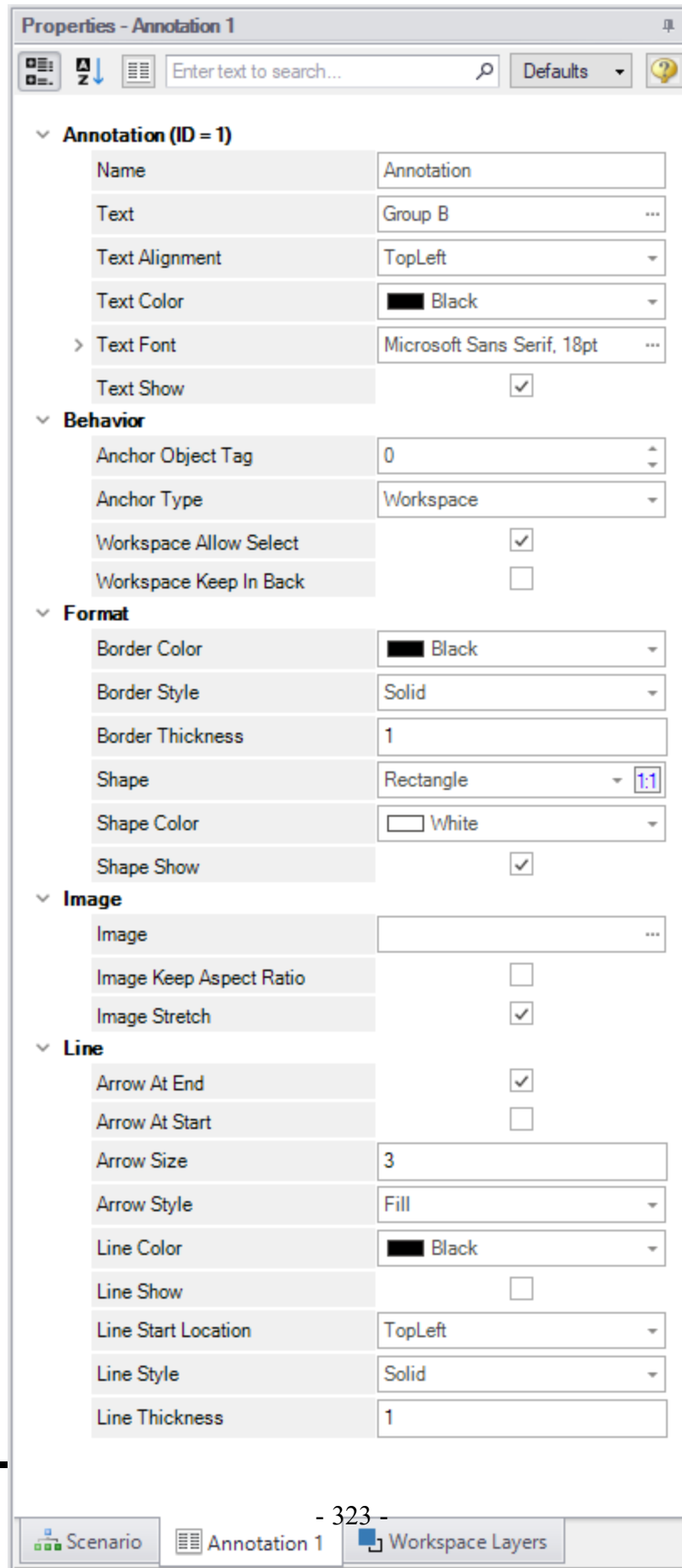


Figure 4: Annotation view for the Properties in the Quick Access Panel

Workspace Layers Tab

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

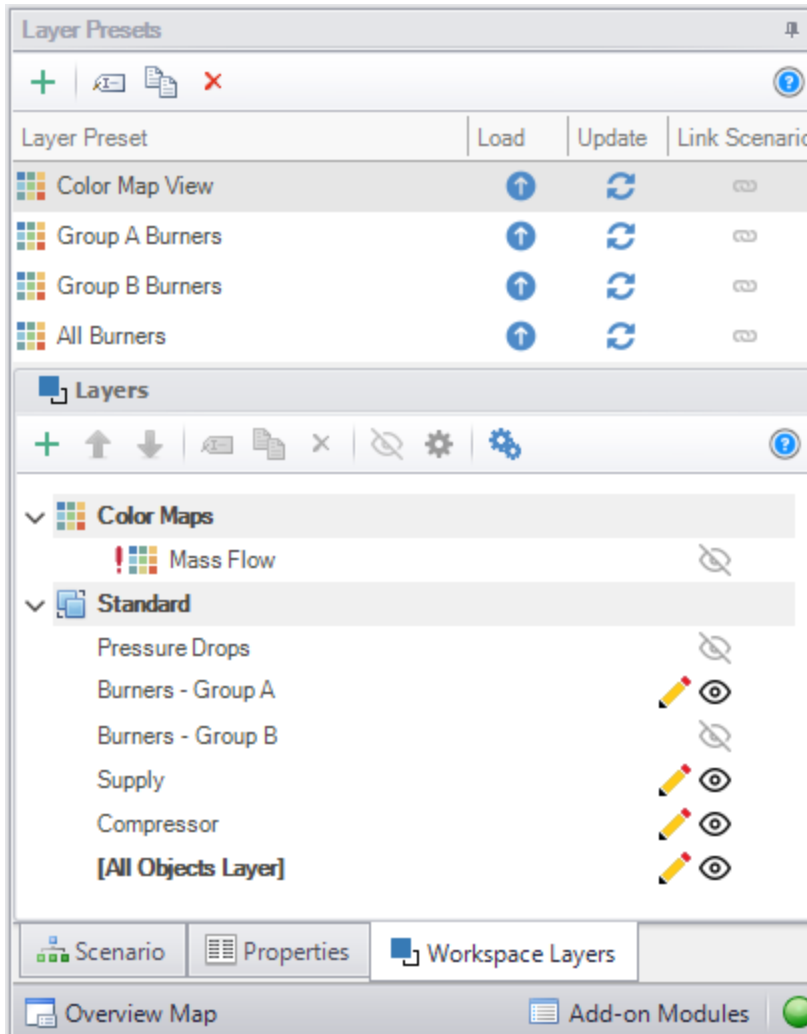


Figure 5: View of the Workspace Layers panel structure

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.

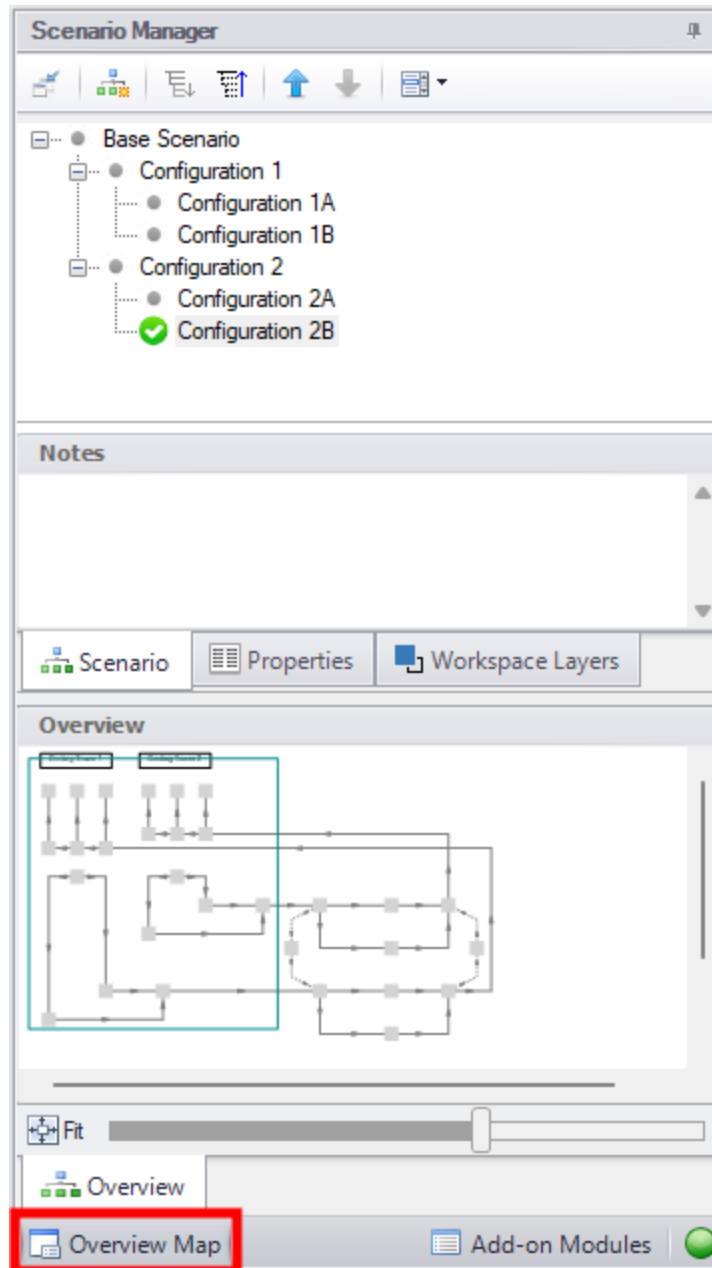


Figure 6: Workspace Overview Map

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.

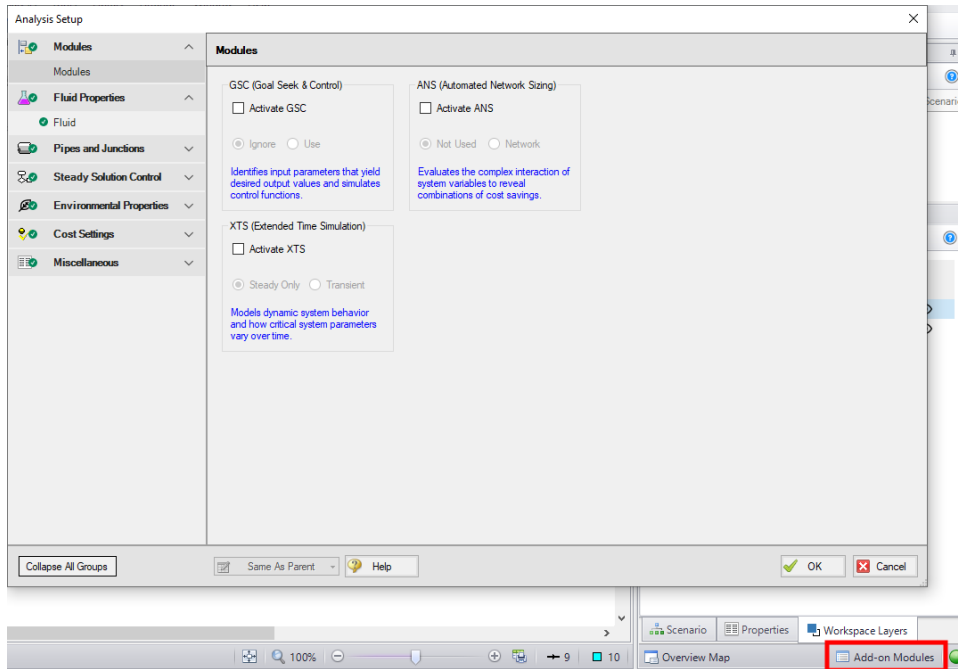


Figure 7: Activating the Modules from the Quick Access Panel

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.

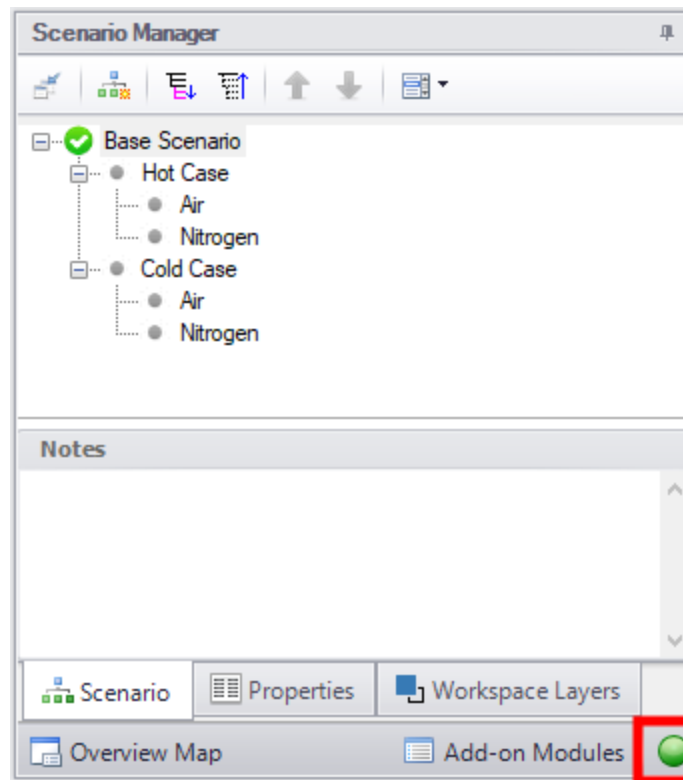


Figure 8: The Model Status Light is located at the bottom-right of the Quick Access Panel

Minimizing the Quick Access Panel

By default, AFT Arrow 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 Arrow 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.

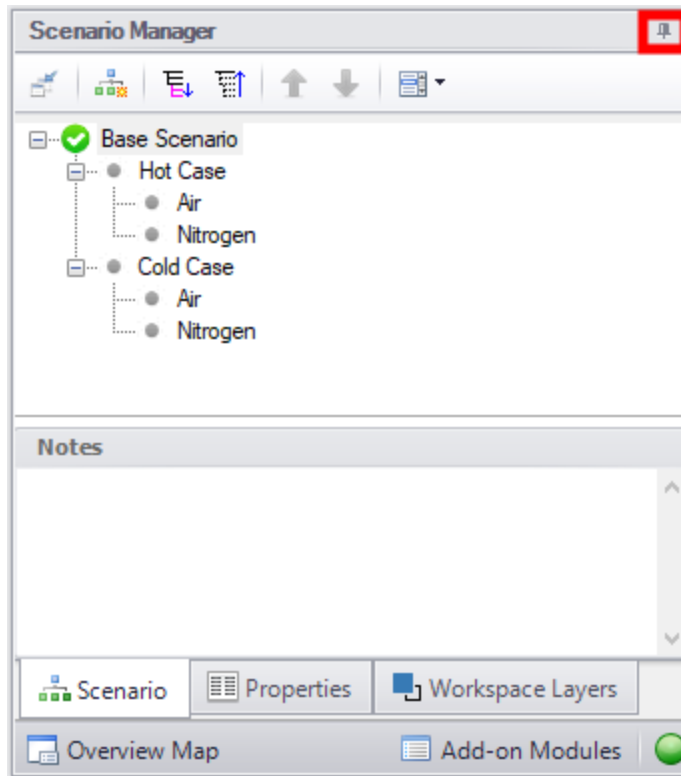


Figure 9: The Workspace with the Quick Access Panel expanded

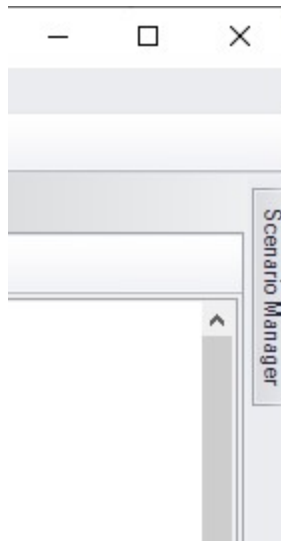


Figure 10: The Workspace with the Quick Access Panel collapsed

Related Topics

Interface and User Options.....	305
---------------------------------	-----

Model Status Light

The light at the right of the Status Bar located at the bottom of the AFT Arrow window, which indicates whether or not the model is ready to run.

Clicking on the Model Status Light will show the [Analysis Setup window](#).

Related Topics

Interface and User Options.....	305
---------------------------------	-----

I

Solution Progress Window

Solution Progress indicates the current state of the solution iterations.

The most out of [tolerance](#) junction or pipe is listed for the appropriate tolerances, along with general information about the solution method and related settings.

Note: Both absolute and relative criteria are always displayed. This can be helpful for selecting the appropriate tolerance criteria on difficult to converge models.

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.

When the model solution is converged, options to view [Output](#), [Graph Results](#), and [Workspace](#) become available.

Pausing the iterations allows the modification of certain [Steady Solution Control](#) parameters during the run, as well as viewing the iteration history. These options are available under Other Actions.

Related Topics

Interface and User Options.....	305
Iterations Panel.....	144
Review the Iteration History.....	742

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).

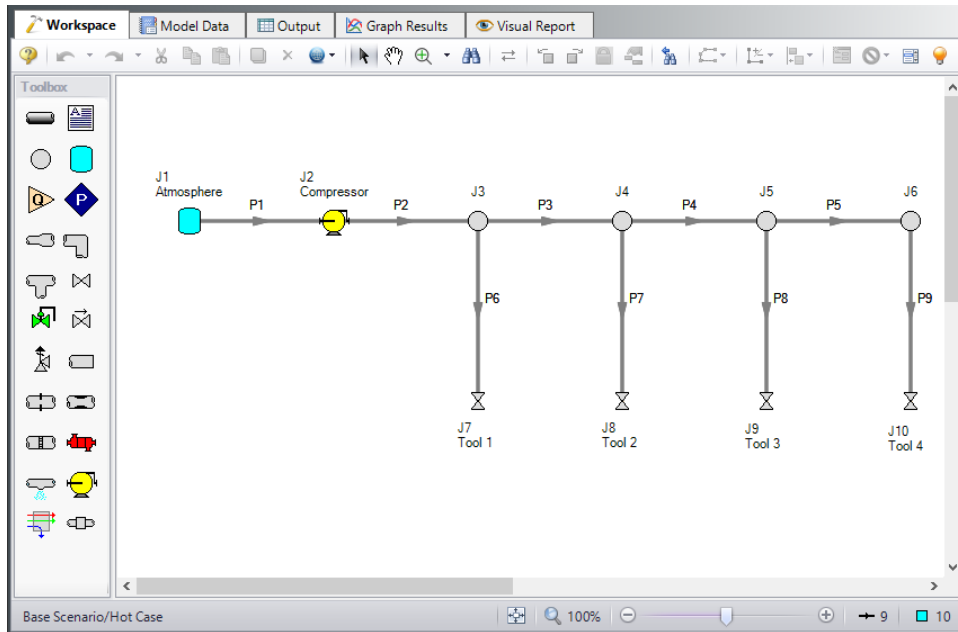


Figure 1: The Workspace window is where you assemble your model

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

Interface and User Options.....	305
Menus and Toolbars.....	305
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353

Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

Related Topics

Interface and User Options.....	305
Menus and Toolbars.....	305
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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.

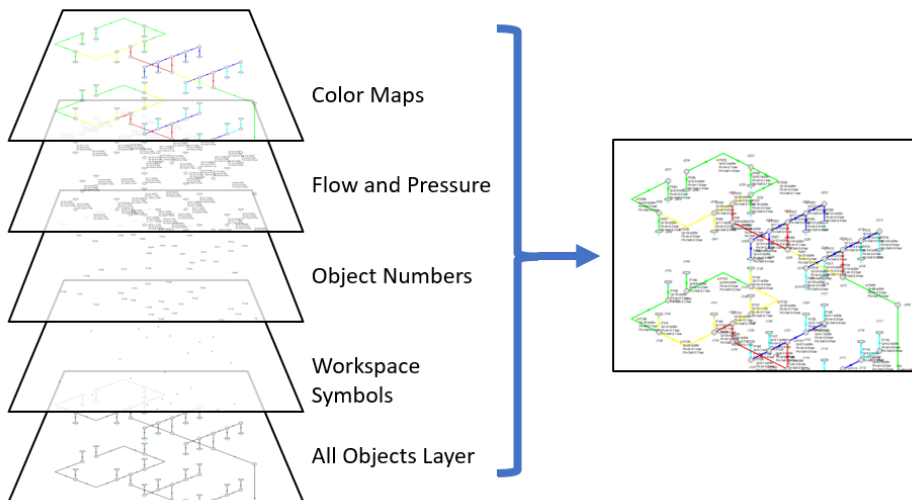


Figure 1: Information from all currently visible layers combines to form the view displayed on the Workspace

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.

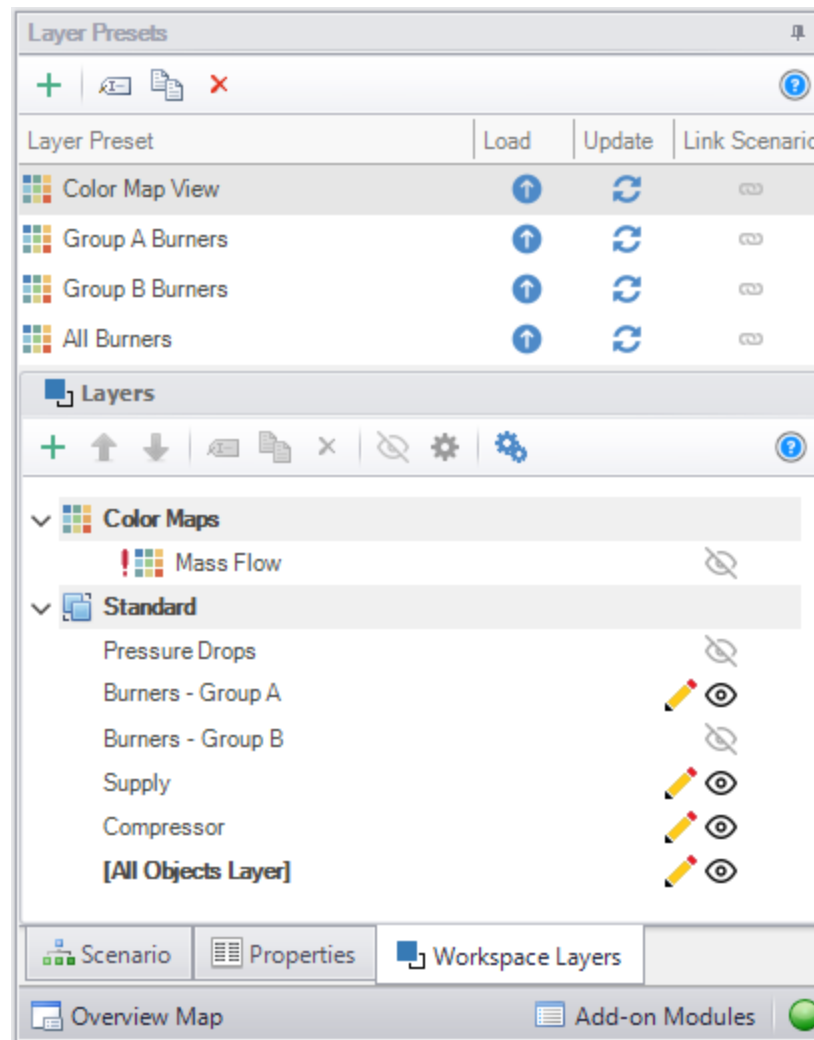


Figure 2: View of the Workspace Layers panel structure

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

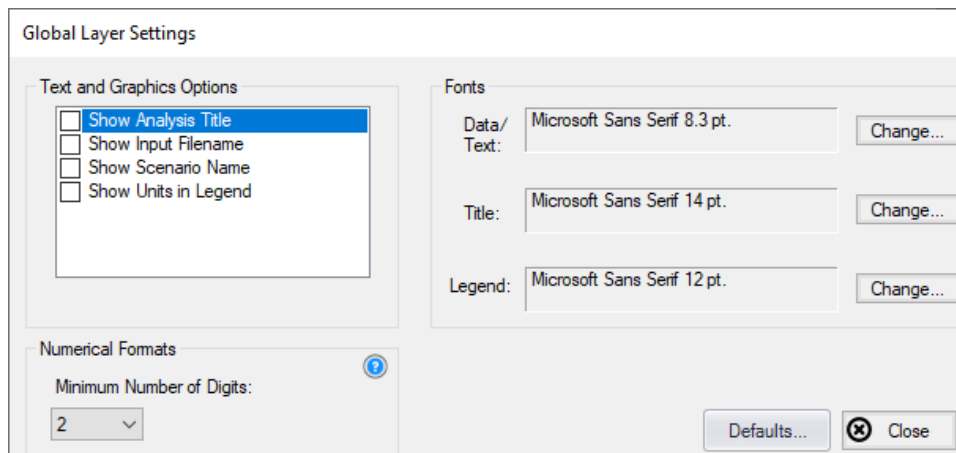


Figure 3: Global Layer Settings window

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](#).

(XTS Module Only) For using [XTS in the Workspace](#), the Show Time for Displayed Results option shows the time associated with the current transient results on the Workspace. This is particularly useful for printing.

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.

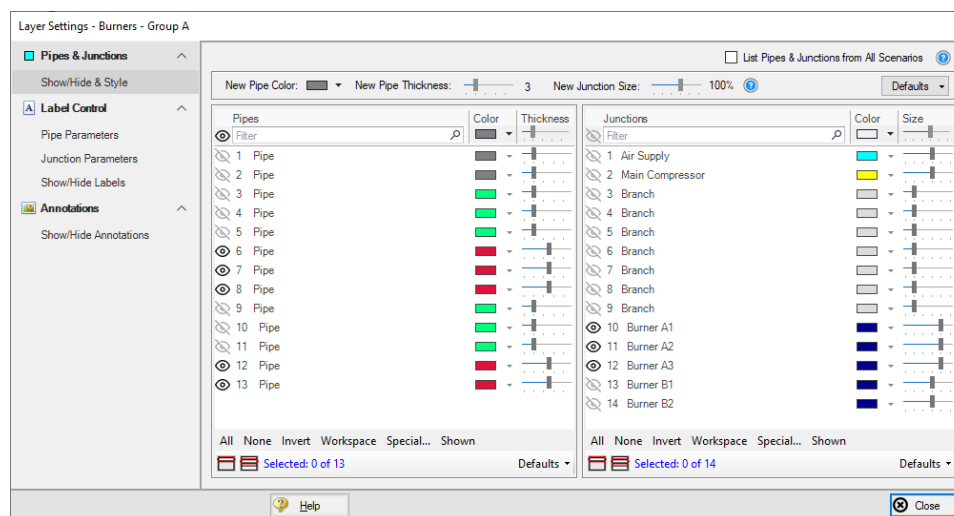


Figure 4: The Show/Hide & Style panel allows the appearance and visibility of pipes and junctions to be managed

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.

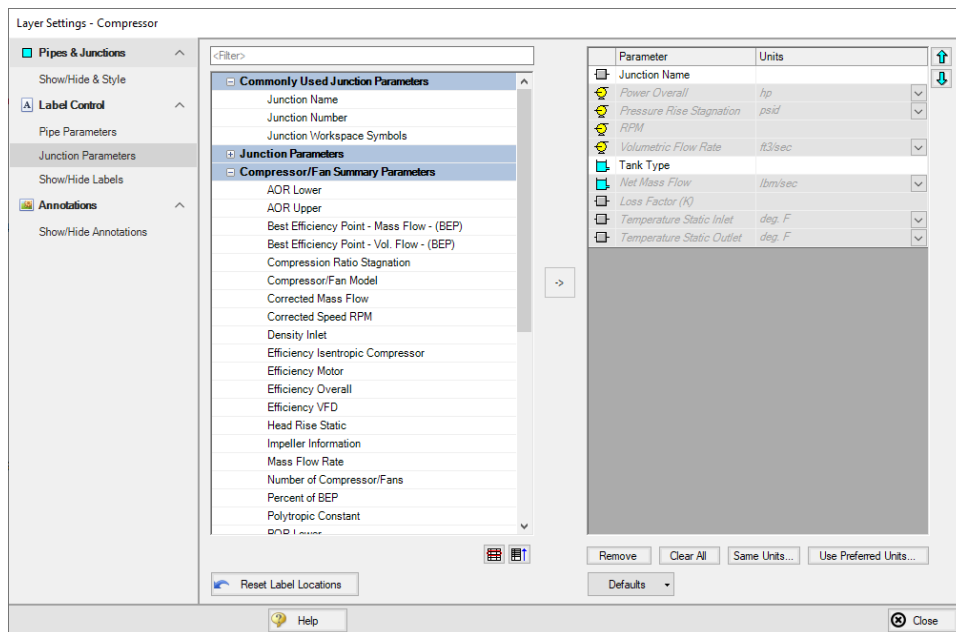


Figure 5: The Junction Parameters panel within the Label Control group of Layer Settings

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.

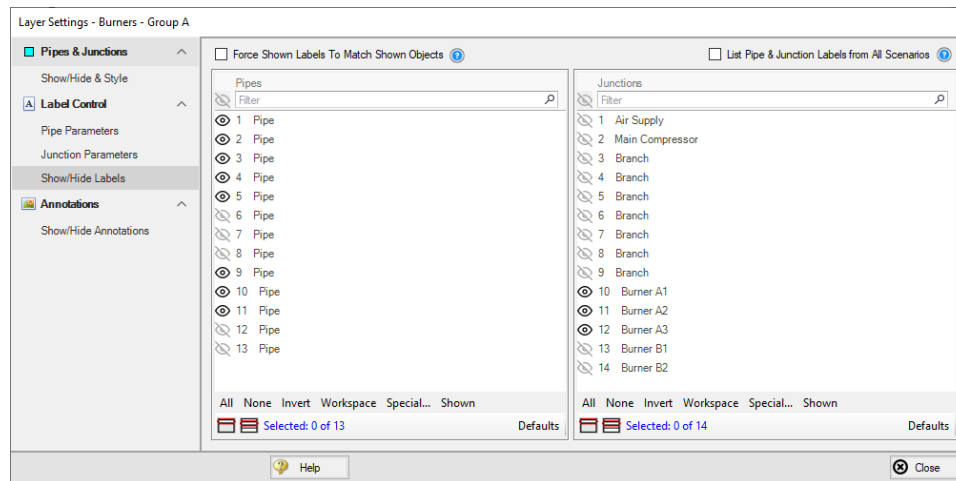


Figure 6: The Show/Hide Labels panel determines which objects will have visible labels for a given layer

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

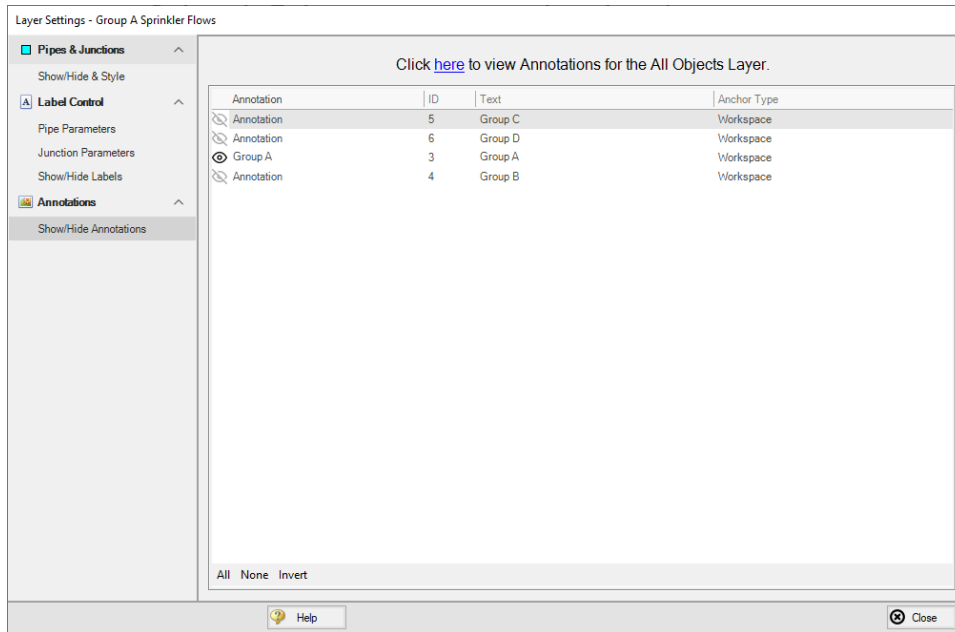


Figure 7: The Show/Hide Annotations panel shows all annotations that exist in the model file and allows their visibility to be controlled on a per-layer basis

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

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.

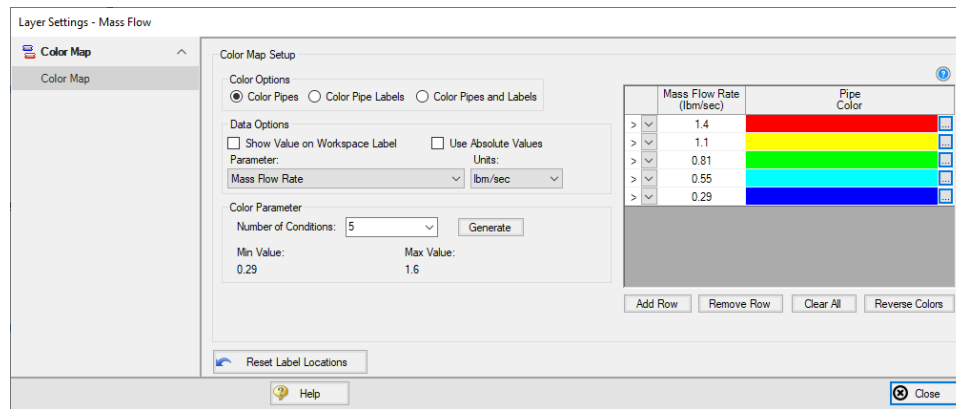


Figure 8: Layer Settings window for a Color Map

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. $>$, \geq , $=$, \leq , $<$) 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.

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

Workspace Pan.....	185
Moving Pipe and Junction Labels.....	186
Workspace Window.....	329
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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.

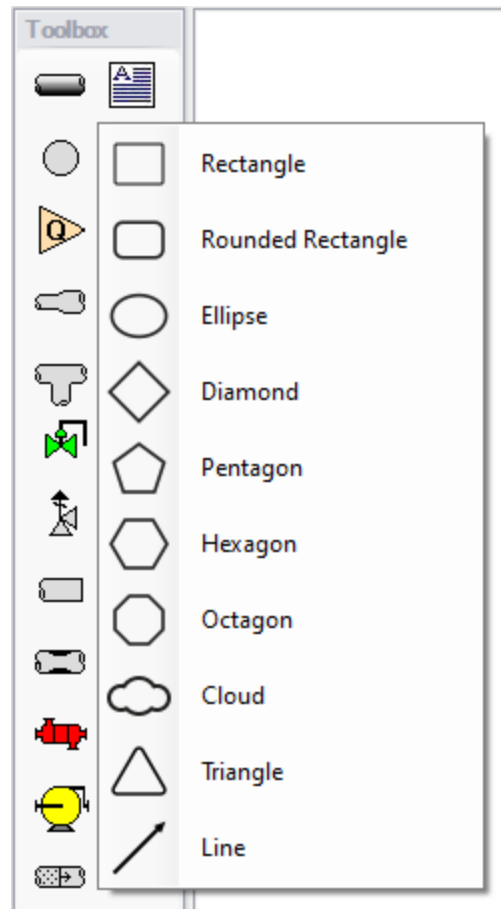


Figure 1: After clicking the Annotation Tool, select the shape for the new annotation

Properties

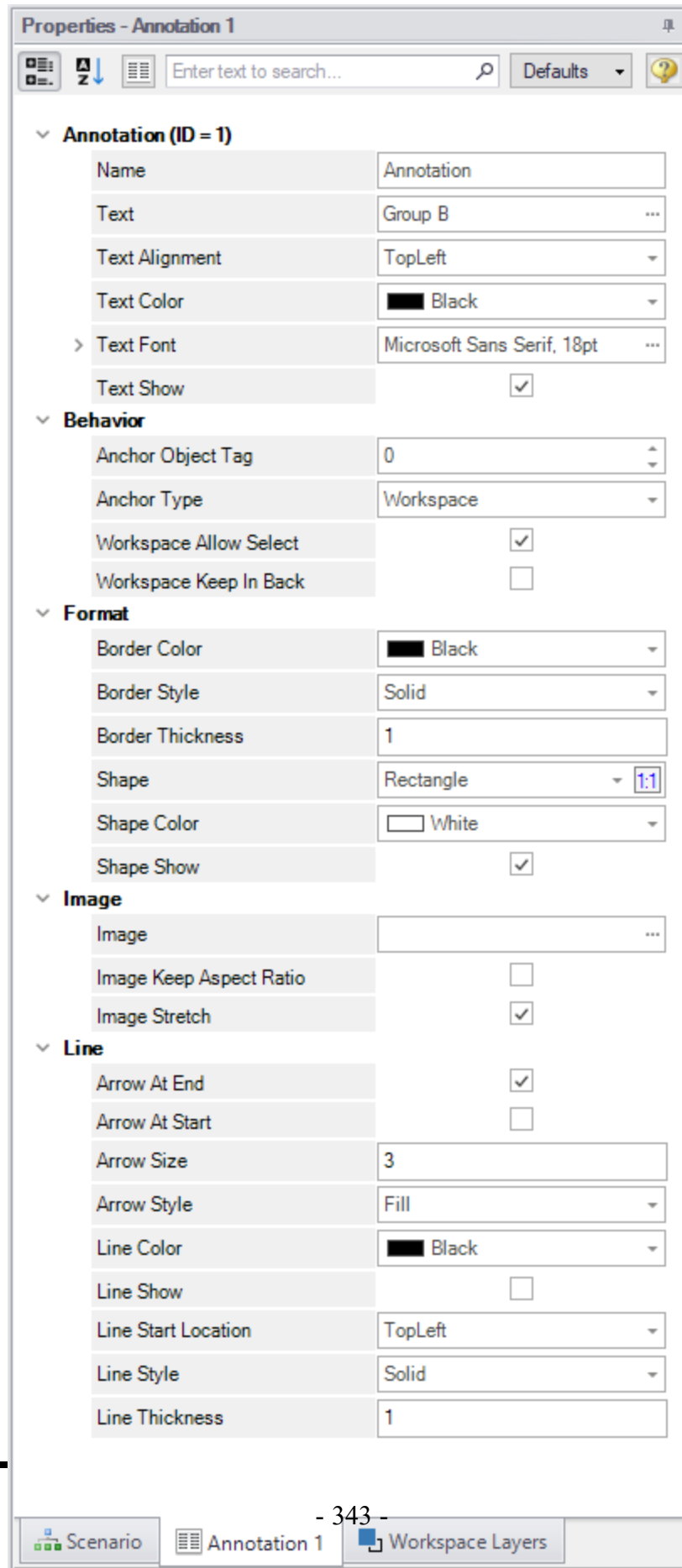


Figure 2: Annotation view for the Properties in the Quick Access Panel

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](#).

Related Topics

Toolbox.....	171
Creating Pipes - the Pipe Drawing Tool.....	173
Creating Junctions.....	177
Keyboard Shortcuts for Workspace.....	350
Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346

Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Extended Model Check.....	353
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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.

Arrow 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.

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

Merging Models

Two models can be merged together using the Merge command on the File menu. Here's how:

1. Open the first model
2. Choose Merge on the [File menu](#)
3. 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. In the [Scenario Manager](#) select the child scenario you want to merge.
3. Right-click the selected scenario and choose the Save Scenario to File Without Children.
4. Enter a name and click OK. This process creates a new model with the desired scenario as the base scenario.
5. Open the original model and merge in the new model file created in Step 4.

Related Topics

Workspace Window.....	329
-----------------------	-----

Workspace Layers.....	331
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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.

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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

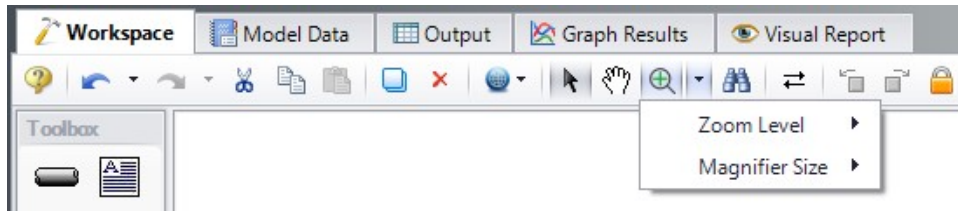


Figure 1: Magnifier tool on the Toolbar

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

Duplicate Special

Duplicate Special allows the model to be copy & pasted, while simultaneously applying other helpful settings as explained below. Additional options will be available when the [ANS module](#) is activated.

- **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.
- **Make Dependent Design Case** - (ANS Only) - Sets the duplicate objects as a [Dependent Design Case](#) of the duplicated model.
- **Keep Common Size Groups** - (ANS Only) - Only available if "Make Dependent Design Case" is checked. This adds the duplicate objects to the same [Common Size Groups](#) that the original objects were included in.

- **Clear Design Requirements** - (ANS Only) - This clears all [Design Requirements](#) from the newly created objects.
- **Link GSC Variables in New Pipes/Jcts** - (XTS Only) - Variables associated with duplicated objects are duplicated and linked together within the [Variables Panel](#).
- **Duplicate Goals** - (XTS Only) - Goals associated with duplicated objects are duplicated.

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353

Undefined Objects Window

This window is launched by clicking the "Define All Pipes and Junctions" item in the [Analysis Setup window](#) when there are undefined objects in the model.

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

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Raw Data Changes.....	353

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.
- 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.

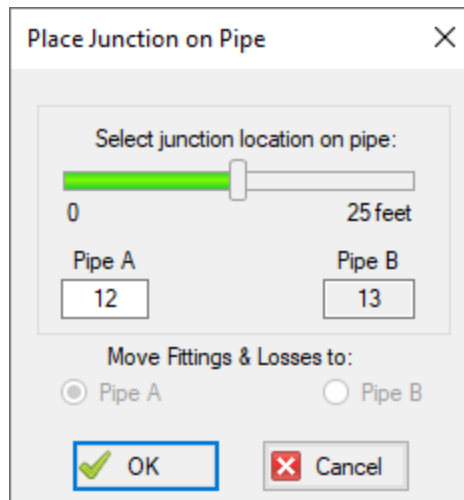


Figure 1: Split pipe keyboard shortcut

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
- When [ANS](#) is active, go to the Sizing window using CTRL + N

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

Toolbox.....	171
Creating Pipes - the Pipe Drawing Tool.....	173
Creating Junctions.....	177
Annotations.....	341
Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Extended Model Check.....	353
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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.

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Raw Data Changes.....	353
Duplicate Special.....	348

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 pump curves or junctions resistance curves, and later change that x-y data. The question is whether you want to have only the current junction you are working on updated, or all junctions that use that same base data.

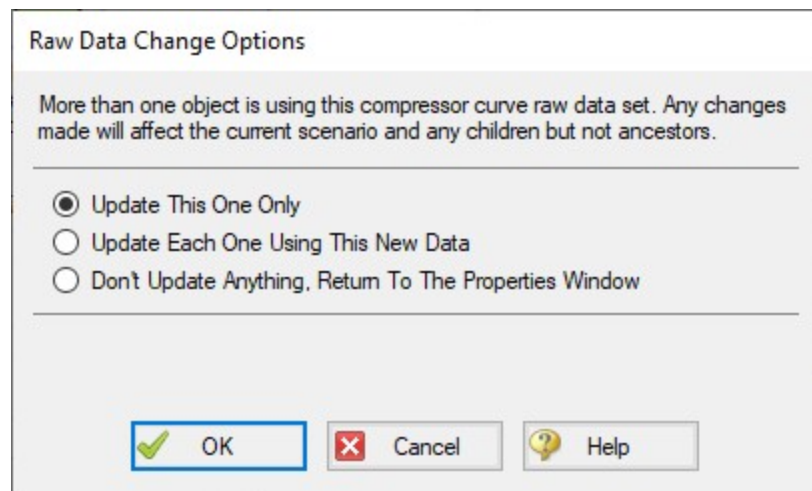


Figure 1: Raw Data Change Options

Related Topics

Workspace Window.....	329
Workspace Layers.....	331
Merging Models.....	346
Renumbering Workspace Objects.....	345
Scale/Flip Workspace.....	347
Magnifier Tool.....	347
Annotations.....	341
Extended Model Check.....	353
Keyboard Shortcuts for Workspace.....	350
Undefined Objects Window.....	349
Duplicate Special.....	348

Model Data Window

The Model Data window offers a text-based perspective on all engineering information in the pipe flow model. This window is 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 resize 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 resize the text displayed on screen.

Workspace Model Data Output Graph Results Visual Report

General Notes

Segments Per Pipe= 2
Mach Number Increment= 0.01
Pressure Tolerance= 0.0001 relative change
Mass Flow Rate Tolerance= 0.0001 relative change

Pipes Pipe Fittings & Losses Pipe Detail Summary

Pipe	Name	Pipe Defined	Length	Length Units	Hydraulic Diameter	Hydraulic Diam. Units	Friction Data Set	Roughness	Roughness Units	Losses (K)	Initial Flow	Initial Flow Units	Junctions (Up, Down)
1	Pipe	No	1	feet	2.067	inches	Standard	0.0018	inches	0			1, Nor
2	Pipe	No	25	feet	2.067	inches	Standard	0.0018	inches	0			None,
3	Pipe	Yes	25	feet	2.067	inches	Standard	0.0018	inches	0			3, ↓

Branch Compressor/Fan Tank Valve

Branch	Name	Object Defined	Elevation	Elevation Units	Initial Pressure	Initial Pressure Units	Database Source	Special Condition	Boundary Flow (+ = in/- = out)	Boundary Flow Units	(Pipe #) K In, K Out
3	Branch	Yes	0	feet				None			(P2) 0, 0
4	Branch	Yes	0	feet				None			(P3) 0, 0
5	Branch	Yes	0	feet				None			(P4) 0, 0

Base Scenario/Hot Case 100%

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 Loss Data table, accessed with an adjacent tab. The Pipe Loss Data 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.

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 the far left column where the pipe or junction ID numbers are shown. Alternatively, you can use the [Global Edit](#) windows opened from the Edit menu.

- **GSC Module tables** - When using the [GSC Module](#), if GSC data has been entered in the Goal Seek and Control Manager, the variable and goal data will be displayed in the General section in a "Goal Seek & Control" table.
- **XTS Module tab** - When using the [XTS Module](#), if transient events have been defined for any junction, the Initiation of Event settings and raw transient data will be listed for each transient event in the Junction section in a Transient Data tab.

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 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 View menu (or Toolbar).

The Model Data window contents can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Saved directly to an Adobe PDF file
- Sent to the printer

Related Topics

Interface and User Options.....	305
Menus and Toolbars.....	305
Model Data Control.....	356
Bill of Materials.....	358

Related Blogs

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

Model Data Control

The Model Data Control window allows you to customize what data to show for pipes and for each junction type (see Figure 1). You can also specify which pipes and junctions to show in the Show Selected Pipes/Junctions tab. The window can be opened from the [Tools menu](#) (or Toolbar).

Pipe Data Display

The data to be shown for 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.

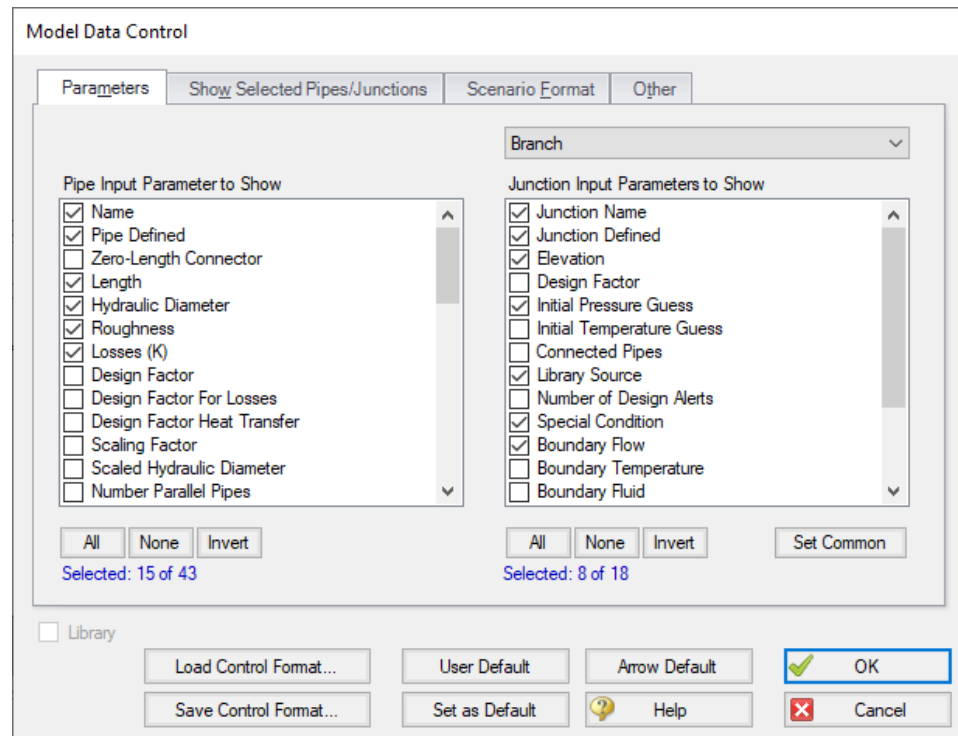


Figure 1: The Model Data Control window allows selection of the parameters to show in the Model Data window

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

It is possible to specify whether junction curve data is to be displayed with the Model Data, whether to display [Design Alerts](#), and whether to include a [Bill of Materials](#).

Scenario Format

A powerful Model Data feature when using scenarios is to show all direct ancestor data in the Model Data window 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.

Related Topics

Model Data Window.....	354
Bill of Materials.....	358

Bill of Materials

A bill of materials is a summary of all materials, components, and fittings that have been specified for a project.

A bill of materials can be added to the Model Data Summary section by opening Model Data Control, and checking "Include Bill of Materials" on the Other tab.

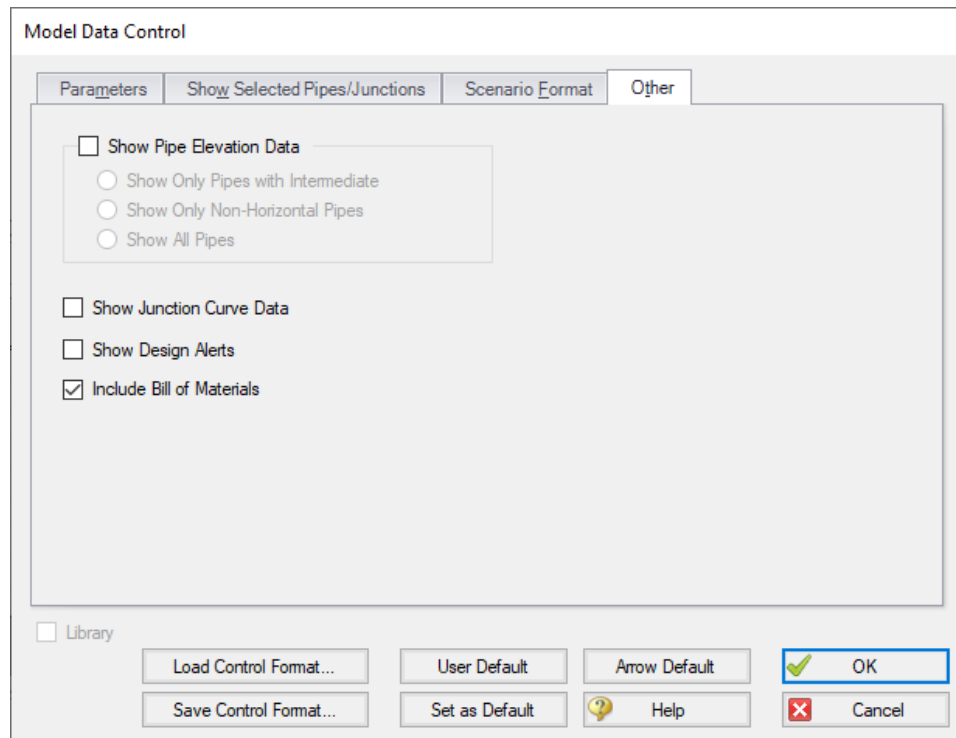


Figure 1: Include Bill of Materials

Related Topics

Model Data Window.....	354
Model Data Control.....	356

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 run, 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. [Output Control](#) lets you specify the parameters, their units, and the order of their display in the Output window.

Data in the Output window can be copied to the clipboard, saved to a text file, exported to a file in a format for direct importing into a Windows spreadsheet package, saved directly to an Adobe PDF file, or sent to the printer. The printed output follows the same flexible, organized, and accessible format as the Output window itself.

The 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.

Interface and User Options

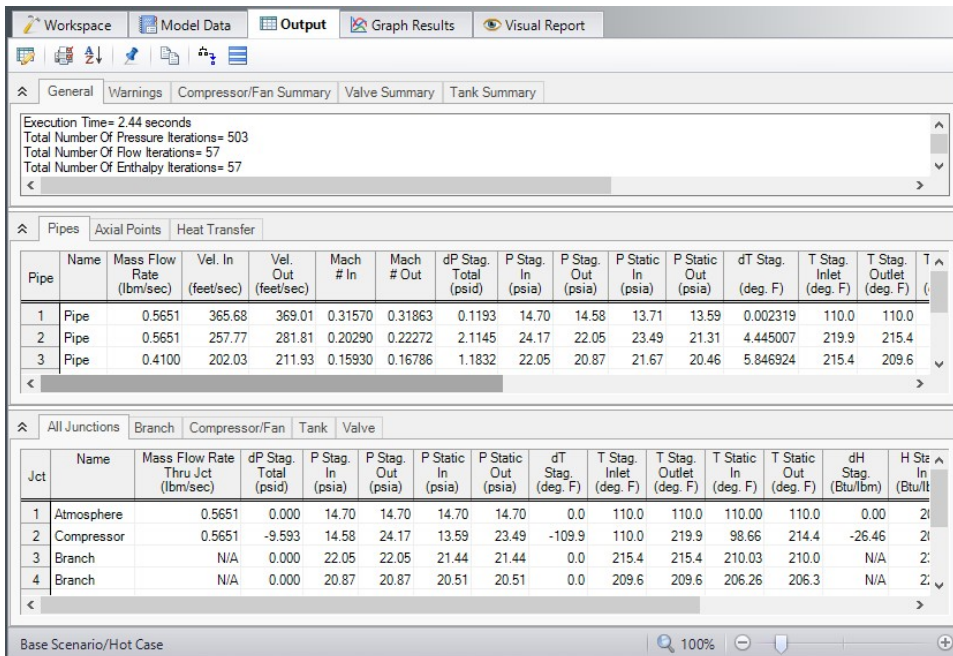
The Sort window opened from the [Arrange menu](#) (or Toolbar) offers the ability to sort the Pipe Results or Junction Results tables according to the values in any of the columns.

The [Transfer Results to Initial Guesses](#) feature on the Edit menu (and Toolbar) sets all pipe and junction initial 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 the right mouse button down 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 resize 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 resize the text displayed on screen.



The screenshot shows the 'Output' window with the following data:

Execution Time= 2.44 seconds
Total Number Of Pressure Iterations= 503
Total Number Of Flow Iterations= 57
Total Number Of Enthalpy Iterations= 57

Pipes Table:

Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T
1	Pipe	0.5651	365.68	369.01	0.31570	0.31863	0.1193	14.70	14.58	13.71	13.59	0.002319	110.0	110.0	
2	Pipe	0.5651	257.77	281.81	0.20290	0.22272	2.1145	24.17	22.05	23.49	21.31	4.445007	219.9	215.4	
3	Pipe	0.4100	202.03	211.93	0.15930	0.16786	1.1832	22.05	20.87	21.67	20.46	5.846924	215.4	209.6	

All Junctions Table:

Jct	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. Inlet (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/lb)
1	Atmosphere	0.5651	0.000	14.70	14.70	14.70	14.70	0.0	110.0	110.0	110.00	110.0	0.00	21
2	Compressor	0.5651	-9.593	14.58	24.17	13.59	23.49	-109.9	110.0	219.9	98.66	214.4	-26.46	21
3	Branch	N/A	0.000	22.05	22.05	21.44	21.44	0.0	215.4	215.4	210.03	210.0	N/A	2:
4	Branch	N/A	0.000	20.87	20.87	20.51	20.51	0.0	209.6	209.6	206.26	206.3	N/A	2:

Figure 1: The Output Window

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](#).
 - **Add Note** - A text note can be added to any Output cell. Cells with notes will be marked with a red triangle in the top corner. Hovering over a cell with a note allows the note to be viewed, or edited by clicking in the note.
 - **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 model input, all output windows are erased. This prevents you from changing your input, forgetting to rerun it, and mistaking the previous Output window for the current model's results.

Results Diagrams

Results diagrams are available for both compressor/fan results (see Figure 2) and pipe heat transfer results(see Figure 3). 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 for compressors/fans or the heat transfer tab for pipes with heat transfer.

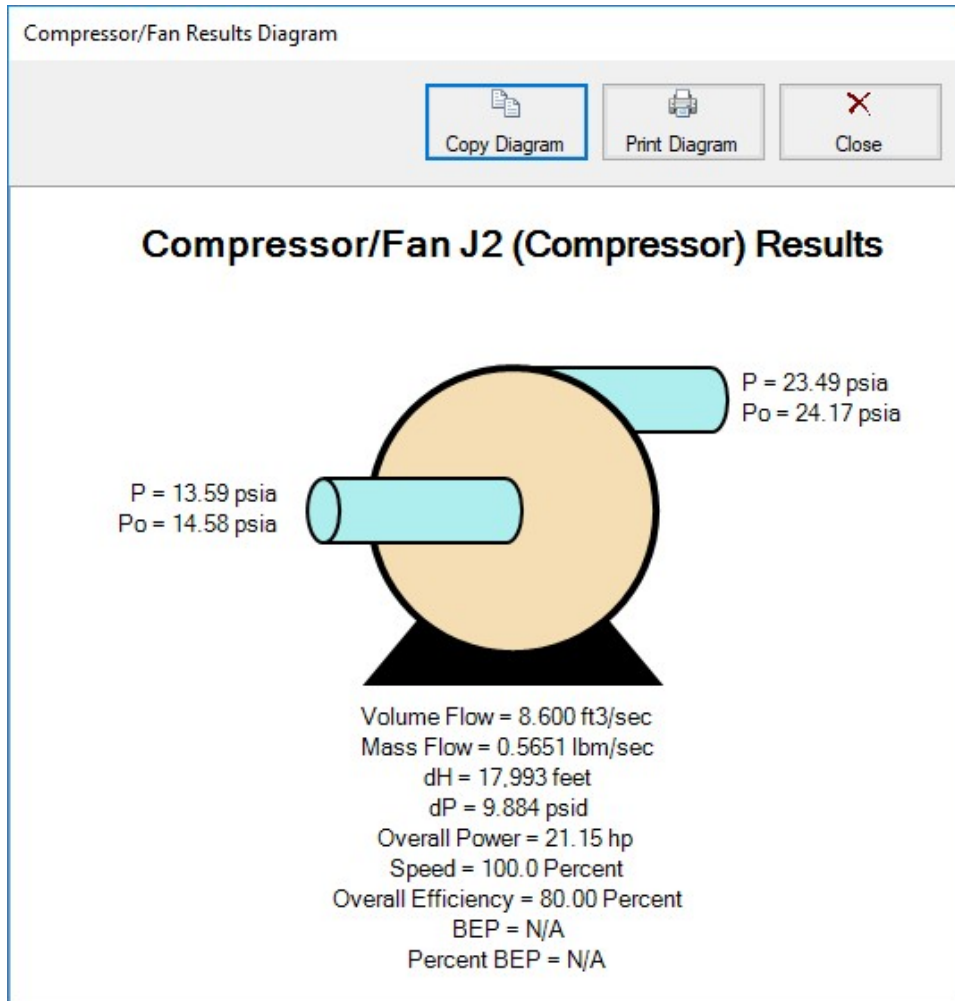


Figure 2: Compressor/Fan Results Diagrams

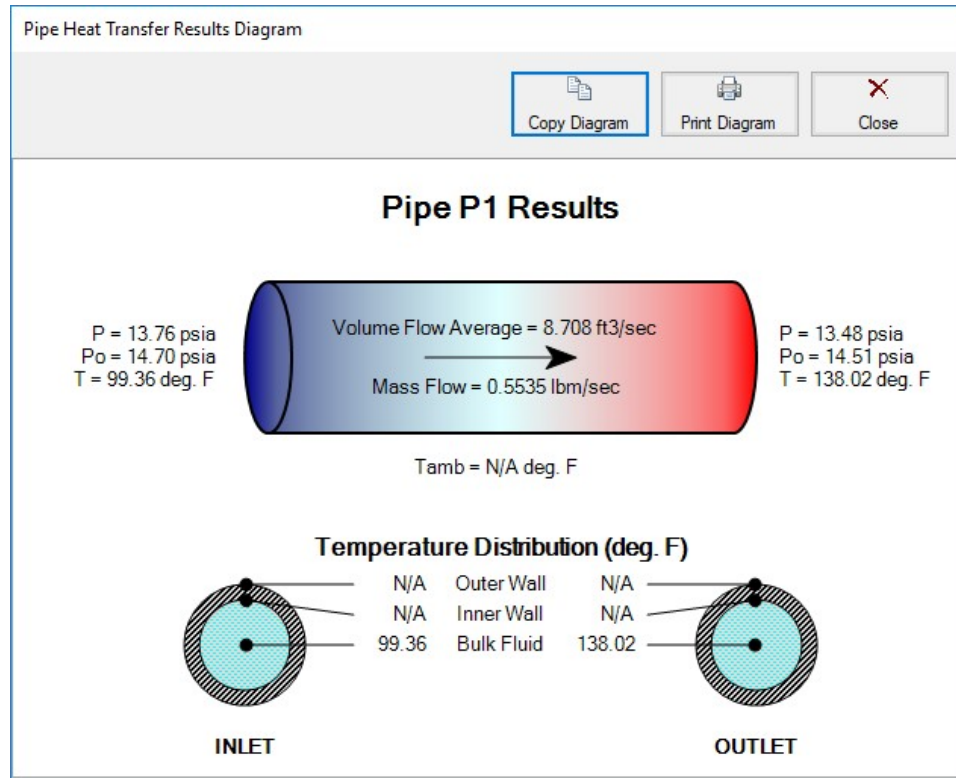


Figure 3: Pipe Heat Transfer Results Diagram

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
- 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

Interface and User Options.....	305
Menus and Toolbars.....	305
Output Control.....	364
Output Parameters.....	370

Applied Standards..... 371
 Use the Output Window Sort Feature..... 737

Output Control

The Output Control window (see Figure 1) allows you to customize the parameters shown on the Output window. The control features are grouped on numerous tabs.

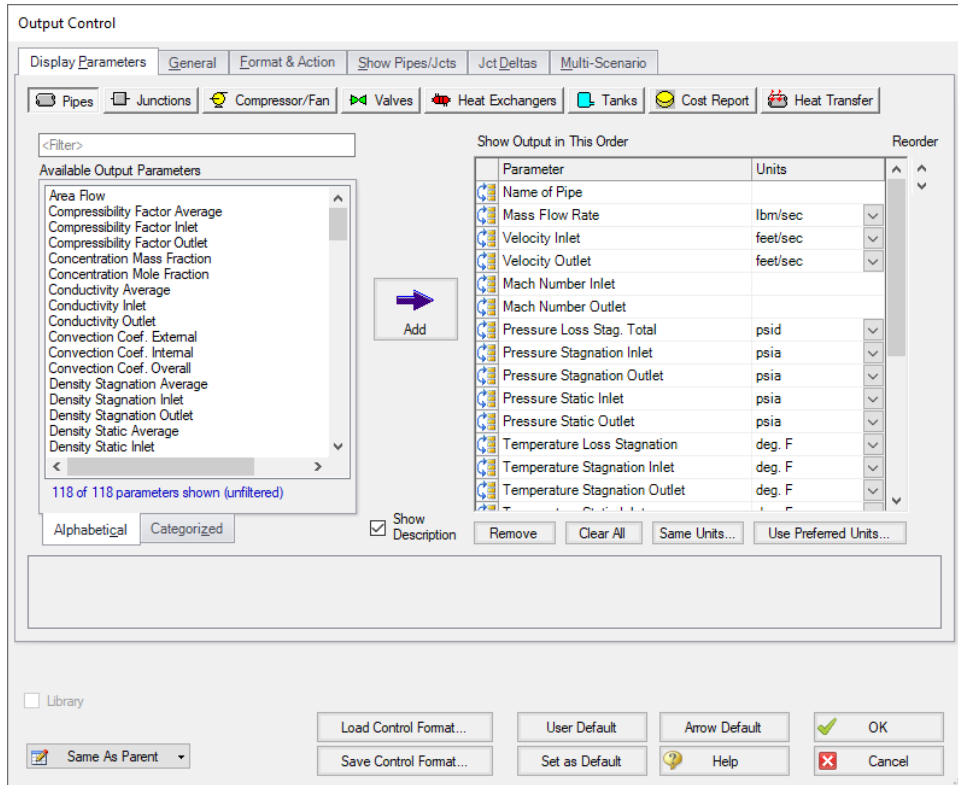


Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe parameters are shown. The Junction and other parameters function the same.

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. It is also possible to search the available parameters using the Filter box above the list.

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 dropdown 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 Heat Transfer Output**
- **Cost Report**

In addition if the [ANS module](#) is active the Pipe Sizing tab will be available, as well as Design Requirements and Library tabs for each pipe/junction which has design requirements or libraries assigned to them.

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, the iteration time for AFT Arrow to arrive at a converged solution is reduced. The initial guess can also be transferred manually by Selecting the [Transfer Results to Initial Guesses](#) from the Edit menu. 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 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
 - 9.876E-02
- **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
 - 9.876E-02
- **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.

Options to adjust the colors for the headers and design alert highlighting are also available.

Cost Report Formatting

The [Cost Report](#) monetary numerical format can be specified.

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 folder 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

Junction Deltas

You can have AFT Arrow calculate the difference between two junctions for a particular parameter. These results are displayed in the General Section. A list of Junction Deltas is created by selecting the upstream and downstream junctions, the parameter and units of interest, then adding it to the list by clicking the Add >> button. Some parameters will require that either the inlet or outlet value for the junction be specified. The list will contain the parameter symbol and the two junction numbers with "in" or "out" in parentheses. Different parameters can be used for each item in the list so you can, for example, choose pressure difference between junctions 2 and 10 and also temperature difference between junctions 45 and 16.

Multi-Scenario

You can view output from multiple [scenarios](#) concurrently in the Output window. To do this, select the Multi-Scenario tab. This tab is only visible when at least one child scenario exists. From this tab, choose which scenarios you would like to view concurrently by clicking on the Select Scenarios button located at the bottom right of the Scenario Order and Colors box under the Multi-Scenario tab. The color of each scenario is automatically generated and can be regenerated by selecting "Reset All Colors". You can manually select the background color for the Output data of a given scenario by clicking on the ... button located in the row corresponding to the scenario.

You can also order the scenarios to appear in the Output window according to your preferences. To do this, arrange the scenarios in the order you would like them to appear by using the turquoise arrows at the bottom left corner of the Scenario Order and Colors box. Then, select how

you would like the data to be displayed by clicking the appropriate choice under the Display Current Scenario section. Figure 2 shows the Output window with multiple scenarios displayed.

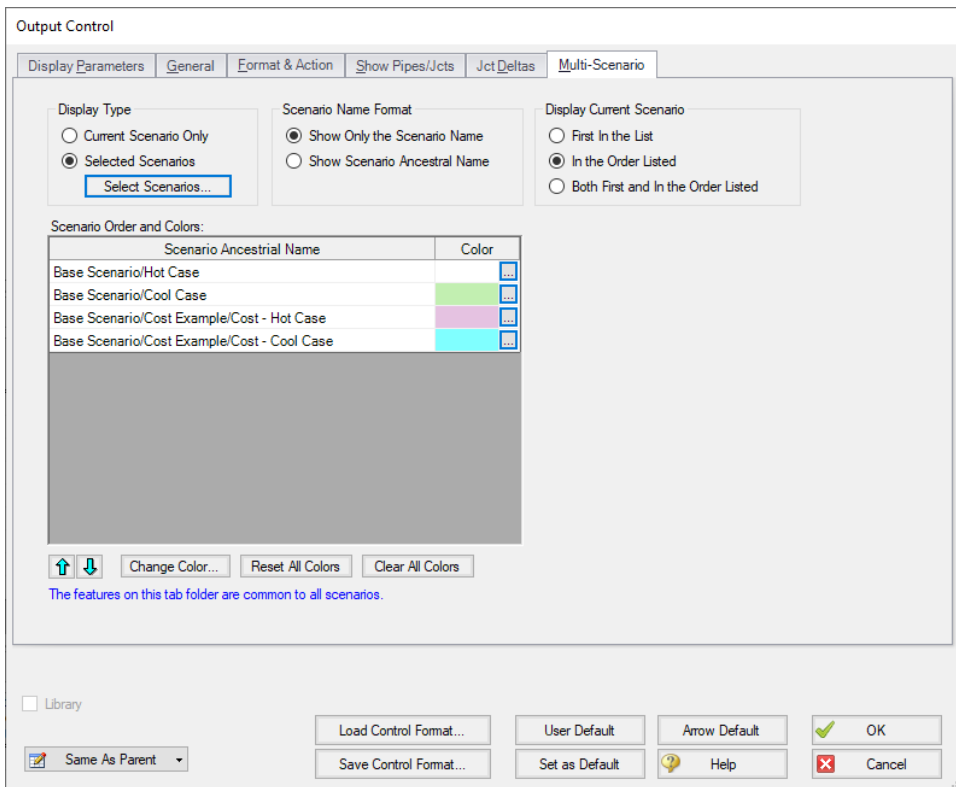


Figure 1: Output Control window selections for multi-scenario output

The screenshot shows the Output window with two tables. The top table is for Pipes and the bottom table is for Junctions. Both tables display data for multiple scenarios (Hot Case, Cool Case, Cost - Hot Case, Cost - Cool Case) and are color-coded by scenario.

Pipe	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 - Hot Case	Pipe	0.5700	369.2	372.7	0.3188	0.3218	0.1217	14.70	14.57	13.70	13.56	2.382E-03
1 - Cool Case	N/A	0.5980	310.3	312.8	0.2977	0.3001	0.1054	14.70	14.59	13.82	13.70	4.139E-03
1 - Cost - Hot Case	N/A	0.5665	366.7	370.0	0.3166	0.3195	0.1200	14.70	14.58	13.71	13.58	2.344E-03
1 - Cost - Cool Case	N/A	0.5979	310.3	312.8	0.2976	0.3001	0.1054	14.70	14.59	13.82	13.70	4.127E-03
2 - Hot Case	Pipe	0.5700	260.4	280.3	0.2050	0.2233	2.126	24.13	22.01	23.44	21.26	14.95
2 - Cool Case	N/A	0.5980	217.0	235.4	0.1917	0.2083	1.853	23.96	22.11	23.36	21.45	1.413
2 - Cost - Hot Case	N/A	0.5665	258.5	281.6	0.2035	0.2230	2.119	24.16	22.04	23.47	21.29	6.949
2 - Cost - Cool Case	N/A	0.5979	217.0	235.3	0.1917	0.2083	1.853	23.96	22.11	23.36	21.45	1.413

Jct	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)
1 - Hot Case	Atmosphere	0.5700	0.000	14.70	14.70	14.70	14.70	0.000	110.0	110.0	110.0
1 - Cool Case	N/A	0.5980	0.000	14.70	14.70	14.70	14.70	0.000	1.343E-05	1.343E-05	1.343E-05
1 - Cost - Hot Case	N/A	0.5665	0.000	14.70	14.70	14.70	14.70	0.000	110.0	110.0	110.0
1 - Cost - Cool Case	N/A	0.5979	0.000	14.70	14.70	14.70	14.70	0.000	1.343E-05	1.343E-05	1.343E-05

Figure 2: Output window with multiple scenarios displayed

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, uncheck 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. Arrow has built-in default parameters, units and settings which you can choose by clicking the Arrow Default button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of Arrow'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 and junction deltas) 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, allowing standardized reporting.

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 Topics

Output Window.....	359
Output Parameters.....	370
Applied Standards.....	371
Use the Output Window Sort Feature.....	737
Note No Scenarios have been set in the Multi-Scenario Output display.....	725

Output Parameters

A variety of parameters are available to display in the Output window depending on the junctions and loss models used in the model.

A description of each parameter may be seen by double-clicking the header in the Output Window table, and is also displayed below the Output Parameters table when that parameter is selected in the [Output Control window](#) (Figure 1).

Additional information can be found below for some of the output parameters.

Output Window Table Header:	Flow Area
Visual Report Abbreviation:	A
Description:	Pipe cross-sectional flow area

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 Flow Coefficients](#) topic.

Equivalent Orifice Area/Diameter

The Equivalent Orifice Area and Equivalent Orifice Diameter is calculated using the sharp-edged [orifice loss model](#) with an assumed discharge coefficient of 0.6. The equivalent orifice calculations assume that flow is choked at the orifice.

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 spray discharge. 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

Output Window.....	359
Output Control.....	364
Applied Standards.....	371
Use the Output Window Sort Feature.....	737

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.

Related Topics

Output Window.....	359
Output Control.....	364
Output Parameters.....	370
Use the Output Window Sort Feature.....	737

Graph Results Window

The Graph Results window, shown in Figure 1, allows you to generate high quality printed graphs.

The Graphing panel offers [several types of standard graphs](#) for compressor/fan vs. system, profile, and transient ([XTS Module Only](#)) information. Users can also control graph content to plot information for multiple parameters, pipes, paths, or scenarios simultaneously.

Results can be graphed as linear or log versus linear.

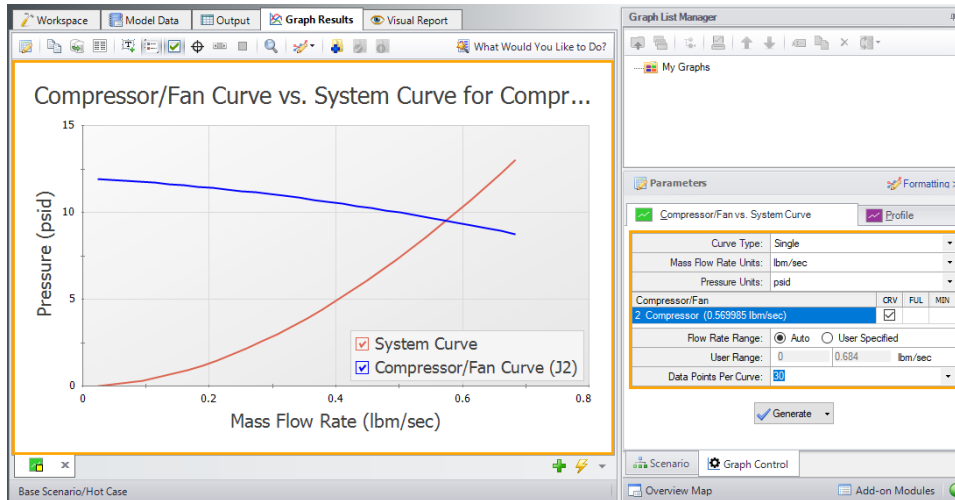


Figure 1: The Graph Results window offers full-featured plot generation

The Graphing panel lets you control the content of the graph and offers several types of standard graphs.

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 a 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 Item in Current Tab** – Regenerates the graph in the currently active tab.
- **Load Graph Item in New Tab** – Creates a new tab and regenerates the graph there.
- **Load Folder items** - Regenerates all the Graphs in a Folder in separate tabs, in the same tab side by side, or in the same tab as stacked graphs, depending on the selection.

- **Keep Folder Display Layout** - Saves the current layout when multiple graphs are loaded on the same tab.
- **Reset Folder Display Layout** - Resets the Graph List Folder to the default layout when multiple graphs are loaded on the same tab.
- **Close Open Folder Items** - Closes any graphs that were opened from the current folder.
- **Delete Folder Items** - Deletes all graphs in the current folder.
- **Print Preview All Graphs in Folder** - Displays a print preview of all items in the current folder.
- **Move Graph Item Up** – Moves the selected Item up one level.
- **Move Graph Item Down** - Moves the selected Item down one level.
- **Rename Graph Item** - Changes the name of the Item.
- **Duplicate Graph Item** – Creates a copy of the selected Item with a new name.
- **New Folder** – Creates a new Folder where more Graphs can be saved to.
- **Delete Graph Item** – Removes Item from the Graph List Manager.
- **Move Graph Item To Folder** – Moves the selected Item to a previously existing Folder or a new Folder.

Graph Results Toolbar

The Graph Results Toolbar also has many options that are available 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 outlined and current Graph with the Graph in the Graph List that has a gray background
- **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
 - Right-clicking on the x-y data set provides further options to find the maximum or minimum in the data set, create an annotation on the graph for a data point, or to copy an individual data point
- **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 datapoint.
- **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.
- **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" on the toolbar.

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:

- **Green** – Compressor/Fan vs. System
- **Profile** – Purple
- **Blank Graph** – 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.

Cross-Plotting Features

The graphical data can be exported to a file for later import as a cross plot against results for a different model. For example, let's say you would like to see the effect on the system pipe pressure drop when you change a valve loss factor. You could run the Solver for one value of K and export the results to a file. Then you could rerun the Solver for a new value of K and import the results from the previous case for cross plotting. These features are accessed using the Save Graph Data and Import Graph Data items on the Graph Results File menu.

The Graph Results window is always erased when you make changes to model input. Rerun the model to graph the new results.

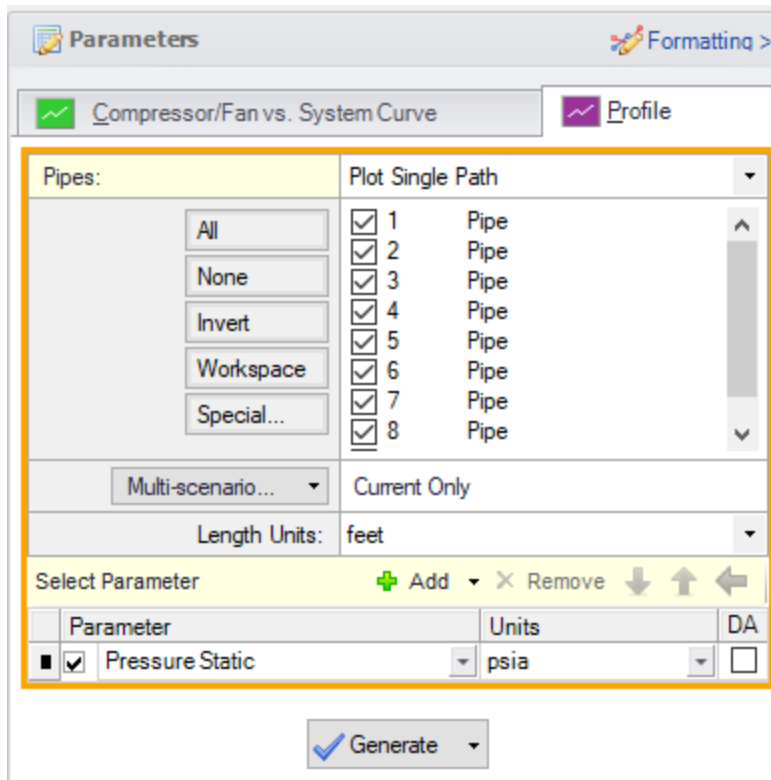


Figure 2: The Select Graph Parameters window controls the Graph Results window

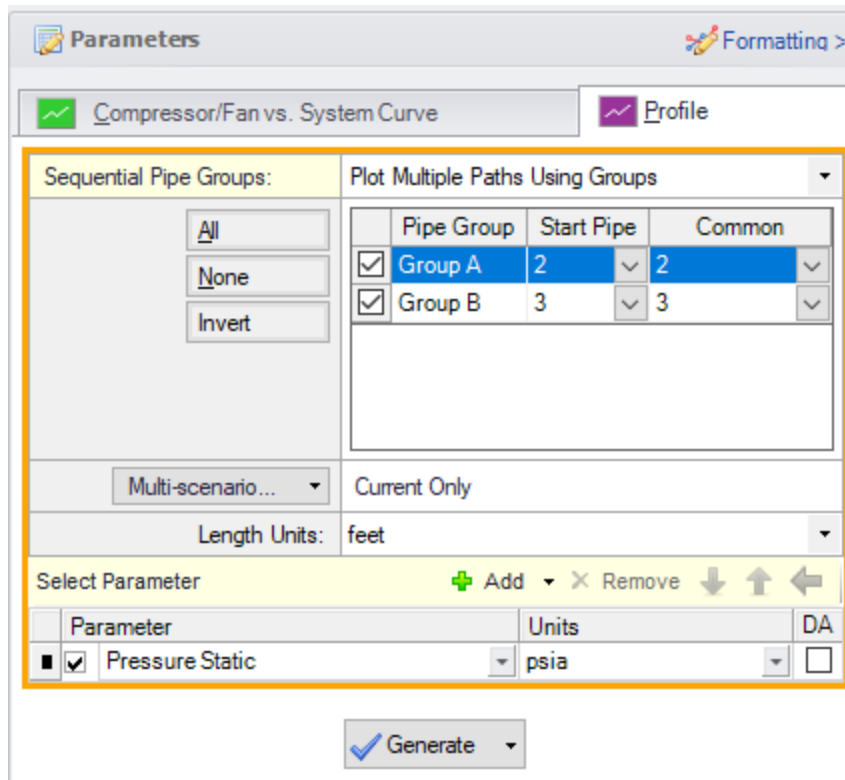


Figure 3: Multiple paths can be cross-plotted using predefined groups

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 directly to Excel using the [Excel Export Manager](#)
- Exported to a delimited file suitable for direct importing into spreadsheet software

Related Topics

Interface and User Options.....305

Menus and Toolbars.....	305
Quick Graph Formatting Window.....	378
Graph Types.....	378
Compressor Fan vs. System Curve Methods.....	381
Basic Graph Tutorial.....	382
Multi Scenario Compressor Fan Vs. System Curves Tutorial.....	390

Related Blogs

[Quick Guide to Custom Output](#)

Quick Graph Formatting Window

The Quick Graph Formatting window is accessed by clicking the Formatting button in the Quick Access Panel, or from the Tools menu.

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

Changes can be undone by using the Reset button at the bottom of the window, or can be returned to the default formatting by using the Arrow Default or User Default buttons at the bottom of the window.

Related Topics

Graph Results Window.....	371
Graph Types.....	378
Compressor Fan vs. System Curve Methods.....	381
Basic Graph Tutorial.....	382
Multi Scenario Compressor Fan Vs. System Curves Tutorial.....	390
XTS Quick Graph.....	609

Graph Types

AFT Arrow offers several types of graphs to help users better understand and interpret their data. Each type of graph is given a brief description below.

Compressor/Fan vs. System Curve

This graphing option allows you to plot a Compressor/Fan curve versus a system curve for a single Compressor/Fan or group of Compressor/Fans.

Note: The Compressor/Fan Curve vs. System Curve is only available if there is a Compressor/Fan in the model

You can create Compressor/Fan vs. system curves for complex parallel flow networks. Composite Compressor/Fan curves for Compressor/Fans in parallel or in series can also be also generated. If you create a composite Compressor/Fan curve, you can also show single Compressor/Fan curves on the same graph. If efficiency data exists you can cross-plot the efficiency curve.

If POR or AOR have been defined in [Compressor/Fan Configuration](#), these items can be displayed on the Compressor/Fan vs. System graph.

Compressor/Fan vs. System curves can be displayed in two ways when parallel Compressor/Fans are selected. The first method is the "traditional" method, used primarily in earlier versions of AFT Arrow. The second method is called the "enhanced" method which attempts to determine the flow split on the Compressor/Fans and adjust the system curve accordingly. It also attempts to create a better composite curve when parallel Compressor/Fans are dissimilar. For more information see [Compressor/Fan vs. System Curve Methods](#).

Compressor/Fan vs. System curves for different scenarios can be plotted together on one graph by adding individual curves to a Multi-Scenario Compressor/Fan Vs. System Folder, then loading them together in a single Multi-Scenario graph. For detailed information on how to do this see [Multi-Scenario Compressor/Fan vs. System Curves Tutorial](#).

Profile

This graph category will allow you 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 and inform you which pipes are preventing the path from being continuous. The independent variable in these graphs is the length along the pipe(s).

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 that only a single path to be specified.

A profile graph can also be generated directly by selecting a single, continuous flow path in the Workspace, then right-clicking the Workspace and choosing the desired parameter under "Generate Profile Graph."

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.

Multi-Scenario Graphing Features

Starting with Arrow 9, users can plot information from multiple scenarios in the same graph for profile plots (or Transient plots with the [XTS Module](#)). Note [Multi-Scenario Compressor vs System curves](#) are generated using a different method.

In the Select Graph Parameters window, the Multi-Scenario button allows users to select the scenarios in the model they wish to plot for a particular graph. The same pipe layout must exist for all scenarios being plotted in a graph. Design alerts cannot be included in multi-scenario plots.

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, Arrow 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 1 below.

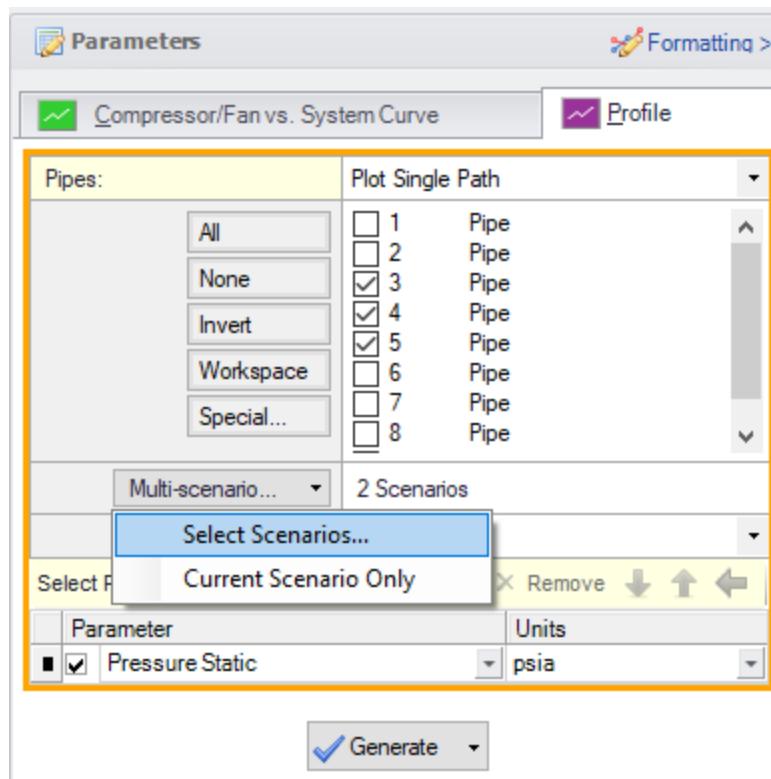


Figure 1: Graph Parameters window showing Multi-scenario selection options

Related Topics

Graph Results Window..... 371

Quick Graph Formatting Window.....	378
Compressor Fan vs. System Curve Methods.....	381
Basic Graph Tutorial.....	382
Multi Scenario Compressor Fan Vs. System Curves Tutorial.....	390

Compressor Fan vs. System Curve Methods

When plotting compressor/fan and system curves with two or more parallel compressors/fans, AFT Arrow offers two methods: Enhanced and Traditional.

It should be understood by the engineer that it is not possible to create an unambiguous composite compressor/fan and system curve for parallel compressor/fan systems. The reasons why are discussed here.

First, let's discuss the pressure curve. A composite compressor/fan curve shows a single pressure curve with values at particular flow rates. However, when there are parallel compressors/fans with unequal pressure curves or even equal pressure curves with unequal flow distribution, it is not clear what flow rate to use in the composite pressure curve. Further, with unequal flow distribution, different pressures for each parallel compressor/fan can result. But the composite pressure curve shows only a single pressure for any given flow. What pressure is this? The maximum, minimum, arithmetic average, flow rate weighted average, or some other value? A composite curve has uncertainty over what is meant by the composite flow rate and composite pressure.

Composite system curves have similar issues to composite pressure curves. The entire system performance is boiled down to a single pressure vs. flow rate curve. What flow rate is being referred to and what pressure is being referred to which goes with a given flow rate?

AFT Arrow attempts to create as meaningful and helpful composite compressor/fan and system curves as possible, but the fact remains that the curves AFT Arrow creates are ambiguous because of the nature of composite pressure vs. system curves.

Enhanced

The "enhanced" method was introduced in AFT Arrow 4.

For composite pressure curves, it assumes that the parallel compressor/fan pressure curves are added to the right with respect to flow rate. No attempt is made to apportion flow splits among the compressor/fan curves. With the pressure curves added to the right, there is a unique pressure value for every flow rate. In reality, the parallel compressors/fans may not work this way, depending on the flow split among the compressors/fans, whether or not the pressure curves are identical, and whether there are check valves at each compressor/fan discharge. This is, however, a reasonable approximation and represents the historical way that parallel compressor/fan pressure curves are added together.

No matter what method is used for creating a composite system curve, composite system curves are created by fixing some flow rate to each compressor/fan and then determining the required pressure from the compressor/fan to deliver the assumed flow rate.

The enhanced system curve takes the flow split at the operating point and assumes that the flow split remains the same at all flow rates. It collects the required pressure for each compressor at each flow rate and then averages these pressures together arithmetically.

If the compressor/fan pressure curves are identical and the flow split among compressors/fans is proportional, this method will give the same results as the Traditional method.

Traditional

The "traditional" method was used in AFT Arrow 2.0 and 3.0. The traditional method assumes that the flow splits equally among all compressors/fans at all flow rates, regardless of whether compressor/fan curves are identical or not.

The composite pressure curve is created by taking the composite flow rate, splitting it equally among each compressor/fan, obtaining the pressure for each compressor/fan at the split flow rate, and then taking the arithmetic average of the compressor/fan pressures.

The composite system curve is obtained by taking the composite flow rate, splitting it equally among each compressor/fan, running the system model with these flow rates, and then taking the arithmetic average of the required pressures for each compressor/fan.

Related Topics

Graph Results Window.....	371
Quick Graph Formatting Window.....	378
Graph Types.....	378
Basic Graph Tutorial.....	382
Multi Scenario Compressor Fan Vs. System Curves Tutorial.....	390

Basic Graph Tutorial

Generating Graphs is a powerful tool for analysis. AFT Arrow has special tools to allow the user to create Graphs that have the ability to be regenerated even if the output data has changed due to model changes. Below are step by step instructions about how to make and manage graphs from one of the example models.

Create and run a fully defined model. For this example the "Compressed Air System.aro" model was used. The file should be at this location by default C:\AFT Products\AFT Arrow 10\Examples\US - Compressed Air System.aro. The scenario was Base Scenario/Hot Case. It is recommended to copy this file to your desktop and work this tutorial from the copied file as to not save over your example files.

To skip to different sections of this tutorial, use the links below:

- [Compressor/Fan vs. System Curves and Saving Graphs](#)
- [Profile Graphs](#)
- [Displaying multiple graphs simultaneously](#)
- [Creating graphs from Workspace](#)

Compressor/Fan vs. System Curves and Saving Graphs ([return to top](#))

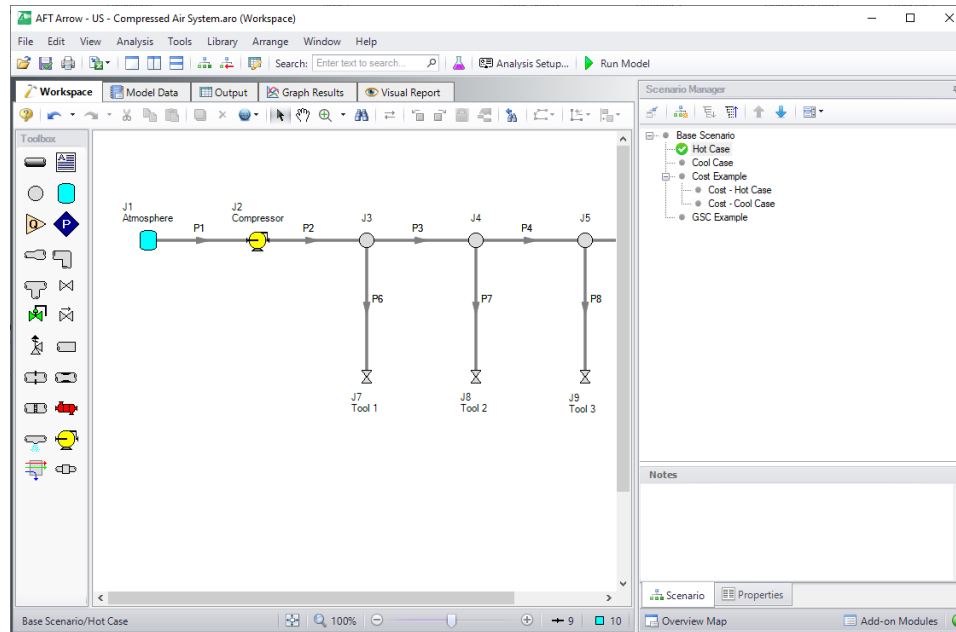


Figure 1: Fully defined model that can generate output and graphs

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 Compressor/Fan vs. System Curve. The Compressor/Fan vs. System Curve is one of several different types of Graphs that can be made. This Graph is created by first selecting the Compressor/Fan vs. System Curve tab and selecting the desired parameters (for the example the defaults have not been altered) in the Parameters area. After the parameters have been selected, click Generate, and you will see the graph 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.

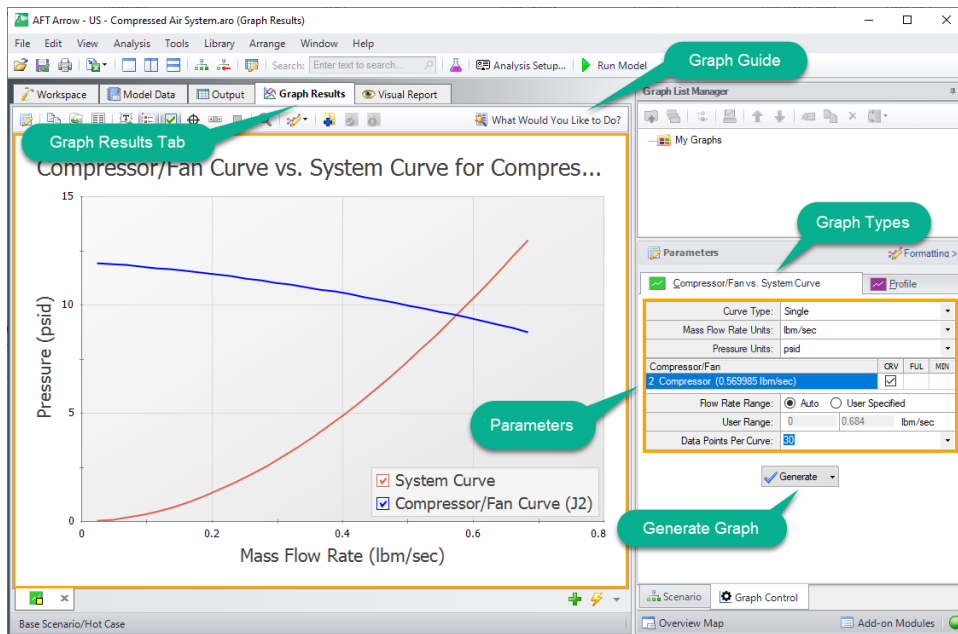


Figure 2: Graph Results Primary Window

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. Name the Graph “Compressor vs. System Curve” (this option and many others can be accessed by right-clicking on the Graph at any time).

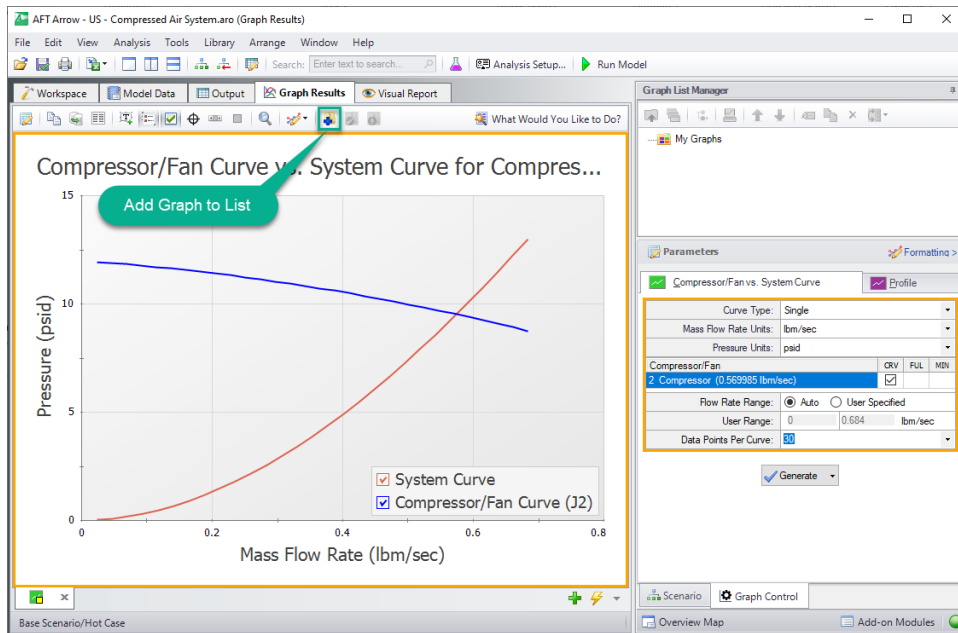


Figure 3: Compressor/Fan vs. System Curve

The Graph has now been added to the list and will be easily regenerated with the same format if the output changes. The name of the added Graph can be seen in the Graph List Manager. 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. This indicates that that Graph has been added to the list, current, and that the displayed parameters are the ones used to generate the Graph.

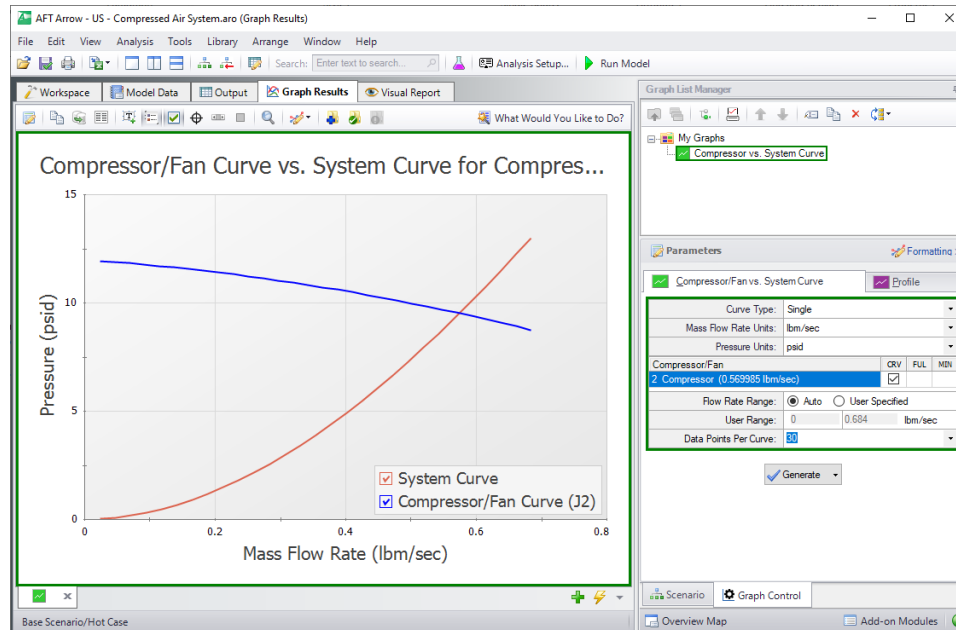


Figure 4: Compressor vs. System Curve added to the list under "My Graphs"

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. This indicates that a change has been made to a graph in the graph list. Another indicator that the Graph is not updated is the small yellow square over the icon on the tab. In this example, the title has been changed from "Compressor/Fan Curve vs. System Curve for Compressor/Fan 2" to a shorter title of "Compressor vs. System Curve" (this was done by right-clicking on the title and changing the text). These changes are not saved to the Graph unless "Update Current Graph in List" is selected.

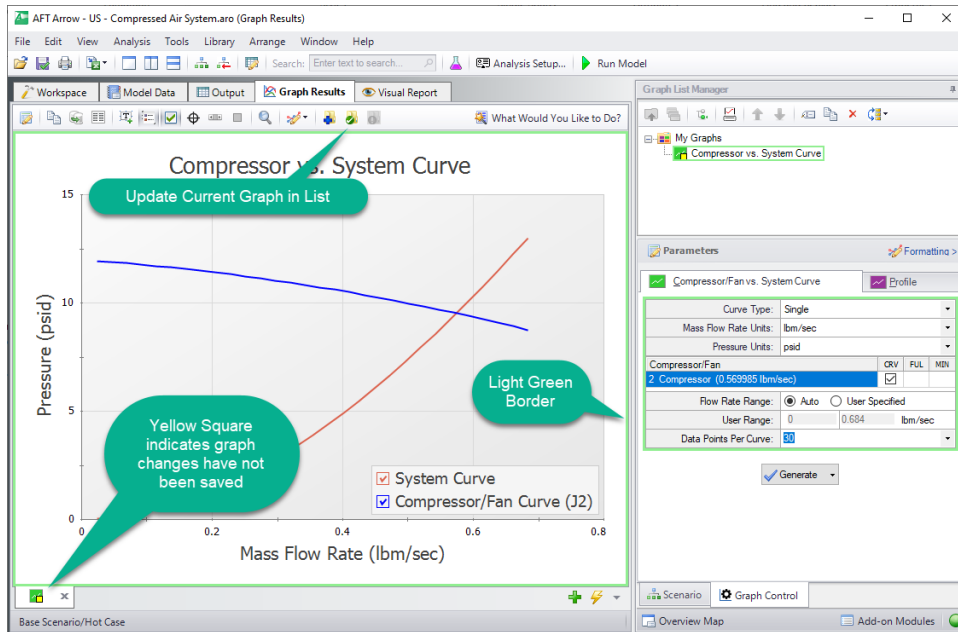


Figure 5: Compressor vs. System Curve with formatting changes

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).

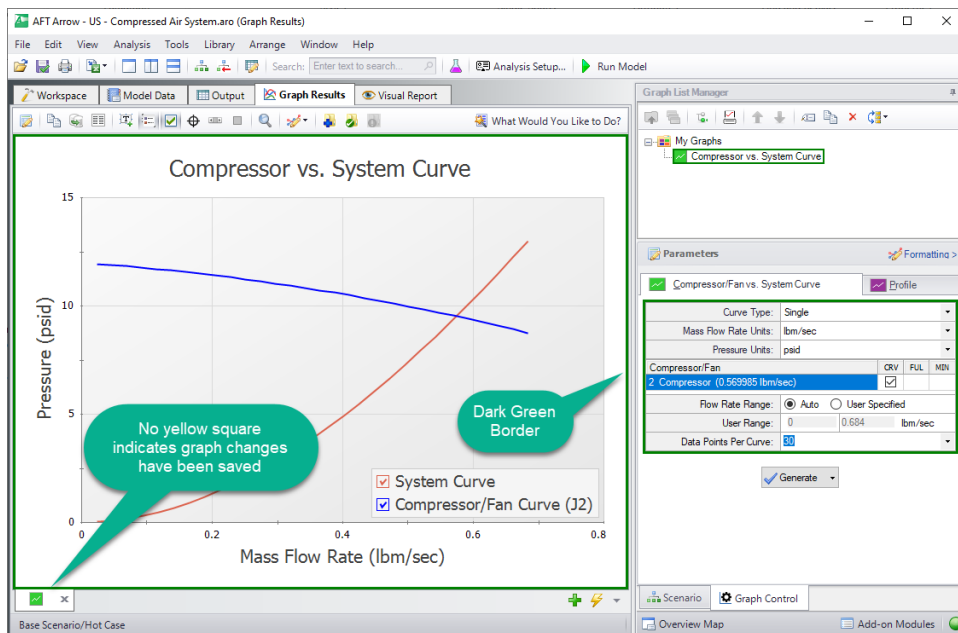


Figure 6: Compressor vs. System Curve with updated formatting changes

Profile Graphs ([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 left. This Graph will be a “Profile” Graph that displays Stagnation Pressure along the length of the flow path from the Tank to Tool #4. Select pipes 1,2,3,4,5, and 9 or highlight the flow path on the Workspace and change the parameter to “Pressure Stagnation” with units of length in feet and units of pressure in psia.

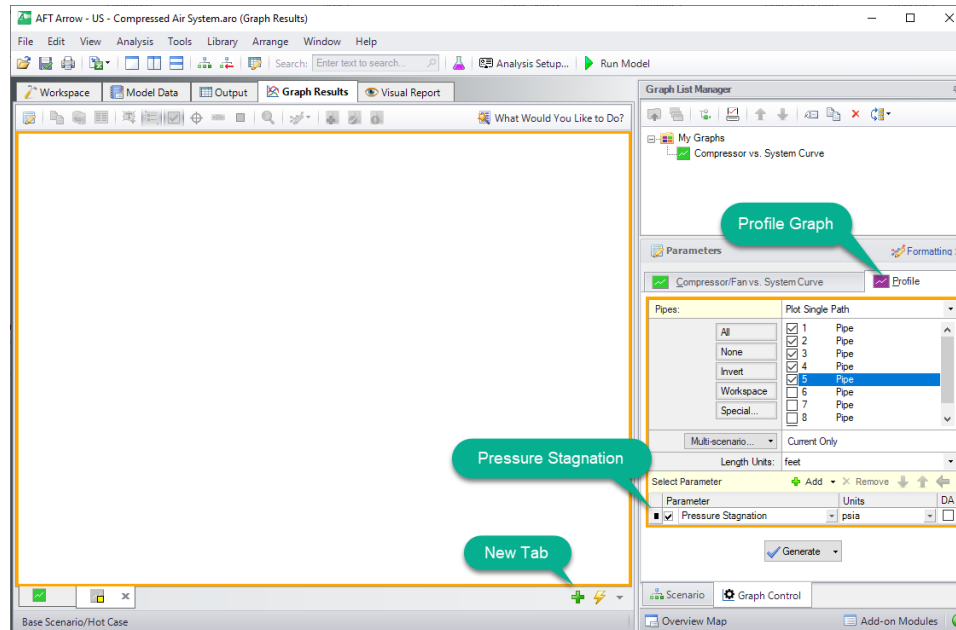


Figure 7: Generating a Profile plot with Pressure Stagnation

This Graph shows the pressure rise caused by the compressor that is very close to the tank and then a decline in pressure that results from major losses caused by friction in the pipeline. This Graph can also be added to “My Graphs” by selecting “Add Graph to List” on the toolbar. Name it “Stagnation Pressure Profile”.

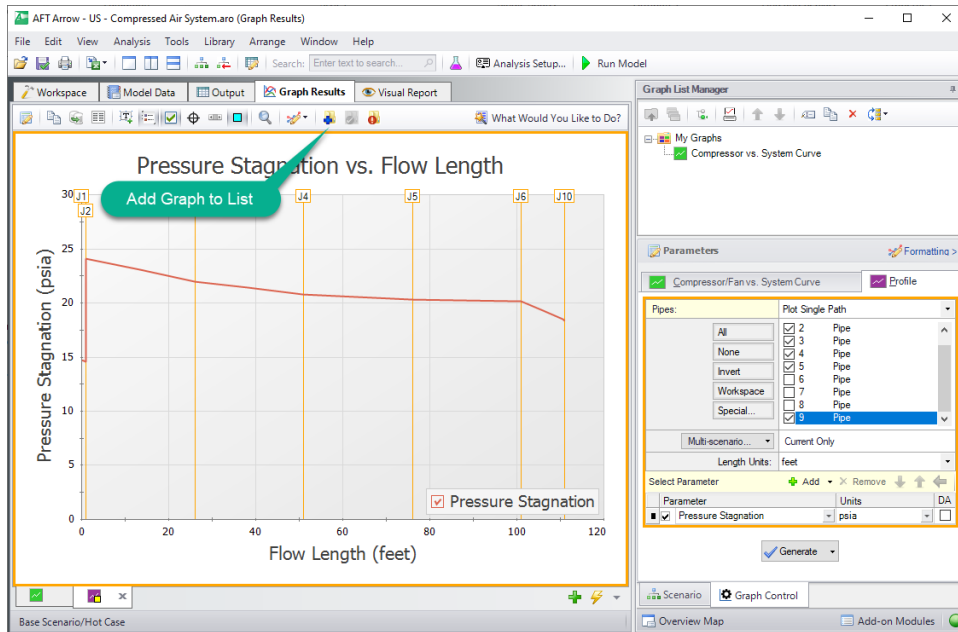


Figure 8: Pressure Stagnation vs. Flow Length

The Graph is now under “My Graphs” and the two Graphs can be switched quickly between by either selecting the Graph from the list, by clicking on the corresponding tab, or using Graph Tab Selection.

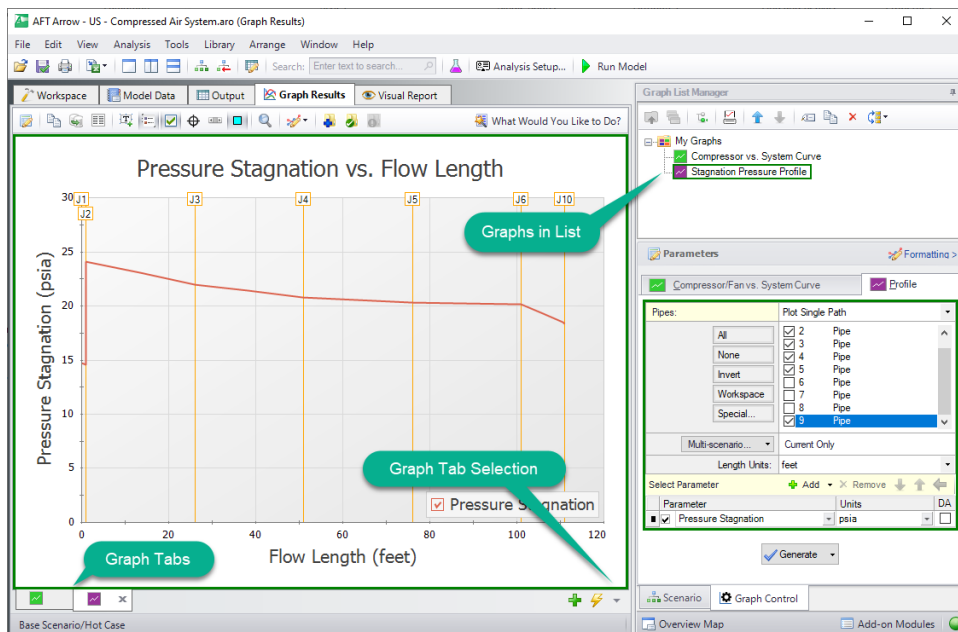


Figure 9: Stagnation Pressure Profile added to the list under "My Graphs"

Displaying Multiple Graphs Simultaneously ([return to top](#))

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 select the “My Graphs” folder. The option to “Load Folder Items” in “Separate Tabs” and “Same Tab” are now available on the toolbar. For this example select “Load Folder Items” and then “Same Tab”.

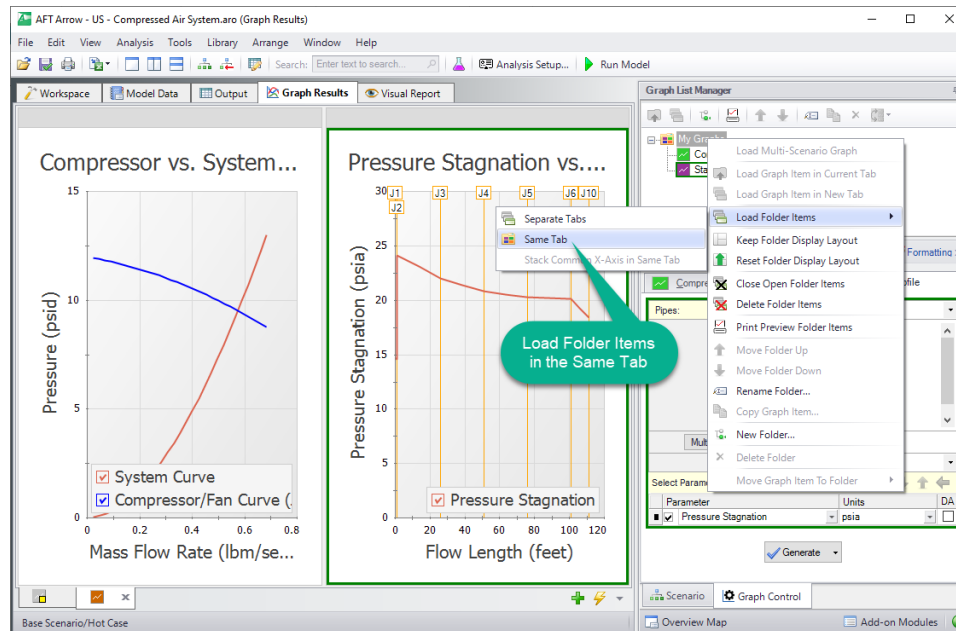


Figure 10: Both Graphs loaded into the same tab

It is also possible to create more Graph Folders other than “My Graphs”. The saved Graphs can then be organized into different folders that can independently be generated into their own tabs or a whole folder of Graphs can be generated into one tab.

Creating Graphs from the Workspace ([return to top](#))

Note that some graphs can also be generated directly from the Workspace after running the model. To generate the Pressure Stagnation vs. Flow Length graph from the Workspace, select the flow path on the Workspace (you can select multiple pipes by holding the SHIFT button on the keyboard while selecting each pipe). Then, right click on the Workspace and select the Generate Profile Graph option, then choose Pressure Stagnation (see Figure 11).

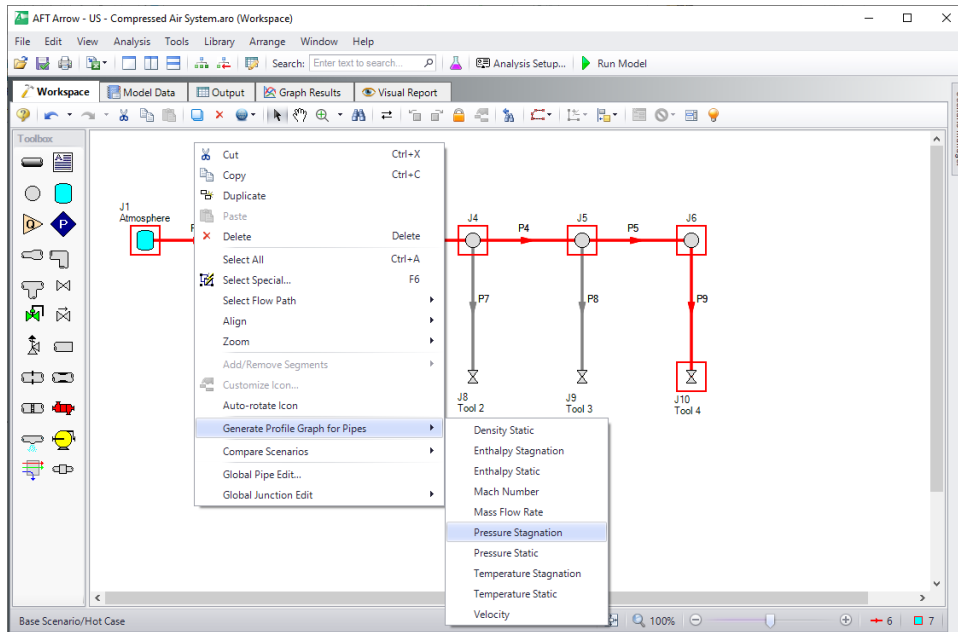


Figure 11: Several graphs can be generated directly from the Workspace

Related Topics

Graph Results Window.....	371
Quick Graph Formatting Window.....	378
Graph Types.....	378
Compressor Fan vs. System Curve Methods.....	381
Multi Scenario Compressor Fan Vs. System Curves Tutorial.....	390

Multi Scenario Compressor Fan Vs. System Curves Tutorial

When a model file has multiple scenarios that have compressors/fans in them, a graph containing multiple compressor/fan curves and/or system curves can be generated for the purpose of comparison. To create a multi-scenario compressor/fan vs. system curve graph, two options are available:

- **Standard Method:** Save compressor/fan vs system curve graphs for each individual scenario to be included in the graph. This allows the user to define settings such as the flow range for the compressor/fan vs system curve on an individual scenario basis, but is more time consuming.
- **Batch Run Method:** Create a template compressor/fan vs system curve graph, then set up a batch run to automatically generate the compressor/fan vs system curves for all scenarios to be included in the graph. This allows for a faster process.

Standard Method

1. In the Graph List Manager select "New Folder". Provide a name for the folder, and check the box to "Make Multi-Scenario Compressor/Fan Vs. System Folder", as is shown in Figure 1. This folder will be used to save the curves from the individual scenarios that will be included in the multi-scenario graph.

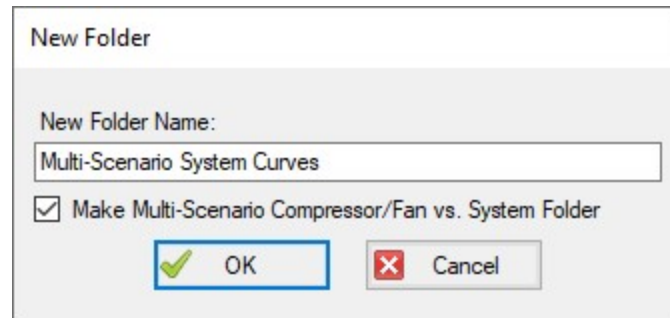


Figure 1: Creating a Multi-Scenario Compressor/Fan Vs. System Folder

2. Load the first scenario you want to add to the graph, and run the scenario so that Output is present. Generate a Compressor/Fan vs. System Curve for the scenario.
3. Add the Compressor/Fan vs. System Curve to the Multi-Scenario Compressor/Fan Vs. System Folder. This can be done by selecting the Multi-Scenario folder in the Graph List Manager, then clicking the Add Graph to List button, and naming the graph. The graph should now be shown subordinate to the folder, with the scenario name listed next to it in red, as is shown below.

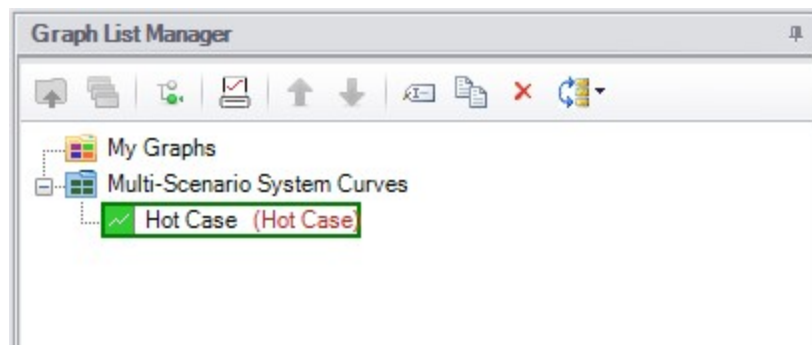


Figure 2: The name of the scenario that each Compressor/Fan vs. System Curve was created in is displayed in the Graph List Manager

4. Repeat steps 2 and 3 for all scenarios that will be included in the graph.
5. When all of the individual Compressor/Fan vs. System curves have been added to the folder, right-click on the folder name and choose "Load Multi-Scenario Graph," as is shown in Figure 3.

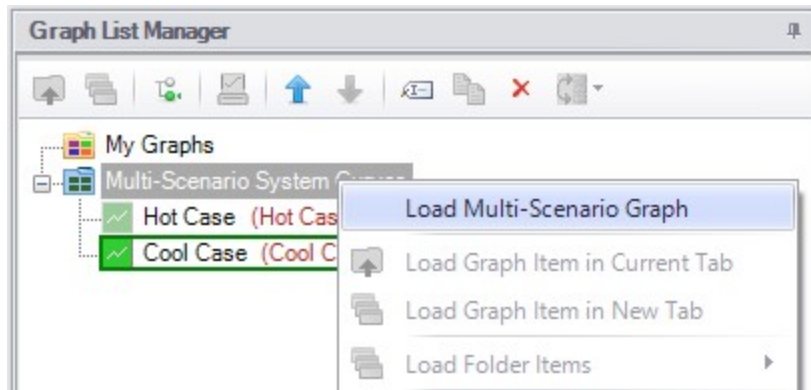


Figure 3: A Multi-Scenario Graph can be loaded from a Multi-Scenario graph folder

6. In the Multiple Scenario Graph Selection window select which curves you would like to include in the graph, then choose Generate Graph as is shown in Figure 4.

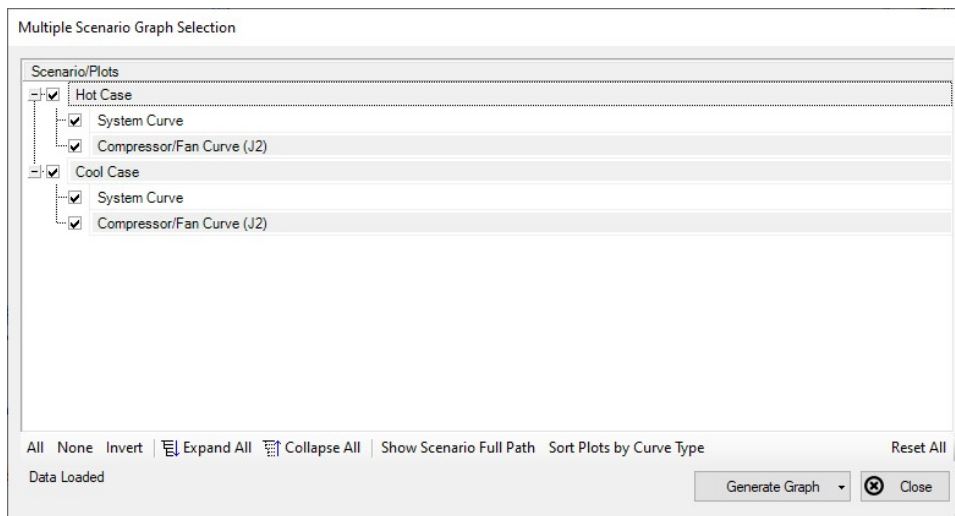


Figure 4: Multiple Scenario Graph Selection

7. The selected curves will now be displayed on a single graph, as is shown in Figure 5.

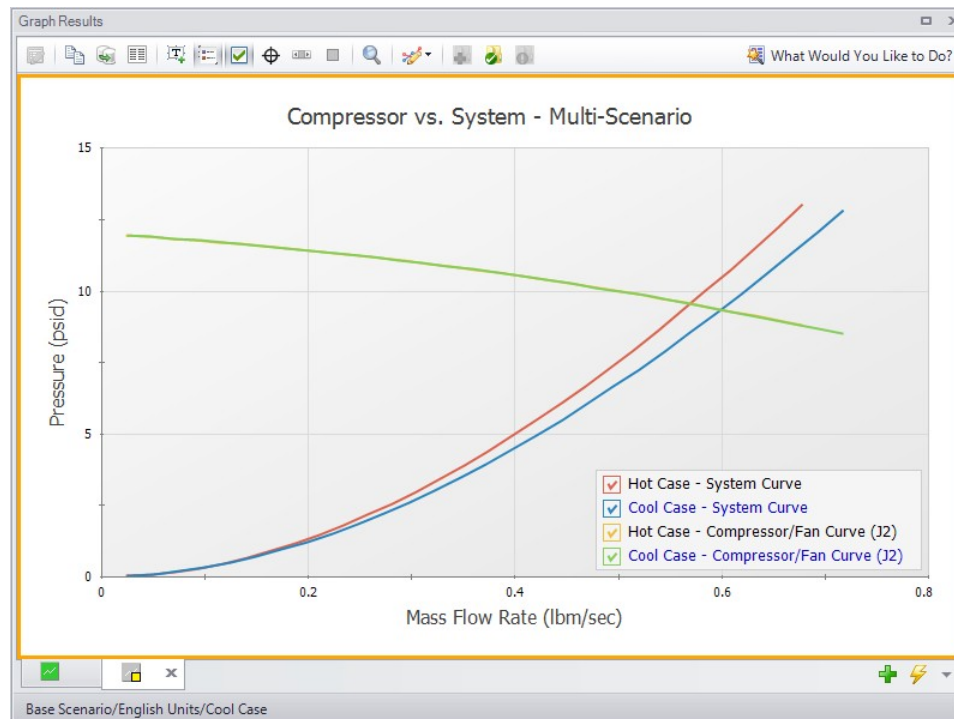


Figure 5: Multi-Scenario Compressor/Fan vs. System curve

Batch Run Method

To speed up the process outlined above when making Multi-Scenario Compressor/Fan vs. System curves for more than a handful of scenarios, the Batch Run tool can be used. Follow the steps below to create the curves.

1. In one of the scenarios being included in the Multi-Scenario Compressor/Fan vs. System curve, generate a single scenario Compressor/Fan vs. System curve and save it as a standard Graph List item. This curve will be used as a template for all the other scenarios, and should be for a compressor/fan common to all scenarios.
2. Launch the Batch Run window found in the File menu. Select the option to have the Batch Runs create Multi Scenario Compressor/Fan vs. System curves, and select the template Compressor/Fan vs. System curve as shown in Figure 6. If you have multiple templates, all options will be shown for selection.

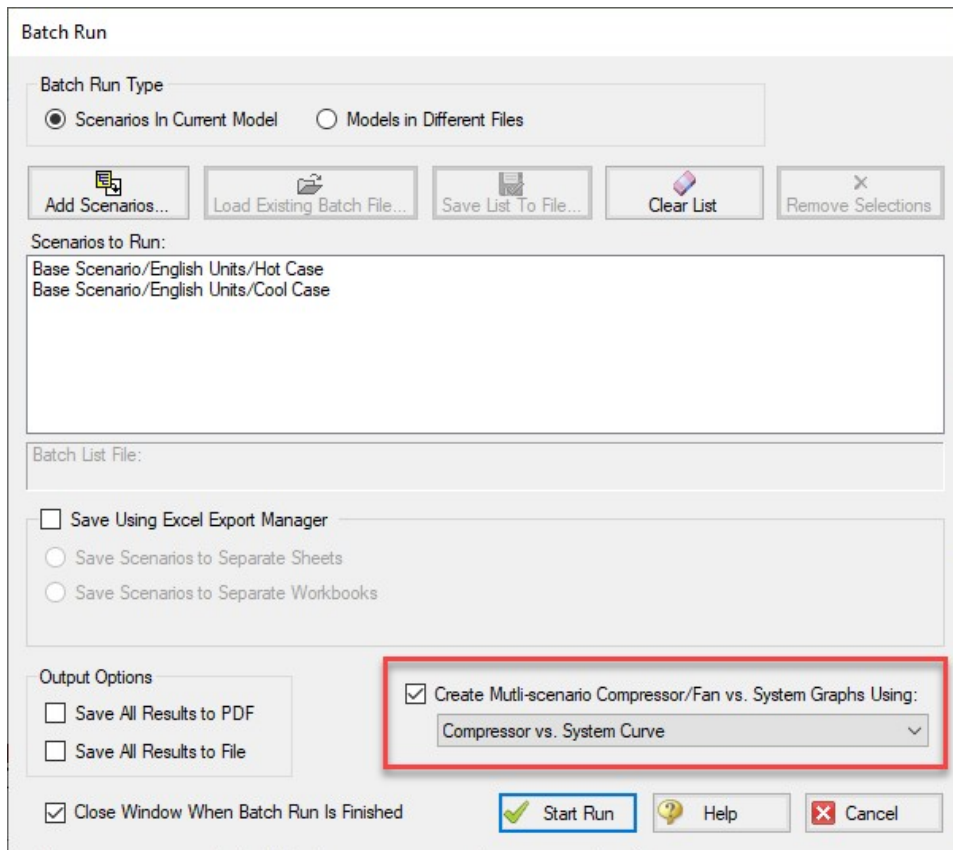


Figure 6: Multi-scenario batch run option enabled in the Batch Run window

3. Add the desired scenarios to run to the list and start the run.
4. Once the run is complete, navigate to the Graph Results tab. In the Graph List Manager, there will be a Multi-Scenario Compressor/Fan System Curve folder named with the date and time of the run. Follow steps 5 and 6 above to load the graph results.

Related Topics

Graph Results Window.....	371
Quick Graph Formatting Window.....	378
Graph Types.....	378
Compressor Fan vs. System Curve Methods.....	381
Basic Graph Tutorial.....	382

Related Blogs

[Getting there is graph the fun! - Multi-Scenario Pump vs. System Curves](#)

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
- Enable orthogonal drawing tips
- Enable Confirm Before Clearing Results
- Enable highlighting for required fields in the Properties windows, and choose the highlight color
- Options for opening models with scenarios
- Set a visual theme for the interface
- Select a location for the Scenario Manager
- Choose the number of data points to show in input tables
- Setting parameters for "button flashing" for curve fit buttons on junction Properties 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 Arrow 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.

Related Topics

Interface and User Options.....	305
Unit System.....	396
Workspace.....	398
Parameters.....	401
Files.....	402
Sizing Preferences.....	404

Unit System

You can set the units you prefer to use in AFT Arrow 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 used prior to making changes to the preferred Units.

Preferred Units

To customize the units used by AFT Arrow, first select Preferred Units under the Unit System section in the User Options window under the Tools menu.

AFT Arrow 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

Related Topics

User Options Overview.....	395
Workspace.....	398
Parameters.....	401
Files.....	402
Sizing Preferences.....	404

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, Arrow 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 Arrow ending at Version 4 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 panel 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 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. Arrow'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, Arrow 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 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.

Related Topics

User Options Overview.....	395
Unit System.....	396
Parameters.....	401
Files.....	402
Sizing Preferences.....	404

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 Control Valve Failure option allows you to choose the default behavior if a control valve is calculated to have failed. This option can be changed on an individual basis in the Control Valve Properties window. See [Control Valve Failure](#) for more information.

The Default Design Factors box allows you to add a default Design Factor to each pipe and junction shown in the list on the Pipes and Junctions window. Setting a default design factor for pipes and junctions 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.

Related Topics

User Options Overview.....	395
Unit System.....	396
Workspace.....	398
Files.....	402
Sizing Preferences.....	404

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 Arrow 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 Backup Locations. Browse to the file location in which you would like to save these settings. Once you have selected the desired file location, click OK. Note that the User File can be saved in multiple locations. To add additional locations, repeat the process for each folder that you would like to save the settings.

Backup Settings

A backup file is a copy of a previously saved version of your *.aro file that can function as a complete, standalone model. These files are used as a safety measure to provide you with one or more previous versions of the model in the case that the current model is deleted or damaged.

To customize the amount of backup files (i.e. the number of previously saved versions of the model) and the frequency that the model is saved, open the Backup Settings window, which is located under the Files section of the User Options window.

From the Backup Settings window, choose the total number of backup files you want to save by clicking on the dropdown box next to Number of Backup Files Created and specify the total number of files. After the model has been saved initially, subsequent instances of saving the model will create a new backup file each instance until the number of backup files saved equals the number of backup files specified in Backup Settings. The most recent backup file will always contain the file extension *.bak001. The file extension of earlier versions is numbered in chronological order with earlier versions receiving larger numbers in the file extension. Therefore, given any model file, the backup file with a file extension of *.bak002 will always contain the model file saved after (more recently than) the backup file containing the file extension of *.bak003. Likewise, for the same model, the backup file with the file extension of *.bak003 will always contain the model file saved after (more recently than) the backup file containing the file extension of *.bak004. This extension naming convention applies to any number of backup files.

If you want Arrow to automatically save the model file periodically when the file is open, tick the box next to Automatically Save Model and select the frequency in the dropdown box.

If you need to use a backup file as a standalone model, add the extension .aro to the backup file (the file name should now appear as *.bak00x.aro), and open the file in Arrow.

You can opt to save no backup files by choosing "0" in the Number of Backup Files Created box.

Encoding

The File Encoding option can be set to "Unicode" in order to use characters outside of the standard english keyboard such as diacritics, cyrillic, etc for text fields such as the pipe/junction name, notes, etc.

Related Topics

User Options Overview.....	395
Unit System.....	396
Workspace.....	398
Parameters.....	401
Sizing Preferences.....	404

Sizing Preferences

The Sizing Preferences section provides options to customize the colors used in the Sizing window, set the location of the Sizing Navigation Panel, and enable options for creating continuous/discrete candidate sets.

Note: This section of User Options will only be shown if the [ANS module](#) is activated in Arrow.

Color Options

The "Sizing Option" parameter colors affect the background colors used for the corresponding columns in the "Options to Minimize Monetary Cost" table on the [Sizing Objective panel](#) and the "Sizing and Cost Options" table on the [Size/Cost Assignments panel](#).

The assigned background colors will also change the background of the Sizing Level status button in the [Sizing Navigation Panel](#).

The "Common Size Group" parameter colors affect the background colors used in the "Pipe Grouping" table on the Size/Cost Assignments panel, as well as the headers used in the "Assign Candidate Sets to Pipes" section on the [Candidate Sets panel](#) and the "Assign Design Requirements to Pipes" section on the [Design Requirements panel](#).

Sizing Navigation Panel Preferences

This section allows the user to set the default location for the Sizing Navigation Panel. By default the Sizing Navigation Panel will be located at the bottom of the Sizing window.

Sizing Panel Preferences

This section provides users with the option to show the discrete/continuous option on the Candidate Sets panel. By default this option will be hidden and all Candidate Sets will be defined as discrete.

In general the continuous candidate set option should only be used if the Sizing Type is set to Continuous, and [discrete sizing](#) does not need to be performed for the pipes.

Related Topics

User Options Overview.....	395
Unit System.....	396
Workspace.....	398
Parameters.....	401
Files.....	402

File Import and Export

This section focuses on the various file types that can be imported and exported from AFT Arrow. 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	406
Importing CAESAR II Neutral Files	411
Importing GIS Shapefiles	415
Importing and Exporting EPANET Files	417
Importing and Exporting from Other AFT Products	421
Exporting to Excel	422
Importing from Excel	428

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 Arrow 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 hydraulic 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 hydraulic 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, 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.

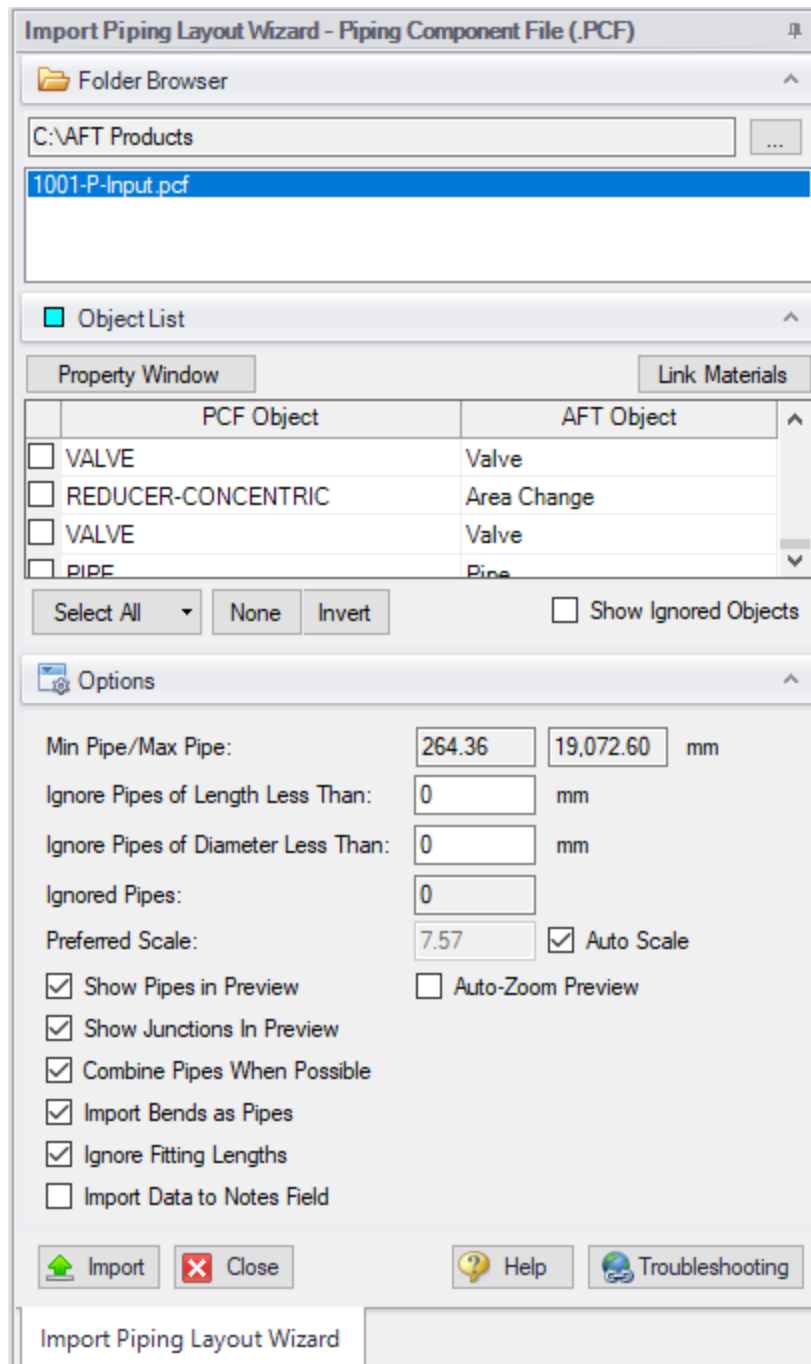


Figure 1: Piping Component File Import

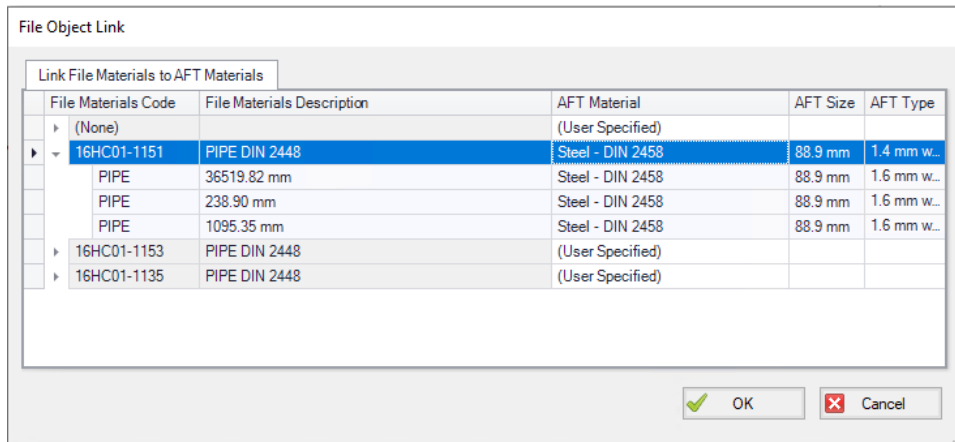


Figure 2: File Object Link window for Piping Component File Import

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: Modeling bends as explicit junctions is recommended in Arrow for more accurate modeling of compressible flow.

- 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 Topics

File Import and Export	406
Importing GIS Shapefiles	415
Importing CAESAR II Neutral Files	411
Importing and Exporting EPANET Files	417
Importing and Exporting from Other AFT Products	421
Importing from Excel	428
Exporting to Excel	422

Related Blogs

[Meet the Import Piping Layout Wizard](#)

Importing CAESAR II Neutral Files

AFT Arrow includes the ability to import CAESAR II Neutral Files (.cii), providing a quick way to get started with the hydraulic modeling portion of a project. Files can be imported into existing models.

Selecting File > Import Piping Layout From > CAESAR II Neutral Files (.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.
 - **CAESAR II Info** - Detailed information from the neutral file can be viewed for objects in the neutral file, including information on physical properties, allowable stresses, 3-dimensional orientation, etc. for the imported objects.
- **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 hydraulic 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

pipng 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.

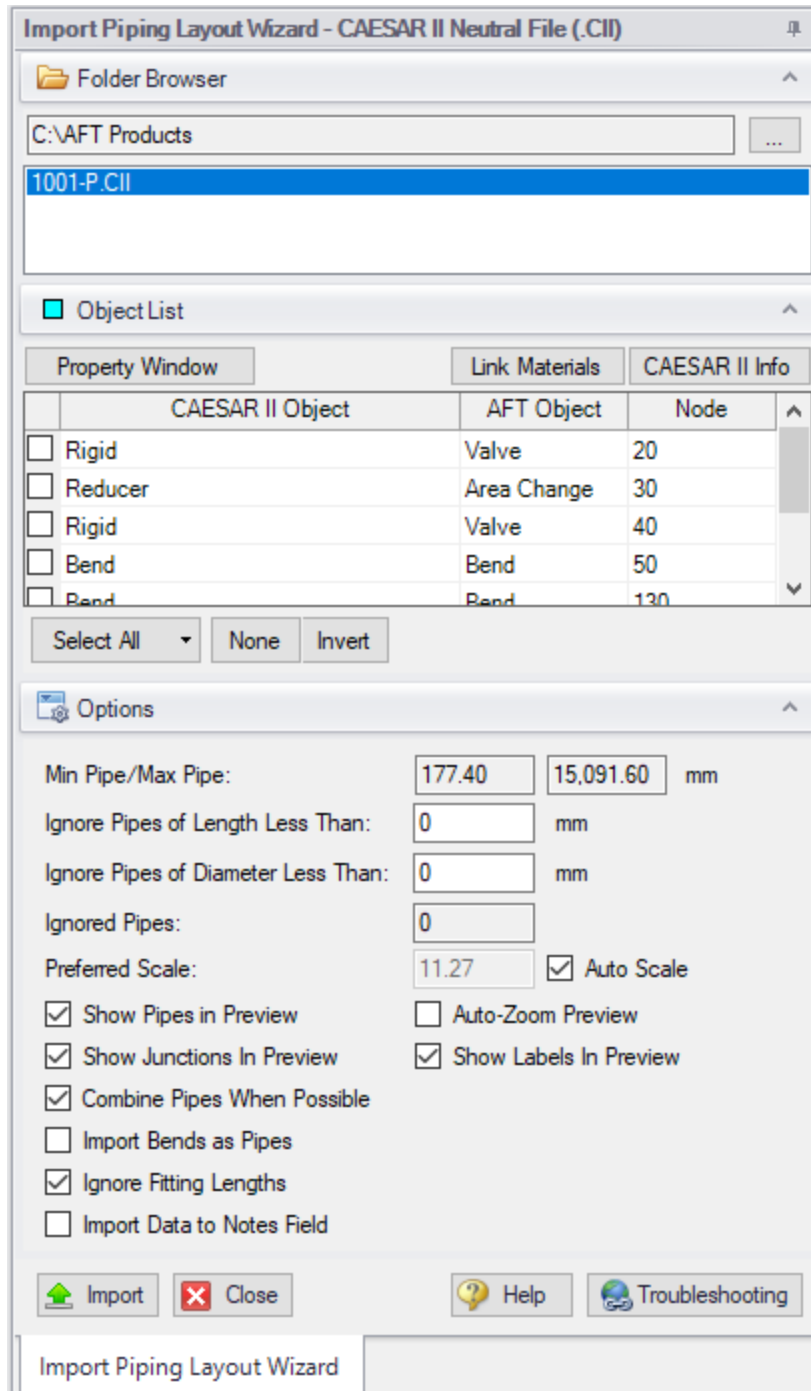


Figure 1: CAESAR II Neutral File Import

Related Topics

File Import and Export406

Importing Piping Component Files	406
Importing GIS Shapefiles	415
Importing and Exporting EPANET Files	417
Importing and Exporting from Other AFT Products	421
Importing from Excel	428
Exporting to Excel	422

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 AFT Arrow 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 Model > GIS Shapefile (SHP). This opens the "Select GIS Files to Import" window 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.

Object 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 (Grid Points/Units)

Scaling only applies when no length is mapped to the AFT "length" field or when no length data exists in the shapefile. Scaling does not alter the Workspace layout or visual length of the pipe. It only adjusts the pipe length property that is calculated by the coordinates being imported from the shapefile.

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 is applied to the automatic calculation of the pipe length property. This window does not change the visual length of the pipe drawn on the workspace. This window also allows the viewing of the coordinates of specific points and parts of different shapes within the shapefile.

Select Mapping

The Data Mapping window allows the user to select what data fields have relevant data for AFT Arrow to import.

- Pipes can map:
 - **Labels** - Names
 - **Length** - Mapping this field will utilize included user shapefile data and override the automatic calculation of pipe length. The scaling window settings will have no effect when this field is mapped.
 - **Friction** - Absolute Roughness, Relative Roughness, Explicit Friction Factor, Weymouth, and Panhandle friction models are available.
 - **Inner Diameter**
- Branches can map:
 - **Labels** - Names
 - **Elevation**
- Compressors/Fans can map:
 - **Labels** - Names
 - **Elevation**
 - **Assigned Flow** - Imports as a Fixed Flowrate compressor/fan
- Tanks can map:
 - **Labels** - Names
 - **Elevation**
- Valves can map:
 - **Labels** - Names
 - **Elevation**

Related Topics

File Import and Export.....	406
Importing Piping Component Files.....	406
Importing CAESAR II Neutral Files.....	411
Importing and Exporting EPANET Files.....	417
Importing and Exporting from Other AFT Products.....	421
Importing from Excel.....	428
Exporting to Excel.....	422

Related Blogs

[Meet the Import Piping Layout Wizard](#)

Importing and Exporting EPANET Files

AFT Arrow 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 hydraulic 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 hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
- **Preferred Scale** - The wizard will Auto Scale the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
- **Show Pipes in Preview** - Display the pipes in the Workspace preview.
- **Show Junctions in Preview** - Display the junctions in the Workspace preview.
- **Import Data to Notes Field** - Add the raw data from the originating file to the Notes tab of the corresponding object.
- **Auto-Zoom Preview** - This option can be enabled to automatically zoom out so that the whole model can be seen in the workspace, which can be convenient for large systems.

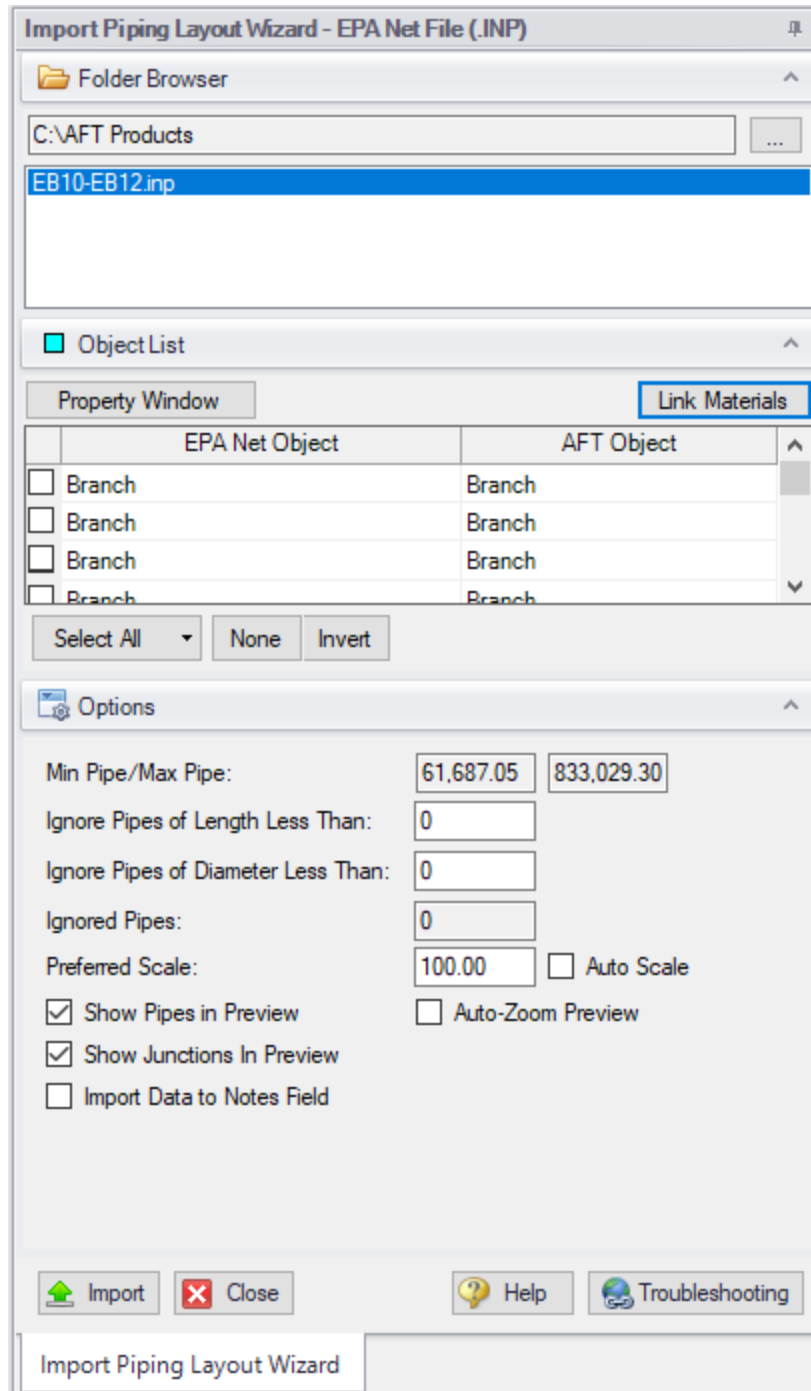


Figure 1: EPANET Import Dialog

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** - If XTS is on, finite-volume tanks will be imported as finite tanks. Otherwise, they will be imported as infinite tanks.
- **Reservoirs** - EPANET reservoirs are imported as infinite tanks in Arrow.
- **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 Arrow. 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.

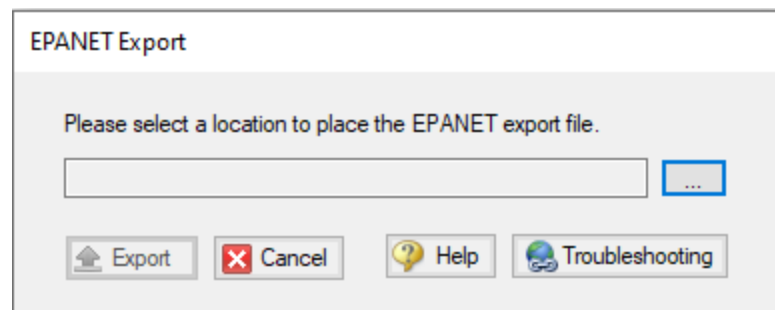


Figure 2: EPANET Export Dialog

Related Topics

File Import and Export.....	406
Importing Piping Component Files.....	406
Importing GIS Shapefiles.....	415
Importing CAESAR II Neutral Files.....	411
Importing and Exporting from Other AFT Products.....	421
Importing from Excel.....	428
Exporting to Excel.....	422

Related Blogs

[Meet the Import Piping Layout Wizard](#)

Importing and Exporting from Other AFT Products

Models made in other AFT products such as AFT Fathom can be opened directly in AFT Arrow 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 heat transfer calculations for a gas model originally built in Fathom.

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 to "All", as is shown below in Figure 1, or to ".xtr" for xStream models.

When the model is opened in AFT Arrow 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 Arrow 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 Arrow. If a junction is defined using a feature not available in AFT Arrow, 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.

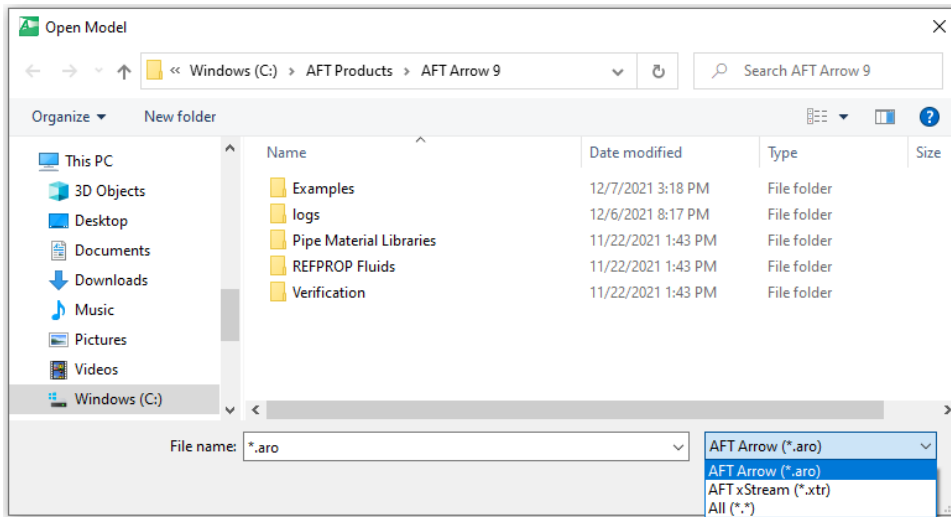


Figure 1: File extensions options for Open File window

Related Topics

File Import and Export	406
Importing Piping Component Files	406
Importing GIS Shapefiles	415
Importing CAESAR II Neutral Files	411
Importing and Exporting EPANET Files	417
Importing from Excel	428
Exporting to Excel	422

Exporting to Excel

AFT Arrow 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.

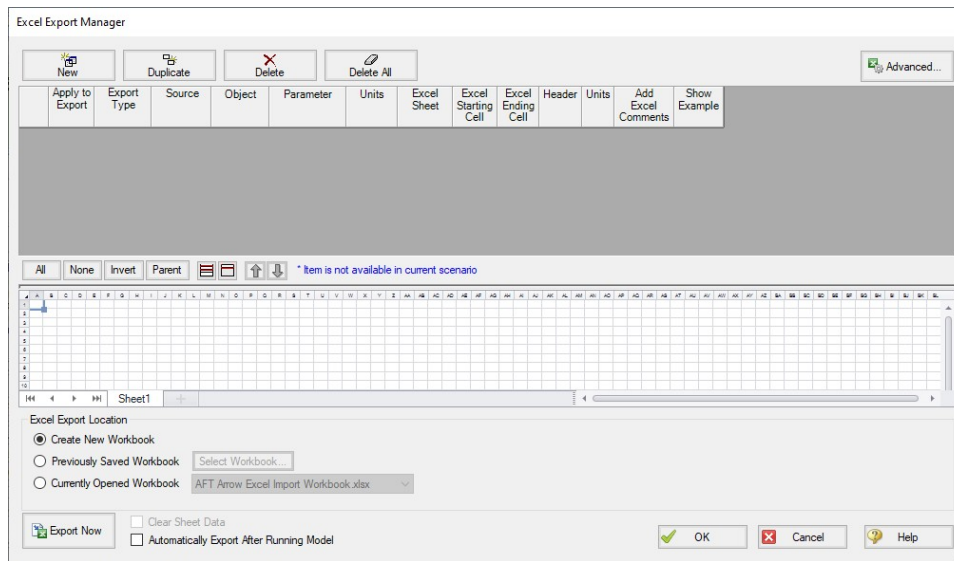


Figure 1: The Excel Export Manager

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.

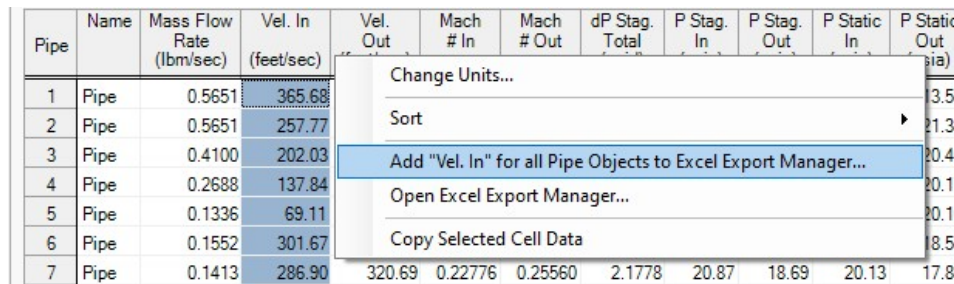


Figure 2: Adding items directly from the Output

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 Pipe section of the output.
 - **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/Fan junction in the model, Compressor/Fan Summary is available.
 - **Graph** - From a specified [Graph List Item](#).
- **Object** - The object in particular that is desired. If Column or Table are selected for Export Type, this selection will be fixed to ALL.
- **Parameter** - The parameter in particular that is desired. If Row, Table or Graph are selected for Export Type, this selection will be fixed to ALL.
- **Units** - The units of the exported value. This is available only for Single Value and Column Export Types. Some parameters do not have units and this will display N/A.
- **Excel Sheet** - Which Excel Sheet to export to. This value can be edited by the user - if it is desired to export to a named sheet this can be entered here.
- **Excel Starting Cell** - The upper-left cell to export the data to.
- **Excel Ending Cell** - The automatically calculated lower-right cell of the data, which depends on the Export Type, export options, and the number of parameters or objects in the model. It is important to recognize that the Excel Starting Cells can be defined in a way that data gets overwritten. The Ending Cell column is a handy reference to help avoid this.
- **Header** - Optionally export the name of the Parameter(s) related to the item.
- **Units** - Optionally export the name of the Unit(s) related to the item.
- **Add Excel Comments** - Optionally adds an Excel comment to the value cell that contains the Parameter name and unit.
- **Show Example** - Displays an example of how the exported cells will appear.

	Apply to Export	Export Type	Source	Object	Parameter	Units	Excel Sheet	Excel Starting Cell	Excel Ending Cell	Header	Units	Add Excel Comments	Show Example
1	<input checked="" type="checkbox"/>	Row	Pipe	P1 (Upstream of Compressor)	ALL	ALL	Sheet1	A1	AD3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
2	<input checked="" type="checkbox"/>	Row	Pipe	P2 (Downstream of Compressor)	ALL	ALL	Sheet1	B5	AD6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
3	<input checked="" type="checkbox"/>	Column	Pipe	ALL	Mass Flow Rate	lbm/sec	Sheet1	A9	B18	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
4	<input checked="" type="checkbox"/>	Table	Compressor/Fan Summary	ALL	ALL	ALL	Sheet1	D9	Q11	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
5	<input checked="" type="checkbox"/>	Other	N/A	N/A	Scenario Name	None	Sheet1	D13	D13	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	

Figure 3: Example Export

	A	B	C	D	E	F	G	H	I	J	K	L
1		Name	Mass Flow Rate	Vel. In	Vel. Out	Mach # In	Mach # Out	dP Stag. Total	P Stag. In	P Stag. Out	P Static In	P Static Out
2			lbm/sec	feet/sec	feet/sec			psid	psia	psia	psia	psia
3	P1 (Upstream of Compressor)	Upstream of Compressor	0.5651	365.63	368.96	0.31565	0.31858	0.1193	14.7	14.58	13.72	13.59
4												
5			lbm/sec	feet/sec	feet/sec			psid	psia	psia	psia	psia
6		Downstream of Compressor	0.5651	257.76	281.8	0.20289	0.22271	2.1147	24.17	22.06	23.49	21.31
7												
8												
9		Mass Flow Rate										
10	P1 (Upstream of Compressor)	0.5651										
11	P2 (Downstream of Compressor)	0.5651		J2 (Compressor)	Compressor	0.5651	8.5977	9.5922	9.8832	17,987.5957		
12	P3 (Pipe)	0.4099										
13	P4 (Pipe)	0.2685		Scenario: Base Scenario								
14	P5 (Pipe)	0.1336										
15	P6 (Pipe)	0.1553										
16	P7 (Pipe)	0.1413										
17	P8 (Pipe)	0.1349										
18	P9 (Pipe)	0.1336										

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.

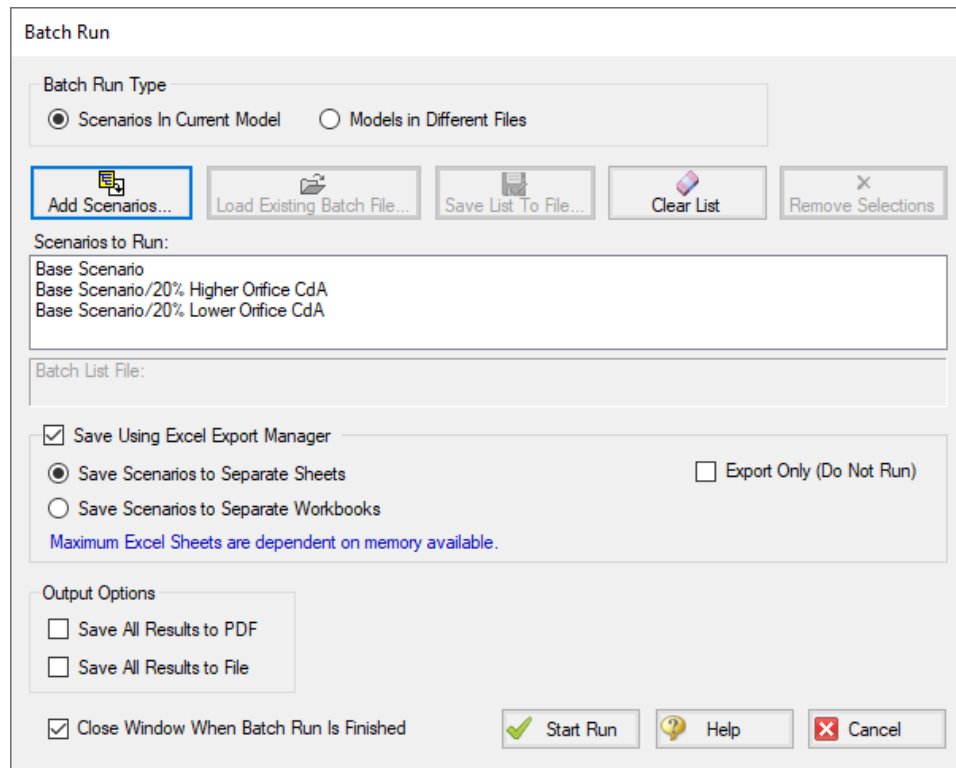


Figure 5: Saving Results From Multiple Scenarios

Related Topics

GSC in Output.....	493
Cost Report.....	485
Output Messages & Warnings.....	709
File Import and Export.....	406
Importing Piping Component Files.....	406
Importing GIS Shapefiles.....	415
Importing CAESAR II Neutral Files.....	411
Importing and Exporting EPANET Files.....	417
Importing and Exporting from Other AFT Products.....	421
Importing from Excel.....	428

Related Blogs

[Quick Guide to Custom Output](#)

[Paint a Clear Picture Faster Than Ever with AFT's New Excel Export Manager](#)

Importing from Excel

A Microsoft Excel spreadsheet can be used to make changes to objects in an AFT Arrow 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 Arrow will search for a worksheet named "AFT Transfer". There can be other worksheets in the workbook, but AFT Arrow 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	HEX heat rate set	Yes	Heat Exchanger	1	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	Yes	Heat Exchanger	2	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	Yes	Heat Exchanger	3	Heat Rate	Set equal to value	3700	Average Heat
5	FCV setpoint set	Yes	Control Valve	15	Control Setting	Set equal to value	300	
6	FCV setpoint reduced by 20 lbm/sec	Yes	Control Valve	16	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increased by 10%	No	Control Valve	16	Control Setting	Change by percent (+/-)	10	
8	Fan Speed Set	Yes	Compressor/Fan	22	Speed (%)	Set equal to value	90	
9								

Figure 1: Sample AFT Transfer Excel Sheet

Row 1 and column A are not read by AFT Arrow - 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 Arrow installation. By default the file path is: C:\AFT Products\AFT Arrow\AFT Arrow Excel Import Workbook.xlsx, or the spreadsheet can be downloaded [here](#).

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.

Table 1: Object Types, Associated Parameters, and Change Codes

Pipe	Area Change	Assigned Flow	Assigned Pressure	Bend
S Additional K Factor	S CdA (for sonic choking)	S Design Factor	S CdA	S CdA (for sonic choking)
S Ambient Fluid Velocity	S Design Factor	S Flow Rate	S Design Factor	S Design Factor
S Ambient Temperature	S Initial Guess Pressure	S Initial Guess Pressure	S Elevation	S Initial Guess Pressure
S Buried Pipe Depth	S Initial Guess Temperature	S Initial Guess Temperature	C Junction Name	S Initial Guess Pressure
S Design Factor - Pipe Fittings & Losses	S Elevation	S Elevation	S Pressure	S Initial Guess Temperature
S Design Factor - Pipe Friction	C Junction Name	C Junction Name	I Sizing/Cost Setting	S Elevation
S Design Factor - Pipe Heat Transfer	I Sizing/Cost Setting	I Sizing/Cost Setting	S Temperature	C Junction Name
S External Heat Flux/Tracing Flux	S User Specified K	I Special Condition		I Sizing/Cost Setting
S External Heat Rate		S Temperature (Static)		S User Specified K
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)				

<p>S Noncylindrical Flow Area</p> <p>S Noncylindrical Wetted Perimeter</p> <p>C Pipe Name</p> <p>S Rectangular Duct Height</p> <p>S Rectangular Duct Width</p> <p>S Roughness Value (for unspecified friction models)</p> <p>S ID Reduction (Scaling)</p> <p>I Sizing/Cost Setting</p> <p>S Soil Surface Temperature</p> <p>I Special Condition</p>				
<p>Branch</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p> <p>S Source/Sink Flow</p> <p>S Source/Sink Temperature/Enthalpy</p> <p>I Special Condition</p>	<p>Check Valve</p> <p>S CdA (for sonic choking)</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>S Forward Velocity to Close Valve</p>	<p>Compressor/Fan</p> <p>S Control Setpoint</p> <p>S Delta Head/Pressure to Re-Open Combined Check Valve</p> <p>S Design Factor</p> <p>S Fixed Flow Rate</p> <p>S Fixed Speed (%)</p> <p>S Forward Velocity to Close Combined Check Valve</p>	<p>Control Valve</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>S Loss When Full Open</p> <p>S Setpoint</p> <p>I Siz-</p>	<p>Dead End</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p>

	C Junction Name S Loss Value I Sizing/Cost Setting I Special Condition S Xt Value	S Initial Guess Pressure S Initial Guess Temperature S Elevation C Junction Name S Nominal Efficiency S Polytropic Constant I Sizing/Cost Setting I Special Condition	ing/Cost Setting I Special Condition S Xt When Fully Open	
General Component S CdA (for sonic choking) S Design Factor S Initial Guess Pressure S Initial Guess Temperature S Elevation C Junction Name S K Factor I Sizing/Cost Setting I Special Condition	Heat Exchanger S Controlled Outlet Temperature S Design Factor - Friction Loss S Design Factor - Heat Transfer S Heat Flow Into System S Initial Guess Pressure S Initial Guess Temperature	Orifice S Area/Diameter S Design Factor S Exit Pressure S Exit Temperature S Initial Guess Pressure S Initial Guess Temperature S Inlet Elevation C Junction Name S Loss Value S Outlet Elevation I Sizing/Cost Setting	Relief Valve S Cracking Pressure S Design Factor S Exit Pressure S Exit Temperature S Exit Temperature S Initial Guess Pressure S Initial Guess Temperature S Elevation C Junction Name S Loss Value I Sizing/Cost	Screen S Design Factor S Initial Guess Pressure S Initial Guess Temperature S Elevation C Junction Name S K Factor I Sizing/Cost Setting

	S Elevation S Heat Transfer Area C Junction Name S K Factor S Overall Heat Transfer Coefficient S Secondary Fluid Flow Rate S Secondary Fluid Specific Heat S Secondary Fluid Inlet Temperature I Sizing/Cost Setting I Special Condition		Setting I Special Condition	
Separator S Initial Guess Pressure S Initial Guess Temperature S Elevation C Junction Name I Sizing/Cost Setting	Spray Discharge S CdA (for sonic choking) S Cd (Discharge Coefficient) S Design Factor S Dis-	Tank S Design Factor S Elevation S Fixed/Initial Volume C Junction Name S Pressure I Sizing/Cost	Tee or Wye S Design Factor S Initial Guess Pressure S Initial Guess Temperature S Elevation	Valve S CdA (for sonic choking) S Design Factor S Elevation S Exit Pressure S Exit Tem-

	<p>charge Flow Area</p> <p>S Exit Pressure</p> <p>S Exit Temperature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Number of Holes</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>	<p>Setting</p> <p>S Temperature</p>	<p>C Junction Name</p> <p>I Sizing/Cost Setting</p>	<p>perature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>C Junction Name</p> <p>S Loss Value</p> <p>S Open Percentage</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p> <p>S Xt Value</p>
<p>Venturi</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p>				

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
Control Valve	None	0
	Closed	1
	Fully Open - No Control	2
General Component	None	0
	Closed	1
Heat Exchanger	None	0
	Ignore Heat Transfer	2
Jet Pump	None	0
	Closed	1
Pump	None	0
	Pump Off No Flow	1
		2
	Pump Off With Flow Through-	

Junc-tion	Special Condi-tion	Valu-e
	h	
Relief Valve	None	0
	Failed	2
	Open	5
	Ignore Relief Valve	
Spray Dis-charge	None	0
	Closed	1
Valve	None	0
	Closed	1
Weir	None	0
	Ignore Weir	5

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 Arrow 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 res-

	istance 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 Pipe Edit](#) or [Global Junction Edit](#) to an arbitrary value before the changes are made from Excel.

Related Topics

File Import and Export	406
Importing Piping Component Files	406
Importing GIS Shapefiles	415
Importing CAESAR II Neutral Files	411
Importing and Exporting EPANET Files	417
Importing and Exporting from Other AFT Products	421
Exporting to Excel	422

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](#) to and from other model files.

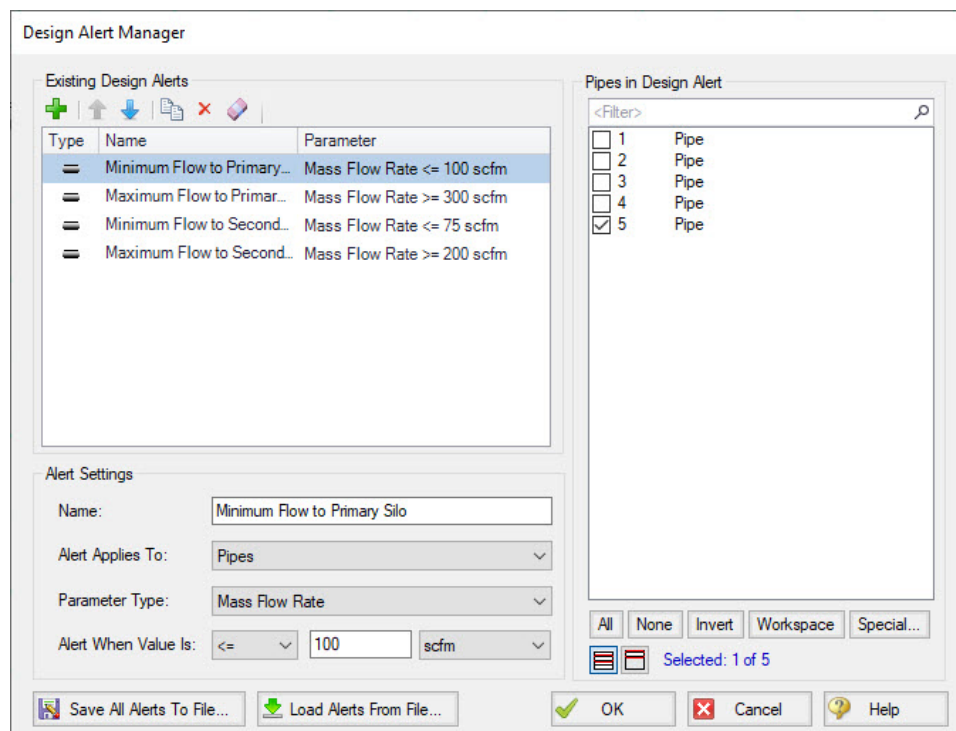


Figure 1: Design Alert Manager

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 alter 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.

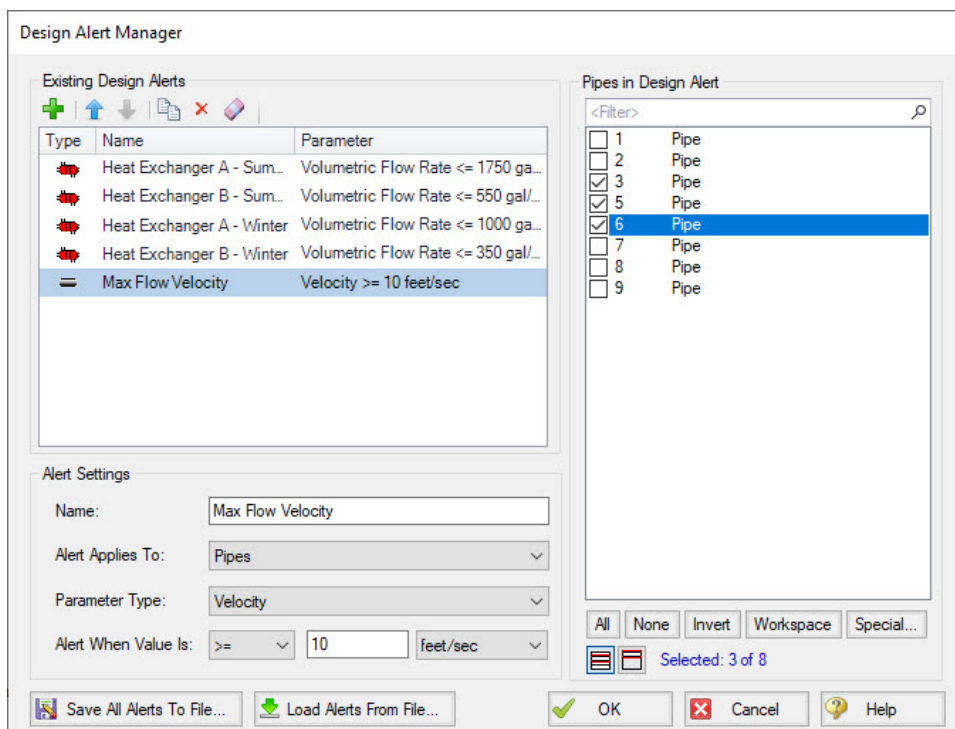


Figure 2: Creating a Design Alert

Pipes	General Junctions	Compressors/Fans	Valves	Heat Exchangers
Enthalpy Stagnation	Pressure Loss Stagnation Total	Mass Flow Rate	Cv	Heat Rate
Enthalpy Static	Pressure Loss Static Total	Percent of BEP	Flow Area	Log Mean Temperature Difference
Mach Number	Pressure Stagnation Inlet	Power Overall	K	Mass Flow Rate
Mass Flow Rate	Pressure Stagnation Outlet	Pressure Discharge Stagnation	Kv	Pressure Drop Static
Pressure Gradient Stag. Total	Pressure Static Inlet	Pressure Discharge Static	Mass Flow Rate	Temperature Inlet Static
Pressure Stagnation	Pressure Static Outlet	Pressure Rise Static	Open Percentage	Temperature Outlet Static
Pressure Static		Pressure Suction Stagnation	Pressure Drop Stagnation	
Temperature Stagnation		Pressure Suction Static	Pressure Static Inlet	
Temperature Static		Speed	Pressure Static Outlet	
Velocity				
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 'Pipe Design Alerts':

- 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 of output data for pipes. The 'Velocity' column is highlighted in yellow for pipes 112, 113, and 114, indicating they exceed the design alert.

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 window, a status bar indicates 'Design Alerts Exist' and shows a search icon and '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](#) help topic.

Related Topics

Design Alerts Importing & Exporting445
 Design Alerts From Output 444

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...

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)
0.1193	14.70	14.58	13.71	13.59	0.002319	110.0	110.0	98.86
2.1146	24.17	22.05	23.49	21.31	4.445068	219.9	215.4	214.39
1.1832	22.0							6 212.06
0.5303	20.8							2 208.02
0.1385	20.3							4 200.85
2.5327	22.0							9 207.92
2.1778	20.8							9 202.79
2.0010	20.3							6 194.89

Figure 1: Output Context Menu

Create Design Alert automatically defines a Design Alert based on the selected value and quantity:

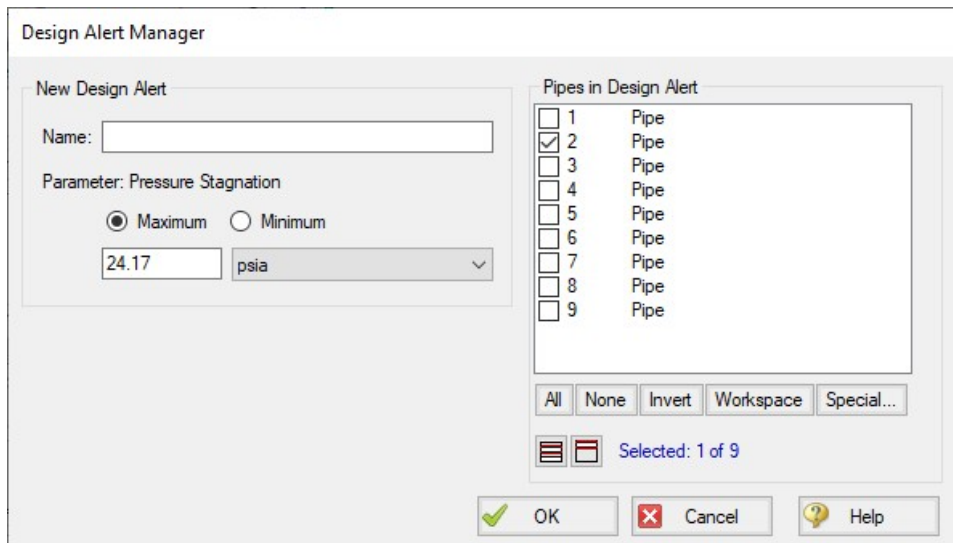


Figure 2: Automatically defined Design Alert

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](#).

Related Topics

Design Alerts.....	441
Design Alerts Importing & Exporting.....	445

Design Alerts 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.

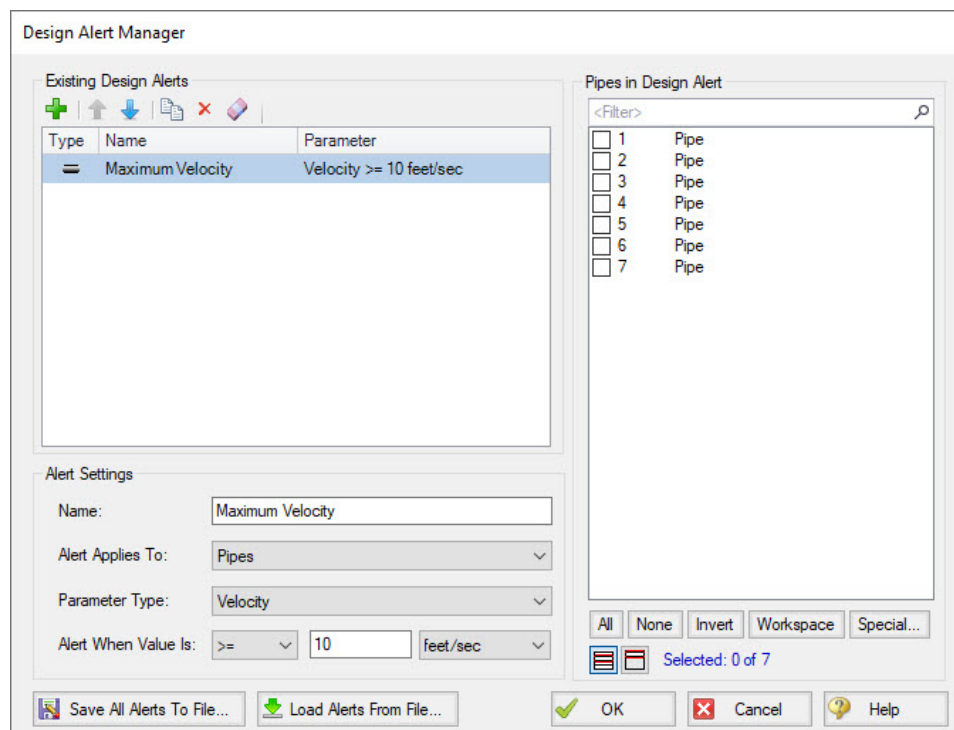


Figure 1: A design alert for velocity to be exported

- 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.

Related Topics

Design Alerts	441
Design Alerts From Output	444

Scenario Manager

Scenario Manager (in the [Quick Access Panel](#)) is a powerful tool for managing variations of a model, referred to as scenarios. It allows you to quickly test the same model with selected differences (such as pump 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 specifications
- 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 also identified in AFT Arrow's [Status Bar](#)). At the bottom, there is a place where you can record notes related to that scenario (the notes also propagate 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.

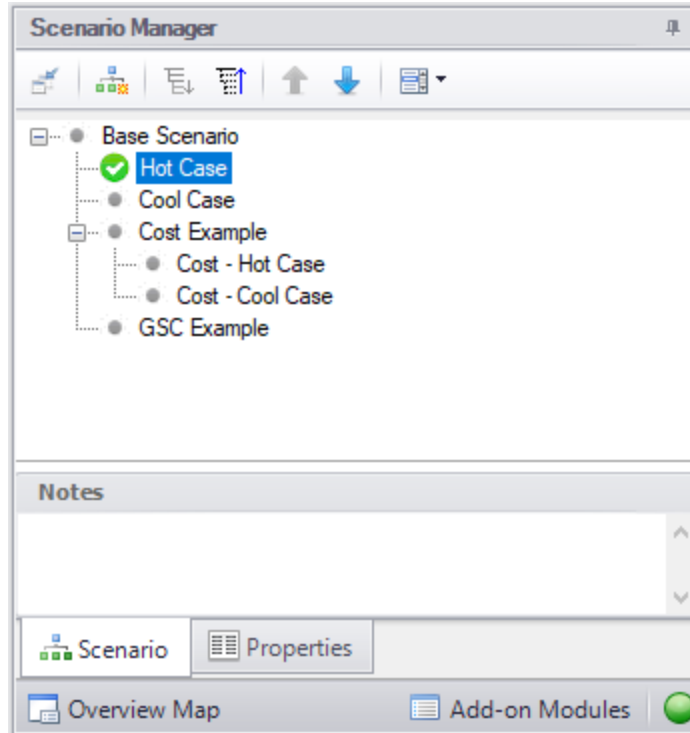


Figure 1: Scenario Manager in Quick Access Panel

Related Topics

Note No Scenarios have been set in the Multi-Scenario Output display.....	725
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

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 loaded as the current scenario by double-clicking on it.

Figure 1 illustrates a model with a Base Scenario with three children, '2 inch PVC Pipe', '3 inch PVC Pipe' and '4 inch PVC Pipe', representing three different size pipe runs. Each of these, in turn, has two children, 'Max Demand' and 'Min Demand', for the two operating cases.

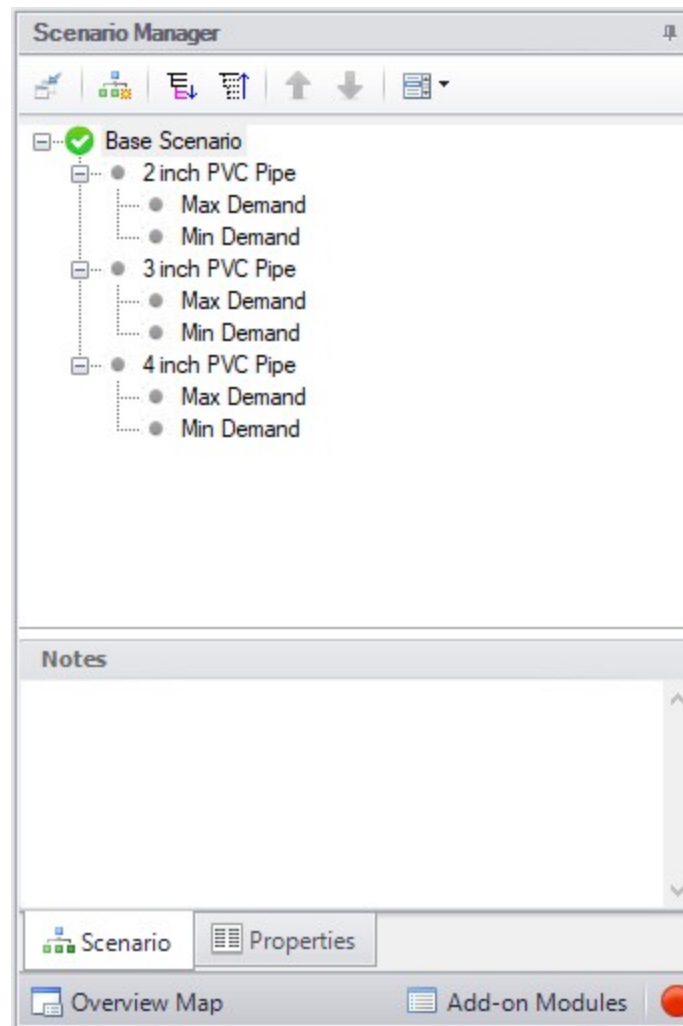


Figure 1: Scenario Manager in Quick Access Panel

'2 inch PVC Pipe', '3 inch PVC Pipe' and '4 inch PVC Pipe' were created by first selecting the Base Scenario then clicking on Create Child. 'Max Demand' and 'Min Demand' were, in turn, created by selecting each of the above children scenarios for the three different pipe run sizes and then clicking on Create Child.

Note: Alternatively, one could have made the Base Scenario current, created the '2 inch PVC Pipe' child and then selected it and created the 'Max Demand' and 'Min Demand' scenarios below it. Then, with '2 inch PVC Pipe' still selected, clicked Clone With Children twice and renamed the new clones '3 inch PVC Pipe' and '4 inch PVC Pipe'.

The 'Max Demand' and 'Min Demand' 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.aro :: 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 Arrow'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

Differences between scenarios can be viewed in two different ways. All data associated with any selection of scenarios can be viewed using the [Scenario Comparison tool](#). Alternatively, 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.

Output from different scenarios can also be displayed in the [Output window](#) using [Output Control](#) features.

Fast Scenario Changes

AFT Arrow 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 [Tools menu](#) or the [Common Toolbar](#).

Related Topics

Scenario Manager.....	447
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

Modifying Individual Scenarios

When first created, a scenario is identical to its parent. To modify or run a scenario, it must first be made the current Workspace scenario by selecting it within the Scenario Manager window, then clicking on the Load As Current Scenario button. (The name of the current scenario is displayed in both the Scenario Manager window and in AFT Arrow's [Status Bar](#).) Once a scenario is the current scenario, changes may be made to differentiate it from its parent using any of AFT Arrow'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 Properties](#) - fluid selected, system data
- [Solution Control](#) - solution method, tolerances, relaxation

- [Output Control](#) - Output window content and formatting
- [Cost Settings](#) - application of cost data
- [Goal Seek & Control](#) (GSC Module Only) - Application of variables and goals can change between scenarios. The actual variables and goals are the same in all scenarios.
- [Libraries Manager](#) - The connected library content can be varied between scenarios
- [Model Data Control](#) and [User Options](#) applies to all scenarios.
- [Sizing Settings](#) (ANS Module Only) - All input in the Sizing window. Candidate Sets and Design Requirements are the same in all scenarios, but they may be applied differently.

Related Topics

Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

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 on the Scenario Format tab 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 (Figure 2). 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 it's 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 Analysis Setup, Output Control, 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.

Related Topics

Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

Running Models in Batch Mode

AFT Arrow [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 settings.
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 Arrow 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 Arrow 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.

The entire batch run can be canceled at any time by clicking the Cancel button in the [Solution Progress window](#).

Note: You can view output from multiple scenarios concurrently in the [Output window](#).

Related Topics

Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

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.

Related Topics

Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Disconnected Scenario.....	457
Scenario Comparison Tool Overview.....	457

Disconnected Scenario

A special type of scenario that is no longer connected to the [Scenario Manager](#), created from the [Tools Menu](#) or from the Toolbar in the Output window. A disconnected scenario itself breaks the inheritance of normal child scenarios so that it is truly standalone scenario. *No* changes made to any parent scenario will propagate to this scenario. This is useful to preserve certain settings from a different scenario.

Disconnected Scenarios are denoted with (*).

Related Topics

Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Scenario Comparison Tool Overview.....	457

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 Topics

Accessing Scenario Comparison Tool.....	458
Modify Comparison.....	459
Scenario Comparison Grid.....	461
Scenario Manager.....	447
Creating, Organizing, and Editing Scenarios.....	448
Modifying Individual Scenarios.....	451
Scenario Inheritance.....	452
Running Models in Batch Mode.....	455
Open Model Using Last Scenario.....	457
Disconnected Scenario.....	457

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](#) with 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 with 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 Grid.

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 Topics

Scenario Comparison Tool Overview.....	457
Modify Comparison.....	459
Scenario Comparison Grid.....	461

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.

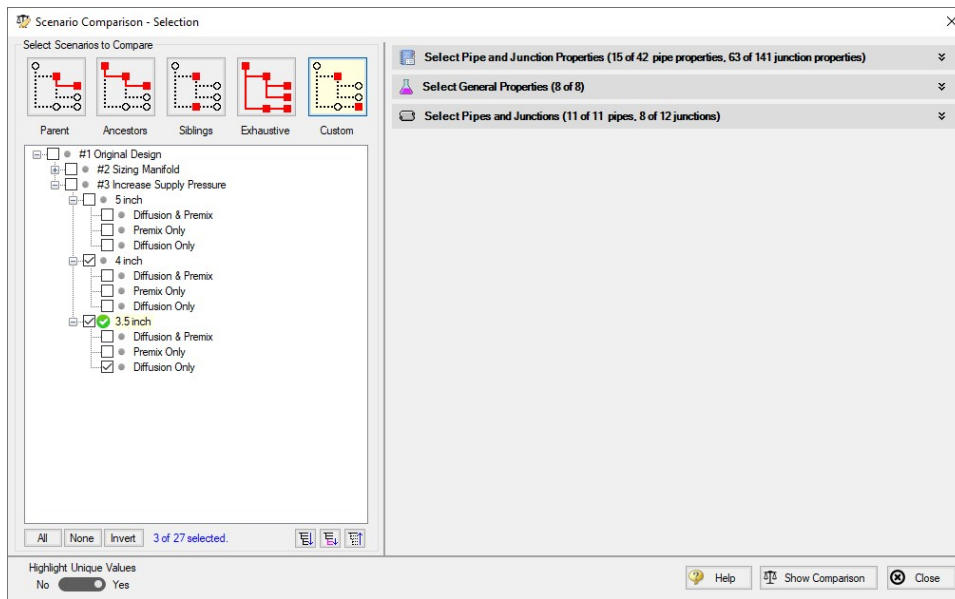


Figure 1: Scenario Comparison window options to Modify Comparison

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.

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 Topics

Scenario Comparison Tool Overview.....	457
Accessing Scenario Comparison Tool.....	458
Scenario Comparison Grid.....	461

Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

Scenario Comparison Grid

Choosing the Compare Selected Scenarios menu item from the right-click or Tools menu will bring the user directly to the Scenario Comparison Grid based on any selected Workspace objects and scenarios. The settings for the scenario comparison can be seen/modified by clicking [Modify Comparison](#) at the bottom of the window.

The tabular data from the report can be copied to and pasted to a separate file or worksheet using the Copy button at the bottom of the window.

Item Show	Parameter Show	Scenario		
		3.5 inch	4 inch	Diffusion Only
P4	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P6		No Differences		
P7	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P9		No Differences		
P10	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
Junctions (8 of 12 compared)				
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
J7 (Tee or Wye)		No Differences		
J8 (Premix Gas FCV 1)	Special Condition	None	None	Closed
J10 (Bend)		No Differences		
J11 (Premix Gas FCV 2)	Special Condition	None	None	Closed

Figure 1: Default Scenario Comparison Grid 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. 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		
		3.5 inch	4 inch	Diffusion Only
P4	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P5	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P6		No Differences		
P7	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P8	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P9		No Differences		
P10	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
P11	Hydraulic Diameter	3.548 inches	4.026 inches	3.548 inches
	Size	3-1/2 inch	4 inch	3-1/2 inch
Junctions (8 of 12 compared)				
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
J7 (Tee or Wye)		No Differences		
J8 (Premix Gas FCV 1)	Special Condition	None	None	Closed
J10 (Bend)		No Differences		
J11 (Premix Gas FCV 2)	Special Condition	None	None	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
At Least 1 Parameter Difference	Different Values Across Scenarios	Yes

Copy Help Modify Comparison Close

Figure 2: Scenario comparison showing differences only. Each unique value is highlighted with a different color to indicate which scenarios have differences from the currently active scenario, which is the 3.5 inch scenario for this example.

Related Topics

Scenario Comparison Tool Overview.....	457
Accessing Scenario Comparison Tool.....	458
Modify Comparison.....	459

Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

Cost Calculation

AFT Arrow has the powerful ability to account for both recurring and non-recurring costs in your model.

The most common and easiest to account for recurring cost is [Energy Cost](#). It is often desirable to reduce the electricity usage of compressors in a system, and AFT Arrow makes measuring this value easy.

In addition to energy usage, AFT Arrow can track other user-defined costs such as maintenance, installation and material cost with a [Cost Library](#).

Cost Calculation Procedure

Depending on what types of costs or what components of a model you wish to include this process may change.

Note: This process only applies if the [ANS module](#) is not enabled. To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

1. [Enable Cost Calculation](#) from the [Cost Settings panel](#) in Analysis Setup.
2. Specify system life and energy cost in [Cost Settings](#).
3. Junctions or pipes you wish to include in the cost calculation must be added to the Cost Report with the Cost tab in the object's properties window.
4. If you are calculating all costs (not just Energy), you will need to create [Cost Libraries](#). To represent more complex Energy Cost, such as varying cost over time, you will need to create an [Energy Cost Library](#).
5. Within the Cost Libraries, specific recurring or non-recurring costs will need be [defined](#).
6. These costs can be further refined to adjust automatically based on diameter, time, power, or C_v with [Scale Tables](#).
7. Review the application of costs with the [Cost Application Manager](#) and the [Cost Summary Window](#). It is important that the costs are being applied as desired.
8. Run the model and view the cost totals with the [Cost Report](#).

Related Topics

Cost Settings Group.....	149
Cost Settings Panel.....	150
Cost Application Manager.....	470
Scale Tables.....	472
Select Size From.....	474
Cost Definitions.....	475
Cost Libraries.....	478
Cost Summary Window.....	483
Cost Report.....	485

Enabling Cost Calculation

Note: This process only applies if the [ANS module](#) is not enabled. To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

Enable Cost Calculations from the [Cost Settings panel](#) in Analysis Setup. The following options are available:

- **Do Not Calculate** - Selected by default.
- **Compressor/Fan Energy Only** - Only calculate the costs associated with the power consumption of compressors. The energy cost and system life must be entered in the Cost Settings panel.
- **Calculate** - Enable full calculation of costs. In addition to defining Cost Settings, [Cost Libraries](#) must be defined to use this feature.

Including Objects in Cost Report

Workspace objects must be included in the Cost Report to have their cost calculated. This allows only analyzing the cost of certain parts of the model - useful if only one compressor is being replaced, or if an expansion to an existing system is being considered. To include the objects in the Cost Report simply select Include Cost in Report on the object's Cost tab, as shown in Figure 2. Finer detail in how the Cost is included can be set by clicking the [Cost Application](#) button.

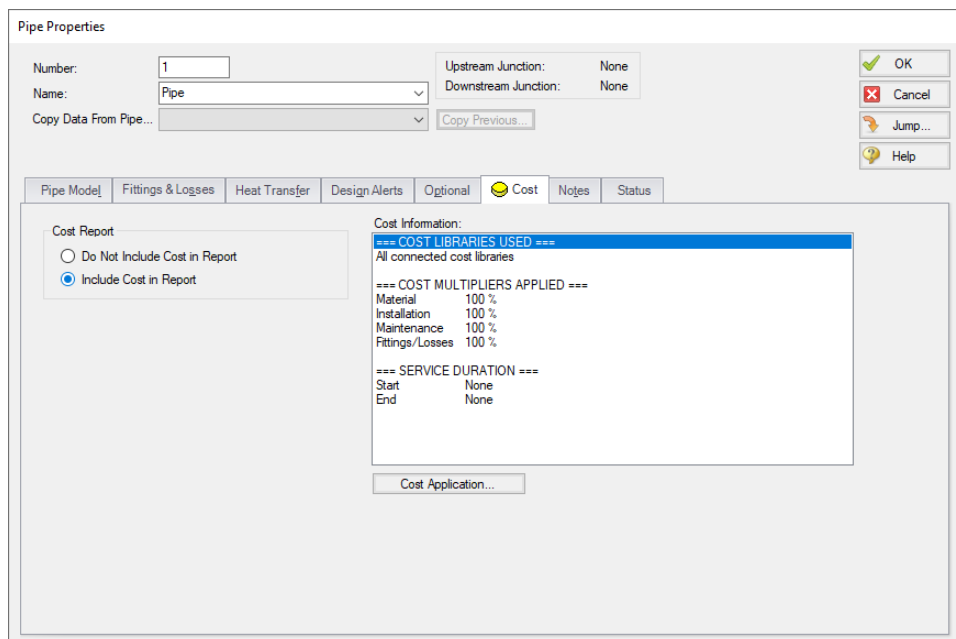


Figure 1: Include Cost in Report

Energy Cost Library

Energy Cost Libraries allow users to input time varying energy costs. These energy costs can then be assigned to [compressor/fans](#), so that compressor/fan operation/energy costs can be calculated.

Creating an Energy Cost Library

When opening the Energy Cost Library window from the Library menu, you will be greeted with Figure 1. An Energy Cost Library either needs to be created or opened.

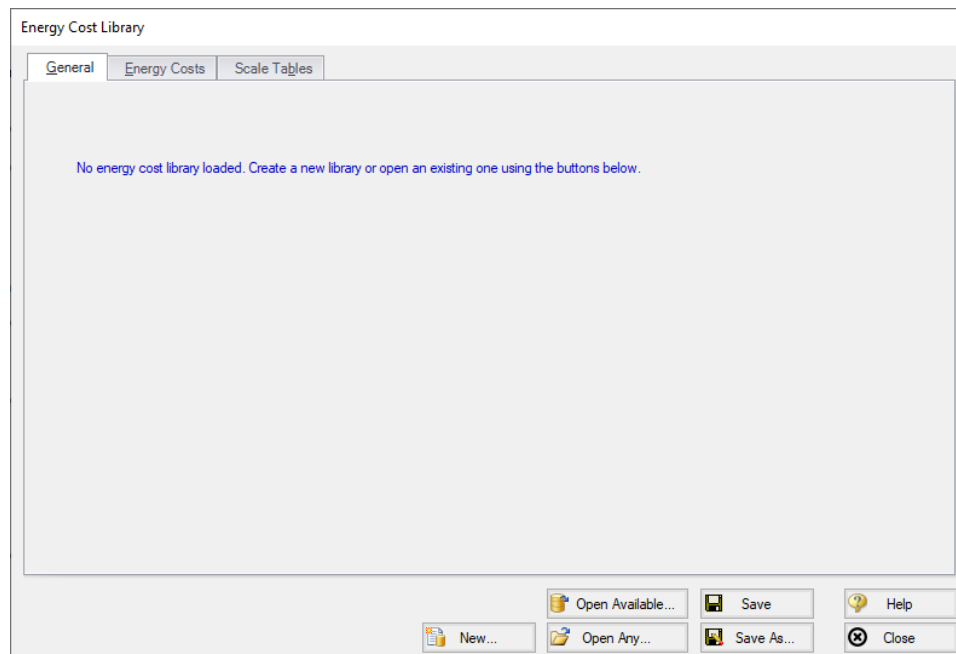


Figure 1: Energy Cost Library Window

To create a new Energy Cost Library, click the New button. The Description field is mandatory, and the notes are optional. Select an alternate Monetary Unit if desired. Finally, save the library by clicking Save As and enter name for the *.ecd file. AFT Fathom will also prompt if the library should be connected and available - typically this is desired so the library can be used in the currently open model.

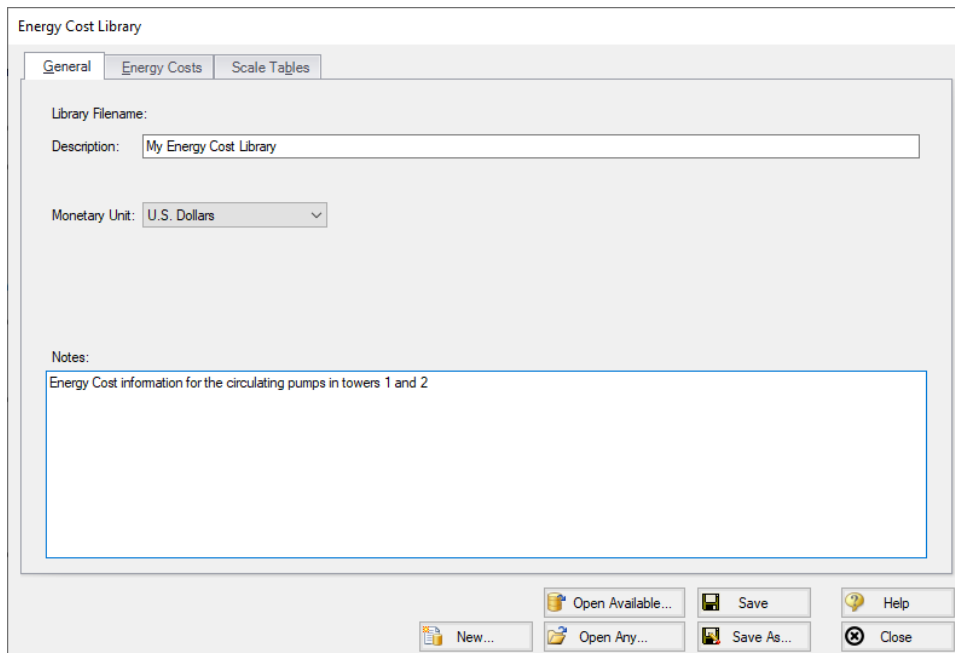


Figure 2: New Energy Cost Library

Defining Energy Costs

Once an Energy Cost Library is opened, the Energy Costs tab allows costs to be added. Select New Cost to add costs to the library. In Figure 3, costs have been added to represent a varying electricity cost over night and day. If the Cost Time Period exceeds the time defined in the Energy Cost Library, the pattern will start over at zero. That is, for the above example if the Cost Time Period was one year the defined 24 hour cost cycle would repeat 365 times.

The Cost Scaling column allows a Scale Table to be applied to a cost, as discussed below.

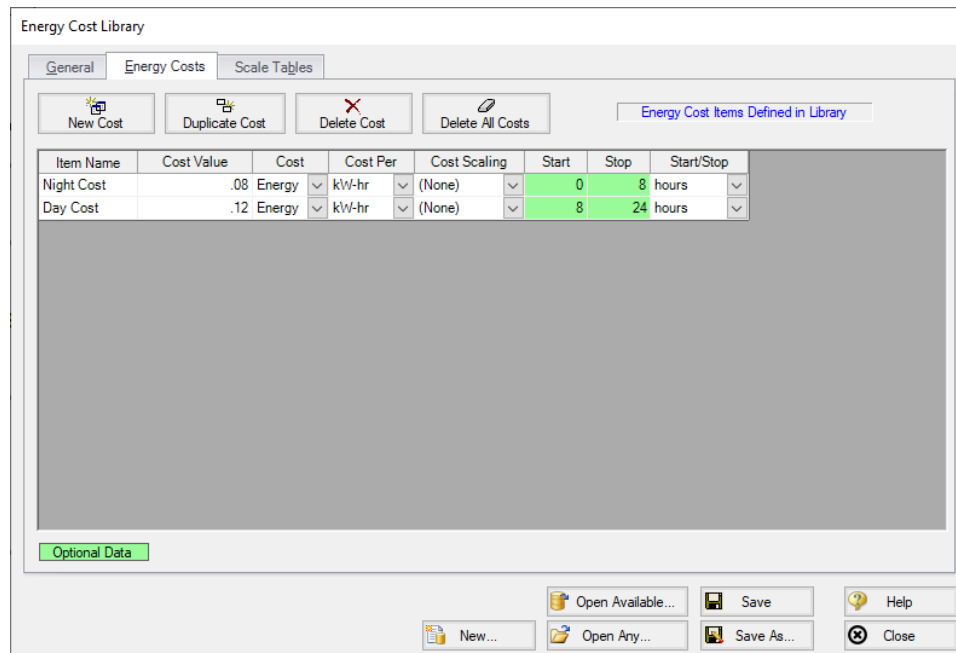


Figure 3: Energy Costs

Scale Tables

You can add a time-based multiplier to an Energy Cost Library on top of the normal Energy Costs if desired. The Scale Table can be used, for example, to represent the increasing cost of electricity year by year. See [Scale Tables](#).

For energy cost libraries, the Scale Tables are limited to entering a multiplier over time.

Applying Energy Costs in the Model

The following must be completed to view the Operation/Energy column in the Cost Report:

- The [Cost Settings panel](#) in Analysis Setup must have the following definitions:
 - Cost Calculation set to either Pump Energy Only or Calculate
 - Cost Definition checkbox enabled for Operation/Energy
 - Energy Cost set to Use Energy Cost Libraries
 - Cost Time Period must have a non-zero System Life
- Pump junctions must have the appropriate selection from the [Library Jct](#) dropdown
- Cost tab of each Pump Properties window must be set to Include Cost in Report
- [Output Control](#) must have the Operation Costs parameter added under the Cost Report

Library Connection

Creating a new Energy Cost Library will prompt connecting it automatically. If for whatever reason an existing Library is not connected in can be manually connected with the [Library Manager](#).

Cost Application Manager

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

This Cost Application Manager, accessed from an object's Cost tab, allows the user to do the following:

- Choose which cost libraries will be applied to each pipe and junction
 - Note that if multiple libraries are applied, costs from ALL of the selected libraries will be included.
- Choose when each pipe's or junction's life will start and/or end
- Specify fractional cost multipliers to be applied (for cases such as compressor energy when the compressor operates only part of the time)

The Cost Application Manager can also be accessed from the Tools menu - in which case it will show data for all Workspace objects.

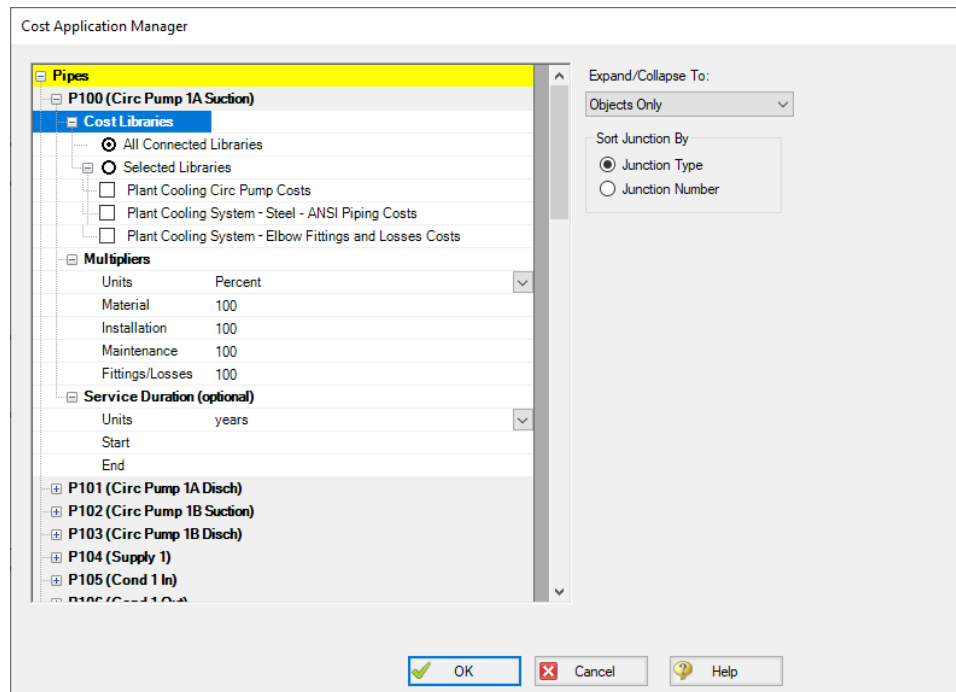


Figure 1: Cost Application Manager

How Duplicate Costs are Handled

To maximize flexibility for the user, multiple cost libraries can be connected to the same engineering library. What happens if there is cost data for the same engineering item in each library? For example, assume an engineering library, compressor.dat, exists with a specific compressor "Model 100A".

Two cost libraries exist based on compressor.dat:

- maxcost.cst ("Model 100A" installation cost = \$1000)
- bestcost.cst ("Model 100A" installation cost = \$750)

If both of these libraries are connected, and All Connected Libraries are applied to the compressor, AFT Arrow will interpret the cost for "Model 100A" as the sum of the two: \$1750.

There will be occasions you want cost items to be summed, and other cases where you do not. Obviously, if you have two libraries, one that tracks the maximum cost and the other that tracks the "best" cost of an item, you want to use only one cost at a time and not have the costs summed. This can be accomplished with the Cost Application Manager, by selecting only the desired library to apply.

However, perhaps the two acquisition costs do not represent the same item, but rather represent partial costs of that item that you want to maintain separately. When managed in this way, you want the costs to be summed.

Related Topics

Cost Calculation.....	465
Scale Tables.....	472
Select Size From.....	474
Cost Definitions.....	475
Cost Libraries.....	478
Cost Summary Window.....	483
Cost Report.....	485
Custom Monetary Units.....	484

Scale Tables

- Scale tables have a type (time or size) and a format (multiplier or cost). "Size" can refer to diameter, power, or C_v .
- Diameters can be quickly found and entered using the [Select Size From](#) button.
- The type of scale table defines the parameter that controls the variation in cost. For example, increasing diameter would generally cause an increase in cost, or energy prices may be projected to increase or decrease with time.
- Time type tables only accept the multiplier format.
- All scale tables require both a cost and a multiplier to determine the final cost value. Multiplier format tables require a base cost in the cost definition, cost format tables require a base multiplier.
- One scale table can be applied to multiple cost items - pipe materials, junctions, or fittings & losses.

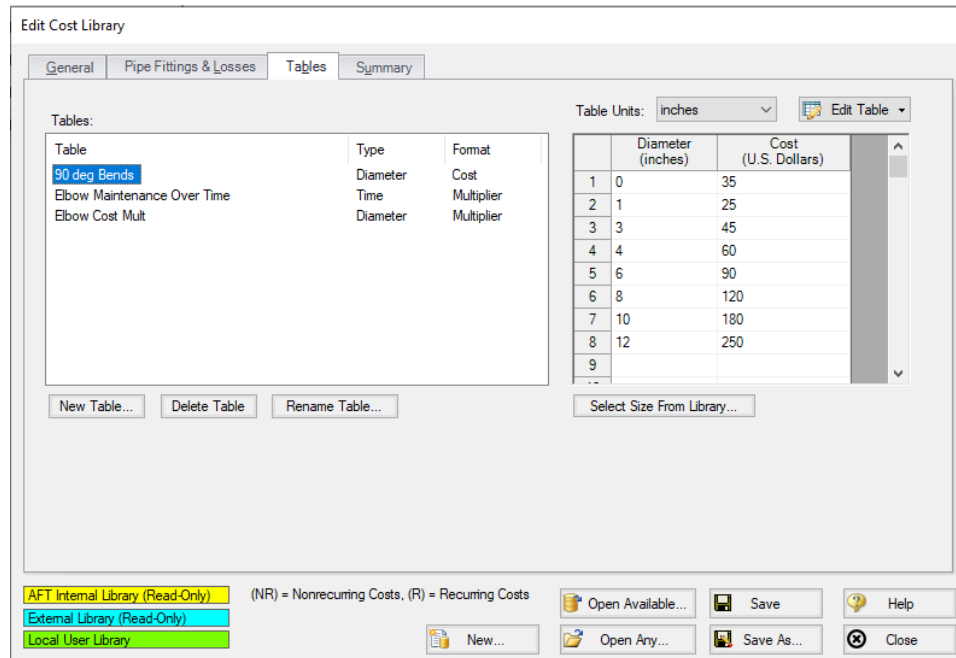


Figure 1: Scale table example of a table of costs (Diameter inches vs U.S. Dollars)

Energy Cost Libraries

Changes in [Energy Cost](#) over time can be captured with a scale table. Only time type scale tables can be applied to an Energy Cost Library.

Cost Libraries

Scale tables are applied in the cost definition area of the [Cost Library window](#). Pipe Materials only accept time scaling tables for recurring cost (shown in Figure 2)

	Cost #1	Cost #2	Cost #3
Description	1" Sch 40 Material	1" Sch 40 Installation	1" Sch 40 Maintenance
Cost Type	Material (NR) ▾	Installation (NR) ▾	Maintenance (R) ▾
Cost (U.S. Dollars)	2.2	7.2	
Cost Per	Length ▾	Length ▾	Time per Length ▾
Cost Per Units	feet ▾	feet ▾	years ▾
Cost Per Units #2			feet ▾
Time Scaling Table			Labor Inflation ▾

Figure 2: Time Scaling table applied to recurring cost

Fittings & losses and junction non-recurring costs can accept size scaling tables. Only one table can be applied to one cost. The selection of a table of multipliers will require a base cost, and the selection of a table of costs requires a base multiplier (default of 1). This is demonstrated below.

	Cost #1	Cost #2
Material	Steel - ANSI	Steel - ANSI
Material Size	All Sizes	All Sizes
Material Type	All Types	All Types
Use Size Table	Table of Multipliers	Table of Costs
Cost (U.S. Dollars)	52	
Cost Per	Item	
Cost Per Units		
Multiplier		1
Size Scaling Table	Elbow Cost Mult	90 deg Bends
Time Scaling Table		Low Maintenance Over Time

Figure 3: Applying size tables to fittings & losses

Recurring costs can include both a size scaling table and a time scaling table. This would allow you to specify one base cost and use scale tables to represent both a range of sizes and an increasing cost over time, for example. Specifying both types of tables would be useful for a sizing calculation with the [ANS module](#), where cost is being minimized by varying the size.

Related Topics

Cost Calculation	465
Cost Application Manager	470
Select Size From	474
Cost Definitions	475
Cost Libraries	478
Cost Summary Window	483
Cost Report	485
Custom Monetary Units	484

Select Size From

The Select Size From button opens the Nominal Diameter window, which allows you to quickly obtain the inner diameter for a nominal size and schedule. The Diameter window is called from the [Scale Tables window](#).

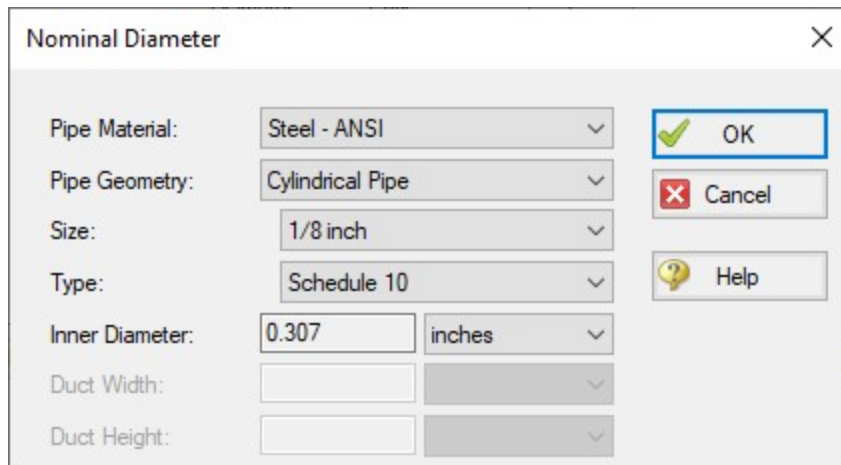


Figure 1: Nominal Diameter selection

Selecting a size and clicking OK enters the Inner Diameter into the currently selected Scale Table row.

Related Topics

Cost Calculation	465
Cost Application Manager.....	470
Scale Tables.....	472
Cost Definitions.....	475
Cost Libraries.....	478
Cost Summary Window.....	483
Cost Report.....	485
Custom Monetary Units.....	484

Cost Definitions

Depending on the contents of the associated Engineering Library there are tabs for Pipe Materials, Junctions, and Pipe Fittings & Losses. Each of these tabs serves the same purpose in slightly different ways - allowing the definition of costs for a certain component.

First, select a component of a certain type or size in the upper-left of the window. Then, click on the New Cost button. A new cost item is created in the table below as a new column. Some cells are displayed in black, which means that are not relevant to the type of entry selected.

Costs can be:

- Material (NR)
- Installation (NR)
- Maintenance (R)

where the (NR) and (R) indicate Non-Recurring and Recurring. Non-Recurring Costs are calculated as a sum total for the model, whereas Recurring costs require a time period and a total system life, as defined in the [Cost Settings panel](#) of Analysis Setup.

These costs are on either a per-item, per-length, or time basis. The costs can further be adjusted with [Scale Tables](#).

Pipe Materials Costs

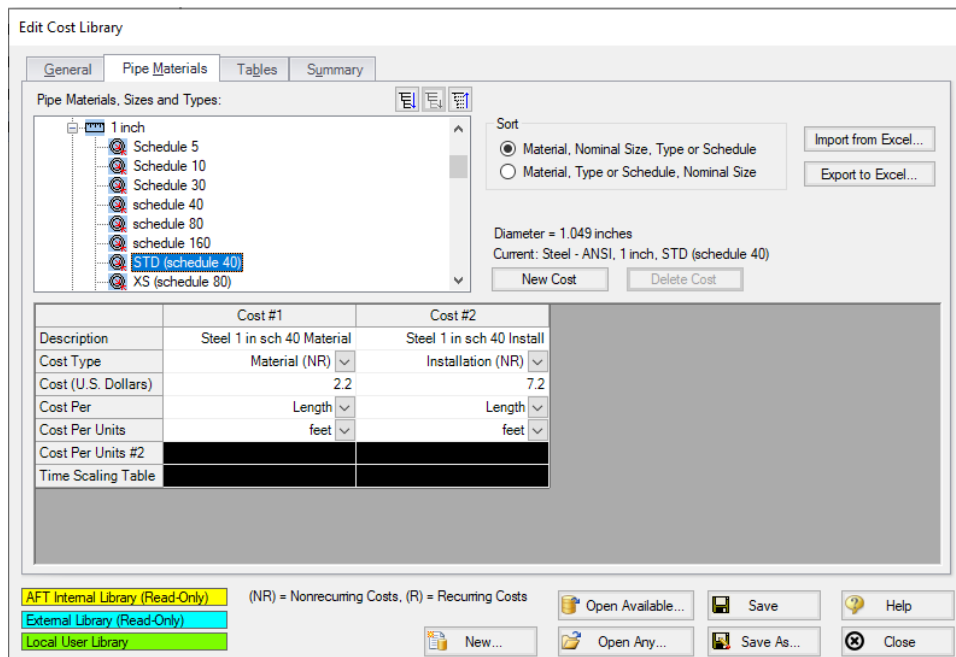


Figure 1: Cost Definitions for a Pipe Material

The Pipe Materials tab shows all the pipes in the engineering cost library. Costs can be entered at several levels. You can enter costs at the material level, the nominal size level, and finally at the type (e.g. schedule) level. Costs entered at the material apply to all pipes nominal sizes and types in that material type. Costs entered at the nominal size level apply to all schedules within that nominal size. Costs entered at the type level apply only to that type.

To make this process easier, values can be imported or exported to . The easiest way to import cost data from Excel is to first export the desired pipe Sizes and Types. Only the currently selected Sizes and Types will be exported (if the Pipe Material is selected, all will be exported) with any currently defined cost information included.

	A	B	C	D	E	F
1	Compressed Air Example Pipe Costs					
2	Cost Units:	U.S. Dollars				
3	Length Units:	feet				
4				===== COST PER UNIT LENGTH =====		
5	Material Name	Size	Type	Material	Installation	Maintenance
6	Steel - ANSI	1/8 inch	STD (schedule 40)			
7	Steel - ANSI	1/4 inch	STD (schedule 40)			
8	Steel - ANSI	3/8 inch	STD (schedule 40)			
9	Steel - ANSI	1/2 inch	STD (schedule 40)			

Figure 2: Excel Sheet for Importing Cost Definitions

The empty columns for cost per unit length can be filled in where cost data is required - it is not necessary to supply data for every cost, or for every Size/Type. After saving this sheet, the data can be imported directly with "Import from Excel."

Recurring costs for Pipe Materials can be modified with a Time [Scaling Table](#).

Junctions Costs

Costs can be entered for all junction types.

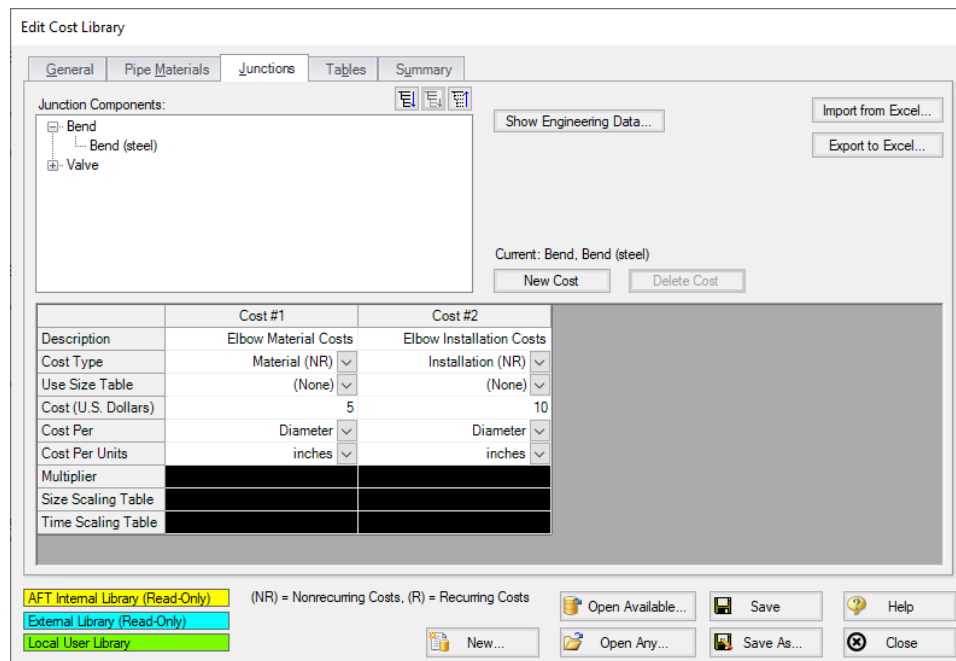


Figure 3: Cost Definitions for a Bend

Above is an example of entering cost data for a bend junction. Junction costs can also be entered at a higher level. For example, you can apply a cost to all Smooth Flanged Elbows.

Junction costs can be scaled with a Table of Multipliers, Table of Costs, or a Time Scaling Table.

The behavior of certain junctions varies slightly:

- **Compressors/Fans** - Cost can be entered per item (compressor) or per Power. This will allow an approximate scaling for the size of the compressor on a more reasonable basis than diameter.
- **Control Valves** - Costs can be entered per item or diameter. With a Scale Table, costs can be scaled based on C_v .
- **Other Junctions** - Costs can be entered per item or diameter. Scaling tables can also be used on a diameter basis.

Pipe Fittings & Losses Costs

Cost specification functions the same as non-compressor junctions, with the exception that fitting & loss costs can be associated with a particular pipe material and size. Since the pipe fitting & loss engineering library is included only in the AFT Arrow internal library, cost data for pipe fittings & losses can only be entered when the cost library is associated with the internal engineering library – unless you add your own custom fittings & losses to an engineering library. Then cost libraries can be associated with that library as well.

Related Topics

Cost Calculation	465
Cost Application Manager	470
Scale Tables	472
Select Size From	474
Cost Libraries	478
Cost Summary Window	483
Cost Report	485
Custom Monetary Units	484

Cost Libraries

AFT Arrow can track user-defined costs such as maintenance, installation and material cost with a Cost Library.

Note that AFT Arrow is not designed as a cost estimator. Instead, user-defined cost data can be used to compare different designs to determine which one has a lower cost. Like all parts of the model, the results of a cost calculation depend on how the costs have been defined by the user.

Relationship to Engineering Libraries

AFT Arrow uses Engineering Libraries to create collections of standardized components.

Cost Libraries must be created based on an engineering library to provide user-defined cost values for components in the engineering library. There can be [multiple cost libraries](#) for each

engineering library. This allows cost data sets to be maintained separately from the engineering data. This can be useful for maintaining costs in different currencies or differences based on geographical location or times of the year.

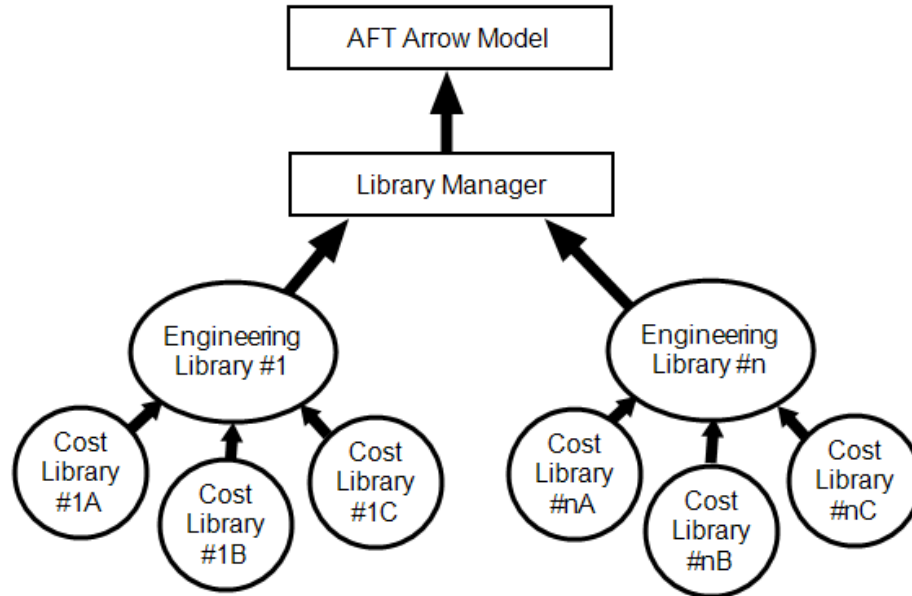


Figure 1: Relationship between Cost and Engineering Libraries

Like Engineering Libraries, Cost Libraries must be connected to be used. See [Library Browser](#) for information on how to connect libraries. Newly created cost libraries are connected automatically.

Edit Cost Library Window

The Edit Cost Library window is opened from the Library menu. This window allows you to create a cost library for items that are in a specific [engineering library](#). Costs can be entered for pipes, junctions, and pipe fittings & losses.

On the General tab, a cost library can be created with the New button, or an existing library can be accessed with one of the Open buttons.

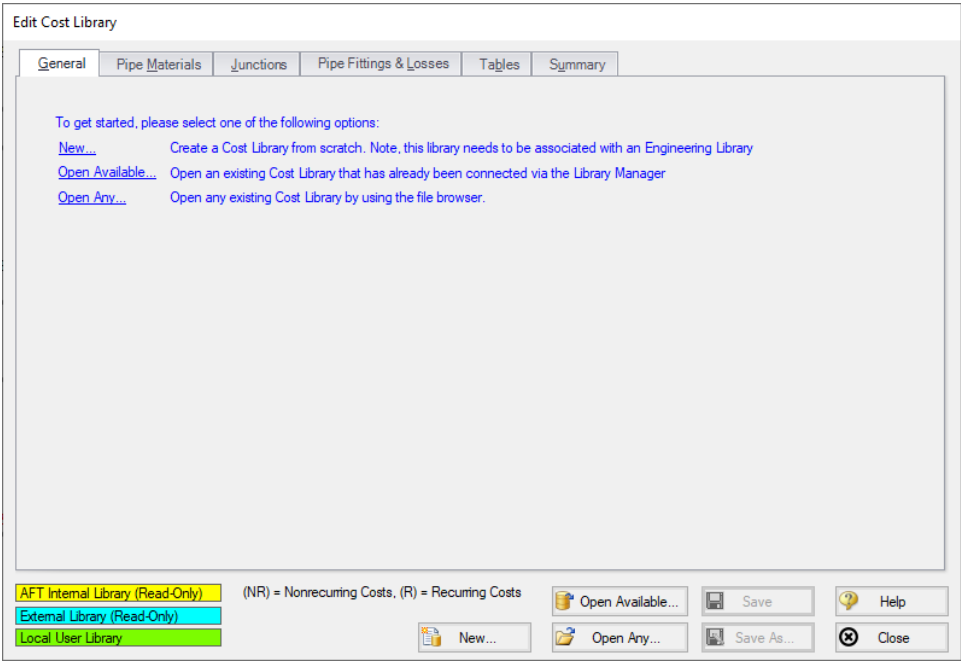


Figure 2: General tab of Cost Library Window

Clicking New brings up the following window:

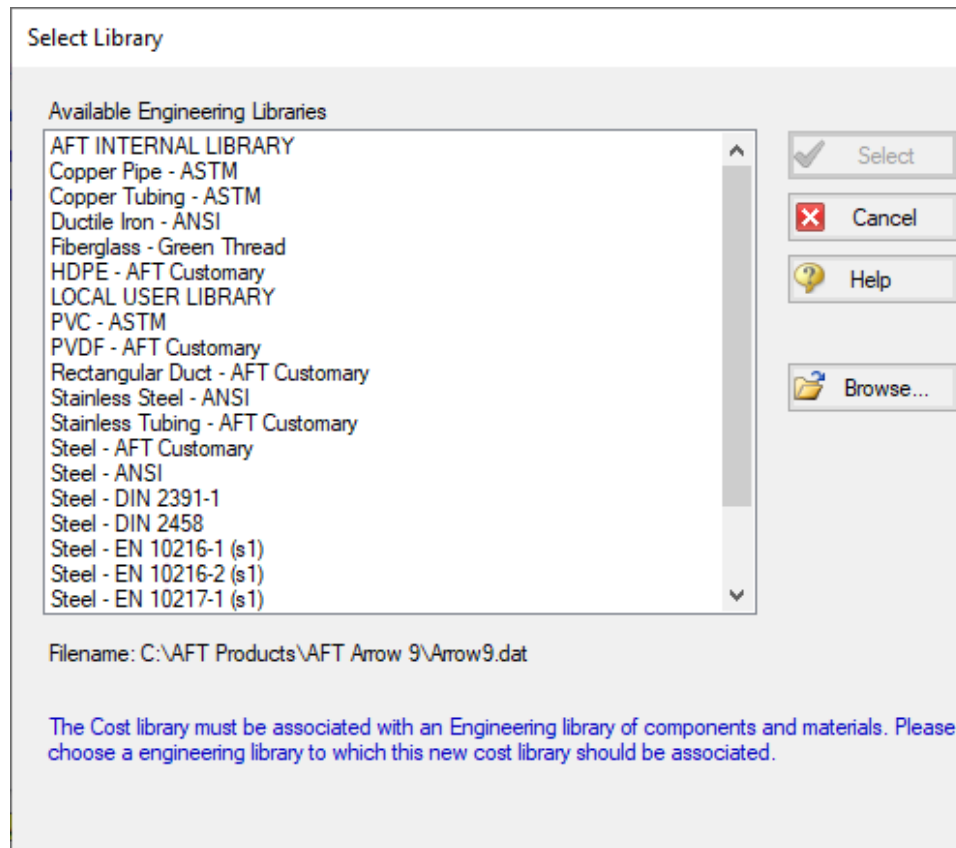


Figure 3: Select Library Window

All Cost Libraries need to be associated with an Engineering Library. After selecting a library, the Edit Cost Library Window will change to allow you to enter information into the newly created Cost Library.

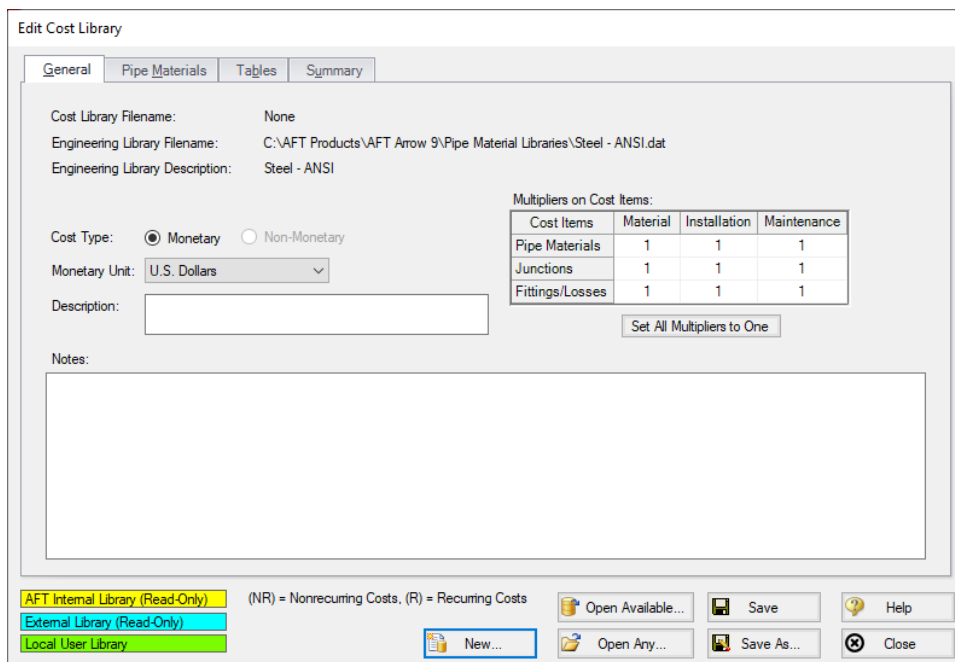


Figure 4: Cost Library Window - New Library

A Description is required for all Cost Libraries, which is displayed as the library's name in the Library Manager. Multipliers can be set for all items in the cost library using the Multipliers on Cost Items table. Notes are optional.

Cost Definitions

Depending on the contents of the associated Engineering Library there are tabs for Pipe Materials, Junctions, and Pipe Fittings & Losses. Each of these tabs serves the same purpose in slightly different ways - allowing the definition of costs for a certain component. See [Cost Definitions](#) for a detailed explanation.

Summary

The summary tab allows for an easy way to quickly review what has been defined in a cost library. This is different than the [Cost Summary Window](#), as this summary is a listing of all items defined as costs, whereas the Cost Summary Window lists objects in the model with costs applied to them.

The costs are grouped by type, and only defined costs are shown. An example of defined costs on a Pipe Materials library is shown below.

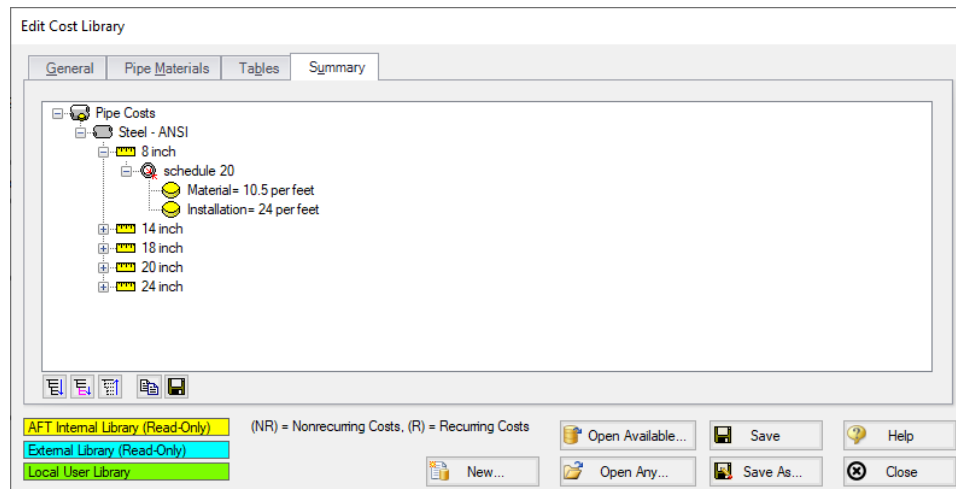


Figure 5: Cost Summary

Related Topics

Cost Calculation	465
Cost Application Manager	470
Scale Tables	472
Select Size From	474
Cost Definitions	475
Cost Summary Window	483
Cost Report	485
Custom Monetary Units	484

Cost Summary Window

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The Cost Summary window, opened from the Tools menu shows all connected [cost libraries](#) and [energy cost libraries](#), and which pipes and/or junctions are obtaining costs from each library.

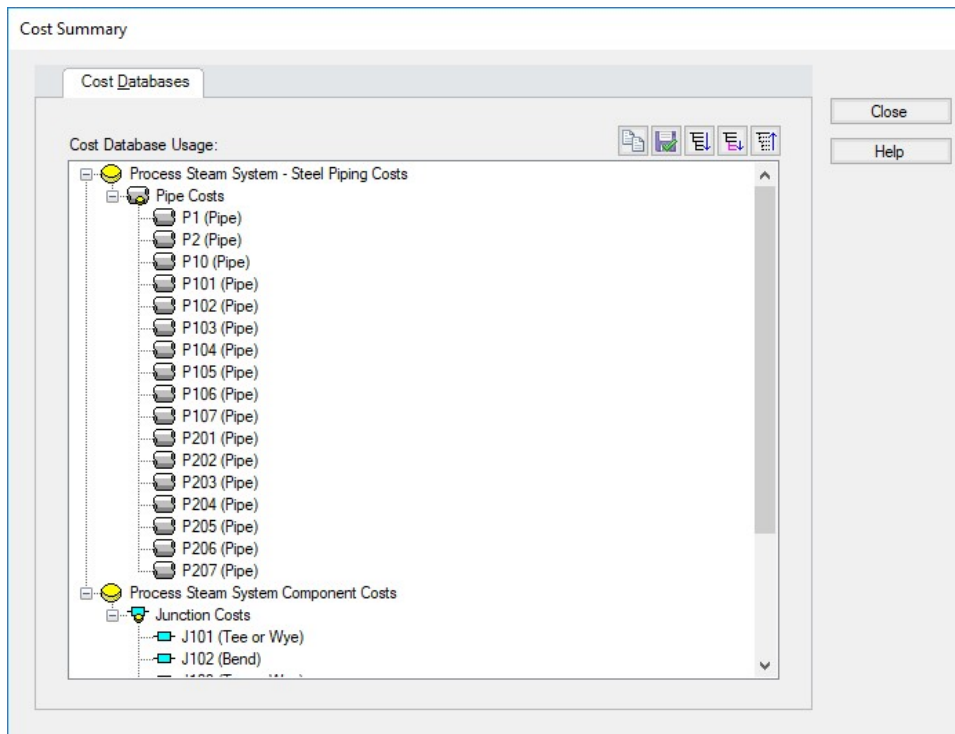


Figure 1: Cost Summary Window

Related Topics

Cost Calculation	465
Cost Application Manager.....	470
Scale Tables	472
Select Size From.....	474
Cost Definitions.....	475
Cost Libraries.....	478
Cost Report.....	485
Custom Monetary Units.....	484

Custom Monetary Units

New Monetary Units can be added with the following steps:

1. Open [User Options](#) from the Tools menu
2. Under Unit Systems, select Show/Hide Unit Items
3. In the Unit Family Type list box, select Monetary Units
4. The Custom Units button will appear on the right, select New
5. Enter the name, description and conversion to U.S. Dollar in the appropriate fields.
6. Click OK

The new Monetary units can be specified as your default units and used in [Cost Libraries](#). If you do have a mixture of cost libraries with different Monetary Units, the conversion factors to U.S. Dollars will need to be updated before the cost is calculated.

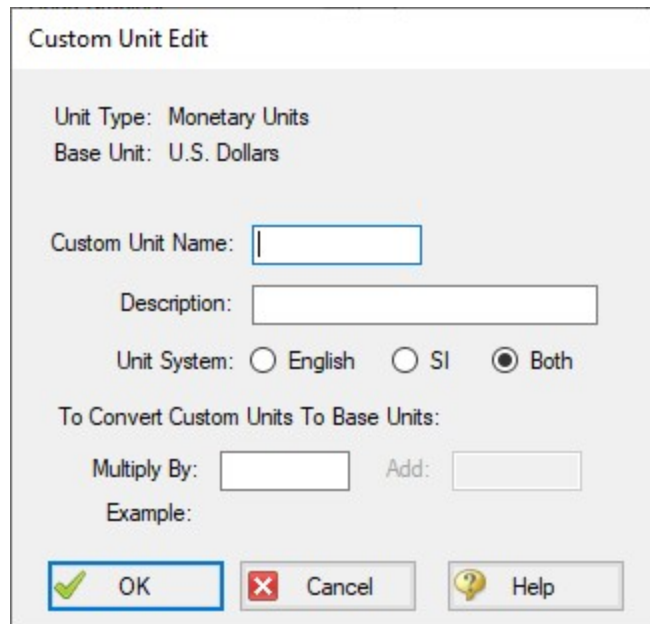


Figure 1: Custom Unit Edit Window

Related Topics

Cost Calculation.....	465
Cost Application Manager.....	470
Scale Tables.....	472
Select Size From.....	474
Cost Definitions.....	475
Cost Libraries.....	478
Cost Summary Window.....	483
Cost Report.....	485

Cost Report

The Cost Report is a detailed list of all costs for the system displayed on the Cost Report tab in the General section of the [Output window](#). The content of the Cost Report can be customized in the [Output Control window](#) on the Display Parameters tab and the Format & Actions tab.

The grand total cost is shown on the top line, with subtotals for each category on the second line. In the main table area costs for each pipe, junction, and pipe fitting & loss included in the cost calculation are shown.

Output Control Format & Action allows you to hide items that have zero cost, group costs by pipe, or show only the subtotals for each category.

General Warnings Cost Report Compressor/Fan Summary Valve Summary Tank Summary									
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/ Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									97,449
Total of All Shown Costs			10,124	4,463	14,587	0	82,861	82,861	97,449
Pipe Subtotal			824	863	1,687	0	0	0	1,687
P1	Pipe	Pipe	7	7	14	0	0	0	14
P2	Pipe	Pipe	170	169	339	0	0	0	339
P3	Pipe	Pipe	170	169	339	0	0	0	339
P4	Pipe	Pipe	170	169	339	0	0	0	339
P5	Pipe	Pipe	170	169	339	0	0	0	339
P6	Pipe	Pipe	34	45	80	0	0	0	80
P7	Pipe	Pipe	34	45	80	0	0	0	80
P8	Pipe	Pipe	34	45	80	0	0	0	80
P9	Pipe	Pipe	34	45	80	0	0	0	80
Branch Subtotal			600	900	1,500	0	0	0	1,500
J3	Branch	Branch	150	225	375	0	0	0	375
J4	Branch	Branch	150	225	375	0	0	0	375
J5	Branch	Branch	150	225	375	0	0	0	375
J6	Branch	Branch	150	225	375	0	0	0	375
Compressor/Fan Subtotal			1,500	500	2,000	0	82,861	82,861	84,861
J2	Compressor/Fan	Compressor	1,500	500	2,000	0	82,861	82,861	84,861

Figure 1: The Cost Report is displayed in the General section of the Output window

Using the Library Sources Tables

AFT Arrow displays the source of all cost data in a run, and you are strongly encouraged to review this data to verify that the solution was based on the appropriate cost data.

In the [Output window](#), the sources of all cost data are displayed for each pipe and junction in the Library Sources table. Figure 2 shows the Library Sources for an example case.

Workspace Model Data Output Graph Results Visual Report				
General				
Pipes Axial Points Heat Transfer Library Sources				
Pipe	Name	Pipe Material	Pipe Installation	
1	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
2	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
3	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
4	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
5	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
6	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
7	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
8	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	
9	Pipe	Compressed Air Example Pipe Costs	Compressed Air Example Pipe Costs	

Figure 2: Library Sources table for pipes

Related Topics

Exporting to Excel.....	422
GSC in Output.....	493
Output Messages & Warnings.....	709
Cost Calculation.....	465
Cost Application Manager.....	470
Scale Tables.....	472
Select Size From.....	474
Cost Definitions.....	475
Cost Libraries.....	478
Cost Summary Window.....	483
Custom Monetary Units.....	484

Add-on Modules

AFT Arrow has three optional add-on modules which extends modeling capabilities into other areas. The modules allow multi-variable goal seeking and control simulation, full system sizing, and time varying simulation. The [Modules panel](#) within the Modules group in the [Analysis Setup](#) window allows you to activate and enable any of the three AFT Arrow add-on modules. An additional license to the base AFT Arrow license is required to use a module. These modules include:

- [GSC \(Goal Seek & Control\)](#) - Identifies output parameters that yield desired output values and simulations control functions.
- [ANS \(Automated Network Sizing\)](#) - Evaluates the complex interaction of system variables to reveal combinations of cost savings.
- [XTS \(Extended Time Simulation\)](#) - Models dynamic system behavior and how critical system parameters vary over time.

Related Topics

Modules Group.....	77
Modules Panel.....	78

GSC - Goal Seek & Control

What is the GSC Module?

An optional add-on module to AFT Arrow which allows users to perform multi-variable goal seeking and simulate system control functions.

Many engineering modeling tasks involve more than just directly solving a system. In some cases a manual, “cut and try” method of changing input variables to achieve desired operating results is required. When a single parameter is being changed, the manual process (while tedious and time consuming) can be used successfully. However, varying two or more parameters at the same time quickly becomes impractical.

The GSC module automates the process of changing input variables to achieve desired design goals. For single variable situations, the GSC module offers the advantage of being much faster than manual methods. Further, it provides a practical tool for solving cases when there are two or more variables.

How Does the GSC Module Work?

The GSC module employs numerical optimization technology. The optimization engine employed by the AFT Arrow GSC module uses state-of-the-art optimization technology licensed from Vanderplaats Research and Development, the leading company in optimization tech-

nology. However, rather than minimizing some function value such as system cost or weight, the GSC module satisfies equality criteria using “goal programming” methods.

The core hydraulic solution in AFT Arrow is performed by the Hydraulic Solver. In standard AFT Arrow usage, the Hydraulic Solver is called once to solve a system. The GSC module’s Numerical Optimizer is used to call the Hydraulic Solver repeatedly, thus solving a series of models with different inputs. The Numerical Optimizer adjusts the user specified [Variables](#) in order force the Hydraulic Solver output values to agree with the user specified [Goals](#). Figure 1 depicts the relationship between the different components.

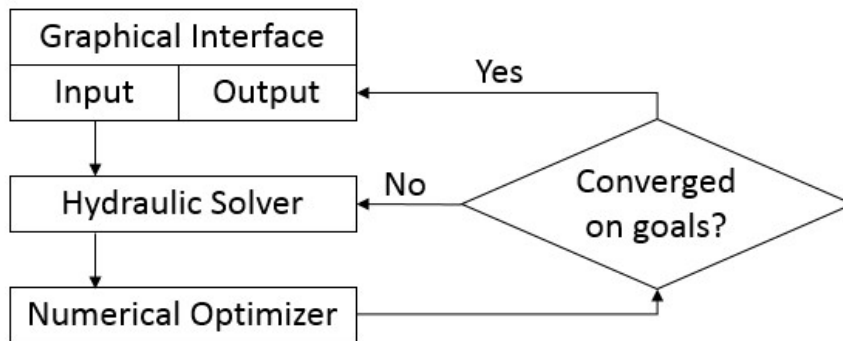


Figure 1: The GSC module flowchart shows how the Hydraulic Solver is called repeatedly in an iterative loop

Using the GSC Module

After AFT Arrow is loaded, the GSC module can be activated or deactivated for use from the Startup window, Tools menu, or [Quick Access Panel](#). Whether or not GSC is activated impacts the Analysis menu, Model Data window and Output window.

If the GSC module is active, the user can still run models without goal seeking. This is selected under Goal Seek & Control on the Analysis menu. Hence there are three possibilities for GSC.

- GSC is not active
- GSC is active and goal seeking is ignored
- GSC is active and goal seeking is performed

Related Topics

Modules Panel.....	78
Add-on Modules.....	489
ANS - Automated Network Sizing.....	500
XTS - Extended Time Simulation.....	586

Goal Seek and Control Group.....	100
Variables Panel.....	101
Goals Panel.....	106
Numerical Controls Panel.....	111
Goal Seek & Control in Model Data.....	491
GSC in Output.....	493
GSC in Scenarios.....	493
Changing Input Values to GSC Results.....	494
Goal Seek and Control Troubleshooting.....	495

Goal Seek & Control in Model Data

When the GSC module is active, the [Model Data window](#) displays a list of all variables and goals in the General section. This is useful for documentation purposes. See Figure 1.

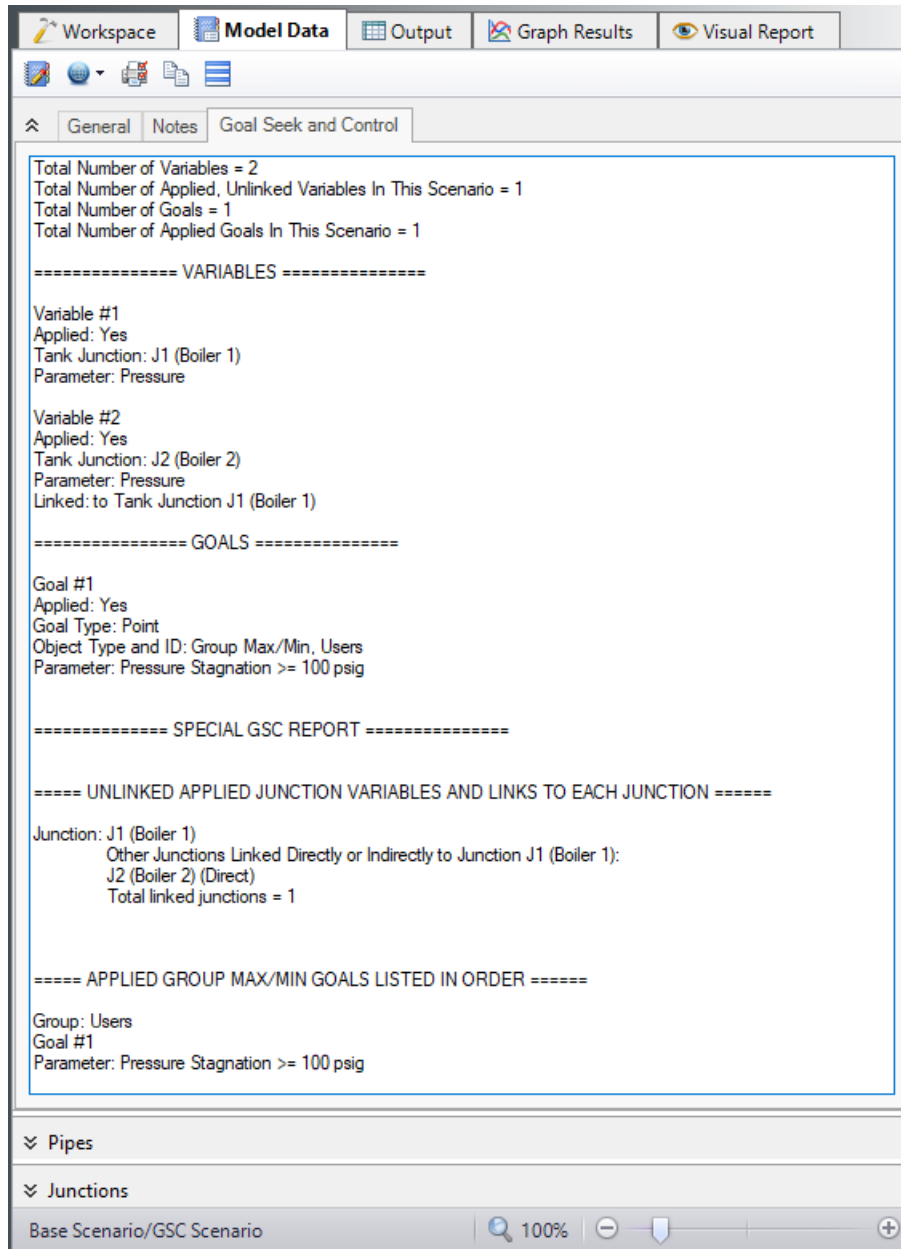


Figure 1: Model Data window shows the variable and goal input data

Related Topics

GSC - Goal Seek & Control.....	489
GSC in Output.....	493
GSC in Scenarios.....	493
Changing Input Values to GSC Results.....	494
Goal Seek and Control Troubleshooting.....	495

GSC in Output

The results for a Goal Seek & Control run are displayed in the [Output window](#) in the general section in two tables called GSC Variables and GSC Goals (see Figure 1).

The figure shows two screenshots of a software interface. The top screenshot displays the 'GSC Variables' table, and the bottom screenshot displays the 'GSC Goals' table. Both screenshots show a menu bar with options: General, Warnings, Design Alerts, Tank Summary, GSC Variables, and GSC Goals.

Variable	Object Type	Object	Parameter	Value	Units
1	Tank	J1 (Boiler 1)	Pressure	313.5	psia
2	Tank	J2 (Boiler 2)	Pressure	313.5	psia

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Users	N/A	Pressure Stagnation	100	99.99	psig	J108 (User 3)

Figure 1: Calculated GSC Variables table (top) shows values required to achieve goals. GSC Goals table (bottom) shows user's goals and actual values achieved (these should match).

Related Topics

Exporting to Excel.....	422
Cost Report.....	485
Output Messages & Warnings.....	709
GSC - Goal Seek & Control.....	489
Goal Seek & Control in Model Data.....	491
GSC in Scenarios.....	493
Changing Input Values to GSC Results.....	494
Goal Seek and Control Troubleshooting.....	495

GSC in Scenarios

GSC [variables](#) and [goals](#) will appear in all [scenarios](#). In other words, you cannot create a variable or goal that will appear in some scenarios and not others. Deleting a variable or goal in one scenario will delete it in all scenarios. However, the application of variables and goals can differ from scenario to scenario. This allows you to perform goal seeking with different variables and goals being used in different scenarios.

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek & Control in Model Data.....	491
GSC in Output.....	493

Changing Input Values to GSC Results.....	494
Goal Seek and Control Troubleshooting.....	495

Changing Input Values to GSC Results

As discussed previously, the Output window [GSC Variables tab](#) indicates how the input parameters must be varied to satisfy the goals. Once you have these results, you may want to keep and use them in the model. There are two ways to do this: [Transferring Results to Initial](#) and generating a Disconnected [Scenario](#).

Transferring Results to Initial

In general, the Output window supports a feature called "Transfer Results to Initial Guesses". This is available on the Edit menu. When used without the GSC module, it transfers calculated results for flow, pressure and temperature to the initial guess values of the pipes and junctions. When used after a GSC run it gives you three options: to transfer the GSC Variable Data only, transfer the calculated results only, or transfer both.

If you choose either of the first two options that transfer GSC Variable Data, the junction data on the Properties window will be changed to the GSC Variable results. This means that one could then choose to Ignore goal seeking, re-run the model, and obtain the same results as before.

Generating a Disconnected Scenario

After a run is complete, a special type of scenario can be created called a [disconnected scenario](#). A disconnected scenario itself breaks the inheritance of normal child scenarios so that it is truly standalone scenario. This is ideal for GSC results in that it may not make sense to save the GSC results for future use if the model can otherwise be changed in arbitrary ways - if the model were changed the GSC results would be invalidated, meaning the user would need to rerun GSC.

For example, assume a GSC run is made to find the valve open position required to achieve a certain flow. If this result is saved, and then the pipe size is later changed, the valve open position would no longer supply the desired flow.

Creating a disconnected scenario helps prevent changes to the scenario since disconnected scenarios cannot be changed by propagated changes from other scenarios.

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek & Control in Model Data.....	491
GSC in Output.....	493
GSC in Scenarios.....	493
Goal Seek and Control Troubleshooting.....	495

Goal Seek and Control Troubleshooting

- [When Goals Cannot Be Achieved](#)
- [Goal Seek and Control Adjustments](#)
- [What To Do if the Solver Gets Stuck in GSC](#)
- [Goal Seek and Control Inconsistencies](#)

Related Topics

GSC - Goal Seek & Control.....	489
Goal Seek & Control in Model Data	491
GSC in Output.....	493
GSC in Scenarios.....	493
Changing Input Values to GSC Results.....	494

When Goals Cannot Be Achieved

In some cases the GSC module may not be able to achieve a [goal](#). This can happen for the following reasons:

- A goal is not physically realistic
- A better starting point is needed for the input values assigned as GSC Variables
- Hydraulic solution tolerances or optimizer [numerical parameters](#) require adjustment

In such cases, the goal will not be satisfied in the results. In other words, the Actual Goal Value and User Goal Value will differ (Figure 1).

In all cases the user will be warned when goals were not met. Figure 1 shows the Output window with such a warning.

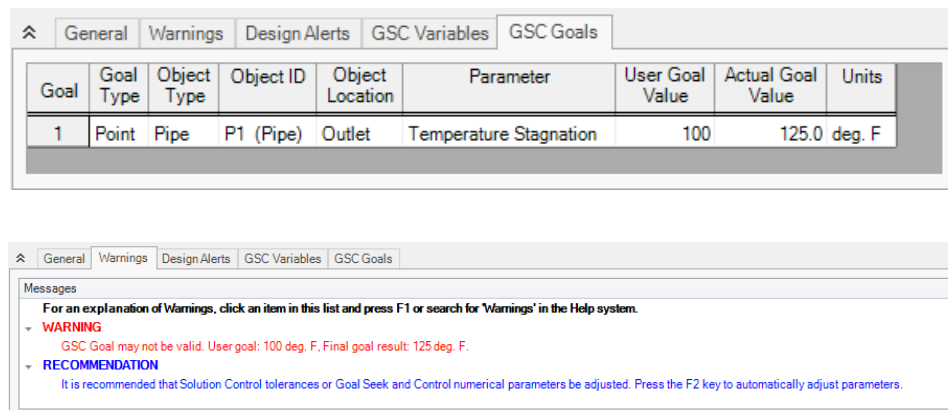


Figure 1: Warning display in Output window when a goal cannot be met

Physically unrealistic goals

It may be possible that a goal is not achievable because it is not physically possible. For example, a compressor/fan may be too small to satisfy a flowrate goal at some pipe.

To determine whether a goal is unrealistic, try changing the goal and rerunning the model. If the goal can be met in some cases and not others, this is an indicator that the original goal was unrealistic.

Better starting point needed

In some cases the goal may be so far away from the original model that the GSC module has difficulty finding a solution. This is not common, but it is worth trying to adjust the starting point of parameters which are being varied.

Changing control parameters

The Numerical Optimizer determines goal search directions by approximating a gradient based on perturbed variable values. By default it uses a central difference approximation, but forward difference can also be used. If the hydraulic solution is not converged to a sufficiently small tolerance, the gradient approximation will not be sufficiently accurate and hence a good search direction cannot be obtained. The end result is that the goal seeking fails.

There are two general areas that can be changed in such cases: the hydraulic solution tolerances in the [Tolerance panel](#), and numerical control parameters in the [Numerical Controls panel](#).

In the next few sections recommendations on how to manually change these parameters will be given. However, before making a manual change it is definitely worth trying the automated parameter adjustment feature.

Figure 1, bottom picture, shows the warning when a goal is not met, and also shows a recommendation and suggests pressing the “F2” key. When doing so, a window appears as shown in Figure 2. This window will offer up to three options for automatic parameter adjustment. It is typically a good idea to accept the default changes, click OK, and rerun the model.

The purpose of the Figure 2 window is to simplify for the user making common changes to resolve goal seeking problems.

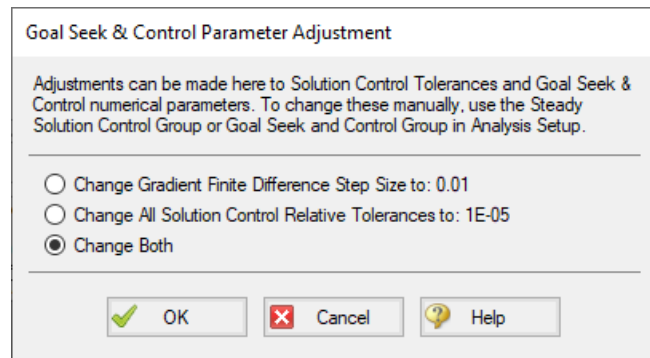


Figure 2: When goal seeking fails, pressing the F2 key will show this window which allows the user to automatically implement recommend numerical parameter adjustments

Caution: The user should be very cautious in manually changing parameters in the Tolerance panel and the Numerical Controls panel. Making changes without a good understanding of the parameters involved can lead to incorrect results.

- **Changing tolerance in Steady Solution Control Group >Tolerance panel-** By default AFT Arrow uses Relative Tolerance criteria set to 0.0001 (see Figure 3). Goal seeking convergence can sometimes be improved by reducing these values to 0.00001 or 0.000001.

See [Tolerance panel](#) for a more in-depth discussion.

- **Changing goal seeking numerical control** - Figure 4 shows the Numerical Controls panel in Analysis Setup. The most useful parameter one can adjust is the Relative Finite Difference Step Size. Using values between 0.01 and 0.001 are recommended. In some cases using a value as high as 0.1 may be tried.
- **Changing Multi-Variable Goal Weighting** - When two or more variables (and, hence two or more goals) are applied, the relative influence of each goal is important. By default, the GSC module "normalizes" each goal. This means that the difference between each goal and the current value is divided through by the goal. A Root Mean Square (RMS) of all normalized values is then taken, and this function is then driven to zero. The normalization approach tends to work better in cases where the numerical values of different goals are very different.

There are two other options available. These can be selected on the Numerical Controls panel (Figure 4). The second option is to not normalize the goals. Each goal is then subtracted from the current value, and an RMS is taken of the differences. This becomes the function that is driven to zero. This method tends to work best when the numerical values of the goals are similar in magnitude.

A final option is for the user to input weighting factors. This approach uses the normalized approach described previously, but multiplies each normalized value by a user specified weighting factor before performing the RMS. The general approach here is if one or more of the goals is not being reached, increase its weighting factor. The weighting factors are entered into a column in the goals table on the [Goals panel](#). This column will become visible when the third option is chosen.

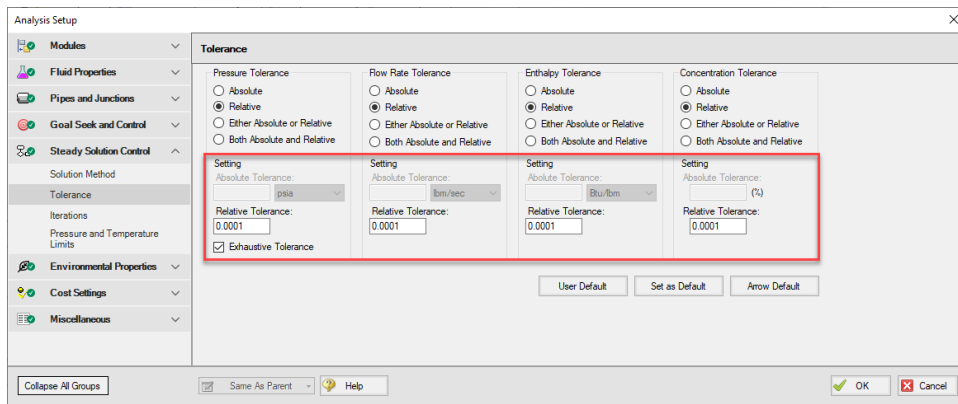


Figure 3: Reducing tolerance criteria may resolve goal seeking difficulties

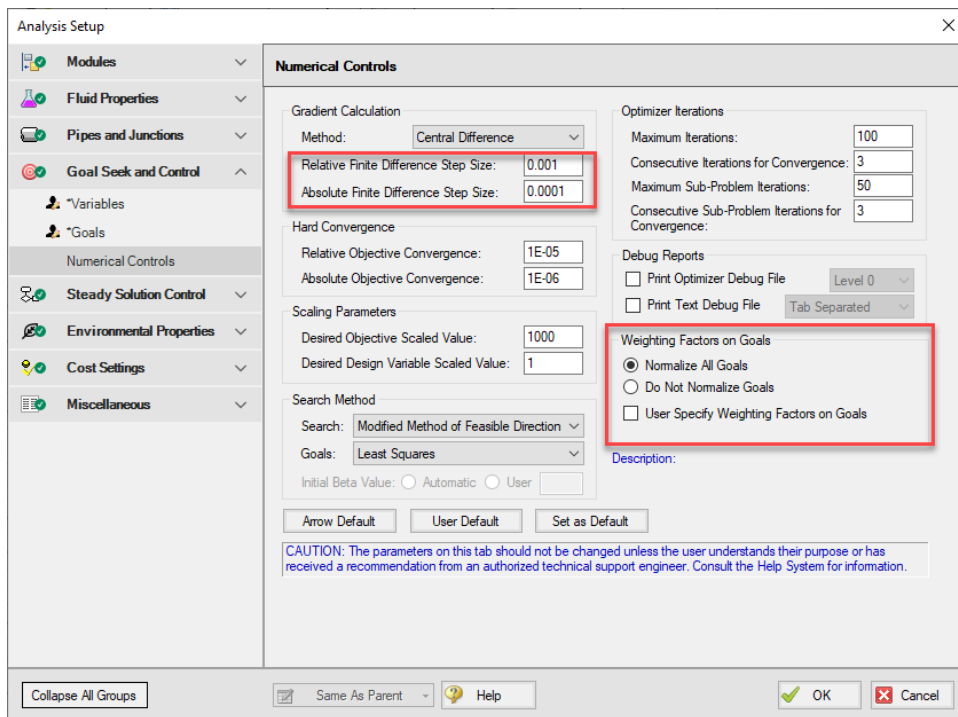


Figure 4: Changing certain numerical control parameters in the Numerical Controls panel may resolve goal seeking difficulties

Related Topics

Goal Seek and Control Troubleshooting	495
What To Do if the Solver Gets Stuck in GSC	499
Goal Seek and Control Adjustments	499
Goal Seek and Control Inconsistencies	500

GSC Goal may not be valid	718
Goals Panel	106

Goal Seek and Control Adjustments

When a GSC run results in the goals not being met, the user may want to adjust GSC search [numerical control parameters](#) or standard [Solution Control](#) parameters. These can be automatically adjusted after a run if a warning has been displayed and the user is prompted to press the F2 function key. When the user does this, recommended changes are offered which the user can choose to have automatically applied.

The changes will affect either the [Goal Seek and Control Group](#) in Analysis Setup and/or the [Steady Solution Control Group](#).

Related Topics

Goal Seek and Control Troubleshooting	495
When Goals Cannot Be Achieved	495
What To Do if the Solver Gets Stuck in GSC	499
Goal Seek and Control Inconsistencies	500

What To Do if the Solver Gets Stuck in GSC

Figure 1 of the [Introduction to the GSC Module](#) depicts the logic of how the Hydraulic Solver and Numerical Optimizer work together. Discussed elsewhere is the issue of [physically unrealistic goals](#). Occasionally the user may have specified a goal which is in fact physically realistic, but in the process of going from a physically realistic starting point to a similarly physically realistic goal, a physically unrealistic system may be proposed by the Numerical Optimizer. What happens in such cases?

When performing goal seeking with the GSC module, AFT Arrow has been designed to gracefully handle such cases. One of two things will typically happen:

- The Hydraulic Solver will experience a computational error
- The Hydraulic Solver will not be able to converge

In the first case, AFT Arrow does not stop and tell the user of the error, but instead sets an internal flag that the system used for the particular iteration that caused the error was a poor system and should not be considered further by the Numerical Optimizer. In other words, it skips the system that caused the error. This will happen in a way which is transparent to the user and requires no intervention.

The second case is of more interest here. In this case the Hydraulic Solver gets stuck on one of the systems proposed by the Numerical Optimizer and cannot progress. The Hydraulic Solver will continue trying to converge until it reaches the iteration limit. Once it reaches that limit, it will react in a similar way to the first case above – it will set a flag and skip that system.

However, the default maximum number of iterations is 50,000 (specified in the [Iterations panel](#)). It is perfectly fine for one to wait until this limit is reached and allow the AFT Arrow to handle the

situation automatically, but depending on the size of the model it may take awhile to get to 50,000 iterations. In such cases it may be desirable to allow AFT Arrow to more quickly conclude the model will not converge by reducing the maximum iterations to something like 5,000.

Tip: If the Hydraulic Solver appears to get stuck while performing a GSC run, you can pause the Solver, open the [Steady Solution Control group](#) from the Other Actions menu, and reduce the [Maximum Iterations](#). The default value is 50,000. Depending on the model size, a reduced value of 5,000 may be appropriate.

Related Topics

Goal Seek and Control Troubleshooting	495
When Goals Cannot Be Achieved	495
Goal Seek and Control Adjustments	499
Goal Seek and Control Inconsistencies	500

Goal Seek and Control Inconsistencies

If an object in the [Goal Seek and Control group](#) is defined with a parameter that has not been defined in that object's property window, an error will appear stating "Junction XX does not have XX parameter modeled. Cannot use as a GSC variable."

In this case, the user may resolve this inconsistency by either selecting a different parameter for the variable in the Goal Seek and Control group, or by changing the specified junction to be defined using the specified variable.

Related Topics

Goal Seek and Control Troubleshooting	495
When Goals Cannot Be Achieved	495
What To Do if the Solver Gets Stuck in GSC	499
Goal Seek and Control Adjustments	499

ANS - Automated Network Sizing

What is the Automated Network Sizing (ANS) module?

Sizing a single pipe or a single compressor for minimum cost is relatively straightforward with manual iteration. When considering a network of pipes, the manual approach of selecting cheaper components one at a time quickly becomes not only impractical, but ineffective. While manual component selection can certainly *reduce* cost, it often does not result in a *minimized* cost.

The path to *minimizing* cost is often unintuitive. Replacing certain components with more expensive ones often results in a lower total system cost. Solving such a problem - with dozens or

perhaps hundreds of variables - cannot reasonably be completed by hand. A computational approach quickly finds ways to reduce cost that may never be found by other means.

In addition, components cannot be arbitrarily changed without impacting overall operation. Any given system has certain requirements that must be met. Reducing pipe sizes too much, for example, may result in flow rates that are unacceptably low. There is a balance between reducing cost and providing the desired operation. The ANS module automates this process, minimizing the cost of complex systems while still meeting requirements with ease.

Note: ANS is not designed to be a **cost estimator**. It instead offers an approach for comparing potential designs using cost as a common denominator. The value of ANS is in the *minimization* of cost, not the precise estimation of total cost.

Using the ANS Module

The ANS module can be enabled from the [Modules panel](#) in [Analysis Setup](#). Activating ANS adds the Automated Sizing item to the Analysis menu, and enabling ANS affects many areas of the AFT Arrow interface.

If the ANS module is active, models can still be run without ANS by selecting Analysis > Automated Sizing > Not Used. It is also possible to run models without sizing by selecting Do Not Size in the [Sizing Method panel](#). These options create three distinct possibilities:

- **ANS is not active** - this is standard AFT Arrow mode.
- **ANS is active, but not sizing** - the [Sizing Window](#) and Sizing panels are visible but will not be used in simulation.
- **ANS is active and sizing** - the Sizing Window and Sizing panels are visible and the settings are applied during simulation.

Examples

ANS is best understood by sizing a model yourself. We have created several examples to help walk you through what is required with ANS: ANS Examples.

Related Topics

Modules Panel.....	78
Add-on Modules.....	489
GSC - Goal Seek & Control.....	489
XTS - Extended Time Simulation.....	586
Automatic Sizing Group.....	152
Sizing Summary Panel.....	153
Sizing Fundamentals.....	502
Sizing Window.....	516
Solution Progress with ANS.....	536

Sizing Results.....	538
Workspace Integration.....	539
Junction Loss Factors.....	542
Improving and Troubleshooting the Sizing Process.....	542

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Sizing Fundamentals

How Does the ANS Module Work?

ANS is built to allow you to approach the automated sizing process in a way appropriate to your system.

You must specify an **Objective - minimizing or maximizing a specific system variable**. Most commonly, the ultimate Objective is to minimize monetary cost.

The pipes to be sized must be defined - any given pipe may be included or excluded from the automated sizing process. It may not be desired to automatically size every pipe in the entire system independently. It is common that groups of pipes - such as those in a header - must be of equal size. This is accomplished by placing them into **Common Size Groups - sets of pipes that are automatically sized but must all be the same size**.

The selection of piping is generally restricted to certain materials, sizes, and types independent of fluid flow concerns. Perhaps certain system piping must be Stainless Steel of any common size, whereas other piping in the system can be regular Steel, but is limited to three on-hand diameters. These are represented with **Candidate Sets** - selections of pipe sizes and types that are allowed.

If the only goal is to minimize cost, the answer is trivial - select the smallest pipe sizes available. Clearly, this is not practical as systems are not built for the sake of being inexpensive, but are meant to accomplish some goal. **These goals, such as minimum flows or pressures, are Design Requirements**. Any number of Design Requirements can be specified in a system, controlling the allowed fluid behavior.

There are several different **Sizing Methods** available. These methods approach the same Objective in different ways, with certain methods being more appropriate for certain systems. Agreement between methods is a strong indication that the ideal sizes have been selected.

Analysis vs. Design

Sizing a system to minimize or maximize a system-wide quantity like monetary cost or pipe weight is a complex process with many considerations. AFT Arrow is foremost an analysis tool,

but the ANS module is a design tool. While closely related, analysis and design are different in many ways.

Engineering analysis is the process of using accepted calculation methods to predict the behavior of a given system. Analysis requires the full specification of the system before calculations are made. For example, fundamental fluid equations can be used to determine the flow in a pipe of a known size.

When a new system is being designed, the pipe size is not known. Instead, it might be desired to find the size necessary to meet a minimum flow. The engineer can make an educated guess, perform the analysis, modify the size, and repeat the process. The analytical methods are used iteratively to arrive at a final design.

Even for a simple problem, the iterative *Specify Model > Analyze > Investigate Results > Adjust Model* method can be very inefficient. For simple systems, it is often possible to solve for a design variable by manipulating the fundamental equations. This would allow the process to be completed in one design iteration.

For complex problems direct solutions are impractical or impossible. Instead, the iterative method can be applied with numerical methods. AFT Arrow already automates the "*Specify Model > Analyze*" portion of the process. AFT Arrow ANS further simplifies the process by using intelligent searching algorithms to automate the "*Investigate Results > Adjust Model*" steps.

By providing ANS with information on the requirements of a system, it offers a true design-oriented approach, automating a complex iterative process.

Related Topics

Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Objective

The **Objective is the goal of the sizing process** - the variable of concern and whether it should be minimized or maximized.

A valid Objective variable must be calculable as a total for any potential system configuration. For example, a system can be sized to minimize or maximize weight because any pipe has a weight measured in a common unit. However, different pipes may have different pressure ratings - it does not make sense to maximize or minimize pressure rating as they cannot be added together and retain the same meaning. Further, the value must be available for all possible configurations - if some pipe sizes are of unknown weight, the sizing cannot proceed effectively because it is unknown whether choosing that size is an improvement or not.

The Objective is defined on the [Sizing Objective panel](#).

Conceptual Example

Consider a simple problem - moving fluid from one large tank to another. The pressures in the tanks are fixed. The fluid must be transferred in a pipe of specialized material, and it is difficult to manufacture this material in small diameters, making them more expensive. The Objective is to select the cheapest pipe possible for the task.

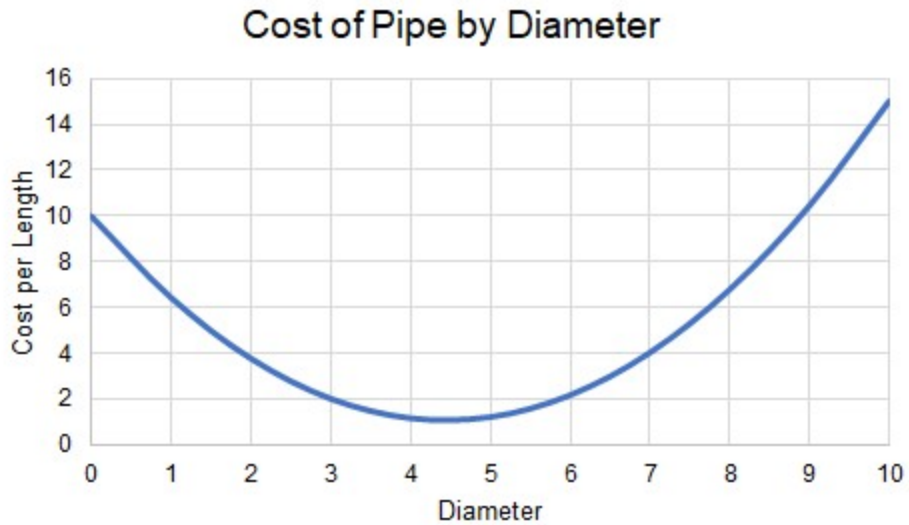


Figure 1: Cost for different diameters of specialized piping

Clearly, there is a minimum cost between diameters "4" and "5."

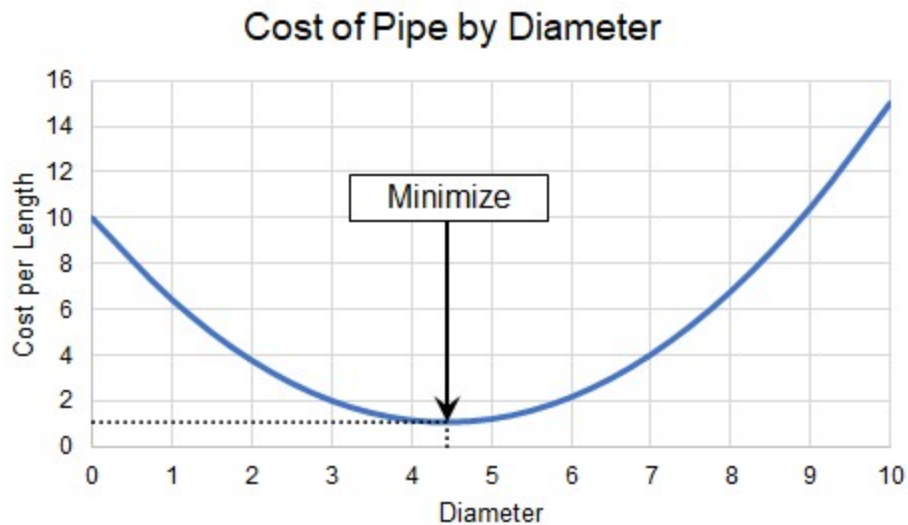


Figure 2: Locating the Diameter with minimum cost

However, what if we are limited by other requirements? For example, we may have a minimum flow rate. This is a [Design Requirement](#), and makes the problem more complex.

Related Topics

Sizing Fundamentals.....	502
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Design Requirement

A real system has **Design Requirements - specific parameters that must be met, regardless of the Objective value**. Targeting an Objective without any other associated requirements is not meaningful. If a system is not attempting to achieve some goal, then the minimum cost is zero as no system is required.

The Objective alone may be "How can I drive my cost to a minimum?" but the question really on the designer's mind is "How can I drive my cost to a minimum while maintaining desired operation?"

Design Requirements are defined on the [Design Requirement panel](#).

Note: There is a distinction between a boundary and a Design Requirement. Boundaries represent a **physically forced** flow or pressure while Design Requirements represent **desired behavior**. See [Boundaries vs. Design Requirements](#) for further discussion.

Conceptual Example

Continuing with [our example](#), we want to minimize the cost of some pipe. However, there is a minimum flow rate. This is a Design Requirement and limits the possible solutions. In our example, flow rate follows a predictable relationship with diameter.

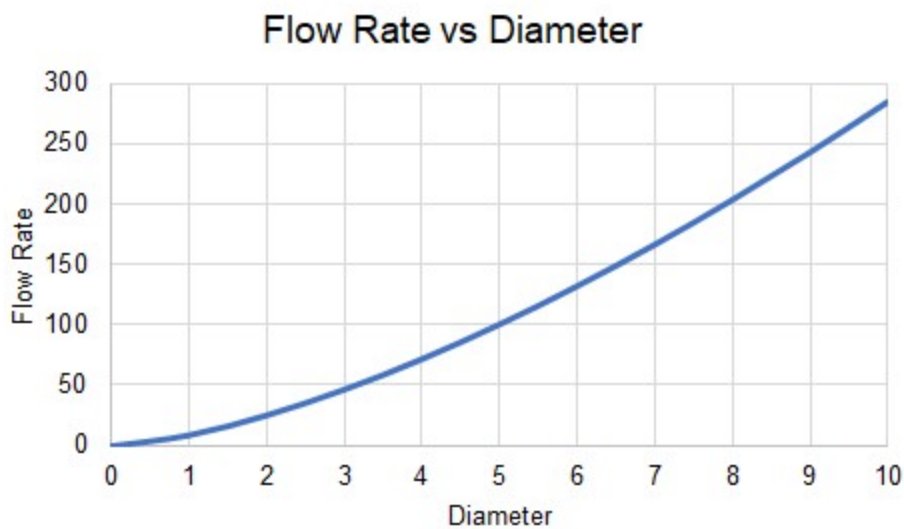


Figure 1: Flow Rates for different diameters, showing the minimum required for our pipe

Our pipe has a Design Requirement that flow rate is at least "150", which indicates a minimum diameter between "6" and "7."

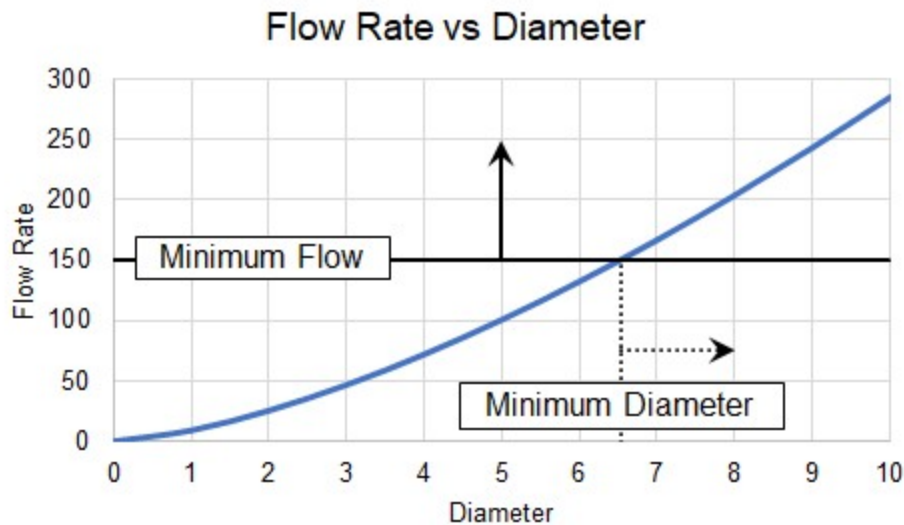


Figure 2: Minimum diameter indirectly imposed by flow rate requirement

Adding Design Requirements may influence the sizing results as this requirement makes certain solutions [Infeasible](#).

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Feasible and Infeasible Designs

If a system has Design Requirements, there will be designs unable to meet the requirements. These designs are termed ***Infeasible*** - configurations that violate Design Requirements. Likewise, configurations that meet all Design Requirements are ***Feasible***.

Feasibility can be determined without any knowledge of the Objective. While it is not directly related to the Objective, feasibility can heavily influence the sizing process. It is typical that the ideal solutions are infeasible because they do not meet the Design Requirements. Instead, the best solution is where the Objective is met in the feasible space. Not all Design Requirements will impact the sizing in the same way.

It is important not to confuse *infeasible* design with *impossible* design. A potential design being *infeasible* does not mean that the system could not be built as specified. In fact, it is possible that the system could run quite well. It *only* means that a specified Design Requirement has been violated.

Conceptual Example

We previously [added a Design Requirement](#) to our pipe, which makes certain designs Infeasible. The minimum flow rate Design Requirement requires a minimum diameter.

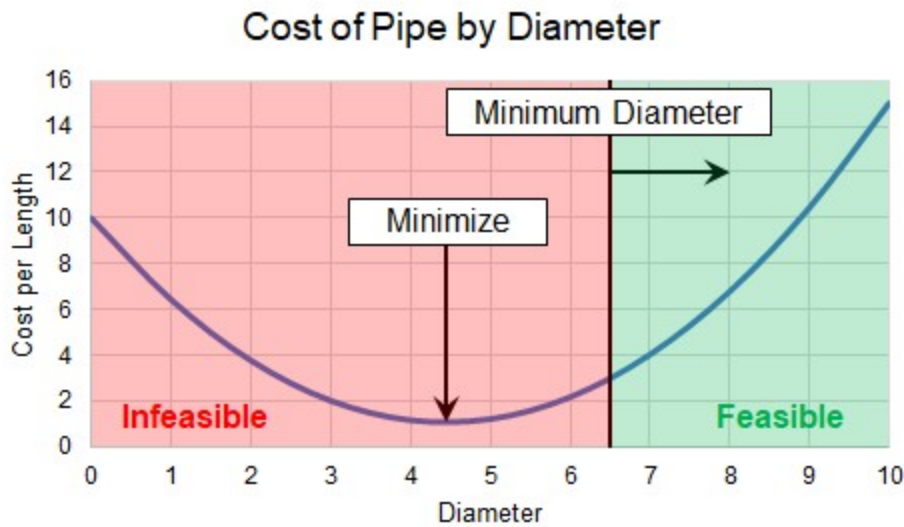


Figure 1: Certain diameters are infeasible when a Design Requirement is added

Clearly, our best case minimum cost is now in the Infeasible region. This would still be the cheapest pipe to select, but would limit flow too much. Instead, we need to find the minimum cost **within the feasible region**.

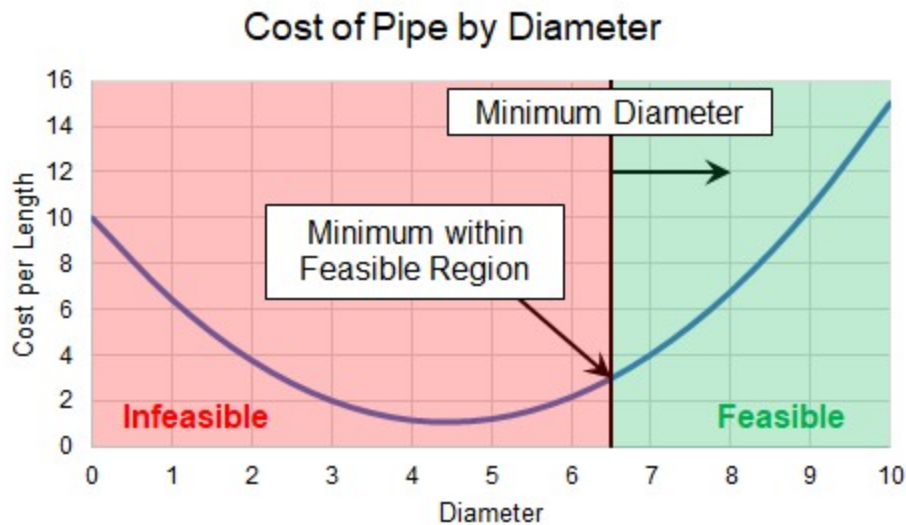


Figure 2: Minimum Feasible Cost

Because our cost only increases with diameters larger than our minimum, the best case cost is now the minimum diameter possible due to the flow requirement. This means that the Design Requirement is [Active](#).

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Active, Inactive, and Violated Design Requirements

Design Requirements may or may not drive the sizing solution. **Design Requirements are Active if they drive the sizing solution. Design Requirements that have no impact on the sizing solution are Inactive.**

There can easily be Design Requirements that, while important for normal operation, do not drive the sizing solution because violating them would not improve the Objective. These Inactive Design Requirements could be removed entirely without changing the sizing solution.

It is not always easy to tell what Design Requirements will be Active or Inactive. A better overall solution can easily often obtained by making a local selection *more* expensive. This means that Design Requirements that would appear to only affect high-cost options may in fact be Active.

Knowing which Design Requirements are Active or Inactive in a sizing analysis can provide valuable insight into a design. Perhaps one Design Requirement is significantly increasing the cost of the system, but is not critical to the operation. In such cases it may be worthwhile to relax the restriction and modify the Design Requirement for a cheaper solution.

A Design Requirement is Violated if it is not met. During the sizing process it is common that Design Requirements will be Violated. However, if a solution exists, they should only be Active or Inactive in the final result.

Conceptual Example

Adding a specific Design Requirement to our pipe [changed the sizing solution](#). What if we had also added a maximum flow rate, effectively imposing a maximum diameter? It will not impact the sizing solution, despite it being an important consideration for our system. Our original Design Requirement is considered Active, because it is driving the sizing solution, while the second Design Requirement is considered Inactive. As before, the absolute minimum cost diameter would Violate our minimum flow requirement, making it infeasible.

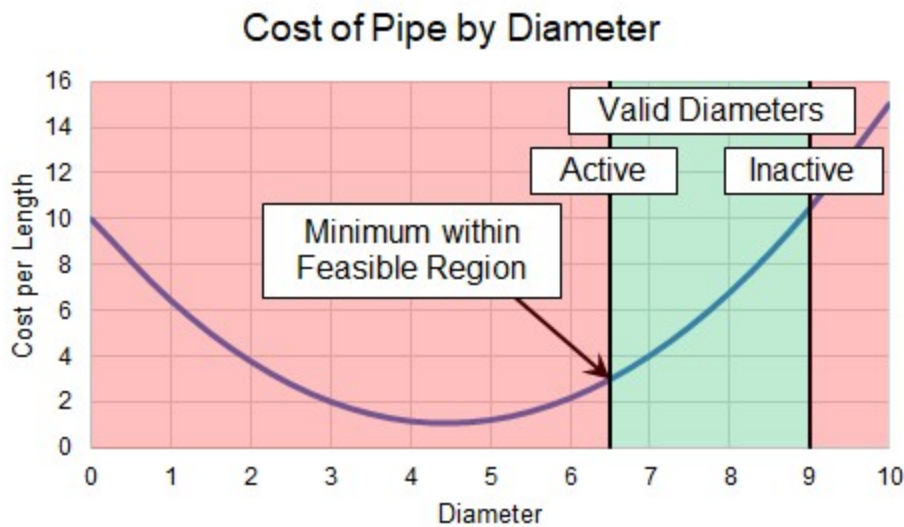


Figure 1: Multiple Design Requirements

Piping is generally not available in any requested size, but instead only in selected, [Discrete](#) diameters, which will also affect the sizing solution.

Related Topics

Sizing Fundamentals..... 502

Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Discrete and Continuous Sizing

Continuous sizing considers any physical diameter in a given range. The ideal solution of a Continuous sizing will contain fractional sizes. However, piping is generally only available in *Discrete* sizes - a selection of individual, unique diameters.

It is assumed during a Continuous sizing that the Objective value varies linearly between intermediate sizes. For monetary cost, this is highly unlikely to be the case - custom pipe of unusual diameter would be very expensive. Continuous sizing is useful because it is generally a far easier problem to mathematically solve. On its own, it will give the engineer a good idea of what the ideal system and best-case Objective value looks like. Coupled with a Discrete sizing method, the Continuous sizing result provides an excellent starting point, making the mathematical solution easier.

Continuous sizing methods also generally require less input as they are less restrictive. Starting the sizing process on a Continuous basis is often a good idea as it can reveal potential problems or areas of improvement earlier in the design process.

The selection between Continuous or Discrete is defined on the [Sizing Method panel](#).

Conceptual Example

[Our example](#) has so far been a Continuous sizing problem - we are able to select any size of pipe to drive the cost to an absolute minimum. However, the piping is only available in Discrete sizes. This fact will impact the sizing solution and the resulting cost.

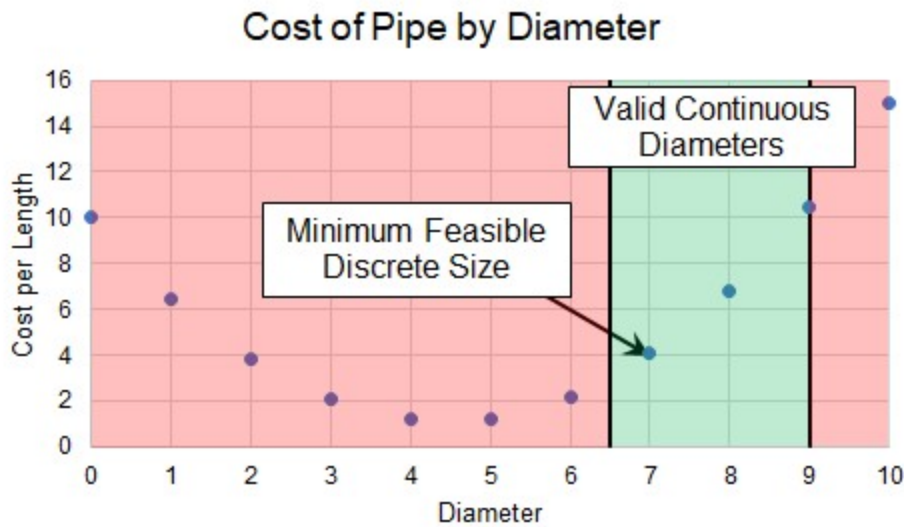


Figure 1: Minimum Feasible Discrete Size

The minimum flow Design Requirement is still considered Active in this case, as it is still driving the sizing solution.

Note: In our example, the cost of diameter "7" is almost double the cost of diameter "6." In such a case, it may warrant further investigation into why the minimum flow rate is required. If the minimum flow was imposed as a rule of thumb or for a non-critical reason, it is well worth considering violating this Design Requirement and selecting size "6" for the cost savings.

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Multiple Operating Cases

Sizing a system has many affects in addition to saving on quantities like weight or monetary cost. The operating capabilities of the system are impacted by any sizing decision. What may work well for one situation may not work well for another situation. Furthermore, the ideal design for a

system that must operate in more than one way may not simply be the ideal design of one of the cases.

In other words, repeatedly sizing the same system for individual multiple operating cases may not result in the ideal design.

By sizing multiple operating cases in parallel, it is possible to account for these variations and still obtain an ideal design in just one sizing simulation. We call these operating cases *Dependent Design Cases*.

This can be enabled on the [Sizing Objective panel](#), with the [Dependent Design Cases panel](#) showing details on the process and model configuration.

Conceptual Example

[Minimizing the size of a pipe](#) with a Design Requirement of a minimum flow is relatively straightforward when only one situation must be considered.

However, consider if our example had multiple operating, or **Dependent Design Cases**. For example, the pressures in the tanks may vary depending on the time of day, the required flow rate may vary with demand, and the environment may vary which could affect the system through thermal affects.

The minimum pipe size found without considering these situations **may violate a Design Requirement** in other cases. Clearly, this cannot be allowed. While some attempt could be made by the engineer to determine the "most severe" case and size for that case, this is often very difficult and can easily neglect potential savings. Instead, by leveraging Dependent Designs, we can account for these variations directly in the model.

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Common Size Group

A **Common Size Group** is a group of pipes that must be of uniform size.

There are two reasons to create Common Size Groups:

- **Physical Requirements** - All of the pipes in the group must be the same size in the field. For example, a long header may be drawn on the Workspace as a series of individual

pipe objects. Adding these individual pipes to a Common Size Group forces the entire header to be a common size. If the pipes were not placed in a Common Size Group, it is likely that different sizes would be selected for different sections of the header. Requirements based on pipe connections are also common - it would not be unusual to require equal diameter piping upstream and downstream of a valve or other junction.

- **Reduction in Complexity** - Every additional independently sized pipe increases the difficulty of the sizing problem. Sizing problems with large numbers of independently sized pipes can take extremely long to solve, often for relatively little benefit. This is especially true when the resulting pipe sizes are similar, or when the cost of a particular pipe has minimal impact on the overall cost. For example, parallel pipes would often be expected to have the same size, and short connector pipes will likely have no significant bearing on overall cost if there are much longer pipes being sized. By adding these types of pipes to a Common Size Group, the solution time can be dramatically reduced.

Common Size Groups are defined in the [Size/Cost Assignments panel](#).

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Initial Costs and Recurring Costs.....	514
Candidate Set.....	516

Initial Costs and Recurring Costs

Systems can be sized for a variety of costs. Most notably, there are **Initial Costs** and **Recurring Costs**. The sum of the Initial Costs and Recurring Costs over the expected system lifetime is known as **Life Cycle Cost**.

Initial Costs include any **one-time** cost that is required to get the system in question operational. Most often, this is a combination of Material and Installation costs.

Recurring Costs include any cost that accrues over time. This could be fixed costs at regular intervals, such as the cost of routine Maintenance. It can also be the potentially variable cost of Energy to run compressors.

Note: Costs must be defined in [Cost Libraries](#), which are available as part of the base application.

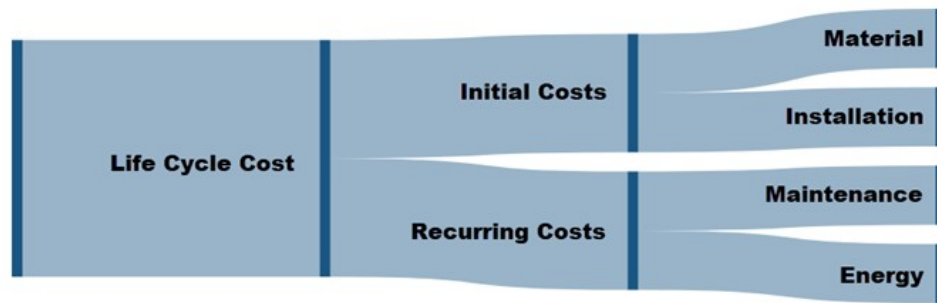


Figure 1: Components of Life Cycle Cost

Keep in mind when considering what costs to include what your most significant cost is likely to be. Systems are often designed to run for many years. This generally means that the total cost of running the system ends up as much more than the cost of building it. Fully defining the sizing input for minimizing Life Cycle Cost is more difficult, but often worthwhile. The inclusion of cost components is defined in the [Sizing Objective panel](#).

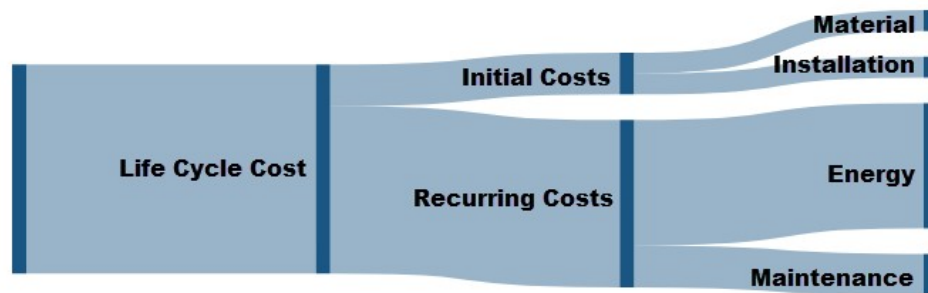


Figure 2: Potential distribution of costs for a long System Life

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Candidate Set.....	516

Candidate Set

A **Candidate Set** is a set of allowed pipe sizes. All pipes that are being sized must be associated with a Candidate Set.

The Candidate Set is required for both [Discrete and Continuous Sizing](#). In continuous sizing it is used to determine a minimum and maximum pipe size. The intermediate sizes are only used if sizing is done on a discrete basis.

Multiple Candidate Sets can exist, but only one can be applied to any given pipe or [Common Size Group](#).

Candidate Sets, and their assignment to pipes, are defined on the [Candidate Sets panel](#).

Related Topics

Sizing Fundamentals.....	502
Objective.....	503
Design Requirement.....	505
Feasible and Infeasible Designs.....	507
Active, Inactive, and Violated Design Requirements.....	509
Discrete and Continuous Sizing.....	511
Multiple Operating Cases.....	512
Common Size Group.....	513
Initial Costs and Recurring Costs.....	514

Sizing Window

The Sizing Window is a sixth [Primary Window](#) that appears when the [ANS module is activated and enabled](#).

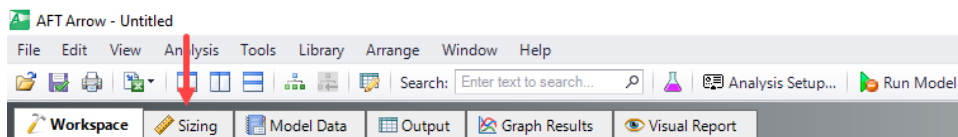


Figure 1: The Sizing Window

The Sizing Window contains all of the information required to define a sizing analysis. Other areas of AFT Arrow interact with the definition of sizing parameters, but ultimately all changes are reflected here. The Sizing Window is further divided into Sizing panels.

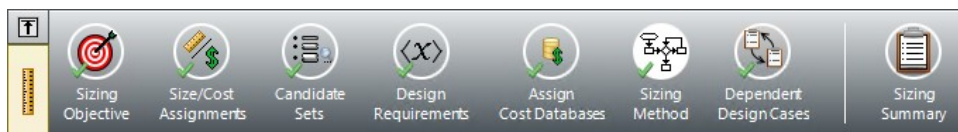


Figure 2: Sizing Navigation

Each Sizing panel serves a certain purpose in the sizing process.

- [Sizing Objective panel](#) - What parameter should be minimized or maximized?
- [Size/Cost Assignments panel](#) - What pipes or junctions should be sized, which should be counted in the cost total, and how should they be grouped together?
- [Candidate Sets panel](#) - What materials, sizes, and types of pipes can be considered in the sizing process?
- [Design Requirements panel](#) - What requirements does the system have?
- [Assign Cost Libraries panel](#) - If optimizing on cost, what is the source of the cost data?
- [Sizing Method panel](#) - What type of sizing should be completed and what method should be used?
- [Dependent Design Cases panel](#) - What operational variations must the sizing analysis account for?
- [Sizing Summary panel](#) - How are the sizing settings currently applied to the system?

Panel Locks

Every Sizing panel can be individually locked to prevent accidental changes.

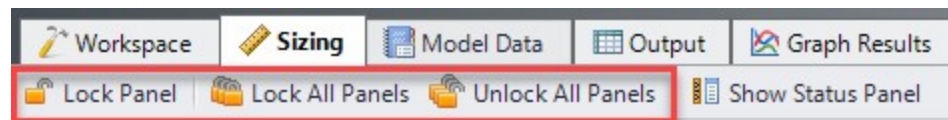


Figure 3: Locking panels

Related Topics

Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Sizing Objective Panel

Sizing Options



- **Perform Sizing** - The best case selection of pipe sizes will be determined based on the various Sizing Window settings.
- **Calculate Cost, Do Not Size** - Pipe sizes will not change from their initial, user-defined size. The costs of the system as-defined will be calculated.
- **Do Not Size** - No sizing or cost calculations will be carried out. This is equivalent to temporarily disabling ANS.

Objective

The [Objective](#) is the goal of the automated sizing process. Any of the listed parameters can be minimized or maximized.

- **Pipe Weight** - Calculated using the pipe material density and dimensions from the internal pipe material library for the pipes being sized.
- **Pipe and Fluid Weight** - Calculated using the pipe material density and dimensions from the internal pipe material library, as well as the fluid density defined in the [Fluid panel](#)
- **Flow Volume** - The total volume of fluid in the pipes calculated from the inner diameter and geometry defined in the [Pipe Properties window](#).
- **Pipe Inner Surface Area** - The internal surface area of the pipes calculated using the inner diameter and geometry defined in the Pipe Properties window.
- **Monetary Cost** - The monetary value of the pipes/junctions calculated from the user provided [cost libraries](#)

Minimizing a physical parameter such as weight, volume, or surface area is often a good approximation of minimizing Initial Cost. These options are also useful in specific cases where monetary cost of the fluid system is not the primary concern.

Other Options

Sizing Lock Options

Lock All Sizing Windows After Run ?

Load Default Set As Default

Multiple Operating Cases

Enable Dependent Design Cases ?

By default, all sizing panels are locked after a simulation is run. Modifying sizing input will require unlocking the panel.

It is possible to model [multiple operating cases](#) simultaneously by enabling Dependent Design Cases. The [Dependent Design Cases panel](#) is hidden unless it is enabled here.

Monetary Cost Options

There are several considerations to be made when considering the total Monetary Cost of a piping network. Is the Initial Cost of the Materials required the only concern? Do Installation Costs have to be accounted for? The true cost of a system could be considered its [Life Cycle Cost](#) - how much does it cost not only to build the system, but to run and maintain it? These are all valid approaches to the challenge of minimizing cost, and each requires different input.

Size for Initial Cost			
Size for Initial Cost, Show Energy Costs			
Size for Initial and Recurring Costs			
User Specified			
Cost Type	Include in Sizing Objective	Cost Report Only	Ignore Cost
Initial Costs			
Material	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Installation	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Recurring Costs			
Energy	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Maintenance	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>

- **Size for Initial Cost** - Material and Installation Costs will be used to size the system.
- **Size for Initial Cost, Show Energy Costs** - Material and Installation Costs will be used to size the system. Energy cost does not impact the sizing, but will be displayed. This may be helpful to determine if it is worthwhile to resize the system considering Recurring Costs.
- **Size for Initial and Recurring Costs** - Energy Cost is considered in the sizing, along with Material and Installation Cost.
- **User Specified** - Custom sizings are possible by selecting the appropriate radio buttons. For example, if Installation Costs are not readily available, these can be ignored by changing that row to Ignore Cost.

Note: The Sizing Level affects the available options. If the overall Sizing Level is "Do Not Size" then no sizing can take place and this will be reflected by forcing all cost rows to Ignore Cost.

- **Warn When Fittings & Losses Costs are Missing** - By default, ANS will warn the user whenever costs are missing for any [Fitting & Loss](#) defined within a sized pipe. For the most accurate solution, these costs should be included, however, they are often small compared to other costs and this option allows ignoring those warnings.

Recurring Cost Options

If any Recurring Costs is specified to be included in the Cost Report or Sizing, additional input is required.

System Life (For Recurring Costs)

System Life: 20 years

Interest Rate: 15 % Per years

Inflation Rate: 0 % Per years

Energy Cost

Use Energy Cost Libraries

Use This Energy Cost Information

Cost: 0.03 U.S. Dollars

Per: kW-hr

Only this fixed energy cost set here will be used. All Energy Cost settings in the Library Manager will be ignored.

Initial Cost Limit

Max Initial Cost: U.S. Dollars

- **System Life (For Recurring Costs)** - The period for which the Recurring Costs are counted. Interest and Inflation Rates can optionally be specified.
- **Energy Cost** - Only enabled if Energy is included in the Cost Report or Sizing.

- **Use Energy Cost Libraries** - Use [Cost Libraries for energy cost](#).
- **Use This Energy Cost Information** - Specify a monetary cost for energy directly. If this is specified, Energy Cost Libraries will not be utilized.
- **Initial Cost Limit** - Limits the total amount allowed for Initial Cost. A higher initial cost may result in a lower Life Cycle cost, but it is sometimes the case that there is an absolute limit on initial spending.

Related Topics

Sizing Window.....	516
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Size/Cost Assignments Panel

The Size/Cost Assignments panel defines which components are to be included or excluded from sizing and cost reporting. Buttons across the top of the panel display selections for a specific component type.

Pipes

The options for pipes include including or excluding them from sizing, including or excluding from cost, including in cost only if size changes, and collecting the pipes into [Common Size Groups](#).

- **Sizing and Cost Options** - This section of the table controls sizing and cost calculation.
 - **Automatically Size** - If either of these columns are selected, this pipe will be included in the sizing analysis.
 - **Always Include in Cost** - The cost of this pipe will always be considered and reported.
 - **Include in Cost if Size Changes** - The cost of this pipe will effectively be considered zero for the initial size. This is useful for representing existing

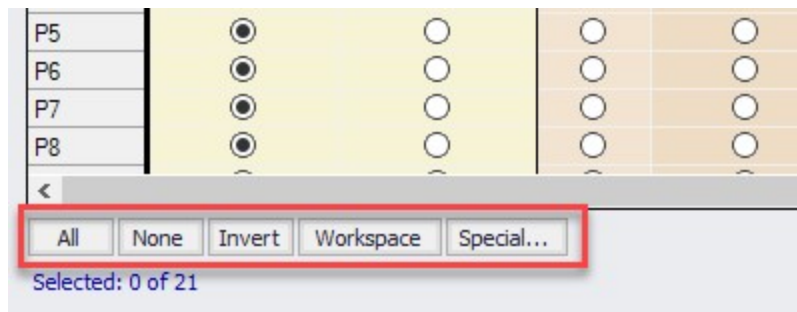
pipes, while considering the cost savings that could result from replacing the existing pipe with one of a more appropriate size.

- **Do Not Size** - If either of these columns are selected, this pipe will not be included in the sizing analysis.
 - **Include in Cost** - The pipe will not be considered for sizing, but the cost of the pipe for the user specified size will be included. This is useful for having a complete picture of the system cost, even if certain pipes must be a given size.
 - **Do Not Include in Cost** - The pipe will not be sized or included in costs.
- **Pipe Grouping** - Pipes can be placed into Common Size Groups here. You can sort this table by clicking on the column headers, or by right-clicking on the headers and choosing Organize Grid by Common Size Group.
 - **Not in Group** - the pipe is not in a group, and is either not sized, or sized independently from any other pipe.
 - **Common Size Groups** - This section is only visible after a Common Size Group is created with the New button. When a Common Size Group is created, it will create a corresponding column underneath the Common Size Groups header. Pipes can then be placed into the Common Size Group by selecting the radio button for the given pipe under the given Common Size Group. Groups can be collected to the top of the table by clicking on their column headers.

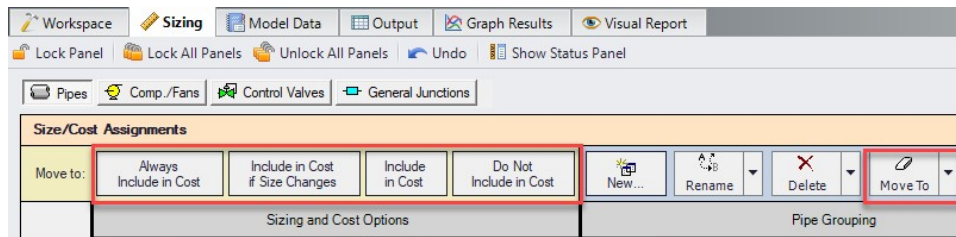
Pipes	Sizing and Cost Options				Pipe Grouping				
	Automatically Size		Do Not Size		Not in Group	Common Size Groups			
	Always Include in Cost	Include in Cost if Size Changes	Include in Cost	Do Not Include in Cost		Main Lines	Main Compressor	HEX A	HEX B
P1	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P2	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P3	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>				
P4	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>				
P5	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>

Note: You can also add Pipes to Common Size Groups [directly from the Workspace](#), and optionally color the Workspace pipes to indicate which pipes are in what groups.

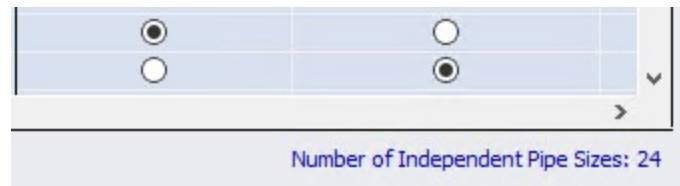
Pipes can be selected by clicking on the row header, and multiple pipes can be selected by holding control or shift. With multiple pipes selected, their sizing or cost options can be reassigned with the buttons near the top of the table. All of the tables displayed in this panel include the [selection features](#) available elsewhere in AFT Arrow.



Selected rows can have their settings changed all at once with the buttons just above the table.



Note the indication of the total number of Independent Pipe Sizes in the lower right. This quantity has important implications on run time and what Search Method will be appropriate for the model



Compressors/Fans, Control Valves, and Junctions

- **Cost Options** - This section of the table controls sizing and cost calculation.
 - **Include in Cost Report and Sizing** - If a junction is included in the sizing calculation, this does not mean that the junction itself is modified as pipes are. Instead, this means that the cost associated with the junction is included in the Objective. This can indirectly affect the size of the junction as the pipes are sized accordingly around it.
 - **Include in Cost Report Only** - The junction cost will not be included in the Objective. However, the cost of the junction will be included in the Cost Report. The sizing of the junction may change as a result of pipe size changes, but the cost impact of this change will not drive the solution.
 - **Do Not Include Cost** - Not included in sizing or costs.

Comp./ Fans	Cost Options		
	Include in Cost Report and Sizing	Include in Cost Report Only	Do Not Include Cost
J7	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
J10	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>

Maximum Cost Groups

Compressors/Fans and control valves can optionally be grouped together in maximum cost groups. This is useful to account for situations where the cost of multiple components is the

same, regardless of how the system operates around them.

For example, it is common to have two identical compressors installed in parallel, with only one operating and the other functioning as a spare. Clearly, the cost of purchasing and installing two identical compressors will be identical regardless of how they end up being operated. The automated sizing calculation will base the cost of the compressor off its required size to maintain the desired flow. However, in the case of a spare there is no flow requested, meaning that the ideal size of compressor is "zero" - no compressor at all! ANS must be told that the spare compressor is the same size as the running compressor. This will force the costs to be the same, accounting for both in the sizing.

In any Maximum Cost Group, the **largest cost** found during sizing will be applied to *all* of the junctions in the group for the given category. There are three categories:

- **Material** - The most common grouping - A set of identical junctions will likely cost the same to procure, and therefore should be placed into a Material Maximum Cost Group.
- **Installation** - Commonly tied to Material - the junctions are likely to cost the same to install.
- **Maintenance** - May or may not be grouped - if all junctions have routine maintenance whether or not they are used, they should be grouped. If maintenance occurs only when the junction is operating for some amount of time, then a spare will have different maintenance costs and they should not be grouped.

Maximum Cost Groups					
Material		Installation		Maintenance	
Not in Group	VendorA	Not in Group	VendorA	Not in Group	VendorA
<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>

Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

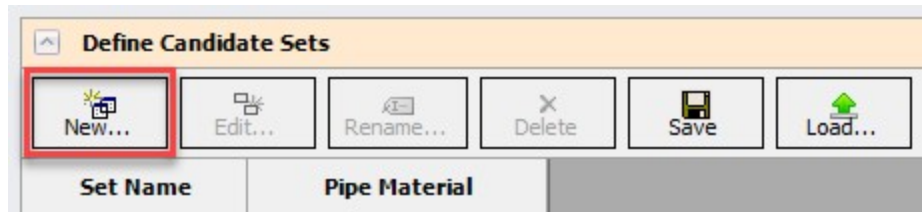
[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Candidate Sets Panel

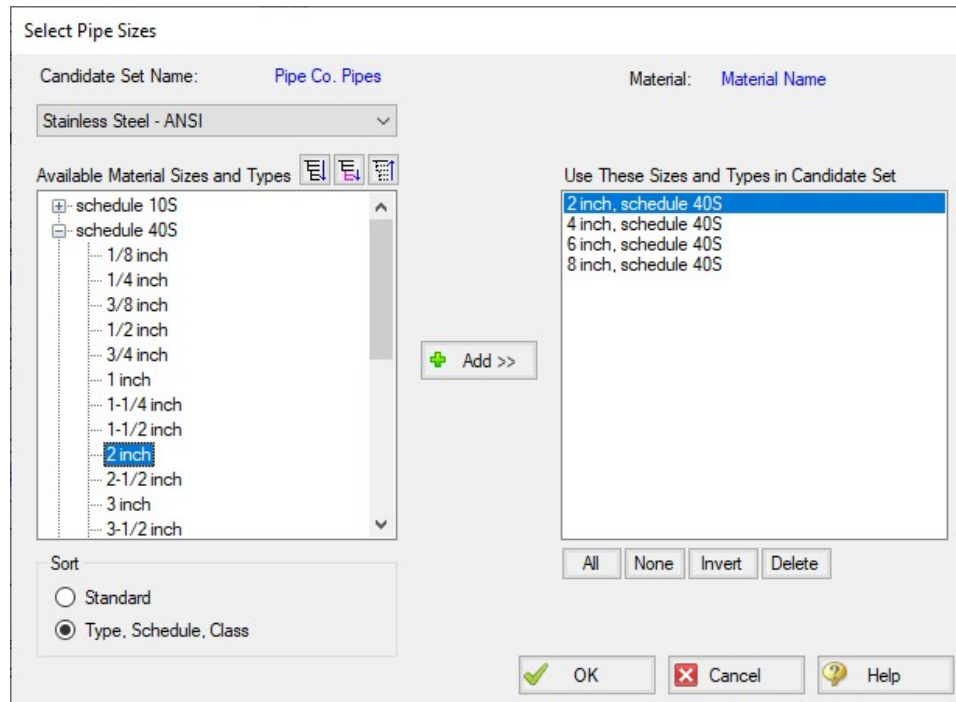
[Candidate Sets](#) are selections of pipe Material, Types, and Sizes that can be considered in the automated sizing process.

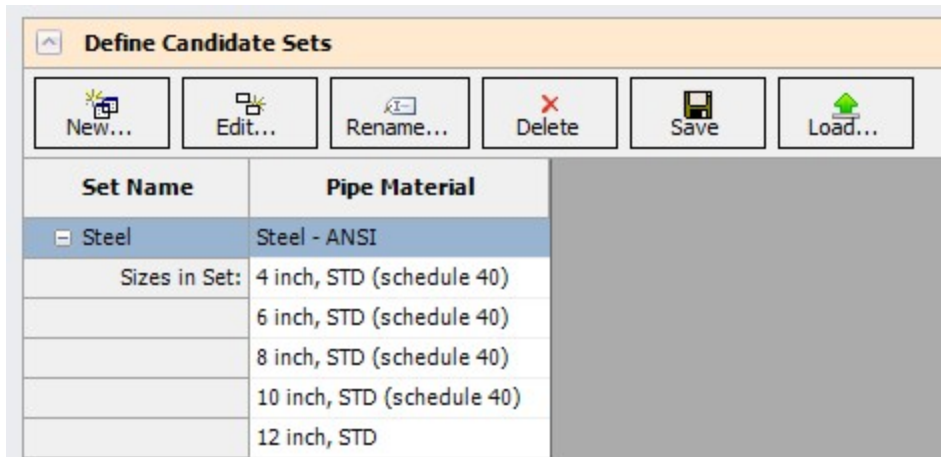
Define Candidate Sets

Before it can be applied to any pipes in the model, a Candidate Set must be defined.



Clicking New will request a name for the new Candidate Set. After entering a name, the Select Pipe Sizes window will appear. In this window, a Material must be selected, and any number of specific Types and Sizes can be added for consideration. It would be common to select commercially available Sizes of a certain Type, but any selection can be made.



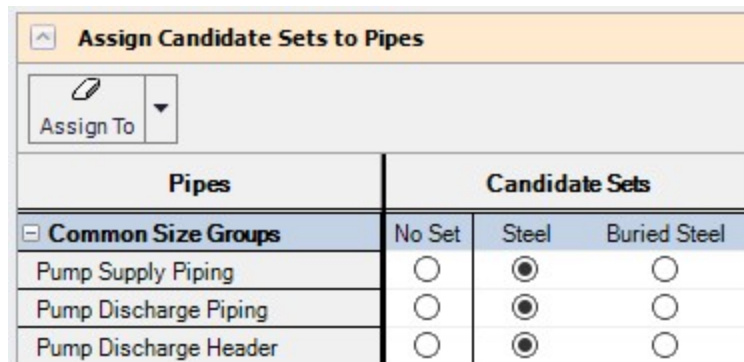


Candidate Sets can be saved to and loaded from a file for distribution or use in other models.

Note: The Candidate Sets defined here are common to all [scenarios](#). They can be applied or not applied in certain scenarios as needed.

Assign Candidate Sets to Pipes

Every pipe that is being sized must have an associated Candidate Set. As Common Size Groups already define every pipe within them as the same Size, Candidate Sets are assigned to the entire Common Size Group. Multiple Common Size Groups or Not in Group pipes can be assigned at once by selecting the rows and using the Assign To button.



Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528

Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Design Requirement Panel

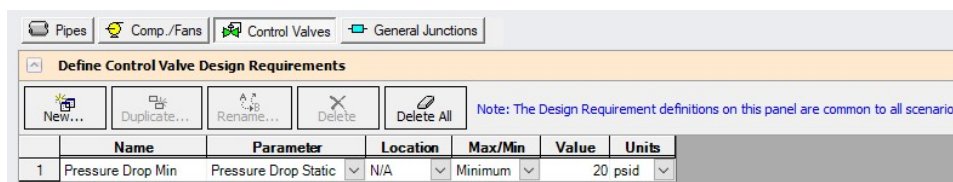
[Design Requirements](#) are of critical importance in a sizing process. Without any Design Requirements, the system will be sized based on cost alone. This will tend to reduce pipe diameters to the minimum size allowed by the [Candidate Set](#). In turn, the fluid behavior of the system may become physically untenable - for example, with extremely high fluid velocities.

Additionally, the very requirements of the system may not be met - certain outflows may require a given pressure and flow and lowering the pipe size too much may mean these needs cannot be met.

Design Requirements ensure that the selected pipe sizes are the minimum cost considering these limitations - the minimum [Feasible](#) cost.

Define Design Requirements

Pipes and Junctions can both have Design Requirements applied to them. The buttons at the top of the panel can be used to switch between Design Requirements for the pipes/junctions. In both cases, individual Design Requirements are defined with a Name, Parameter, Location, and limiting Value.



Note: The Design Requirements defined here are common to all [scenarios](#). They can be applied or not applied in certain scenarios as needed.

Assign Design Requirements

Created Design Requirements can be applied to any number of applicable objects, and a single object can have any number of Design Requirements applied to it. [Common Size Groups](#) are indicated here, and a Design Requirement can be applied to an entire group with ease. If this is not desired, the group can be expanded and individual selections can be made.

Assign Design Requirements to Pipes			
Common Size Groups	Max Velocity (< 150 feet/sec)	Min Velocity (> 50 feet/sec)	Min Pressure (> 18 psia)
+ Compressor Piping	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
+ West Header	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
+ User A	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
- User B	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
P121	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
P122	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
- User C	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
P131	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
P132	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
+ East Header	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
+ User D	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
+ User E	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
+ Not in Group	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Assign Cost Libraries Panel

If Monetary Cost is selected as the [Objective](#), the costs of any components considered in the Objective must be available.

Note: For an accurate analysis, costs are required for **any component that may be considered in sizing**. This means that it is not sufficient to have cost data only for some pipe sizes, or only for certain sized junctions. Every pipe Size and Type considered in a given [Candidate Set](#) needs to have cost information for every parameter considered in the Objective.

Cost and Energy Libraries

Libraries are not discussed in detail here. Both Cost and Energy Libraries are available as a part of the base Application. For a thorough explanation see [Cost Overview](#) and the associated topics. The Creating Libraries for Sizing a System Example is an excellent place to start if you are unfamiliar with Cost Libraries.

Assigning Cost Libraries

Candidate Sets can have any number of Cost Libraries assigned to them. This can be useful as different sources of costs can be distributed among several libraries. For example, different Sizes of pipes may be available only from certain manufacturers, and splitting libraries by material source can provide clarity.

Likewise, "Installation" cost mat have several sub-component costs such as Shipment and Labor. In these cases, There would be an Installation cost for the same Material, Size and Type in two different libraries. Including both of these libraries will use the sum of the two costs in the Objective.

Candidate Sets	Cost Library				
	All	Selected	Pipe Co.	Tube Co.	Shipping and Labor
Steel	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Buried Steel	<input type="radio"/>	<input checked="" type="radio"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Note: Keep in mind when assigning cost libraries that **the automated sizing will consider the sum of costs from all selected cost libraries**. Be careful to include multiple libraries only if every cost would be incurred. Do not include, for example, two cost libraries comparing costs from two different suppliers.

Modifying Cost Calculations

Multipliers on a given cost category for a given component can be applied here. These are useful in a variety of situations. A compressor may be in a difficult to access location, driving up its maintenance cost by some factor. Or, a control valve may only operate a percentage of the time, meaning that less cost is required for maintenance. A pipe that must be buried would have a

higher installation cost. There are many such examples where a specific pipe or junction in the system has a higher or lower cost than the value defined in the library.

Note: Multipliers can also be applied within cost libraries. The multipliers are independent of each other and stack multiplicatively.

If recurring costs are considered, there will be an additional column in this table titled Service Duration. By default, recurring costs are considered for the entire System Life. If certain components should only be considered as incurring cost for part of the System Life, this can be defined here.

Modify Cost Calculations On Compressors/Fans								
Compressors/Fans	Multipliers (percent)				Service Duration (years)			
	Material	Installation	Maintenance	Operation	Full	Partial	Start	End
J2	100	100	100	100	<input checked="" type="radio"/>	<input type="radio"/>	0	20
J234	100	100	70	100	<input type="radio"/>	<input checked="" type="radio"/>	0	10

Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Sizing Method Panel

The Sizing Method panel controls how the automatic sizing will determine the ideal sizes.

Sizing Level

- **Continuous** - Allow the sized pipes to be any diameter
- **Discrete** - Restrict the sized pipes to diameters directly specified in their Candidate Set

[Discrete and Continuous Sizing](#) each have their own advantages. Discrete Sizing will present a set of pipes that are commercially available, as defined by the associated [Candidate Sets](#). Continuous Sizing is generally much easier computationally, and hence faster.

It is good practice to begin with Continuous Sizing and only proceed to Discrete Sizing when the general design of the system has been determined.

Search Method

There are many ways to approach the search for a minimum or maximum.

A foolproof way to determine if a system is really the "best" is to compare every single possible combination allowed. Then, it will be guaranteed that the lowest cost option has been found. Unfortunately, attempting to [brute force](#) the problem is only viable with a very small number of variables. Instead, the solution must be searched for in an intelligent manner, without explicitly checking every possible option.

There are several such methods available in ANS:

- **Continuous Methods** - These methods use [gradients](#) - rates of change of the Objective (total system cost) with respect to one design variable (a pipe diameter). For the most basic system, the gradient will show an increasing cost with increasing pipe size. However, this quickly becomes less rigid when considering Design Requirements and the impacts a certain pipe size has on the system. A smaller pipe size may make the pipe itself cost less, but it may require a larger compressor which costs more, effectively making the gradient show a *decreasing* cost with increasing size. Regardless, if the direction of decreasing cost is known, an intelligent choice to search in that direction can be made. **By intelligently running sizing sub-problems, it is possible to find a discrete solution with continuous methods.** The [Branch and Bound](#) method searches for a discrete minimum by placing additional requirements on certain variables (branching) and analyzing the results of the continuous solution to determine if this section of the design space is worth further investigation (bounding).
 - [Modified Method of Feasible Directions](#) - Reliable and applicable to most problems.
 - [Sequential Quadratic Programming](#) - Frequently the most efficient, but more sensitive to system definition.
 - [Sequential Unconstrained](#) - Sensitive to initial sizes and control parameters, but more capable of handling large numbers of uniquely sized pipes.
- **Discrete Methods** - Some methods are inherently discrete, and can only consider discrete possibilities during the sizing process. These methods are sometimes more adept at finding global minimums, but are also generally much slower than gradient based methods.
 - [Genetic Algorithm](#) - Cross-breeds a random "population" of designs, mimicking biological evolution. Less likely to become trapped in a local minimum, but very slow compared to other methods.

Note: Full understanding of the Search Methods is not required for effective use of ANS. However, it is important to realize that different methods are better suited to different systems. It is recommended to use multiple Search Methods on any system to ensure the best case sizes are located, and to determine if one method is more efficient than the others.

Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Dependent Design Cases Panel.....	532
Sizing Summary Panel.....	534

Related Blogs

[What is the ANS Module Asking For? Introduction to the Sizing window in the ANS Add-on Module](#)

Dependent Design Cases Panel

Dependent Designs are a powerful way of accounting for different [operating cases](#) during the sizing process. This panel displays information on how to create a Dependent Design Case, or if one exists, how the objects are related to their base objects.

Enabling Dependent Design Cases

Dependent Design Cases are enabled by checking the box on the [Sizing Objective panel](#).



Figure 1: The Enable Dependent Design Cases option selected

Note: Once a Dependent Design Case exists, this option cannot be turned off unless the Dependent Design Case is completely removed.

Creating a Dependent Design Case

Once the option above has been enabled, it is possible to create Dependent Design Cases on the [Workspace](#).

This is done simply with [Duplicate Special](#). Select all of the objects to be included in the Dependent Design and select Duplicate Special from the Edit menu, and selecting the special **Make Dependent Design Case** option.

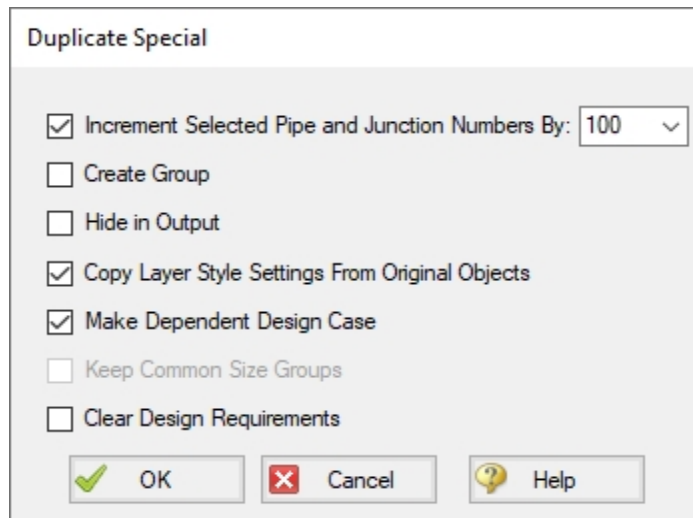


Figure 2: The Duplicate Special window can be used to create Dependent Design Cases

Selecting OK will duplicate all of the objects selected and increment their numbers by the indicated value. In addition, the new objects will be related to their base objects.

Modifying the Dependent Design

Dependent Design objects represent the *same* object that they were created from. Logically, the end result must be that the objects are the same in all dependent designs. It does not make logical sense for pipes to be different lengths, for example. ANS will force a single pipe diameters across all dependent designs.

However, there are many modifications that are reasonable, and they fall into two categories:

- **Modifying the physical configuration of the model** - Environmental changes can affect temperature, changing processes can change the fluid, tank pressure levels can change, compressors may be brought online, valves may change position, and even new pipes may be added to represent a planned expansion. These are just a few examples of a different operating case being the result of a physically changed model.
- **Modifying the [Design Requirements](#)** - The exact same physical system may still have different requirements under different conditions. A plant may have different demands during day or night, or it may be desired to account for future production without modifying the piping layout. These are a couple of examples of changing Design Requirements to create different operating cases.

Note: The only sizing input that can be changed for a Dependent Design are the Design Requirements. Remember that both the base object and Dependent Design object really represent just one physical object - it cannot have different cost or be of a different pipe material in different cases.

Reviewing Dependent Designs

As the model becomes larger and more complex, it may be difficult to tell what objects correspond to others across operating cases. The Dependent Design Cases panel shows a review of how the objects are connected:

Dependent Design Cases		
<small>The grid below shows the relations of Dependent Design Case (DDC) objects to their base objects, as well as their design requirements. Note that Sizing Option, Common Size Group, Maximum Cost Group, Candidate Set, and Cost Libraries/Modifiers will always be synchronized between base objects and their Dependent Design Case objects. This grid is informational only. To change parameters, go to the necessary Sizing Panel.</small>		
Pipes	DDC Pipe?	Assigned Design Requirements
P1	NO	None
P301	YES (P1)	None
P2	NO	None
P302	YES (P2)	None
P10	NO	None
P310	YES (P10)	None

Figure 3: The Dependent Design Cases panel

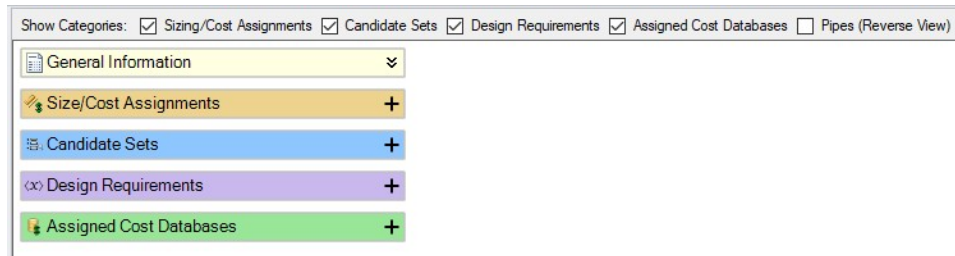
Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Sizing Summary Panel.....	534

Sizing Summary Panel

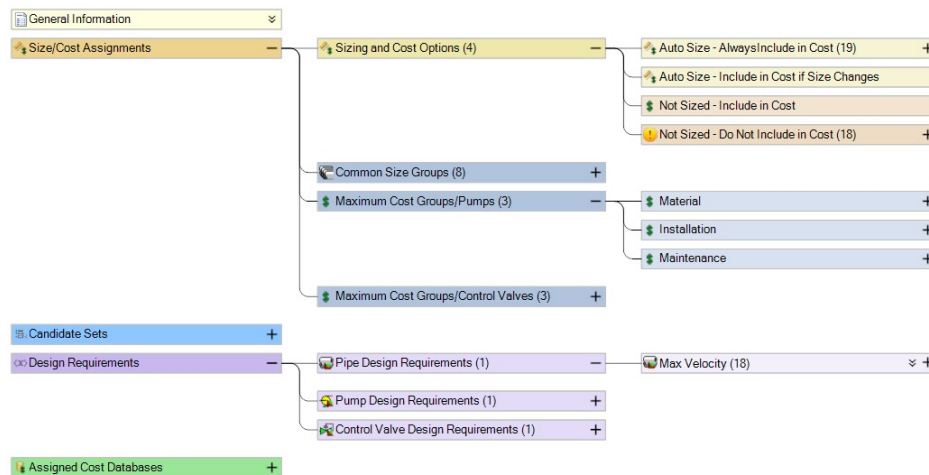
The Sizing Summary panel gives an at-a-glance, but comprehensive, view of how the model has been configured for sizing.

All of the sizing input can be grouped into a few relatively simple categories. However, as the model becomes more complex, the relationships between these inputs can be difficult to visualize. For this reason, the Sizing Summary is a relational diagram with a handful of parent nodes. Nodes can be turned on or off with the checkboxes above the diagram.

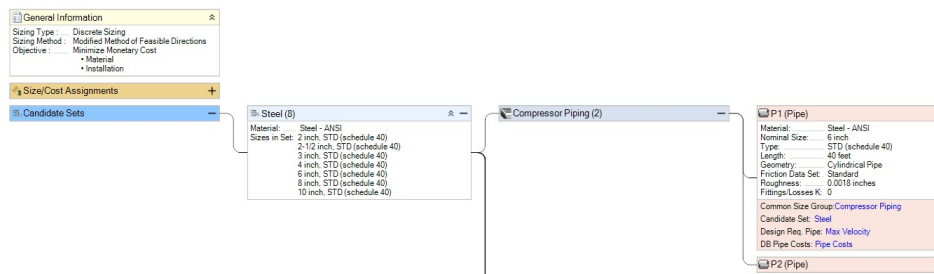


Each node can be expanded to see the related entries.

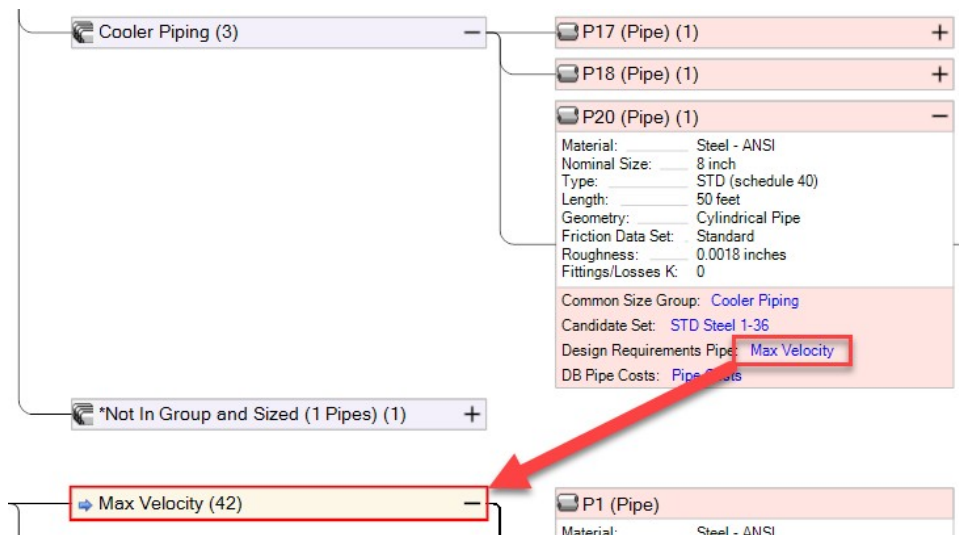
Note: The Summary can be manipulated much like the Workspace - use the scrollwheel to zoom in and out, and pan the view with the scrollbars or by clicking the middle mouse button.



Individual objects can be expanded, showing detailed related information about them.



Clicking on a related category for a specific object will indicate the associated node(s).



Related Topics

Sizing Window.....	516
Sizing Objective Panel.....	518
Size/Cost Assignments Panel.....	521
Candidate Sets Panel.....	525
Design Requirement Panel.....	527
Assign Cost Libraries Panel.....	528
Sizing Method Panel.....	530
Dependent Design Cases Panel.....	532

Solution Progress with ANS

When a model is run with the ANS module enabled, an additional line will be shown in the [Solution Progress window](#) to indicate the progress that the ANS module has made towards reaching the specified goal. There are three fields displayed, as can be seen in Figure 1 below.

The "Best Feasible Cost" and "Current Cost" fields indicate the total cost for the model in the default units, as indicated in the Solution Progress window. The Current Cost field shows the cost calculated for the last design iteration, while the Best Feasible Cost field tracks the cost of the best solution so far that meets all design requirements.

The field on the far right of the Sizing Calls to Solver line tracks the total iterations performed by ANS. For every ANS iteration, ANS makes a call to the hydraulic solver in Arrow to determine the hydraulic solution for the current sizing solution. Thus, this value will always be smaller than the flow and head iterations.

Other Actions

If the solution is paused by clicking the Pause button in the Solution Progress window, there are several options that can be selected from the Other Actions window to assist with convergence.

Selecting [Solution Control](#) allows the Steady Solution Control options to be adjusted before calculations are resumed. This is not recommended unless you have a good understanding of the parameters in Solution Control, or this has been suggested by AFT Technical Support.

The Sizing Control option provides a file with detailed information on the sizing parameters being used by the Sizing Engine. This can provide insight for cases where ANS is not converging, or is taking a long time to converge.

The Accept Sizing option will be available from the Other Actions window if the Genetic Algorithm is selected on the [Sizing Method panel](#). This option allows the user to stop the iterations and accept the design with the Best Feasible Cost as the solution for the model. Accepting the sizing solution early is helpful for models which have been running for a long time, without seeing any significant improvement on the Best Feasible Cost for the model. Rather than spending additional time on find the absolute best design, it may be desired to accept the design that gives the current Best Feasible Cost instead to save time.

Solution Progress - Sizing System...

Maximum Iterations: 1000000 Run Time: 1.90

Solution Method: Length March with Mach # Limit - Length Increment = 2, Mach Limit = 0.01

Relaxation: Automatic

Sizing Method: Sequential Quadratic Programming

Sizing Goal: Discrete - Objective Costs Include: Material, Installation, Operation/Energy

	Absolute Tolerance Max Out of Tol.	Relative Tolerance Max Out of Tol.	Total Iterations
Pressure: 1.0E-04 Relative Change Not used (Absolute Change psia)	0.000E+00	0.000E+00	0
Mass Flow Rate: 1.0E-04 Relative Change Not used (Absolute Change lbm/sec)	0.000E+00	0.000E+00	177
Enthalpy: 1.0E-04 Relative Change Not used (Absolute Change Btu/lbm)	2.428E-01	1.200E-03	176
Concentration: 1.0E-04 Relative Change Not Used (Absolute Change)			

Sizing Calls To Solver: (Objective Cost Units - U.S. Dollars)	Best Feasible Cost	Current Cost	
	165,869	166,092	54

Determining connectivity...
 Checking sizing setup...
 Initializing model...
 Checking if sonic choking exists based on initial guesses...
 Running Solver...
 Initial design is feasible. Cost of initial design is 178,815 (U.S. Dollars) ...
 Performing continuous sizing...

Other Actions ▾ Pause Cancel Graph Results... Visual Report... Output...

Figure 1: The Solution Progress window has an additional line to show information on the ANS Calls to Solver when the Arrow model is run with ANS enabled

Sizing Results

Automatically Selected Pipe Sizes

The pipe sizes corresponding to the best case Objective value are reported in the Output Window.

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
3	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
4	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
5	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026

Cost Report

The Cost Report is displayed in the top section of the Output window. It will display the appropriate variable for the Objective (e.g. kilograms for a Weight sizing). All Cost Libraries that were used in the generation of the report are listed in Library Sources. It is described in detail in [Cost Report](#).

Design Requirement Status

Whether or not a [Design Requirement is Active](#) can have important bearing on system design decisions.

Values in **yellow** are active. Values highlighted in **blue** are Design Requirements that were active during Continuous sizing, and may appear inactive after Discrete sizing. Un-highlighted values are inactive.

Pipe	Velocity Overall Maximum < 150 feet/sec [Max Velocity]
1	107.22
2	85.30
3	84.83
4	89.46
5	45.32
6	128.88

Jct	Pressure Drop Static Minimum > 2 psid [Min Pressure Drop]
10	3.257
11	2.289
13	2.472
14	2.204

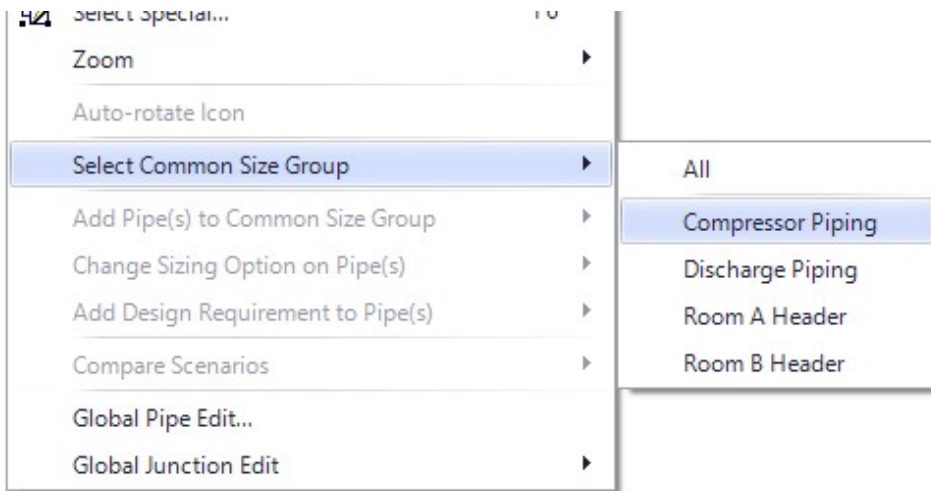
Changing Input Sizes to Resulting Pipe Sizes

- **Set Pipes to Resulting Pipe Sizes** - The [Transfer Results to Initial Guesses](#) takes on additional functionality when ANS is enabled. After a sizing run, you can use this feature to transfer the resulting pipe sizes to the pipes. For certain Search Methods, this can greatly speed up the solution process for future sizing runs.
- **Generating a Disconnected Scenario** - Generating a [disconnected scenario](#) allows the sizing results to be saved as input in a new scenario. This can be useful to keep track of results from different methods, or to tweak the sized solution if required.

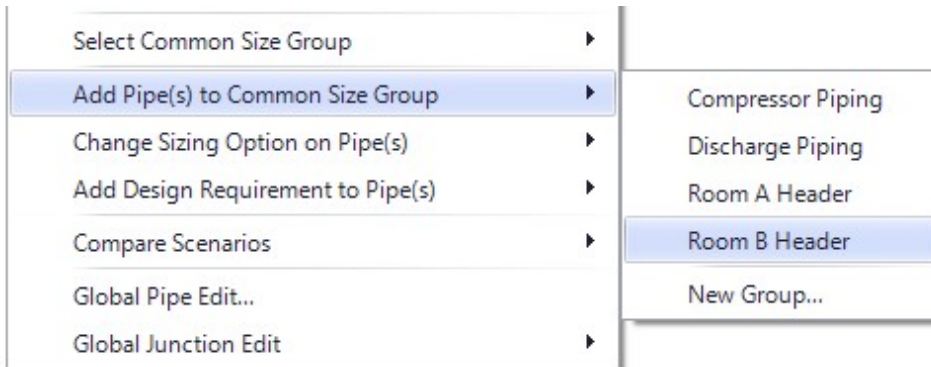
Workspace Integration

Modifying Sizing from the Workspace

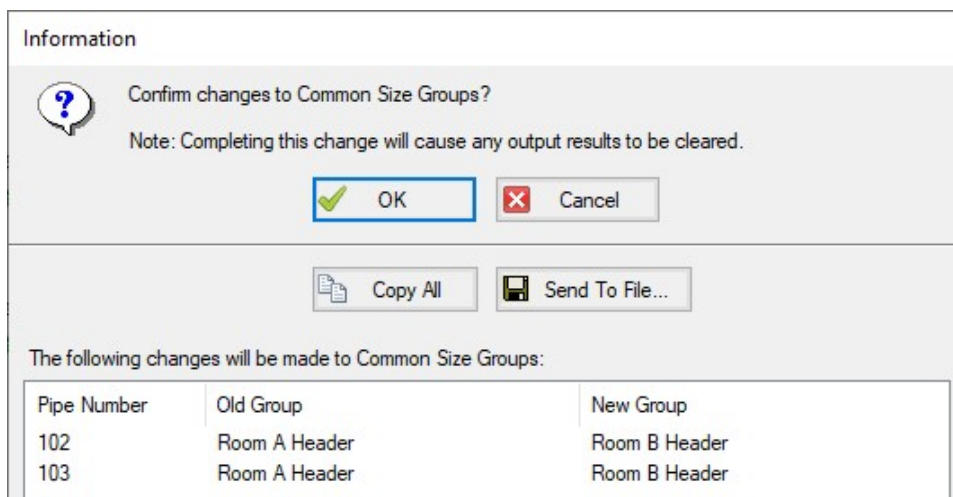
Right-clicking anywhere on the Workspace allows the selection of existing Common Size Groups:



Selected pipes can be moved to a new or existing Common Size Group:



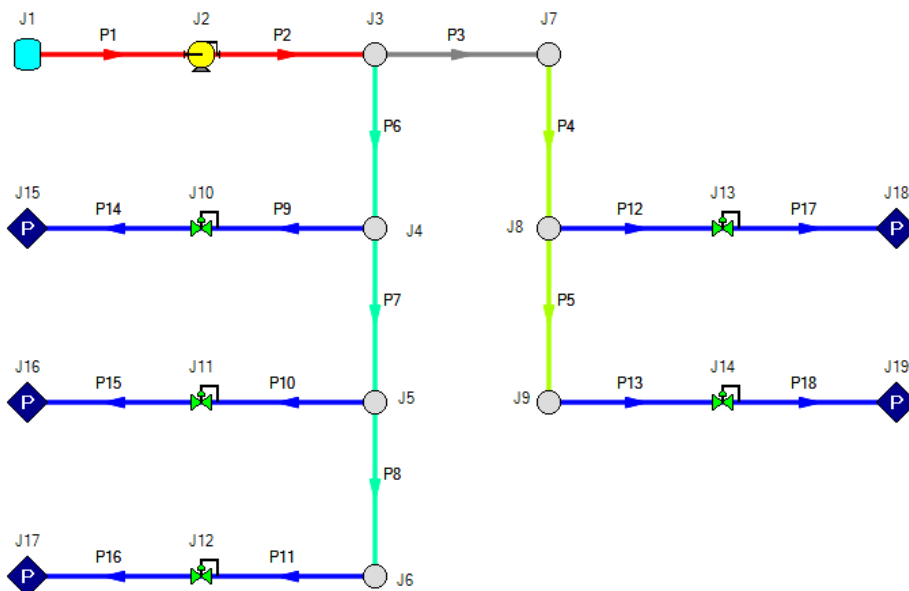
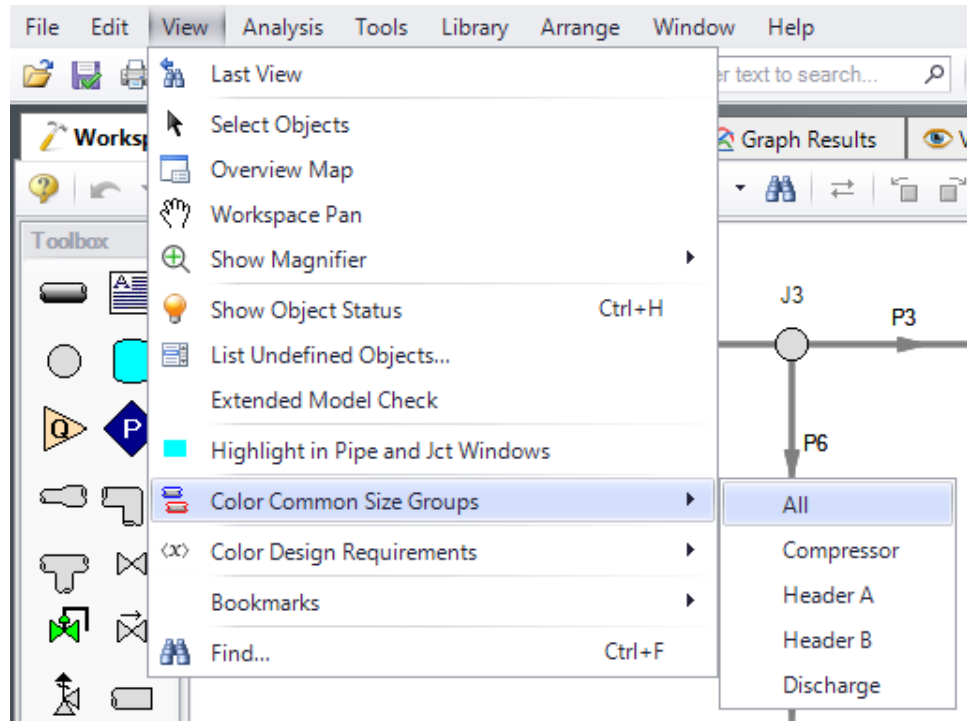
Changes will be previewed before they are applied:



Sizing Level [Assignments](#) and [Design Requirements](#) can also be applied in this manner.

Viewing Assigned Common Size Groups and Design Requirements

Additionally, each Common Size Group or Design Requirement can have a unique color applied to it by selecting the appropriate option under the View menu.



Junction Loss Factors

During the sizing process, the size of junctions will implicitly change along with the size of the neighboring pipes.

Junctions which use handbook loss factors will automatically adjust their loss value when the pipe size changes. For example, a Bend junction defined as a Smooth Bend uses built-in relationships to determine a loss based on pipe area and r/D . As the pipe size changes, this loss value will change automatically.

However, a built-in loss model that updates for different pipe sizes is not always available. Many junctions allow the specification of a Loss Multiplier Table as seen below.

	Diameter	Loss Mult.	K
1			
2			
3			
4			

Diameter Units: inches

Get Nominal Diameter...

Calculate Multipliers

Edit Table

Optional Data

Loss Multipliers

Figure 1: Loss Multiplier Table

With a base loss factor (K-Factor, in Figure 1), ANS can use this table to determine how much to adjust the loss by when changing the associated pipe diameters.

Improving and Troubleshooting the Sizing Process

Check the Continuous Solution First

For small problems with less than ten independently sized pipes, it should be possible to obtain a Discrete solution in a few minutes or less. Larger problems can take a long time to solve, so it is important to point the sizing process in a good starting direction.

The Discrete problem is generally much more difficult to solve than the Continuous, and most of the Discrete methods depend on an initial Continuous solution. Therefore, it is important to ensure the Continuous solution is good. There are a few guidelines to consider:

- **Check the quality of the Continuous Solution before proceeding to the Discrete Solution** - If the Continuous solution is not ideal, then the methods which use the Continuous solution for a starting point will have a poor Discrete solution as well.
- **Ensure the Discrete Solution is worse than the Continuous Solution** - A Discrete solution always places further limits on the solution, and should therefore have a negative

impact on the Objective. If the Discrete solution is better than the Continuous, this means that either the Continuous solution is poor, or that there are numerical issues affecting the solution by some small amount. If the difference is significant, this likely indicates an issue with the sizing process, the solver, or user input.

- **Check for Consistent Continuous Solutions** - Using any of the Continuous methods should give comparable results. If this is not the case, it may indicate that ANS is finding Local Minimums with one or more of the methods.
- **Use the best Continuous Solution to Proceed** - If one of the methods is finding a better Continuous Solution than the others, and the results are reasonable, this result should be used as the basis for the Discrete Solution.

Determine if the solution is a Local Minimum

It is possible for ANS to get stuck in a local minimum and not find the global minimum. If this is happening, there are a few things to try:

- **Change the initial pipe sizes**
- **Use a different Sizing Method** - Pure Discrete Methods are inherently better suited to finding global minimums.
- **Temporarily remove some of the Design Requirements** - This can help identify Design Requirements that are actually driving the sizing process. Having too many Design Requirements can make the system much more difficult to solve with minimal gain.

Use a Different Search Method

The different search methods have different strengths and weaknesses. It is possible that one method will struggle with a problem that others can easily find a solution for.

It is also a good indicator that the best solution has been found if different methods agree.

Use Logical Design Requirements

- **Design Requirements in Closed Areas of a Model** - ANS will attempt to identify Design Requirements that are irrelevant and ignore them during the sizing process. However, depending on ANS to do this is not advised, Design Requirements on closed areas should be removed. For example, it is not meaningful to apply a minimum flow rate requirement on a closed pipe - it is possible that ANS will not find a [Feasible solution](#) if such a Design Requirement is left in place.
- **Conflicting Design Requirements** - It is easy to apply requirements that are impossible to meet. For example, a minimum flow rate greater than an in-line Flow Control Valve set-point, a maximum pressure lower than the connected Assigned Pressure, or a maximum speed on a compressor that has been manually set to a higher speed. With these types of

Design Requirements, ANS will (correctly) conclude that no feasible design exists, and the results may not be meaningful.

Change the Initial Pipe Sizes

The sizing process uses the initial pipe sizes defined by the user as a starting point. It is possible to start the sizing process in an area that poses difficulty. Increasing or decreasing the initial pipe sizes can help.

Simplify the Problem

It is good practice to solve a simpler problem first, before proceeding to a more complex problem.

- **Be sure the model solves as expected before using ANS** - Any issues that are present in the base hydraulic model will also be present in the sizing process.
- **Find a Continuous Solution before searching for a Discrete solution.**
- **Use a small number of Design Requirements**, and add them as necessary. For troubleshooting purposes, it can be useful to remove Design Requirements that are necessary in the final design.
- **Decrease the number of variables** - Adding more pipes to Common Size Groups, and limiting the number of Common Size Groups, can make the problem much easier to solve.

Sizing Components Other Than Pipes

Sizing Compressors/Fans

While compressors are not directly sized by ANS, they are able to be indirectly sized by including an estimation of their cost. Their actual behavior is a result of the system. The appropriate sizing of a compressor requires a Cost Library and understanding of standard [Compressor/Fan Sizing](#).

At the most basic, compressor sizing is simply selecting a compressor that is capable of meeting pressure and flow requirements. Too often this involves selecting a compressor that is larger than required, under the assumption that a bigger compressor will provide for operational margin. Unfortunately, over-sizing a compressor has negative impacts on reliability and efficiency, both of which incur higher cost. Over-sizing could also require the installation of ultimately unnecessary valves or restrictions to bring the flow back down to where it should be.

The ability to appropriately size compressors is built into AFT Arrow. By using the [Compressor Sizing](#) option, AFT Arrow will indicate the amount of pressure required to deliver the specified flow. It is up to the user to find a compressor that operates at its BEP at the calculated flow and head.

The best size of compressor to use when the pipe sizes are allowed to change as well is harder to handle, and requires iteration. See [Sizing Pipes and Compressors Concurrently](#).

Note: Certain compressor configurations require special attention in ANS. See [Special Concerns For Sizing Compressors](#).

Sizing Control Valves

Control Valve sizing is similar - AFT Arrow already has all of the tools needed to size a Control Valve for a specific set of pipes. AFT Arrow will report the Cv or K Factor necessary for a Control Valve to attain a given setpoint. From this value, an appropriate valve could be selected for minimizing cost. It would be typical to select the smallest Control Valve possible while still remaining in a good control range - typically 20-80% open.

Like compressors, selecting the best Control Valve while pipe sizes are changing is more complex. See [Sizing Pipes and Control Valves Concurrently](#).

Sizing Other Junctions

Any other junction can be included in the sizing process as long as cost data exists.

For example, a Bend can have costs associated with it via a [Table of Costs](#). From this table, ANS can determine the cost for a given diameter. ANS will assume that the Bend is of a diameter equal to the upstream pipe inside diameter, and include the corresponding cost in the Objective. If certain components are very expensive compared to piping, this could affect the selection of pipe sizes.

Sizing Ducts

Duct sizing has special considerations discussed in [Sizing Duct Systems](#).

Related Topics

Special Concerns For Sizing Compressors.....	545
Sizing Pipes And Compressors Concurrently.....	547
Sizing Pipes and Control Valves Concurrently.....	551
Sizing Pipes, Compressors, and Control Valves Concurrently.....	555
Sizing Duct Systems.....	556

Special Concerns For Sizing Compressors

There are several special concerns with sizing compressors. For a general overview of the process of sizing compressors see: [Sizing Pipes and Compressors Concurrently](#).

End of Curve Flow Rate

Centrifugal compressors typically have an end of curve, or runout, flow rate. They cannot operate reliably past this point. This can be entered in [Compressor Configuration](#), but this will not actually prevent the model from running the compressor with higher flows. Instead, the curve will be extrapolated past this point and a warning will be generated.

If it is important to prevent the sizing results from considering flows in this region as valid. A [Design Requirement](#) for maximum flow rate equal to the end of curve value should be placed on the compressor.

Partial Service Duration

Without other input, ANS will assume that any compressor included in the sizing operates constantly.

If this is not the case, then either an appropriate Service Duration or Multiplier should be defined in the [Assign Cost Libraries panel](#).

Variable Speed Compressors with Partial Control Applicability

ANS can size compressors with a user defined fixed speed other than 100%, or a controlled compressor that modifies speed to meet a flow or pressure setpoint. However, pressure-type variable speed controlled compressors allow the user to optionally enforce Control only when the actual pressure is above/below a setpoint. This causes an issue with sizing.

If ANS allowed the compressor to change from a fixed 100% speed compressor (non-controlling pressure) to a variable speed compressor (when the setpoint is violated), it would be changing the characteristics of the system. This has a direct impact on the sizing process, so it cannot be allowed - ANS will *always* model this type of compressor as fixed 100% speed.

Either an Always Controlling compressor must be specified, or Design Requirements on pressure must be added to the appropriate pipes.

Unrealistic Sizing Results From Dependent Design Cases

If there are two design cases that have different compressor flow rate requirements, the ideal compressor curve may be unrealistic.

For example, one operating case may require 500 ft³/min, and another may require 750 ft³/min. It is desired to select the ideal compressor. After running ANS, it is determined that the compressor should generate 50 psid in the 500 ft³/min case and 70 psid in the 750 ft³/min case. To meet this requirement, we need a compressor curve that increases from 50 psid to 70 psid with a large increase in flow. This is an unrealistic demand.

To prevent this, a pressure rise Design Requirement can be added to maintain the high flow pressure at 70-90% of the low flow pressure. This will effectively force the sizing to consider only realistically shaped compressor curves.

Related Topics

Sizing Components Other Than Pipes.....	544
Sizing Pipes And Compressors Concurrently.....	547
Sizing Pipes and Control Valves Concurrently.....	551
Sizing Pipes, Compressors, and Control Valves Concurrently.....	555
Sizing Duct Systems.....	556

Sizing Pipes And Compressors Concurrently

The cost of a compressed gas piping system clearly depends on both the cost of the pipes, and the compressor. It is tempting to select compressors and pipes in stages, but this is unlikely to produce the lowest cost solution. Selecting a smaller compressor may not appear to be viable, but changing the pipe sizes may make it viable. Like sizing pipes alone, overall costs can often be decreased by increasing the cost of one or a few components. Attempting to find the lowest cost option manually quickly becomes impractical even with a small number of variables.

Optimal Pumping System Operating Point (OPSOP)

A compressor's BEP is considered the ideal place to run a compressor as the efficiency is highest, and thus cost is lowest. Similarly, any piping system will have a lowest cost design.

Whether a compressor is running at BEP, however, depends on the piping system, and the lowest cost piping system depends on the compressors it contains. What may be the lowest cost piping system for a given compressor, may not be the lowest possible cost option. It is possible that a cheaper compressor could be selected with the correct piping adjustments while still maintaining all Design Requirements.

The point at which no adjustments to compressor or system will result in lower cost is known as the **Optimal Piping System Operating Point (OPSOP)**. One way to determine this point is to exhaustively size the system for different potential sizes of compressors. This is inefficient, but can be used to construct a visual representation of the OPSOP as shown below.

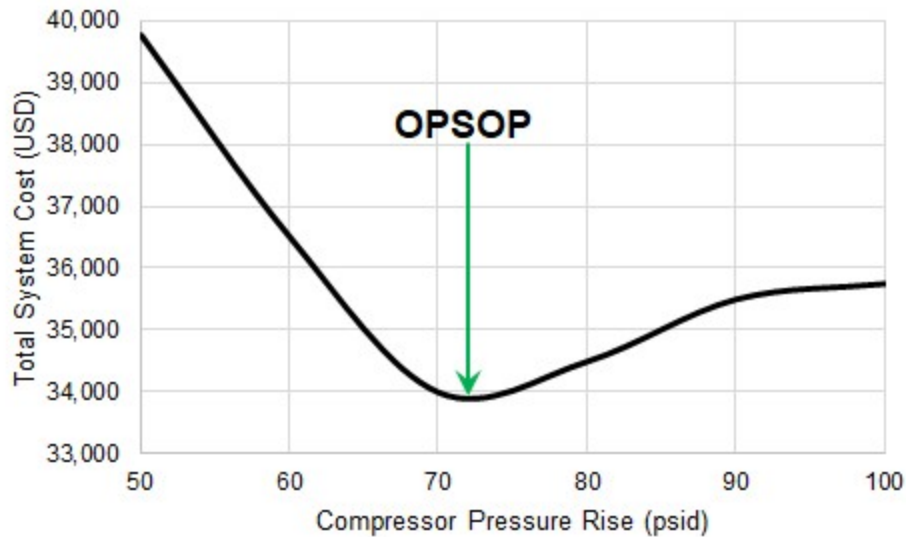


Figure 1: The OPSOP of a given system. Note that every point is an ideal sizing solution for the given compressor size.

If the sizing process is approached in stages, rather than concurrently, it is easy to get a result that says, for example, a 60 psid pressure rise compressor is required. The *best* case system for the 60 psid pressure rise compressor has a minimum cost of about \$36,500. However, allowing the compressor to change as well can result in a system more than 7% cheaper.

Process Outline

1. Find the Approximate Compressor Size
 1. Model the compressor with the [Compressor Sizing](#) option, fixing the flow to the desired amount. AFT Arrow will determine the required pressure rise.
 1. Include an estimate of Nominal Efficiency.
 2. Add the compressor to an [Engineering Library](#).
 1. Attach a [Cost Library](#) to the Engineering Library if one does not already exist.
 2. Create a Power vs. Cost [Scale Table](#) for the compressor with estimated cost values.
 3. Select the appropriate options in the [Sizing Window](#).
 1. The cost for the compressor will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 2. If there are several compressors that will ultimately be of the same type, they should be placed into a Maximum Cost Group.
4. Run ANS.

1. Investigate the resulting compressor size.
 2. Update the Nominal Efficiency based on the result. Also update the cost information if necessary. If the compressor accounts for a large portion of the overall cost, accurate values here are more important.
 3. Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.
2. Compare Candidate compressors
 1. Locate accurate cost information and [compressor curves](#) for the candidates.
 1. Incorporate these values into Engineering and Cost Libraries.
 2. Compare the sizing results for each candidate compressor.

Estimating Compressor Costs

Without knowing specifically what type or size of compressor is going to be included in the system, the compressor behavior must be approximated. The behavior is approximated with the [Compressor Sizing](#) option, indicated with a design flow. We also need to approximate cost. Because energy cost is often the largest component of a compressor's overall cost, and this cost is directly related to the compressor's power, it makes sense to correlate other compressor costs to power as well.

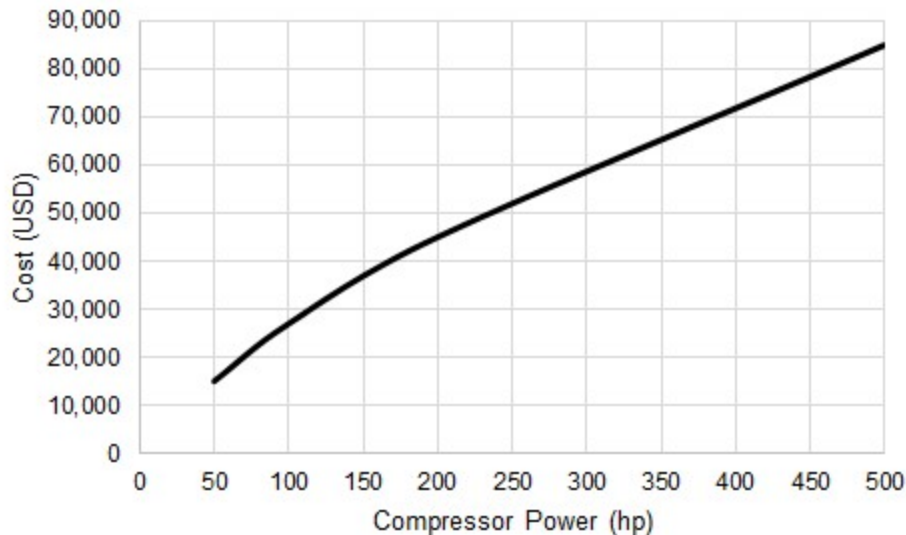


Figure 2 - Total Initial Costs for a series of compressors

Keep in mind that such a comparison is only valid for the selected range of compressors. Different designs of compressors, or compressors from different manufacturers may follow different relationships. One manufacturer may offer cheaper low-power compressors, while their competitor offers cheaper high-power compressors. This cost information is represented in ANS as a Power vs Cost [Scale Table](#). This requires the creation of a [Cost Library](#) including the compressor as a [Custom Component](#).

Because we do not know what specific compressor is going to be used, we also do not know exactly how much it will cost to run or maintain it.

As part of our sizing process, we must assume that the compressor runs at some **Nominal Efficiency**. AFT Arrow will calculate the fluid power required at any given flow and head rise. However, a compressor or motor is not perfectly efficient and will therefore require more power than the fluid power. This affects the compressor selection, and therefore cost, so it is important to include an estimate on efficiency.

Note: It may seem that including a nominal efficiency is not important, as selecting a certain compressor size will minimize the initial compressor cost regardless of this value. However, this does not consider the cost of power over a long period, which is often a driving factor in sizing. Also, it is important to get the relative cost of the compressor correct as this can affect the best sizing. The nominal efficiency will require more power and hence a more expensive compressor - as this is included in the Objective, it is important to account for.

If maintenance costs are expected to be significant, it is important to include an estimate of these costs. Again, this could be accounted for with a Power vs Cost Scale Table.

The Approximate Sizing Solution

When the initial model is defined, the sizing configuration complete, and the compressors appropriately defined with approximate cost information, the automated sizing can take place.

ANS will size the pipes of the system to minimize cost while obeying Design Requirements. It will now include a power-based cost for the compressor. Without cost information on the compressor, ANS would tend to select small pipe sizes to reduce cost without any negative consequence. However, smaller pipe sizes have higher frictional loss, which requires greater pressure rise at the compressor, and hence higher cost. By including compressor cost information, the balance between compressor costs and piping costs can be found.

However, this is still an *estimate* because we are not modeling a real compressor. Fixing the flow and nominal efficiency are good design tools but it is important to represent a real compressor curve not only to account for real effects, but also to ensure that the desired compressor can actually be acquired.

If the compressor costs are a significant portion of the overall cost, the estimates made for cost and nominal efficiency should be refined and the system re-sized before proceeding.

Sizing with a Real Compressor

Now that there is a good idea of the size and cost of compressor we are looking for, we can select a real compressor we may want to use.

It will rarely be clear which compressor is the best choice. First, the exact operating point determined from the approximate solution is likely unavailable in a commercially available compressor. Second, different compressors have different costs - even if a compressor with the exactly

correct specifications is available, it may be much more expensive than a compressor that is slightly mismatched. This cost difference could mean the cheaper compressor is better overall, despite not appearing to be the ideal choice.

The best course is to select several candidate compressors, and size the system independently with each candidate compressor. It is important that accurate [compressor curves](#) and cost information be included for each candidate, as this no longer meant to be an estimation. After testing each candidate compressor, the system with the lowest overall cost can be selected.

If the sizing results differ significantly in this stage this could indicate that the compressor is poorly matched, that there was an error in the process, or that the system is dominated by compressor costs and therefore very sensitive to the estimations.

Related Topics

Sizing Components Other Than Pipes.....	544
Special Concerns For Sizing Compressors.....	545
Sizing Pipes and Control Valves Concurrently.....	551
Sizing Pipes, Compressors, and Control Valves Concurrently.....	555
Sizing Duct Systems.....	556

Sizing Pipes and Control Valves Concurrently

Process Outline

1. Find the Approximate Control Valve Size
 1. Model the Control Valve with the appropriate settings. AFT Arrow will determine the required loss.
 2. Add the Control Valve to an [Engineering Library](#).
 1. Attach a [Cost Library](#) to the Engineering Library if one does not already exist.
 2. Create a C_v vs. Cost [Scale Table](#) for the control valve with estimated cost values.
 3. Select the appropriate options in the [Sizing Window](#).
 1. The cost for the control valve will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 2. If there are several control valves that will ultimately be of the same type, they should be placed into a Maximum Cost Group.
 4. Run ANS.
 1. Investigate the resulting control valve size.
 2. Update the cost information if necessary. If the control valve accounts for a large portion of the overall cost, accurate values here are more important.
 3. Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.
2. Compare Candidate Control Valves

1. Locate accurate loss and cost information for the candidates.
 1. Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate control valve.

Reference Pressure Concerns

It is impossible to generate a unique solution from a system (or part of a system) entirely bounded by fixed flows. For an in-depth discussion of this issue and common areas where it may occur see [Role of Pressure Junctions - Detailed Discussion](#).

Particularly common when sizing a system is the tendency to place a Flow Control Valve immediately downstream of a Compressor modeled as a fixed flow. This situation has no unique solution, as discussed in the [Sizing Compressors or Fans with Flow Control Valves](#) section of the previous topic.

The typical solution to this issue is to temporarily change the Flow Control Valves to Pressure Drop Control Valves where necessary, until a compressor curve is selected and specified. Effectively, this is the equivalent of specifying that our Flow Control Valve has a pressure drop requirement equal to the Pressure Drop Control Valve setpoint.

Control Valves and Design Requirements

ANS will *always* attempt to minimize or maximize the value of the Objective. **ANS does not know if the resulting design is realistic and this has no impact on the sizing.** It can present results that would be physically impossible to attain in the field.

AFT Arrow is no different in this regard - it is possible to specify a system that would require physically unrealistic behavior. For example, with an AFT Standard fluid, the gas temperatures are allowed to drop below saturation temperature. The system will still be solved as though the fluid is completely superheated gas. When possible, there will be Warnings if this type of behavior is detected, but the solver does not make any attempt to prevent this situation. To avoid these unrealistic configurations, the user must inspect the output and determine the changes required on the system to avoid the situation. In other words, the user must adjust the design of the system to meet desired or realistic behavior - a Design Requirement.

As an analysis tool, AFT Arrow does not allow the user to directly impose these requirements. ANS, on the other hand, is a design tool and actively selects inputs to meet certain specifications. It would not be unusual to specify a minimum temperature Design Requirement to prevent the saturation issue.

How does this relate to Control Valves? Control Valves in AFT Arrow will, by default, [force control](#). This means that the system will be able to behave unrealistically in order to meet the setpoint specified, which can be useful in a design stage. **As it would not be useful for a Control Valve to be unable to achieve its setpoint during the sizing process, Always Control (Never Fail) is automatically enabled for sizing.**

Consider a simple case of a Flow Control Valve regulating flow between two reservoirs. It is desired to size this system for lowest pipe weight.

Because of Always Control (Never Fail), the Flow Control Valve is essentially able to act like a compressor - providing "free" pressure to the system in order to meet the setpoint. As the cost of this "compressor" is not considered, the sizing process is trivial - we can simply select the smallest pipe size available, as our "compressor" will simply make up for the increase in pressure loss.

Clearly, it is not realistic for a Control Valve to add pressure to a system. So why don't we allow our Flow Control Valve to not meet its setpoint, as it would in reality? If the Control Valve is allowed to do so, we are essentially stating that the flow it is attempting to maintain is not actually a requirement. If this is the case, then again we will find that the "ideal" sizing solution is to select the smallest pipe size available. The Flow Control Valve will fully open in an attempt to allow more flow, but in the end the flow will be below the setpoint. However, our pipe weight will certainly be minimized.

What is the solution? **We need to add a Design Requirement.** It may seem intuitive to add a minimum flow requirement to our system as we are attempting to control flow. However, this will not solve the problem because the Control Valve will still simply add pressure to meet the setpoint. Furthermore, adding Design Requirements that [conflict](#) with the model itself can cause problems.

Instead, a Design Requirement should be added related to the *position* of the Control Valve. The position can be represented either directly as an Open Percentage if that data is available, or as a form of loss. It is typical that a Control Valve is designed to operate within a certain range of positions - often 20-80% open. It would not be useful to have a Control Valve designed to operate at 100% open, as a slight variation in flow could cause it to not meet its setpoint and open fully.

By adding a Design Requirement on minimum dP/dH or maximum C_v /Open Percentage we prevent unrealistic behavior and return a meaningful sizing result.

Note: It is *always* required to add this Design Requirement for Control Valves in the sizing model - even if the junction is not marked to be Sized in Size/Cost Assignments. Including a Control Valve in Sizing adds its cost to the Objective which can influence the result, which is not always a concern. However, the behavior described above is always true and hence always needs a Design Requirement.

Optimal Control System Operating Point

Just like [sizing pipes and compressors concurrently](#), there is an optimal size of a control valve that minimizes total system cost. Approaching the problem in "stages" rather than concurrently can easily miss the lowest overall cost solution.

Like the compressor process, we do not know specifically what Control Valve is going to be used in the early stages of the process so we need to estimate the cost.

For Control Valves, the logical choice is to estimate with a Cost vs C_v relationship. This information will be represented for the particular component in a Scale Table.

Note: When defining a Cost vs C_v Scale Table, keep in mind that this table is relating the *current* C_v to the cost of the device. For example, if the Control Valve has a Design Requirement to be 80% open, and ANS calculates a corresponding C_v of 120, the cost used from the Scale Table will be that for a C_v of 120, *not* the full-open value of the control valve. This is important to keep in mind when creating the scale table, as manufacturer cost information will often be in the form of full-open C_v vs cost.

The Approximate Sizing Solution

When the initial model is defined, the sizing configuration complete, and the control valves appropriately defined with approximate cost information, the automated sizing can take place.

ANS will size the pipes of the system to minimize cost while obeying Design Requirements - including the Design Requirement on position discussed above. However, this is still an *estimate* because we are not modeling a real control valve.

If the control valve costs are a significant portion of the overall cost, the estimates made for cost should be refined and the system re-sized before proceeding.

Sizing with a Real Control Valve

Now that there is a good idea of the size and cost of control valve we are looking for, we can select a real control valve we may want to use.

It will rarely be clear which control valve is the best choice. First, the exact operating point determined from the approximate solution is likely unavailable commercially. Second, different valves have different costs - even if a valve with the exactly correct specifications is available, it may be much more expensive than one that is slightly mismatched. This cost difference could mean the cheaper valve is better overall, despite not appearing to be the ideal choice.

The best course is to select several candidate valves, and size the system independently with each candidate. It is important that accurate loss and cost information be included for each candidate, as this no longer meant to be an estimation. After testing each candidate, the system with the lowest overall cost can be selected.

If the sizing results differ significantly in this stage this could indicate that the control valve is poorly matched, that there was an error in the process, or that the system is dominated by control valve costs and therefore very sensitive to the estimations.

Related Topics

Sizing Components Other Than Pipes.....	544
Special Concerns For Sizing Compressors.....	545
Sizing Pipes And Compressors Concurrently.....	547
Sizing Pipes, Compressors, and Control Valves Concurrently.....	555
Sizing Duct Systems.....	556

Sizing Pipes, Compressors, and Control Valves Concurrently

Many systems contain both compressors and control valves, which makes the sizing process somewhat more complex.

Like sizing [pipes and compressors](#) or [pipes and control valves](#) concurrently, there are adjustments and estimations that must be made when sizing all three in parallel. The overall goal and considerations that must be taken into account are described in detail in those topics and will not be repeated here. Only the outline of steps will be presented for brevity.

Process Outline

1. Find the Approximate Compressor and Control Valve sizes
 1. Model the compressor(s) with the [Compressor Sizing](#) option, fixing the flow to the desired amount.
 2. Model the Control Valve(s) with the appropriate settings.
 1. If the control valve(s) are Flow Controls downstream of the compressor(s) to be sized, temporarily set them to be Pressure Drop Control Valve with a setpoint equal to the minimum required pressure drop.
 2. Any non-Pressure Drop control valves will require a Design Requirement of minimum dP/dH or maximum C_v /Open Percentage.
 3. Add the compressor(s) and control valve(s) to one or several [Engineering Libraries](#).
 1. Attach a [Cost Library](#) to each Engineering Library if one does not already exist.
 2. Create a Power vs. Cost [Scale Table](#) for the compressor with estimated cost values.
 3. Create a C_v vs. Cost [Scale Table](#) for the control valve with estimated cost values.
 4. Select the appropriate options in the [Sizing Window](#).
 1. The cost for the compressor(s) and control valve(s) will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 2. If there are several compressors or control valves that will ultimately be of the same type, they should be placed into Maximum Cost Groups.
 5. Run ANS.
 1. Investigate the resulting compressor size.
 2. Update the Nominal Efficiency and NPSHR based on the result. Also update the cost information if necessary. If the compressor accounts for a large portion of the overall cost, accurate values here are more important.
 3. Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.
2. Compare Candidate Compressors
 1. Locate accurate cost information and [compressor curves](#) for the candidates.
 1. Incorporate these values into Engineering and Cost Libraries.

2. Compare the sizing results for each candidate compressor.
3. Select the two or three best candidate compressors and compare candidate Control Valves.
 1. If necessary, change Pressure Drop Control Valves back to Flow Control Valves.
 1. This will require a Design Requirement of minimum dP/dH or maximum C_v /Open Percentage.
 2. Locate accurate loss and cost information for the candidate control valves.
 1. Incorporate these values into Engineering and Cost Libraries.
 2. Locate accurate loss and cost information for the candidate control valves.
 1. Incorporate these values into Engineering and Cost Libraries.
 3. Compare the sizing results for each candidate control valve, under each candidate compressor.

Related Topics

Sizing Components Other Than Pipes.....	544
Special Concerns For Sizing Compressors.....	545
Sizing Pipes And Compressors Concurrently.....	547
Sizing Pipes and Control Valves Concurrently.....	551
Sizing Duct Systems.....	556

Sizing Duct Systems

An Engineering library of all possible rectangular duct sizes is needed, as is the case with Cylindrical pipe.

If there are no geometric restrictions, the [Candidate Set](#) should use all of the duct sizes. Certain areas of the system may have maximum width or height restrictions. In these areas, a different Candidate Set should be created from the same Engineering library.

There are two variables for a rectangular duct - height and width, whereas a normal pipe only has one - diameter. ANS treats these variables as independent when solving for the ideal size. This means that duct Candidate Sets must have corresponding widths and heights for every duct dimension.

For example, consider three duct sizes in a Candidate Set: 8x8, 10x10, and 12x12. Because width and height are considered independent by ANS, it is possible that 8 would be selected for width and 10 for height. However, 8x10 is not in the Candidate Set. ANS will warn that the Candidate Set is incomplete when this is detected.

Related Topics

Sizing Components Other Than Pipes.....	544
Special Concerns For Sizing Compressors.....	545
Sizing Pipes And Compressors Concurrently.....	547
Sizing Pipes and Control Valves Concurrently.....	551
Sizing Pipes, Compressors, and Control Valves Concurrently.....	555

Search Methods

ANS includes several different Search Methods for finding the best sizing solution. Some methods are very similar, while others are entirely unique, and each is particularly suited to certain kinds of mathematical problems.

It is impossible to state that one method is always better or worse than another. The quality and speed of each method depends on the specific system being solved and the sizing input. A method that excels for one model may not be the best choice for another model.

Note: All of the methods share a common goal of minimizing (or maximizing) an Objective. The analytical approach driving the sizing (the hydraulic solver) does not vary with the selected method. This means that the best sizing solution across all of the methods is the best overall solution. **It is good practice to test several of these methods for any model.**

- [Genetic Algorithm](#) - Cross-breeds a random "population" of designs, mimicking biological evolution. Less likely to become trapped in a local minimum, but very slow compared to other methods.
- [Modified Method of Feasible Directions](#) - Reliable and applicable to most problems.
- [Sequential Quadratic Programming](#) - Frequently the most efficient, but more sensitive to system definition.
- [Sequential Unconstrained](#) - Sensitive to initial sizes and control parameters, but more capable of handling large numbers of uniquely sized pipes.

Related Topics

Modified Method of Feasible Directions.....	557
Sequential Quadratic Programming.....	559
Sequential Unconstrained.....	562
Genetic Algorithm.....	564

Modified Method of Feasible Directions

A [Continuous Search Method](#) available in the [Sizing Method panel](#).

The Modified Method of Feasible Directions (MMFD) finds the best case design by determining a **search direction** based on the [Objective](#) function and the [active Design Requirements](#). When there are no active Design Requirements, the search direction will simply be in a direction influenced by the current [gradient](#) and previous search directions.

When there is an active Design Requirement, following the direction of greatest change may make the design [Infeasible](#). Instead, the search direction is modified to keep the solution in the feasible region. If a Design Requirement is Violated, the search direction will attempt to return the solution to the feasible region as quickly as possible.

This is the most robust method and can handle a wide variety of problems.

Note: [Branch and Bound](#) rapidly becomes inefficient with an increase in independent variables. For [discrete](#) optimization with more than ten independently sized pipes, the [Sequential Unconstrained](#), or [Genetic Algorithm](#) methods should be considered.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Arrow itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it is important to select an appropriate value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.
- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of **sizing iterations**, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.
 - **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.

- **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of $1,000 \times 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.
- **Sequential Unconstrained Parameters**
 - **Maximum Number of Independent Pipe Sizes for Branch and Bound** - Discrete solutions with this method require the use of Branch and Bound. If the system has more independently sized pipes than this value, the Sequential Unconstrained will automatically be used.
 - **Sequential Unconstrained Parameters** - See [Sequential Unconstrained](#).
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.
- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....	557
Sequential Quadratic Programming.....	559
Sequential Unconstrained.....	562
Genetic Algorithm.....	564

Sequential Quadratic Programming

A [Continuous Search Method](#) available in the [Sizing Method panel](#).

Sequential Quadratic Programming (SQP) finds the best case design by solving a series of quadratic approximations to the [Objective Function](#). Each of these "Sub-Problems" is solved with the [Modified Method of Feasible Directions](#). This method is often the best and most efficient if the problem is numerically well defined. Sizing with this method is always worth checking.

Note: [Branch and Bound](#) rapidly becomes inefficient with an increase in independent variables. For [discrete](#) optimization with more than 10 independently sized pipes, the [Sequential Unconstrained](#), or [Genetic Algorithm](#) methods should be considered.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Arrow itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it is important to select an appropriate value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.
- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of sizing iterations, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.

- **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.
- **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of $1,000 \times 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.
- **Sub-Problem Iterations**
 - **Maximum** - This is the maximum number of approximate problem cycles.
 - **Consecutive for Convergence** - The approximations will be repeated at least this number of times while satisfying the Sizing Level Convergence tolerance criteria.
- **Sequential Unconstrained Parameters**
 - **Maximum Number of Independent Pipe Sizes for Branch and Bound** - Discrete solutions with this method require the use of Branch and Bound. If the system has more independently sized pipes than this value, the [Sequential Unconstrained](#) method will automatically be used.
 - **Sequential Unconstrained Parameters** - See [Sequential Unconstrained](#).
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.
- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....557

Modified Method of Feasible Directions.....	557
Sequential Unconstrained.....	562
Genetic Algorithm.....	564

Sequential Unconstrained

A [Continuous Search Method](#) available in the [Sizing Method panel](#). This method is unique in that it has an internal method to generate [Discrete](#) solutions as well.

Sequential Unconstrained (SU) finds the best case design by solving a series of [unconstrained](#) problems. This simplification is appealing because it is almost always easier to solve an unconstrained problem, making it possible to efficiently solve very large and complex problems. In order to account for the [Design Requirements](#), the [Objective](#) function is artificially increased in [Infeasible](#) regions. This has the effect of driving the unconstrained solution into the feasible region.

Artificially increasing the Objective a large amount on the first cycle creates a numerically ill-conditioned problem which is difficult to solve. Instead, only a small penalty is added initially. This means it is likely the solution on the first cycle will be infeasible. The next cycle increases the penalty factor and uses the results of the previous cycle as a starting point. In this way, the optimum point is moved toward the feasible region in small, numerically stable, steps. Once the result of an unconstrained cycle ends in the feasible region, the process is stopped.

A Hybrid Method

The Modern Penalty Method does not use [Branch and Bound](#) for the determination of Discrete sizes. Instead, it can force the continuous solution into discrete "slots" by creating an artificial set of constraints after the continuous solution is complete. This prevents the need for searching an exceptionally large "tree" that would result with Branch and Bound. However, the method is generally only effective at finding discrete sizes +/-1 size away from the continuous solution.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Arrow itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it is important to select an appropriate

value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.

- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of sizing iterations, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.
 - **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.
 - **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of $1,000 \times 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.
- **Sub-Problem Iterations**
 - **Maximum** - This is the maximum number of approximate problem cycles.
 - **Consecutive for Convergence** - The approximations will be repeated at least this number of times while satisfying the Sizing Level Convergence tolerance criteria.
- **Sequential Unconstrained Parameters**
 - **Maximum Number of Independent Pipe Sizes for Branch and Bound** - Not used when Sequential Unconstrained is directly selected.
 - **Number of Discrete Iterations** - This number of cycles with the artificial discrete constraints in place will be run. If this value is greater than 1, it is possible that discrete sizes more than one size away from the continuous solution can be found. However, this does not force an exhaustive search of all potential discrete sizes in this range.
 - **Continuous Penalties**

- **Initial Multiplier** - This is the penalty applied to the Objective in the first continuous unconstrained cycle.
- **Multiplier on Additional Iterations** - The penalty is increased by this amount each continuous cycle.
- **Discrete Penalties**
 - **Initial Multiplier** - This is the penalty applied to the Objective in the first discrete unconstrained cycle.
 - **Multiplier on Additional Iterations** - The penalty is increased by this amount each discrete cycle.
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.
- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....	557
Modified Method of Feasible Directions.....	557
Sequential Quadratic Programming.....	559
Genetic Algorithm.....	564

Genetic Algorithm

A [Discrete Search Method](#) available in the [Sizing Method panel](#). This method is fundamentally discrete and does not use [gradients](#) in its search.

The Genetic Algorithm is a type of Evolutionary Algorithm - as the name implies, it selects advantageous traits by emulating biological evolution. An initial "population" of potential solutions is generated and gradually improved.

The starting population is an effectively random selection of "individuals". In the context of ANS, an individual is a proposed selection of pipe sizes for the system in question. At the start of the sizing process, these individuals may have an assortment of good and bad traits. A "bad" trait might be one that violates a Design Constraint or increases the cost of the system.

To improve the Fitness of the population there are three major operations: *Selection*, *Crossover*, and *Mutation*.

Selection occurs by determining the "Fitness" of all individuals in the population. Objective value is not the only concern when determining how acceptable any given individual is, so we need a different function to represent this. This function is a Fitness Function. The Fitness of any given individual is measured based on its Objective value and proximity to, or violation of, Design Requirements. Like Darwin's theory of "Survival of the Fittest" the goal in this process is to maximize the Fitness of the population as a whole. With the Fitness known for all individuals, a parent population with high Fitness can be selected.

Crossover occurs by "breeding" the parent solutions to generate new solutions. Some of these solutions will, randomly, have inherited good traits from their parents. Others will have inherited bad traits.

Mutation occurs after crossover, introducing a further element of randomness into the system by changing traits with no regard to the parent traits. This allows the algorithm to consider new solutions that were not previously part of the "gene pool." If no new traits can be introduced, the flexibility of the method is limited. If the mutation rate is too high, however, the good traits will not be carried through to new generations as effectively.

The new population is then combined with the parent population, and the process is repeated.

Note: When sizing with a Discrete method, it is possible to start the search with a Continuous Method. This limits the search space which should reduce the run time. When this option is enabled, the user has control over both the Continuous and Discrete methods.

Sizing Control Parameters

- **Genetic Algorithm Population Size**
 - **Number of Discrete Sizes Around Continuous Solution** - If Start with Continuous Method is selected, this option limits the search space of the genetic algorithm by only considering discrete sizes close to the continuous solution. For example, a Candidate Set may contain 20 unique sizes. Including all of these sizes makes the problem extremely complex, as indicated in [Brute Force Method](#). It is likely that the best discrete solution is near the best continuous solution, so only searching 4 unique sizes (+/- 2) around the continuous solution can dramatically reduce runtime.
 - **Automatic** - Set the User Specified Multiplier to 2 to determine the size of the population to use.
 - **User Specified Multiplier** - Every independently sized pipe in the model has a certain number of possible sizes that it may be, as determined by its particular Candidate Set. Because every independently sized pipe in the model can have a unique Candidate Set, it can also have a unique count of possible sizes. This

multiplier is based on the sum of all of these counts across the model. For example, if there are 2 independently sized pipes with Candidate Sets P1: (4", 6") and P2 (3", 4", 6", 8"), the total count will be 6, and a multiplier of 3 would give a population size of 18.

- **User Specified Population** - The size of the population can also be directly specified. It is important the population size is not too small as this will artificially limit the capability of finding a good solution. However, population sizes too large will not solve in a reasonable time frame.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Genetic Algorithm Seed** - This number is used to generate a "random" population. The same number will always generate the same initial population.
- **Genetic Algorithm Parameters**
 - **Crossover Probability** - The average fraction of the population obtained from crossover. As a probability this does not mean exactly this fraction will be obtained. This value is typically close to 1, meaning that the majority of the population each generation is obtained as children of the parent generation. It is generally less than 1 to avoid losing very good solutions from the parent generation due to poor crossover.
 - **Crossover Distribution Index** - This value is inversely proportional to allowed perturbations caused by crossover. That is, a large value will limit the amount of perturbation generated from crossover, whereas a small value will allow significant perturbation.
 - **Mutation Probability** - The average fraction of the population subjected to mutation. As a probability this does not mean exactly this fraction will be obtained. This value is typically the inverse of the number of independently sized pipes. As the system (and usually, population) gets very large, the need for "additional" randomness decreases. However, for small systems this randomness is important to introduce to prevent a premature solution.
 - **Mutation Distribution Index** - This value is inversely proportional to allowed perturbations caused by mutation. That is, a large value will limit the amount of perturbation generated from mutation, whereas a small value will allow significant perturbation.
- **Genetic Algorithm Convergence**
 - **Consecutive Unimproved Generations** - When no improvements in the Objective occur over this number of generations, the sizing process will be stopped and the results are presented. It is assumed from the lack of progress that the ideal solution has been found. It is impossible to tell in advance how many total iterations this condition may require.
 - **Fixed Number of Generations** - To limit the runtime to a length that can be estimated, the algorithm can be set to a specific count of generations. This should be used with caution as it is possible to end before the best solution has been found, or to continue long after the best solution has been found.

Related Topics

Search Methods.....	557
Modified Method of Feasible Directions.....	557
Sequential Quadratic Programming.....	559
Sequential Unconstrained.....	562

Sizing Theory

Automated Pipe Sizing is a small subset of the mathematical field of Optimization. This is a complex subject that has evolved over many years - the topics here are not attempting to replicate this complexity but instead provide simplified understanding of the methods so that users of ANS can appropriately size systems and adjust models and parameters as needed.

- [Sizing and Optimization](#) - ANS is designed to simplify the optimization problem as much as possible. Those familiar with optimization may find this topic useful to connect common optimization terms to those used in ANS.
- [Boundaries vs. Design Requirements](#) - "Requirements" can be defined in many ways. Boundaries force a certain flow or pressure, whereas Design Requirements place limits on allowable values. While similar, the distinction is important in a sizing context.
- [Physical Parameters vs. Monetary Cost](#) - Monetary Cost sizing is often difficult to the amount of data required. For this reason, minimum cost is often estimated with a physical parameter, such as pipe weight.
- [Brute Force Method](#) - The most basic way to conclusively find a minimum is to exhaustively test all possibilities. This is impossible on a realistic time scale for most systems.
- [Gradient Based Methods](#) - The fastest methods for solving a continuous Objective are based on gradients. Understanding the core of these methods can help in finding better solutions more quickly.
- [Estimating the Gradient](#) - The gradient is only estimated - not analytically determined. This fact means the associated parameters may require adjustment in some cases. Understanding the affect of these adjustments is important.
- [Branch and Bound](#) - Only some methods are inherently suited to discrete sizing. Gradient based methods usually rely on narrowing the solution space and solving a number of easier problems.

Related Topics

Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576

Sizing and Optimization

Techniques for minimizing or maximizing certain parameters - optimization - have been used throughout a variety of industries for many decades. As computers become more prevalent and more powerful, they are able to handle larger problems with more variables and greater complexity. Optimization problems can be enormously complex, and techniques for solving them are almost always iterative. Such problems are generally impractical to solve without a computer.

The selection of pipe sizes to minimize some parameter - often monetary cost - is one specific example of optimization. For the sake of clarity, the mathematical terminology used when discussing optimization problems is compared to the terminology used in the ANS module here.

- **Optimization** - Modifying a system to minimize or maximize some parameter.
 - **In ANS** - This is represented by the name of the module - *Automated Network Sizing*. The sizes of the various network pipes are modified automatically by the application, to achieve some goal.
- **Objective** - The goal of optimization is referred to as the Objective. Minimizing monetary cost is a very common Objective. Others may be minimizing the time to complete an action, or maximizing the amount of material a machine can process.
 - **In ANS** - The term *Objective* is retained.
- **Objective Function** - It is required that the Objective can be calculated for any configuration of the system. This calculation is represented by the Objective Function, much like any other mathematical function. The Objective is to find the global maximum or minimum of this Objective Function.
 - **In ANS** - Is not a user input, and is not visible.
- **Constraint** - Anything that is required by the system regardless of the Objective value. For example, an Objective might be to minimize the amount of cloth needed to make a blanket. The smaller the blanket, the less material it needs, but a tiny blanket is not very useful. Instead, the Optimization might be Constrained such that the blanket must be some minimum area.
 - **In ANS** - These are called **Design Requirements**.
- **Design Variable** - Anything that is allowed to change in order to modify the Objective value. In the blanket example, this might be the width, length, and thread count of the blanket. As with any mathematical problem, the more variables there are, the harder the problem is to solve.
 - **In ANS** - The variables that are allowed to change are the pipe sizes. There are countless ways a fluid system could be Optimized, but pipe sizing is one of the most significant that will impact cost. The total number of design variables in the current analysis is displayed near the bottom of the [Size/Cost Assignments panel](#) as **Independently Sized Pipes**.

Related Topics

Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576
Branch and Bound.....	581

Boundaries vs. Design Requirements

Boundaries

Points in a model that **physically force a given parameter**.

A piping network can be considered to have three variables - *flow*, *pressure*, and the *system*. For analysis, it is required that the user fully define the *system*. The *system* is a collection of components that affect the *flows* and *pressures* - pipes, compressors, valves, etc. Because there are still two variables - *flow* and *pressure* - the user must specify at least one of them as a boundary somewhere in the system.

Note: Due to the nature of fluid analysis, if there is only one boundary it must be a pressure type boundary. See [Role of Pressure Junctions](#).

For a model to be accurate, its *entire* definition must reflect the system it is representing. This includes both the *system* and the *boundaries*. It is obvious that a 120 foot, 12" diameter pipe should be represented in the model with the same length and diameter. However, ensuring the correct selection of boundaries is often less intuitive.

Specifying any given boundary as a fixed pressure will result in some calculated flow at that boundary. Changing this pressure boundary to a flow boundary with the calculated value will not change any of the pressure or flow results in the model. However, this has important physical implications.

For example, consider a pipe bounded by two pressures. Doubling the pipe size will significantly increase flow. However, if a boundary was instead a fixed flow, doubling the pipe size will *not* change the resulting flow but instead change the pressures at the flow boundary. If the boundary in the real system is a large fixed pressure tank, it should not be represented as a fixed flow, because changes to the system will effectively change the pressure required in the tank. In other words, **changing the system changes the physical meaning of the boundary**. If the tank is represented as a fixed pressure, as it behaves in the field, this is not an issue.

This is important in ANS because ANS is **changing the system** as part of the sizing process. Using the incorrect type of boundary can present sizing solutions that would not work in the real system.

Note: It is not uncommon or incorrect to use boundaries to represent a *requirement* in an analysis. This is a common design tool to determine what is required of a boundary. In the above example, it may be desired to know the required tank pressure to drive the required flow. What is important is to recognize whether a junction is representing a requirement or a physical boundary that cannot be changed.

Design Requirements

Design Requirements are **limits on allowable behavior**.

The distinction is important because Design Requirements are *limits* rather than *imposed* values. For example, a Design Requirement may state that a flow must be **at least** 100 feet³/min. The design is considered acceptable if the flow ends up as 120 feet³/min. On the other hand, using a boundary of 100 feet³/min **forces the model always to maintain 100 feet³/min**.

In this manner, Design Requirements are *less* restrictive than boundary conditions. Generally, the less restrictive the problem is, the more flexibility is available to obtain a better sizing result.

Even if it is known that there must be some fixed flow into a tank, using an Assigned Flow boundary at this location may not be the best choice. If the pressure in the tank cannot be changed to the value calculated based on the sizing result, the use of an Assigned Flow here would be incorrect. Instead, it would be appropriate to use an Assigned Pressure and two Design Requirements on flow - a minimum and a maximum, to limit the solution to an acceptable range.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576
Branch and Bound.....	581

Physical Parameters vs. Monetary Cost

There are four physical parameters that can be maximized or minimized in addition to monetary cost:

- **Pipe Weight**
- **Pipe and Fluid Weight**
- **Flow Volume**
- **Pipe Inner Surface Area**

Actual Objective is to Limit the Physical Parameter

Occasionally, the goal of the sizing is to limit the physical parameter. For example, a system to be placed on an aircraft likely has greater emphasis on weight than monetary cost. It is important to keep in mind in such cases that **only the pipes are included in the total Objective value**. Compressors, valves, or other junctions that contribute to weight are not accounted for. This may affect the ideal sizing.

Using a Physical Parameter to Estimate Cost

There are advantages and disadvantages of estimating cost with a physical parameter.

It is generally far easier to use a physical parameter as **cost libraries are not required**. Cost libraries are difficult to locate information for, and can be time consuming to create and manage. Avoiding this greatly simplifies the process.

The main disadvantage is that **only initial pipe costs can be accounted for**. If junctions account for a significant portion of the initial cost, or if the recurring costs from operating them would affect sizing, then this may not result in the best sizing solution.

In general, if the goal is to minimize monetary cost of a system, then monetary information should be supplied. If recurring costs must be included, this is the only reasonable option. However, physical parameters often provide a good estimate of the ideal size for initial cost. It makes sense to start the sizing process with a physical parameter, and proceed to monetary cost as the Objective after larger design issues have been resolved. This is especially true if the potential savings are large.

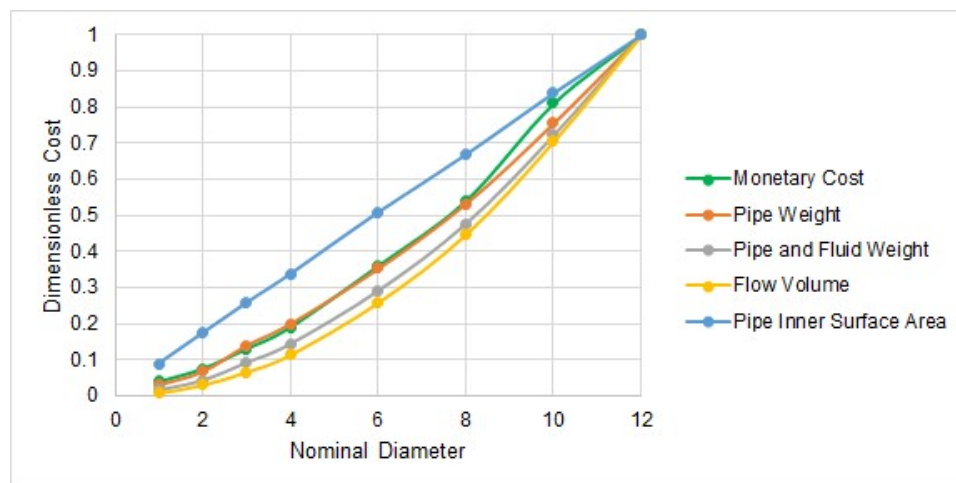


Figure 1: Comparison of Dimensionless Costs for Schedule 40 ANSI Steel water pipe. RS Means monetary cost closely follows pipe weight in this example.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Brute Force Method.....	572
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576
Branch and Bound.....	581

Brute Force Method

The "easiest" way to find the cheapest set of pipe sizes is to test all of the possible combinations exhaustively, and report back the lowest cost option found.

This has the advantage that it is clearly conclusive - there can be no argument that a cheaper alternative exists if every possible alternative was shown to have higher cost. Unfortunately, this method is extremely impractical for any reasonably sized problem. The number of required analyses to find the current value of the Objective is related to the number of design variables and how many unique discrete sizes each variable can have.

In the context of ANS, this is the number of Independently Sized Pipes, and the number of selections in the Candidate Set, assuming there is one set for the model.

$$\text{Unique Hydraulic Models} = (\text{Candidate Set Sount})^{\text{Number of Independently Sized Pipes}}$$

If we have only 3 independently sized pipes, and can only choose from 3 different pipe sizes, then we would need to solve 27 unique hydraulic models, which could be manageable by hand.

However, if we have 12 independently sized pipes, and can choose from 6 different pipe sizes, we suddenly have over **2 billion** unique models to solve. If we could test 50 models every second, this would take over a year to complete - and it is still a relatively small system!

Due to the clear impracticality of this method, it is not used in ANS.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576
Branch and Bound.....	581

Gradient Based Methods

Gradient based methods search for the ideal solution using the *gradient* of the Objective Function. A gradient is a vector function composed of partial derivatives measuring the change of some function with respect to each component. Effectively, a gradient is a derivative of a multi-variable function - **it indicates the magnitude and direction of greatest change** in a multi-variable function.

A gradient of a single variable function is simply the derivative of that function. While most problems requiring complex optimization techniques have many variables, it is easiest to think about a single-variable function, such as elevation related only to distance along some horizontal line:

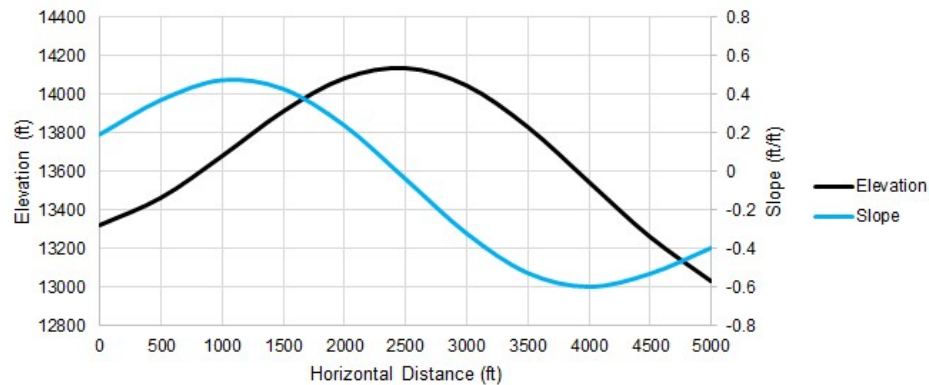


Figure 1: One of many possible elevation profiles of a mountain peak. The slope can be calculated at any point.

The gradient of a mountain's elevation will indicate the magnitude and direction of the elevation change at any given latitude and longitude. In other words, given a specific point, the gradient will tell you how steep the slope is at that point, and which direction it is steepest in. This is very useful for finding the top of the mountain - without being able to see anything but the local value of the gradient, you can make an estimate that the peak is probably in the direction of the slope, and that if the slope is very steep you likely have a long ways to go.

If the entire elevation profile was known ahead of time, finding the peak is trivial. Unfortunately, we do not normally know the entire profile, but instead can only determine its value where we explicitly test it. Stated generally, **the shape and behavior of an Objective Function is not known ahead of time.**

Rather than testing every single horizontal point to find the highest elevation by [brute force](#), we can employ an intelligent gradient based search method. Starting at a horizontal distance of zero, we can measure an elevation and slope:

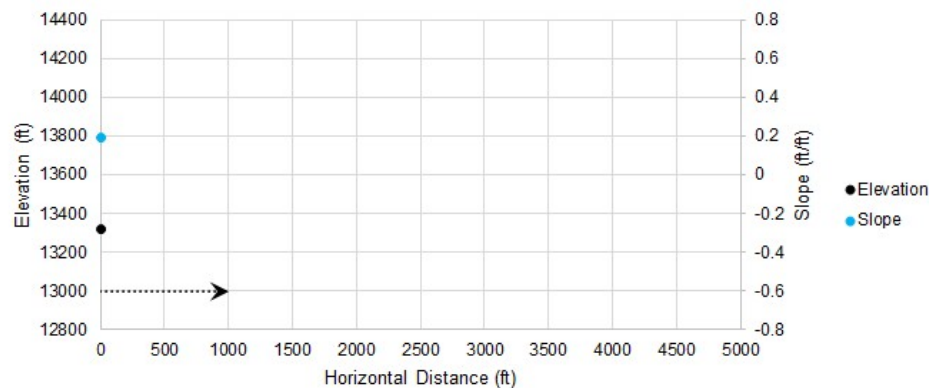


Figure 2: Starting a gradient based search

With the positive slope from our initial condition, we can make an educated guess that the peak is somewhere in the positive direction. We determine where we want to guess and measure our system again at that point:

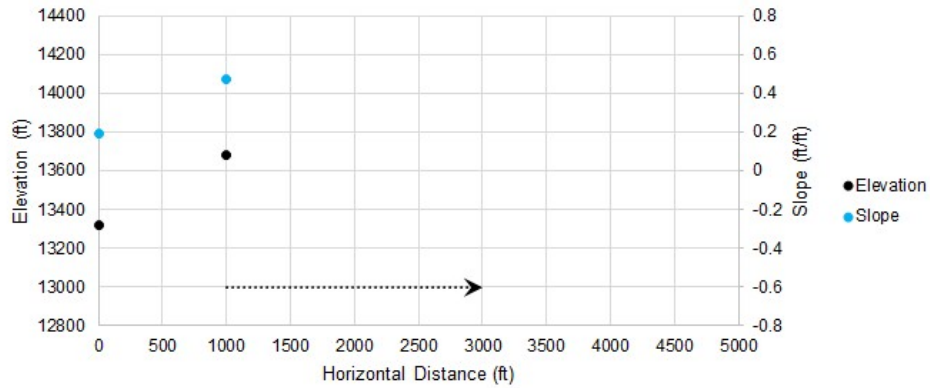


Figure 3: Iteration number 2

Because our slope is higher, we assume that the peak is farther away and adjust our change in distance accordingly. Our new guess has a negative slope, so we will reverse our search direction.

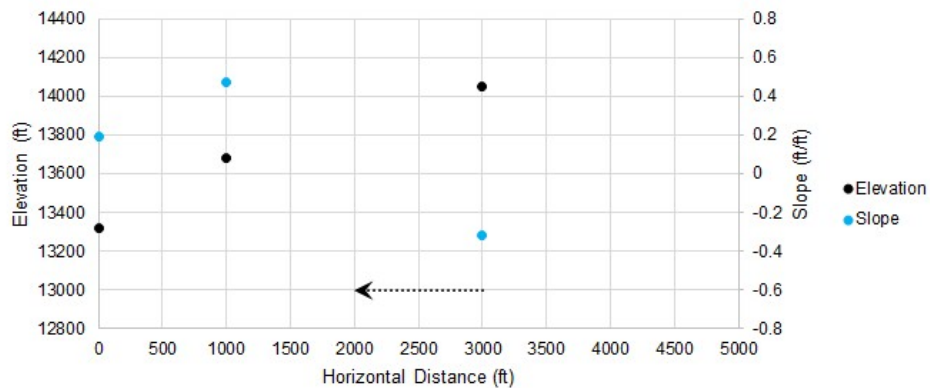


Figure 4: Iteration number 3

We changed sign on slope again, so we reverse direction again. We can use information from previous guesses to narrow down the search range and make a more intelligent guess.

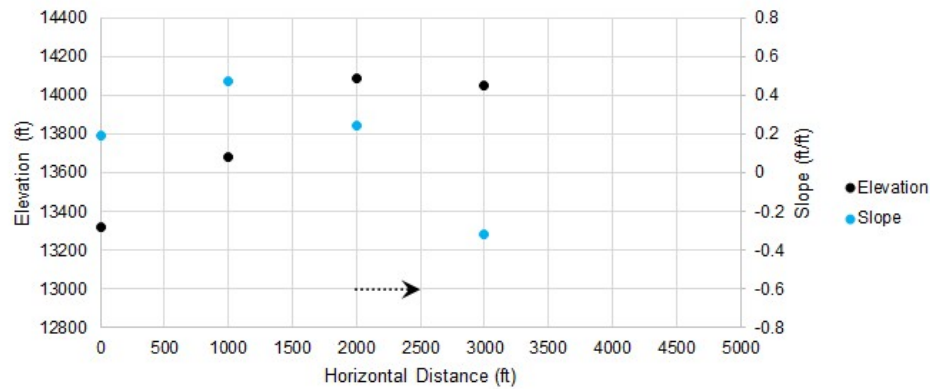


Figure 5: Iteration number 4

As we continue, the adjustments to our guess get smaller and smaller. Once they are small enough, we can consider the search converged, and the peak found.

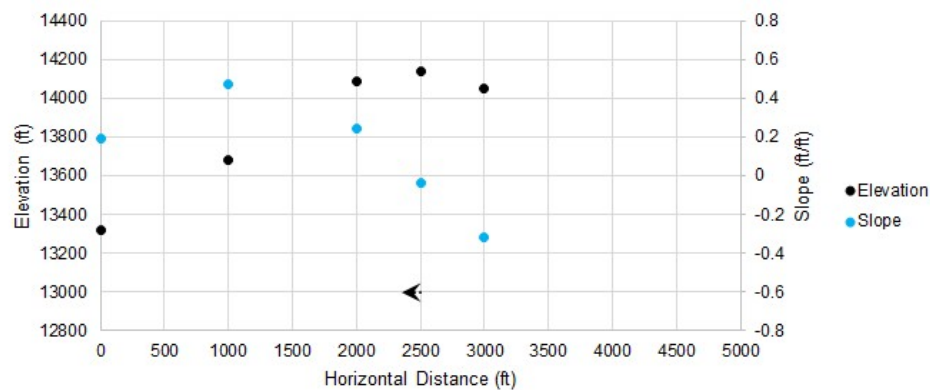


Figure 6: Iteration number 5

In only a few iterations, we have found an excellent approximation of the highest point, without knowing anything about the elevation or slope outside of our selected guess points.

The [search algorithms](#) included in ANS are much more sophisticated than the simple example presented here, but the idea remains the same. Furthermore, the allowable selections are restricted by [Design Requirements](#), which impacts the ideal search direction.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572

EstimatingTheGradient.html.....	576
Branch and Bound.....	581

EstimatingTheGradient.html

Analytically determining the gradient becomes extremely difficult if not impossible for problems with many variables such as sizing a piping network. Instead, the gradient is estimated with a **finite difference**. This is a method for numerically estimating the gradient at a given point.

The test points created with a finite difference method each represent a hydraulic model that must be solved independently. If the model is complex, adding additional variables (unique pipe sizes) or required runs (from a central difference) can make the process take a significant amount of time.

Forward Difference

The Forward Difference estimates the gradient by finding the change in the function value by making a small change in the independent variable.

First, an initial result is found. We want to know the gradient at this point.

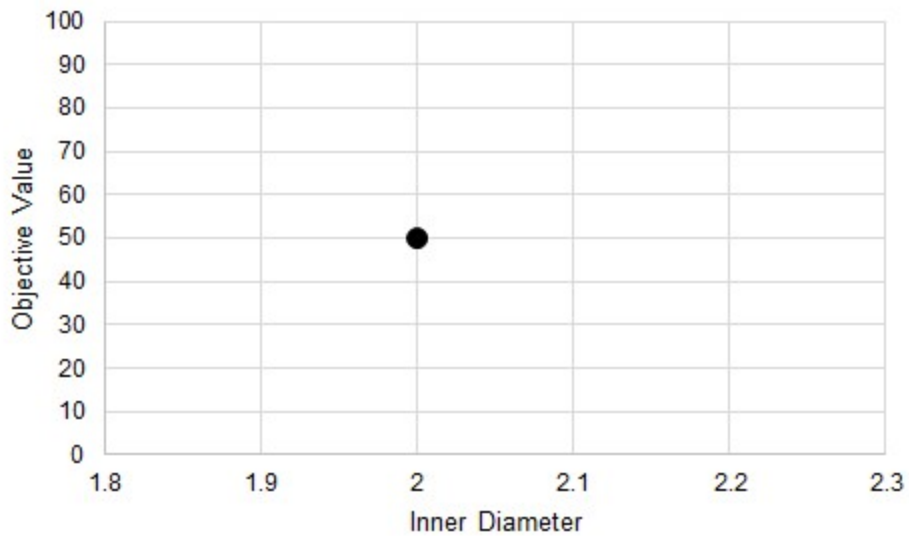


Figure 1: An initial test. It is desired to know the gradient at this point.

The independent variable is perturbed some small amount and a new result found. A line can be drawn through these two points - an estimate of the gradient at the initial point.

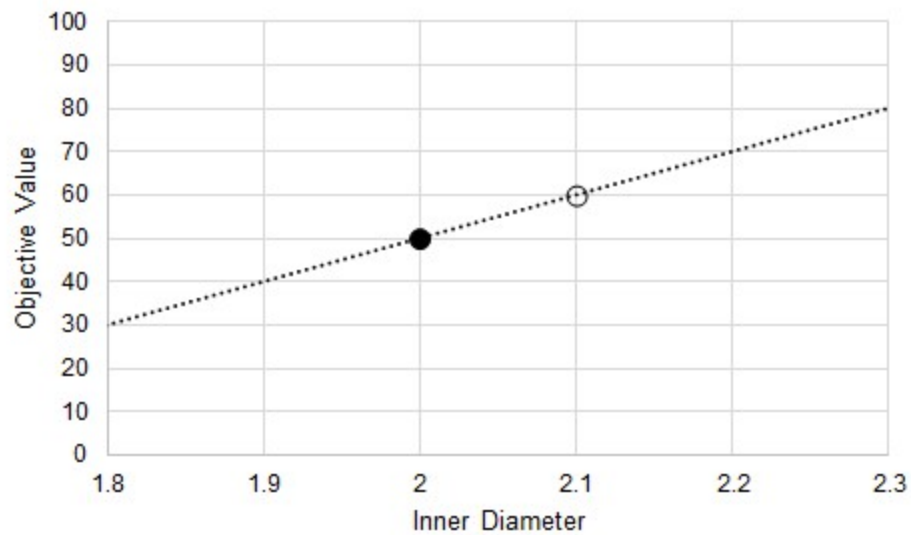


Figure 2: A perturbation. This allows the calculation of an average gradient as an approximation of the gradient at the initial point.

There are two important things to keep in mind:

- This is an average gradient between the two points, but it is **used as an estimate of the gradient at the initial point**. It is not used to represent the behavior between the two points.
- It is only an estimate. **The result depends on the behavior of the Objective Function, and the selection of the test point.**

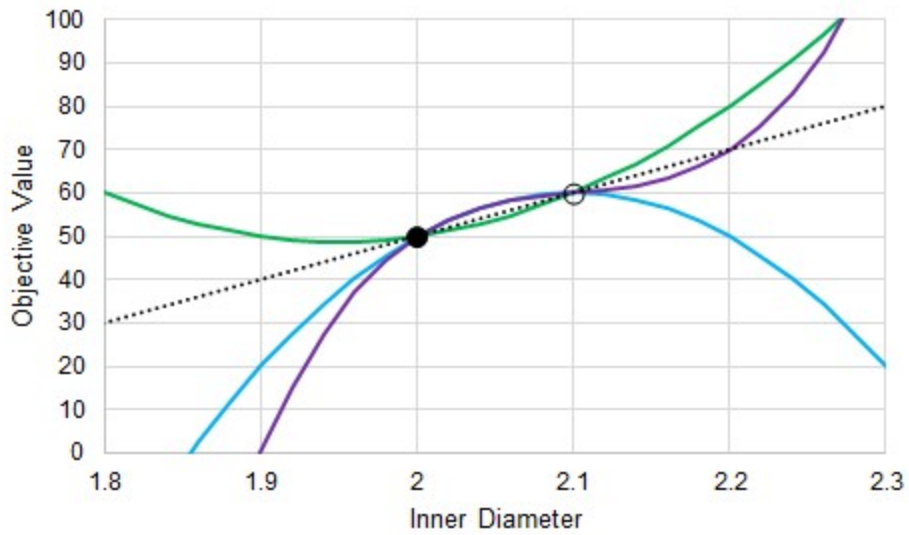


Figure 3: The actual gradient depends on the function. Many possible functions pass through two test points.

Central Difference

The central difference is a very similar process, with generally increased accuracy. We still want to know the gradient at our initial point (Figure 1), but instead of creating one test point, we create two, equidistant from the initial position. This effectively doubles the number of calculations required.

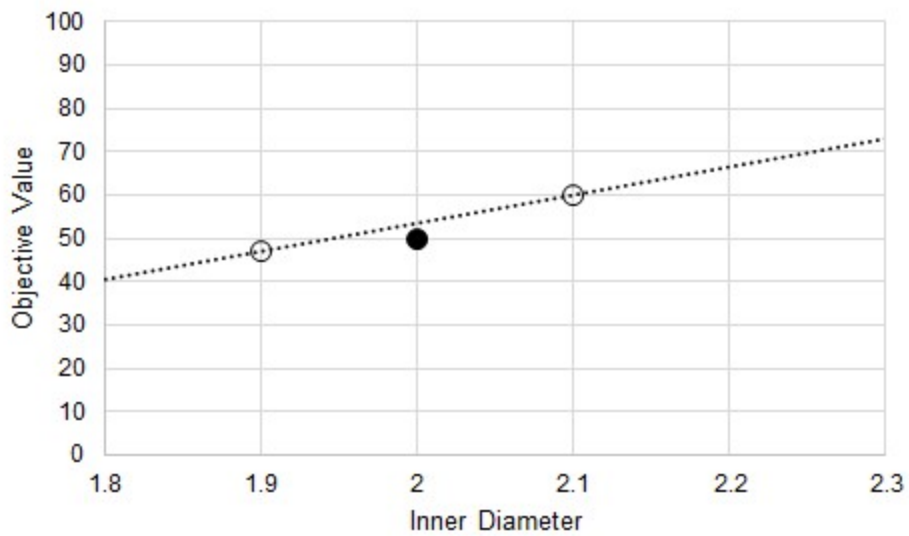


Figure 4: Central Difference estimate of gradient at the initial point

Relationship to Iterative Tolerance

It may seem that simply making the perturbation in the Inner Diameter smaller will increase the accuracy of the gradient and help reduce the issue indicated by Figure 3. If the Objective function was known explicitly, this would be a good assumption. However, in ANS, the results are not known explicitly. Instead, the solution is found via [iteration](#). Iterative methods require some determination of convergence - namely, a *tolerance* on the solution. This means, by definition, that the results found are not the exact, analytical solution. Further, it is impossible to tell how the results may be biased away from the true solution.

When making test points, as before, there is some band of uncertainty in the Objective Value for the point related (but not necessarily equivalent) to the iterative tolerance.

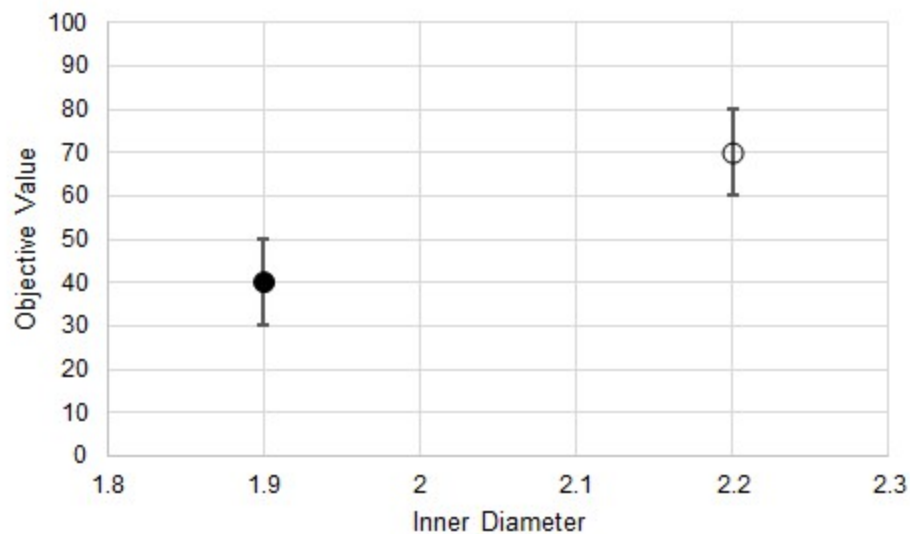


Figure 5: Uncertainty in the Objective Value due to iterative tolerance

We do not know where the "true" Objective Value lands on either of these bands, and we cannot state that the error is in the same direction or is of the same magnitude for both test points. This means that the gradient estimations have some variability as well.

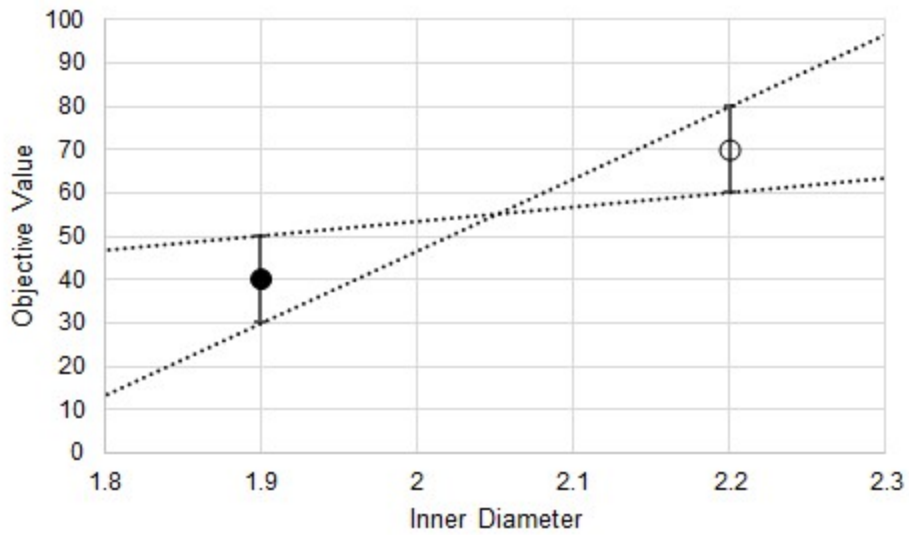


Figure 6: Uncertainty in the estimated gradient

In addition to the uncertainty indicated by Figure 3, there is also added uncertainty from the iterative tolerance.

It can be tempting to reduce the step size to increase gradient accuracy, but the iterative error will not be reduced. This can have dramatic results on the gradient.

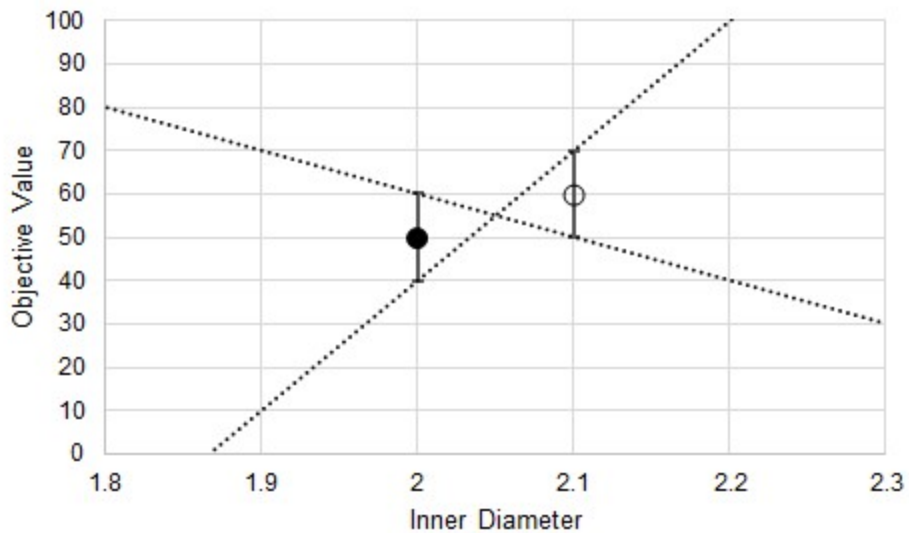


Figure 7: Substantially increased uncertainty in the estimated gradient from decreasing the step size

Now, due to the iterative tolerance, it is possible that the gradient is dramatically affected to the point where even the sign is uncertain.

Note: Making the step sizes too large impacts the gradient negatively in other ways. The critical thing to remember is that the step size should be small enough to capture the changing Objective, but not so small the iterative tolerance impacts the gradient quality.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572
Gradient Based Methods.....	572
Branch and Bound.....	581

Branch and Bound

Piping is almost always only available in Discrete sizes. Determining the best case selection of Discrete sizes requires a different approach than finding the best case Continuous sizes. One such method is **Branch and Bound** which is used for Discrete sizing on the basis of one of the [Gradient Based Methods](#).

Branch and Bound divides the sizing problem into a collection of smaller sizing problems (branches). Solving these "sub-problems" gives information (bounds) about whether or not a given sub-problem is worth further investigation.

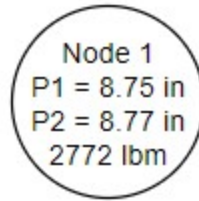
This method is well suited to problems with a relatively small number of variables (unique pipe sizes). For more than 10 variables, the [Sequential Unconstrained](#) method is advised. It is difficult to tell in advance how well any search method will work, and Branch and Bound is no different. It is possible, though unlikely, that the method can effectively reduce to a Brute Force Search. This makes it effective, but potentially very slow.

Example

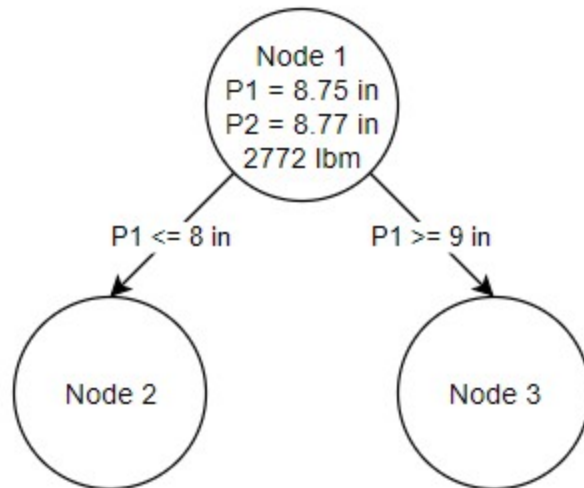
There is a 50 foot pipe followed by a 100 foot pipe, and a 5 psid differential across the pipes. The piping is available in integer inside diameters between 3 and 17 inches, and the pipe walls thickness is always 10% of the inside diameter. Our goal is to minimize the weight of this system, while maintaining a flow of at least 2500 gal/min.

Even though there are only two independent pipe sizes to consider, with 15 possible pipe sizes, there are 225 possible permutations, more than we would like to test by brute force.

We start the process by finding the Continuous Optimum. For this example, this best case is (P1 = 8.75 in, P2 = 8.77 in, Weight = 2772 lbm). It is reasonable to assume that the Discrete Optimum will be near these sizes.

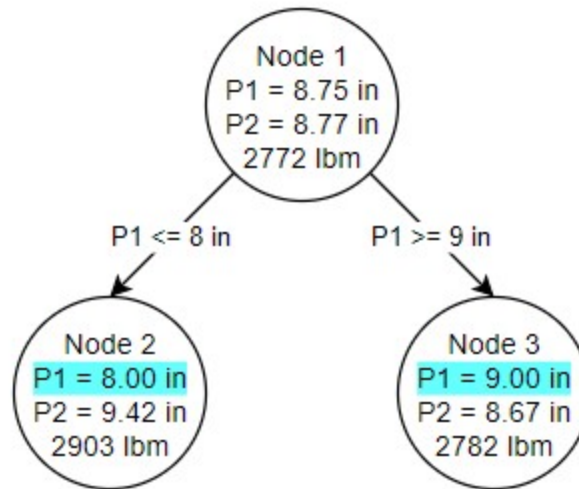


From this first *node*, we want to divide the solution space on our first variable - the diameter of pipe P1. The pipe size cannot be 8.75 inches, it can only be a discrete size, in our case an integer value. Therefore, we *branch* into two sub-problems with pipe P1 restricted to less than 8 or more than 9 inches. We have also established a *bound* on the problem - the solution will never have a feasible weight lower than 2772 lbm.

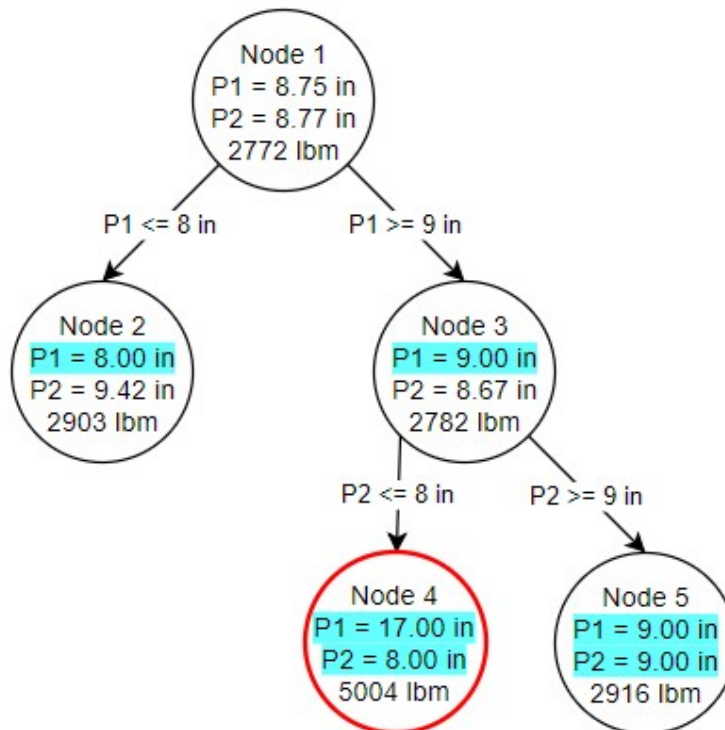


It is important to note that these additional nodes represent **continuous sub-problems**. The pipe size is not forced to be discrete, it only has an additional requirement on it. The resulting pipe size will often be discrete because that continuous diameter is closest to the overall lowest feasible weight.

We can solve each of these independent problems with the same methods we used to solve the first node.

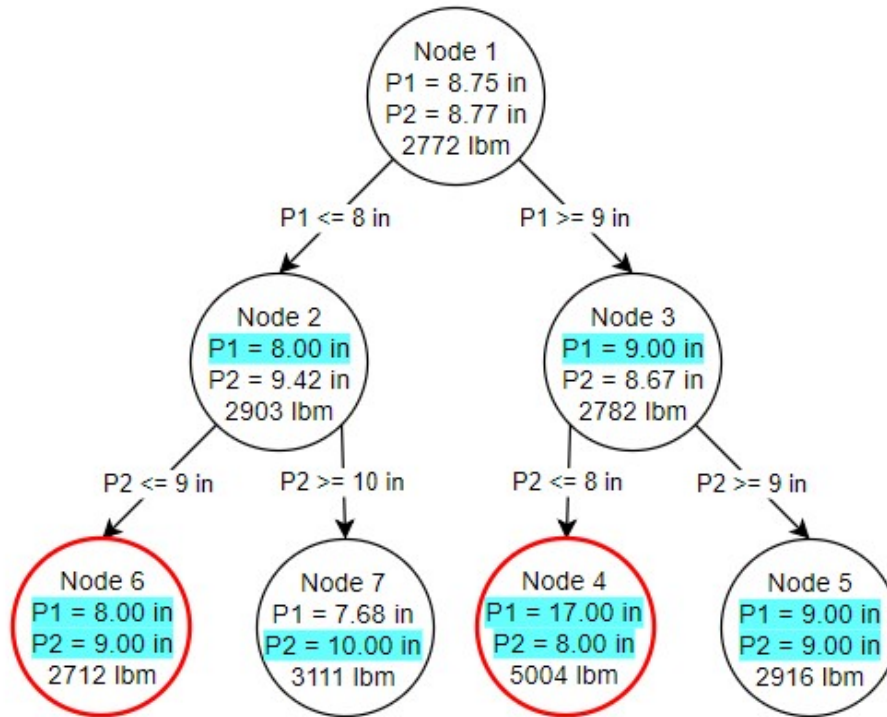


Due to the nature of this particular problem, the ideal pipe sizes for pipe P1 in both Node 2 and Node 3 ended up as discrete sizes. This is not required or enforced by the method. Each of our nodes also has its own bound. This fact is used to determine the location of the next branch. Because Node 3 has a better possible solution (2782 lbm) than Node 2 (2903 lbm), we branch from Node 3. This time, however, we place a restriction on our second pipe's diameter.

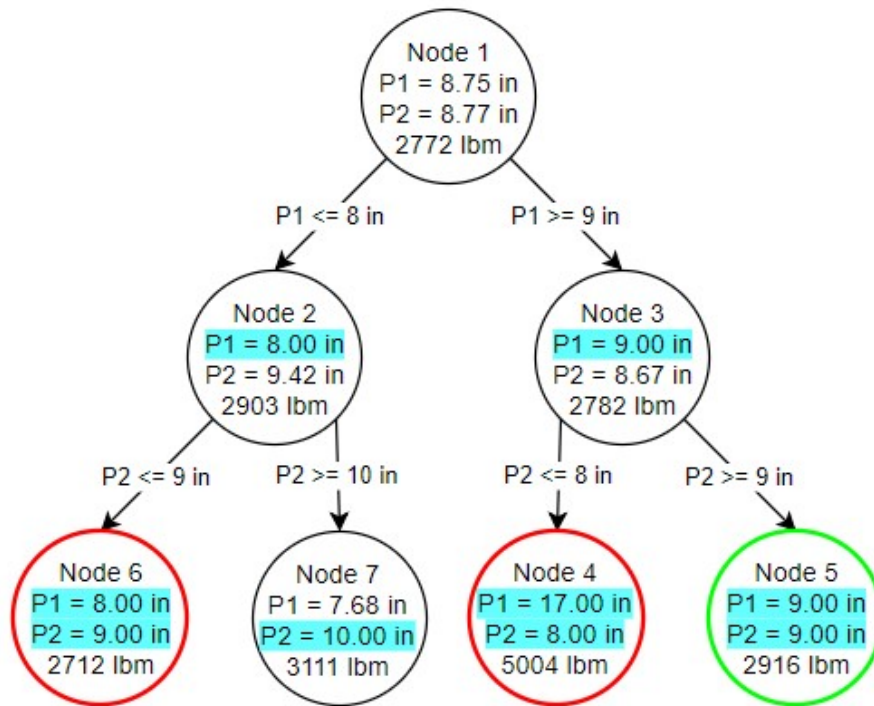


Both of our branches from Node 3 resulted in discrete size selections. However, Node 4 was not able to result in a feasible solution. **Infeasible nodes are valuable as there is no need to branch from them - placing further restrictions on a problem will never make it feasible.** Node 5 on the other hand is both fully discrete and feasible. This is our first **potential discrete**

solution. It is only a potential solution because Node 2 has a lower bound - without branching from Node 2, we do not know if a better discrete solution may exist under it, so branching from Node 2 is exactly how we proceed.



One of our nodes is infeasible, and despite the lower weight cannot be considered. Node 7, however, is feasible but *not* discrete, despite the requirements on diameter on both pipes. We could branch again from Node 7, but there is no need - **we have already found our ideal solution at Node 5**. We know this is the best possible discrete solution because the Node 7 continuous solution has a higher minimum weight than Node 5. No matter how we branch from Node 7, we will never get a better solution than Node 5.



Again, note that the results on Node 5 are a *continuous* solution. It is only because of the requirements we have placed on the problem that the sizes have *resulted* in discrete sizes.

We have managed to find the best possible discrete solution by running seven independent continuous sub-problems, instead of hydraulically solving hundreds of possible configurations. With more independent pipe sizes, the elimination of certain branches often becomes more common, further reducing the cases we need to check.

Related Topics

Sizing Theory.....	567
Sizing and Optimization.....	568
Boundaries vs. Design Requirements.....	569
Physical Parameters vs. Monetary Cost.....	570
Brute Force Method.....	572
Gradient Based Methods.....	572
EstimatingTheGradient.html.....	576

XTS - Extended Time Simulation

What is the XTS module?

An optional add-on module to AFT Arrow which allows users to simulate system transient behavior.

While steady-state modeling answers many design questions, some questions cannot be adequately answered without considering how systems behave over time. Questions such as how long it will take to fill a tank require a dynamic system model. The AFT Arrow XTS module answers such questions.

The XTS module allows you to model transient system behavior. Users can specify the time duration of the simulation, time step size, control system parameters, and how components such as compressors and valves operate over time. Operations such as valve position changes can occur during a specified time schedule, or can occur in response to events in the system thereby simulating control system actions.

Users can also specify tank volume on “finite” tanks, so that tank pressurizing and depressurizing can be simulated based on system pressures and flowrates.

The XTS module is a powerful tool which extends AFT Arrow’s powerful modeling capabilities into the dimension of time.

Note: Transient modeling with XTS does not support heat transfer.

How Does the XTS Module Work?

The XTS module can be described as using a lumped or quasi-steady approximation to time simulation. This means it represents the transient system behavior as a sequence of pseudo-steady-state solutions. In between each pseudo-steady-state solution it adjusts transient parameters, performs mass balances on tanks, and changes component operations as specified by the user.

As a thought experiment, consider a system which you want to simulate for ten minutes in one minute increments. This would require eleven time step solutions (time zero and each minute up to ten). You could manually do this with standard AFT Arrow by running a steady-state model eleven times, and in between each run adjusting the input parameters for the next run based on the results of the previous run. The XTS module automates this manual process.

Besides the automation benefits, the XTS module offers additional benefits such as transient output data management, consolidation and display, and graphing tools to review the transient results.

Using the XTS Module

The user has the option of activating or not activating the XTS module when AFT Arrow first loads. After AFT Arrow is loaded, the XTS module can be activated or deactivated for use from the [Modules panel](#) in the Analysis Setup window or the Tools menu. Whether or not XTS is activated impacts the Analysis menu, Analysis Setup window, Status Bar and multiple other AFT Fathom functions as is discussed in the Modules panel topic.

If the XTS module is active, the user can still run models in Steady Only mode by selecting this under Time Simulation on the Analysis menu. Hence there are three possibilities for XTS:

- XTS is not active
- XTS is active and operated in Steady Only mode
- XTS is active and operated in Transient mode

Types of Transients Modeled

- **Tanks -**
 - Tanks can be modeled as finite tanks with user specified tank volume and initial conditions.
 - Finite reservoirs with known initial conditions allow AFT Arrow to model how tank pressure, temperature, and gas mass change over time to simulate tank pressurization and depressurization and its impact on the flow in the system.
- **Compressor/Fan transients -**
 - [Compressors/Fans](#) can be started or stopped based on [time or event transients](#)
 - Compressor/Fan speed can be changed with time
 - Controlled setpoints can be changed with time
- **Valve transients -**
 - [Valves](#) can be closed or opened based on [time or event transients](#)
- **Control Valve transients -**
 - [Control valve](#) setpoints can be varied based on [time or event transients](#)
- **Other junction transients -**
 - [Spray Discharge](#) junctions can be closed or opened based on [time or event transients](#)
 - [Assigned Flow](#) junctions can have flow and temperature varied based on [time or event transients](#)
 - [Assigned Pressure](#) junctions can have pressure and temperature varied based on [time or event transients](#)
 - [Tank](#) junctions can have liquid level or surface pressure varied based on [time or event transients](#)

- [Heat Exchanger](#) junctions can have a specified heat rate varied based on [time or event transients](#)

Related Topics

Modules Panel.....	78
Add-on Modules.....	489
GSC - Goal Seek & Control.....	489
ANS - Automated Network Sizing.....	500
Transient Control Group.....	99
Simulation Duration Panel.....	99

Related Blogs

[Introducing the New Arrow XTS Module!](#)

[What Is Transient Analysis?](#)

Choosing Time Simulation

([XTS Module Only](#)) When using XTS, the option Time Simulation appears in the Analysis Menu. This drops down to two menu choices: Steady Only and Transient (Figure 1). Users can thus build models with XTS activated that include steady-state and transient simulations.

The Transient option requires the [Transient Control Group](#) to be defined in addition to the other typical settings. The Steady Only option makes the Transient Control Group settings optional.

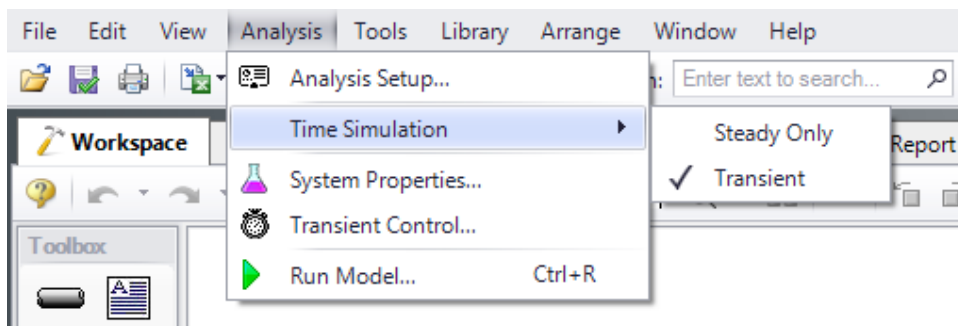


Figure 1: Time Simulation option in the Analysis menu

Junction Transient Data

([XTS Module Only](#)) Many junction types can have active transient data specified. The junction types are listed in Table 1:

Table 1: Junctions that support transient data

Junction Type	Type of Transient
Assigned Flow	Flow rate and temperature
Assigned Pressure	Pressure and temperature
Control Valve	Setpoint
Compressor/Fan	Speed, flowrate, control setpoint
Tank	Temperature, Pressure, Volume, or Heat Rate
Spray Discharge	Flow area
Valve	K , C_v , or open percentage
Heat Exchanger	Heat Rate

The active transient data is entered on the Transient tab (Figure 1). The type of data that can be entered for each junction type is listed in Table 1. If a junction does not have any active transient behavior, there is no need to enter any data on the Transient tab.

In most cases, transient data can be entered as absolute values or as a percentage of steady-state. The distinction is made when choosing between Absolute Values and Relative to Steady-State Value. Some junctions may have additional options, such as the Valve junction (Figure 1).

The first data point always needs to match the steady-state value which is usually entered on the tab at the far left. If, for example in Figure 1, the steady-state data for C_v of 260 is changed to 200, the transient data will also need to be changed. However, if in Figure 1 the Relative option is specified, the first data point would be 100%, and when the C_v is steady-state is changed from 260 to 200 there is no need to change the transient data since the first data point is still 100%. But it is now relative to a different steady-state value.

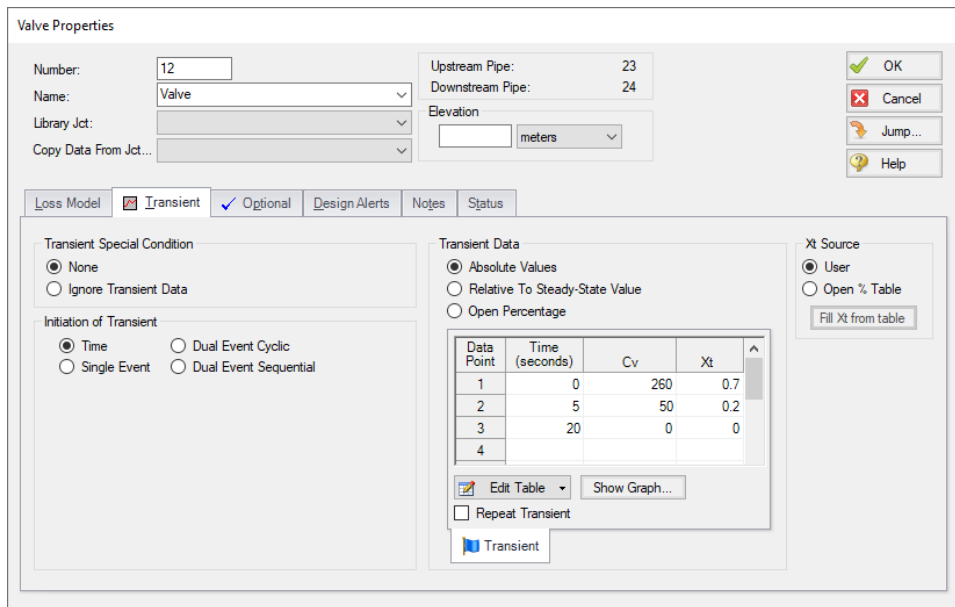


Figure 1: Example of junction transient data

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 Conditions

There may be occasions where, once having entered the transient data, you do not wish the transient to activate for a particular scenario. One option is to just delete the transient data. A second option is to specify the Transient Special Condition as Ignore Transient Data. In such a case, the data can be left in the window for future use, but will not activate during the run.

In Figure 1, there is an area called Initiation of Transient, and the selection is Time. This means that the specified transient is linked to the time of the simulation. Specifically, the valve transient data is activated as soon as the simulation is started. If the transient data was defined to have a constant Cv for one second, the valve does not start to close until the simulation time reaches one second.

Transients can also be initiated based on events. For instance, we can change the valve in Figure 1 to start closing when the pressure at some other system location reaches a certain level. Or maybe when a tank has reached a given pressure. In such a case, the time zero data in Figure 1 does not refer to the start of the simulation, but is relative to when the event occurs. If, for instance, the pressure never reaches the specified value, the valve never closes.

Initiation of Transients

([XTS Module Only](#)) 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.

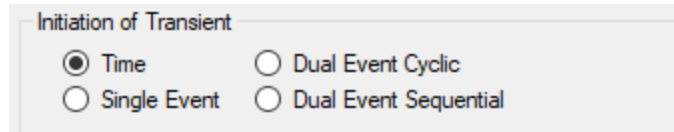


Figure 1: Type of transient initiation

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 [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.

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 pump trip 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.

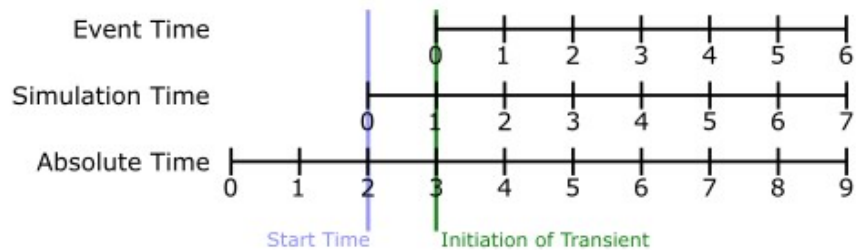


Figure 2: Time basis

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.

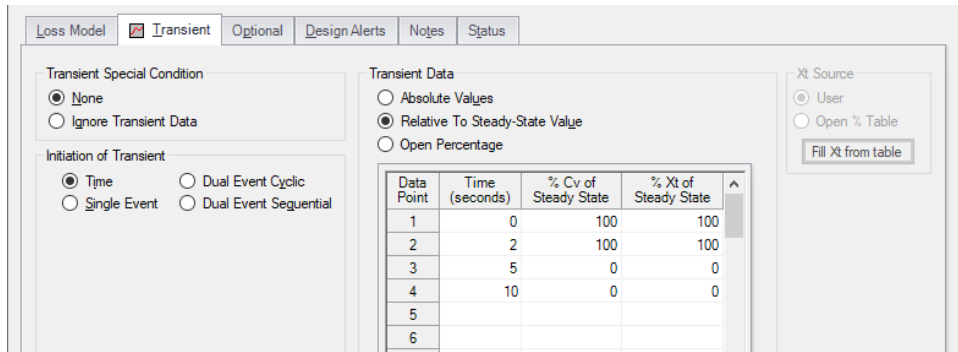


Figure 3: Defining a valve closure at a specific time

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 close when (or if) the pressure at the inlet of P101 exceeds 100 psig. Note that the Transient Data is identical - but the initiation of the event may happen at a very different time.

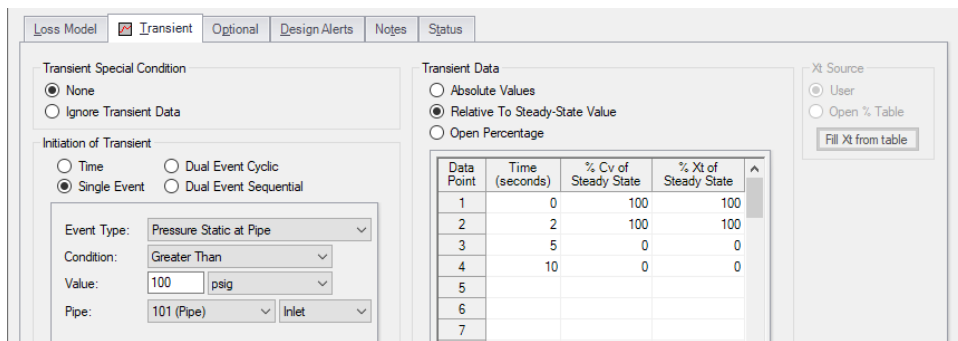


Figure 4: Event definition

Table 1: Event types

** Multi-Condition **	Pressure Stag. Diff. at Junction	Spray Discharge Outflow
Check Valve State	Pressure Stag. Diff. Between Pipes	Tank Pressure
Compressor/Fan Speed	Pressure Stagnation at Pipe	Time Absolute
Control Valve State	Pressure Static at Pipe	Time Difference
Cv	Pressure Static Diff. at Junction	Valve Open Percent
Kv	Pressure Static Diff. Between Pipes	Velocity in Pipe
Mass Flow Rate in Pipe	Relief Valve State	Volumetric Flow Rate in 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 open when a pressure is exceeded, we can tell it to close when the pressure drops below a certain setting.

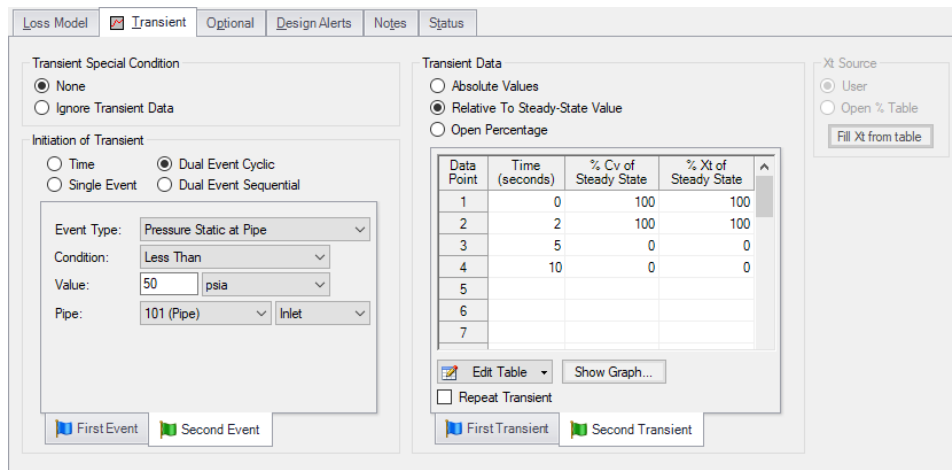


Figure 5: Dual Event set up. Note that a Second Event requires additional data for the Second Transient

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 Arrow 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 two built-in events. The first is that it opens when the set pressure is reached. The second is that it closes again when the pressure falls back below the set pressure. These transients are assumed to be instantaneous unless the Pressure/Pressure Opening/Closing profile is used.

Related Blogs

[And, Not, Or - Starting Transient Events with Complex Logic](#)

Multi-Condition Events

One of the [Event Types](#) available in XTS 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.

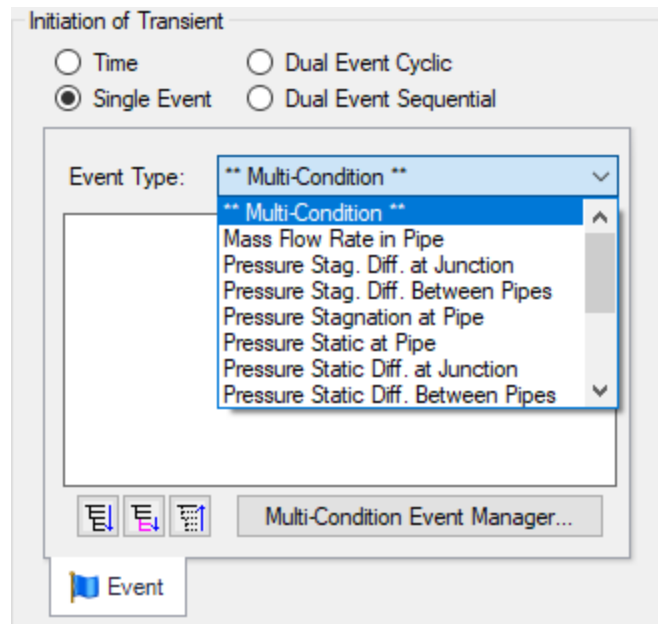


Figure 1: Selecting Multi-Condition Event Type

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.

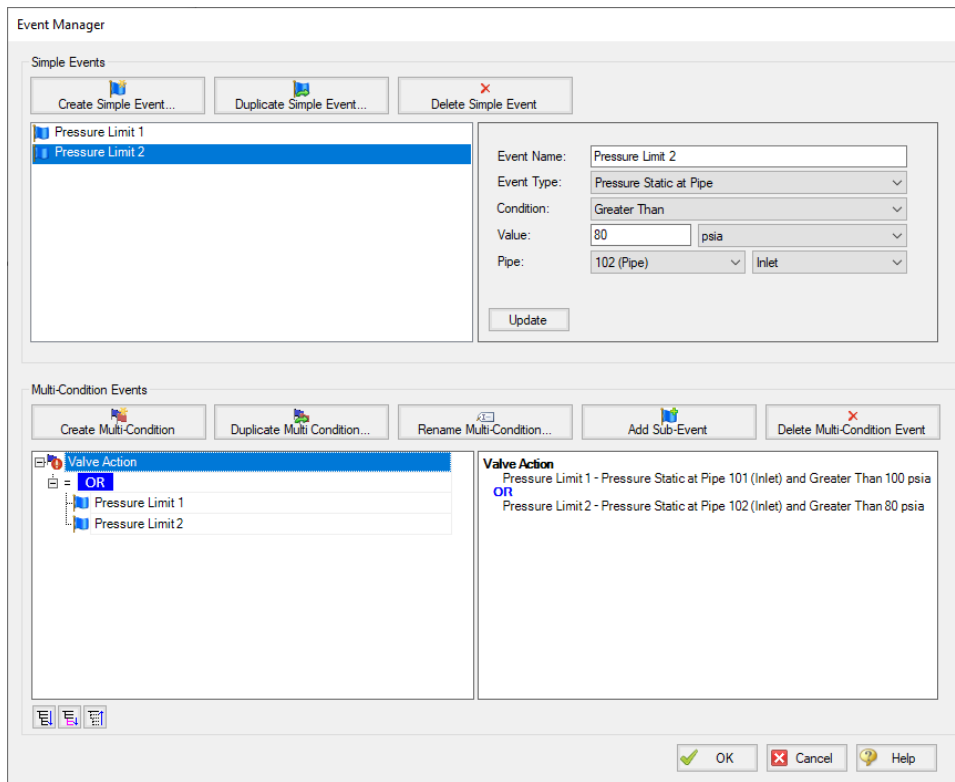


Figure 2: The Event Manager

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:

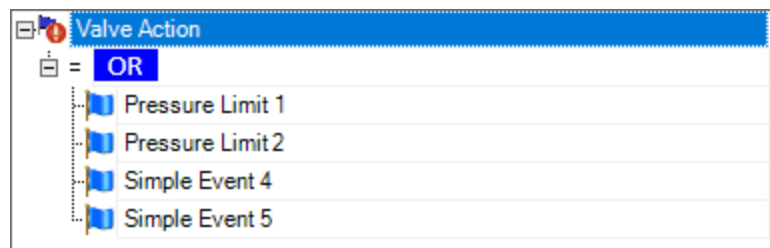


Figure 4: Multiple Events

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.

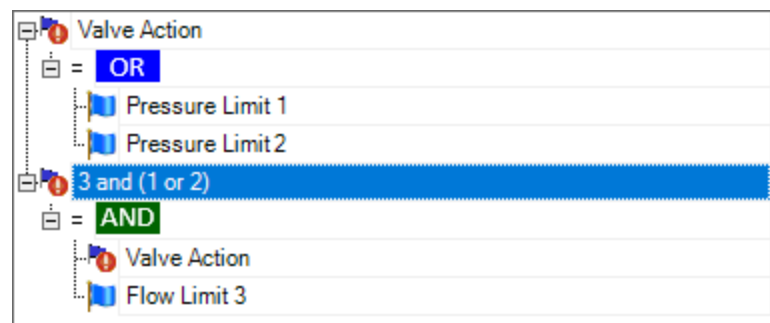


Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))

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.

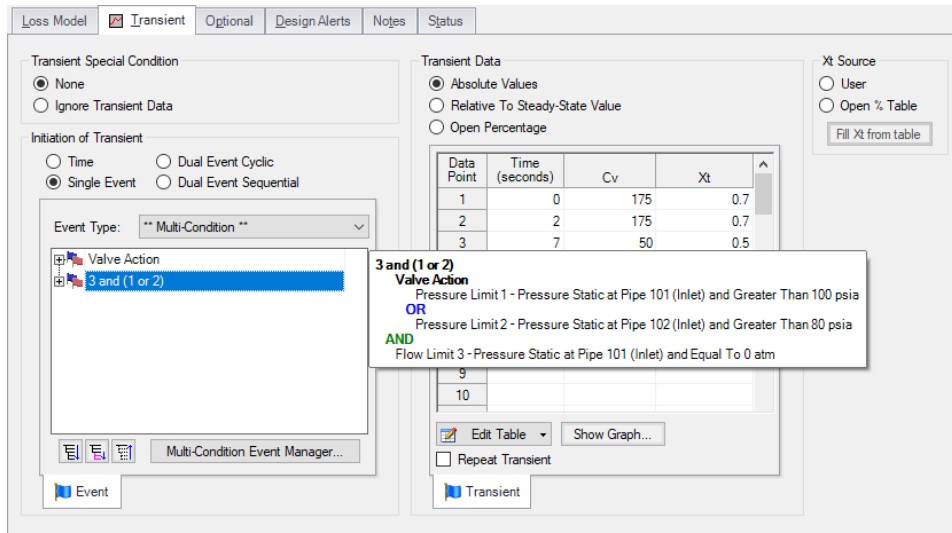


Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event.

Related Blogs

[And, Not, Or - Starting Transient Events with Complex Logic](#)

XTS Special Conditions

(XTS Module Only) Some junctions have [Special Conditions](#) that alter the normal state of the junction.

The junctions that 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](#) window.

Special Conditions with No Transient Data

If a Special Condition is applied to a junction without any transient data, then the transient solution will retain the Special Condition. For instance, if a Special Condition is set for a valve junction, the steady- state solution will solve with the valve closed. In addition, the transient solution

will run the entire simulation with the valve closed as well because there is no transient data entered which implies the valve state is not intended to change.

Special Conditions with Transient Data

If a Special Condition is applied to a [junction that has transient data](#), then the transient solution will ignore the Special Condition. It is then incumbent upon the engineer to relate the Special Condition to the initial transient data.

For instance, assume the user wants to simulate the transient that occurs during a valve opening. The steady-state solution will have the valve closed, and this is modeled by using a Special Condition for the valve junction. In such a case, the C_v data entered on the valve junction's Loss Model tab will be ignored.

On the Transient tab, a transient can be entered. The first data in the table should then be $C_v = 0$, which corresponds to a closed valve. From there the C_v can be increased above zero as desired to open the valve.

If the user were to keep the C_v equal to zero, then that would be equivalent to the previous case where the user set the Special Condition but did not enter any transient data.

Compressor/Fan Special Conditions

Compressor/Fan Special Conditions are slightly more complicated than for valves. In fact, [compressor/fan junctions](#) have two types of Special Conditions. The first is to turn the compressor off and have no flow through it. This is called “**Compressor Off No Flow**”. The second type of Special Condition turns the compressor off but allows flow to go through the compressor. This is called “**Compressor Off With Flow Through**”.

Why would one want to use one Special Condition rather than the other? The simple answer is that the first Special Condition is more appropriate for positive displacement type compressors and the second for centrifugal compressors or fans.

Here's why. When a positive displacement compressor is turned off and has a pressure difference across it, it will usually act like a closed valve and not allow flow to go through it. Thus the first Special Condition would be most appropriate. For instance, assume one wants to model the transient that occurs during the startup of a positive displacement compressor. The best way to do this would be to use the first Special Condition with no flow, and then input a flowrate transient in which the first data point is zero flow.

On the other hand, when a centrifugal compressor or fan is turned off and has a pressure difference across it, in the absence of other valving which prevents flow the compressor or fan will usually allow flow to go through it. Thus the Special Condition that allows through flow would be most appropriate. For example, assume one wants to model the transient during the startup of a centrifugal compressor. One would use the second Special Condition, and, in conjunction with a compressor curve entered at 100% speed, input a speed transient with the initial speed as zero.

Transient Special Conditions

Junctions which accept transient data also support Transient Special Conditions. The default for all junctions is None. The other choice is Ignore Transient Data. This allows one to simulation the static junction without having to delete the transient data.

When a Transient Special Condition is set to Ignore, a # symbol is displayed adjacent to the junction number on the Workspace. This symbol can be customized in the [User Options](#) window.

XTS in Workspace

([XTS Module Only](#)) The transient parameter labels on the [Workspace Layers](#) will change dynamically to be displayed at a selected time. The slider bar and buttons at the bottom of the Workspace function the same as the [Output window time selection controls](#). When the user changes the current time, displayed transient parameters are changed as well. In addition, color maps are updated when in use.

As shown in Figure 1, the current time is displayed in the bottom-left by the time adjustment buttons. The time can also be optionally displayed next to the model as well. The time display on the Workspace can be turned on and off within the [Global Layer Settings](#) window. It is helpful to have the time on the Workspace when it is printed.

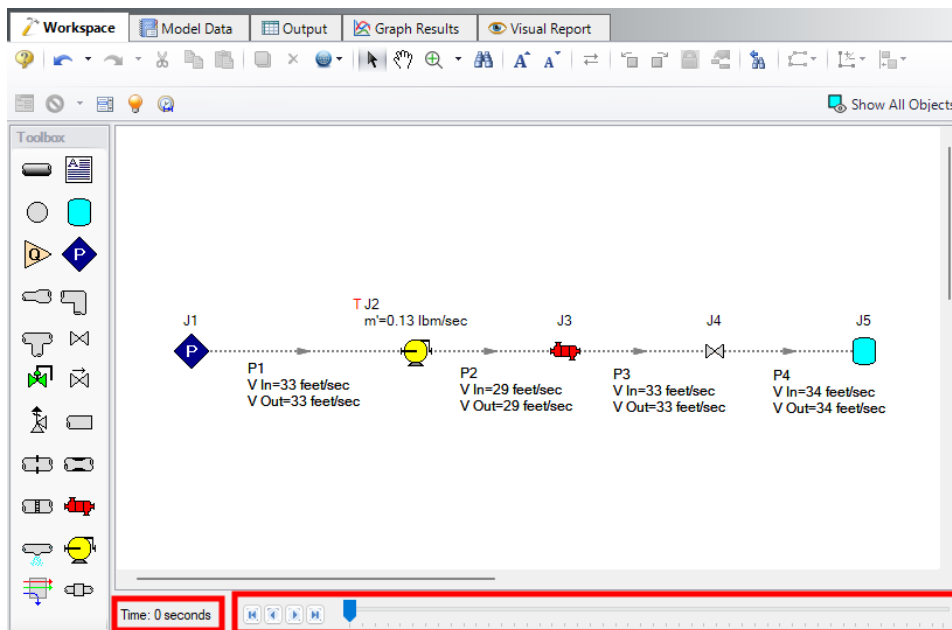


Figure 1: The Workspace has additional time adjustment controls when the XTS Module is in Transient mode

XTS in Model Data

([XTS Module Only](#)) Junction transient data is displayed on the Transient tab in the Junction section of the Model Data window (Figure 1). This tab is only visible if transient input data exists.

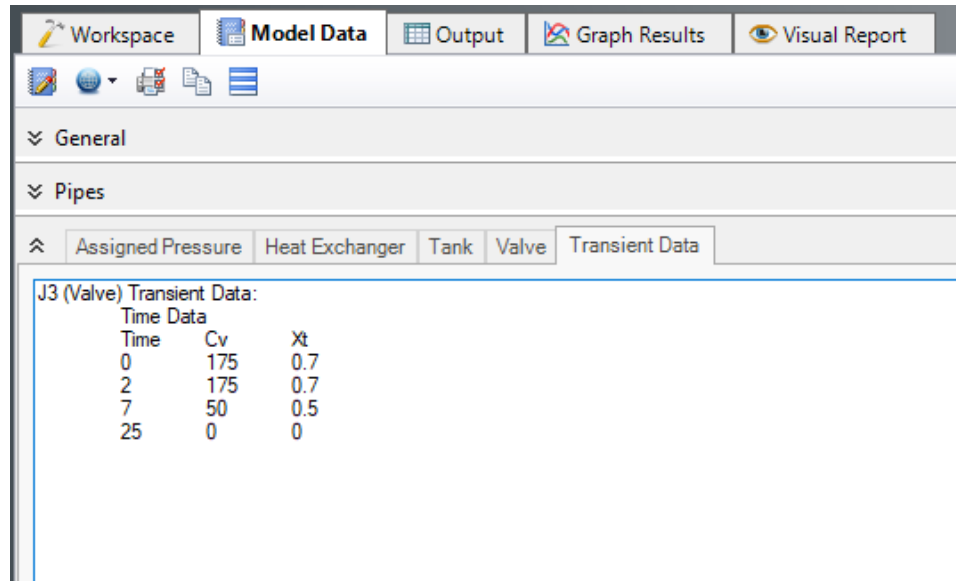


Figure 1: Transient input data is displayed in the Junction section of the Model Data window

XTS in Output

([XTS Module Only](#)) Transient simulations create more output data than steady-state runs. The transient output data is displayed in the [Output window](#) in two general ways.

First, the pipe and junction output tables now reflect output at a specific point in time. In Figure 1 below the displayed results are 10 seconds into the simulation. The time step information can be seen at the bottom left of Figure 1. The forward/reverse buttons and the slider at the bottom allow the user to change the currently displayed time.

Second, the results for all time steps are displayed together in the tables labeled "Transient". Figure 2 shows the same output as Figure 1, except the transient tank and pipe tabs have been selected for display. Here one sees multiple results for each pipe or junction. For example, look at the transient pipe data in the bottom tables. While the table holds data for all pipes, one can see here only part of the data (up to 3 minutes) for the first pipe, Ppipe 1. In the far left column, the first data is the pipe number (#1) and the adjacent number in parentheses is the time for that row (in the units defined for the time step interval - seconds in this case). The time unit of seconds is displayed in the bottom left, and can be changed in the [Output Control](#) time formatting. The data in Figure 2 shows these changes over each time step. The transient tables are organized with a collapsing/expanding capability for each pipe or junction to facilitate data viewing.

Notice that each of the junction Summary reports at the top of Figures 1 and 2 (in the General Section, see Pump, Valve and Reservoir Summary) have a related Transient table that goes with it.

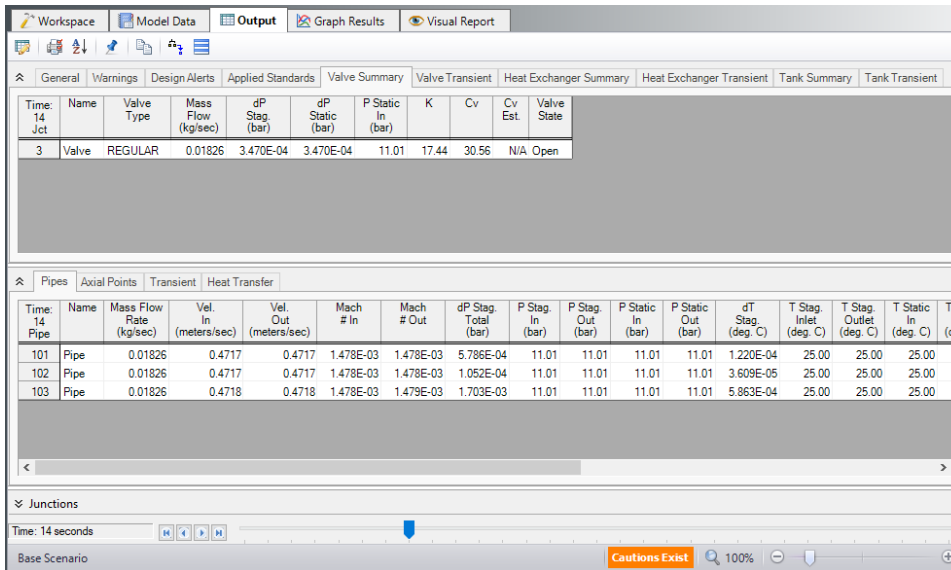


Figure 1: Output window displaying data at 10 minutes

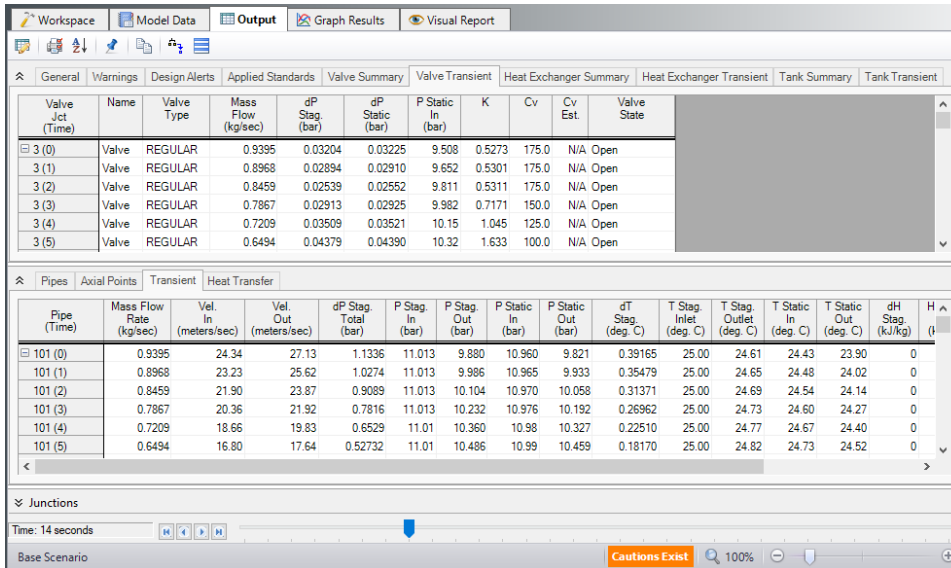


Figure 2: Output window displaying transient data for entire simulation

Related Topics

XTS Quick Graph..... 609

XTS in Graph Results

([XTS Module Only](#)) Transient data can graphed in the [Graph Results](#) window in two general ways: Profile graphs and transient graphs.

Profile Graphs

Profile graphs show results where the distance along the pipe is the independent, x parameter (Figure 1). If the pipes are connected, a sequence of pipes can be graphed in the profile. Profile graphs can only show data at a particular time or show overall maximum and minimum values for all times (Figure 2).

Profile graphs can be created for one or more pipe sequences. Note that the x-axis on profile plots is cumulative distance along the pipe sequence.

Multiple profiles can be plotted simultaneously using the [Groups](#) on the Edit menu.

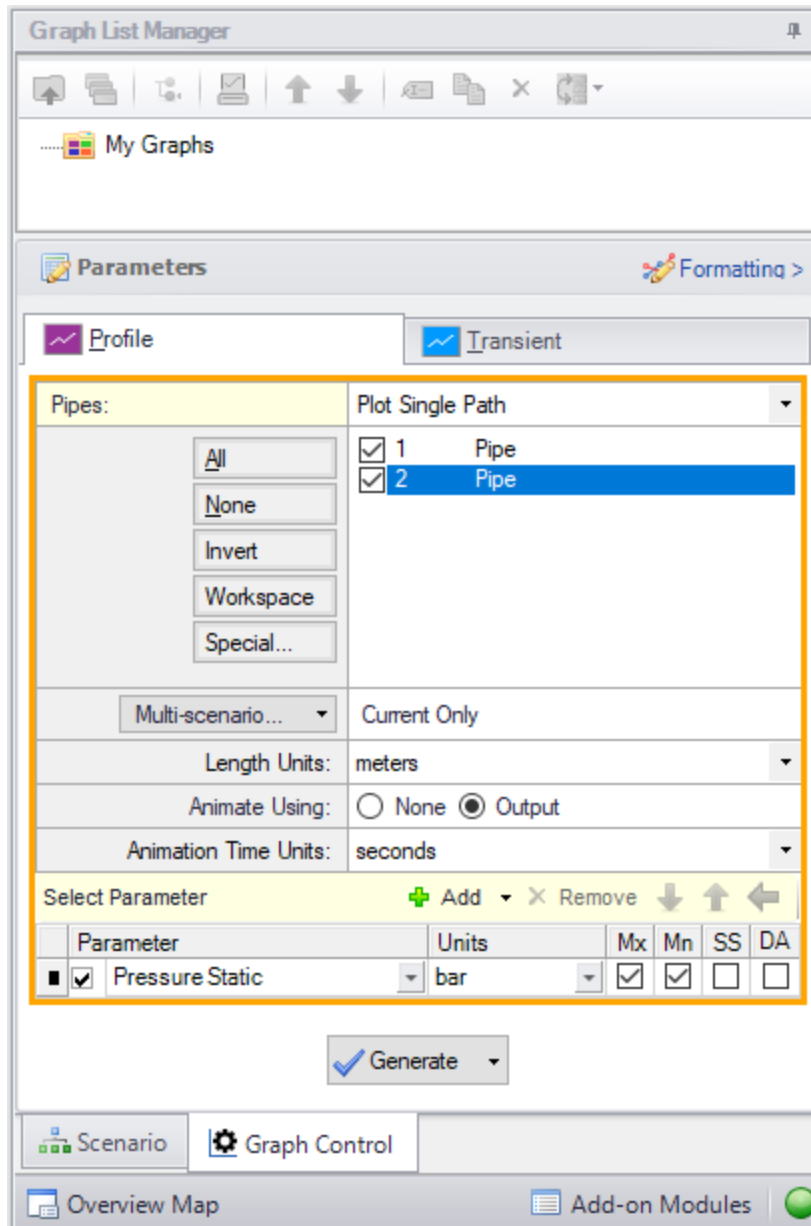


Figure 1: The Graphing panel showing a profile plot for a pipe sequence for maximum and minimum static pressure is to be generated

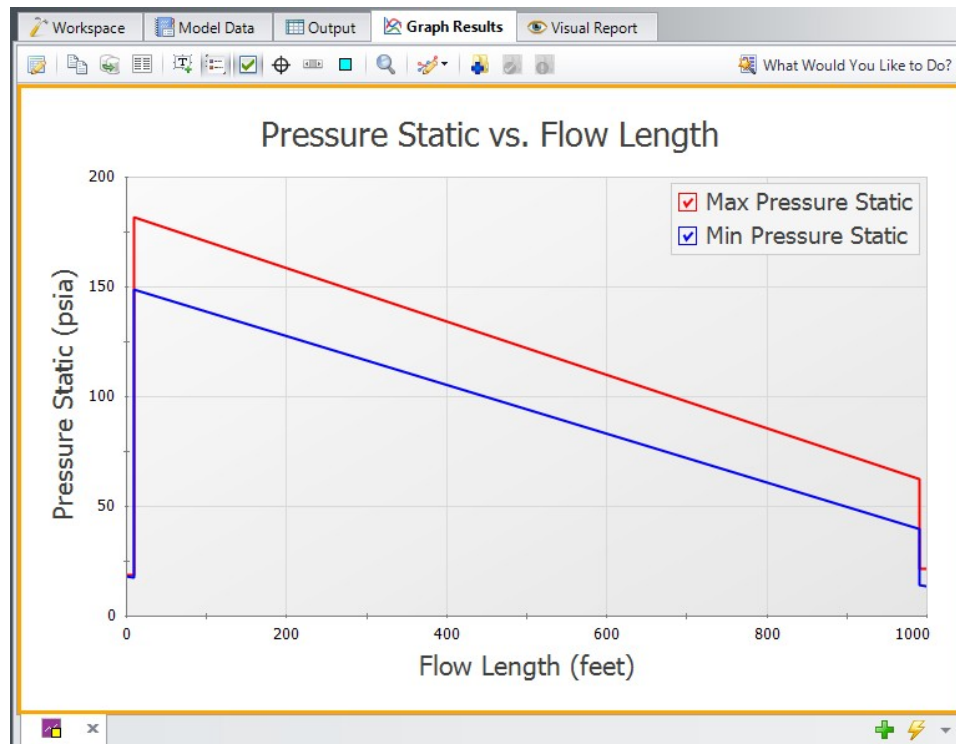


Figure 2: Transient max/min profile plot for pressure as specified in Figure 1

Animation of Output

Another useful feature of profile graphs is that they allow animation of the output results. Animation is selected in the Quick Access Panel (Figure 1) and, when selected, reveals additional control features on the Graph Results window. See [Animating Transient Results](#) for more information.

Transient Graphs

Transient graphs show how parameters vary with time (see Figures 3 and 4). Here the x-axis is time

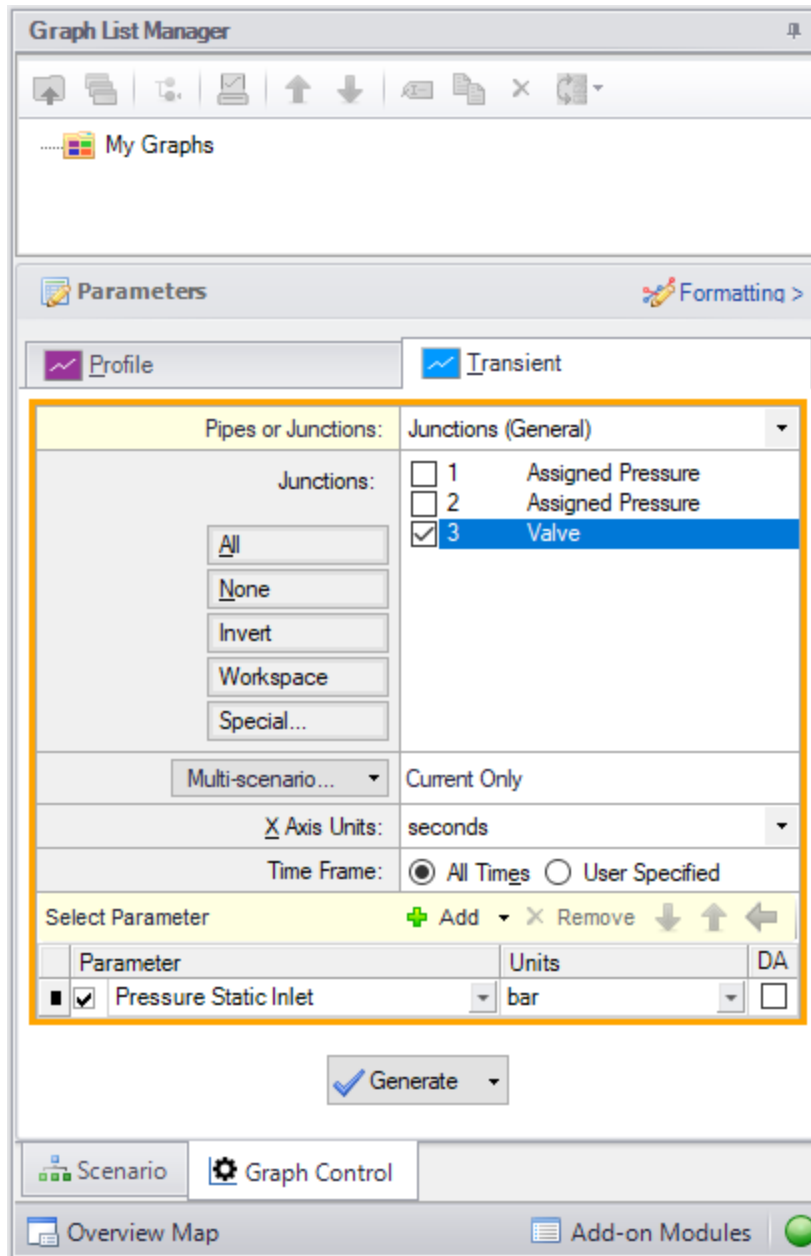


Figure 3: Setting up a Transient graph

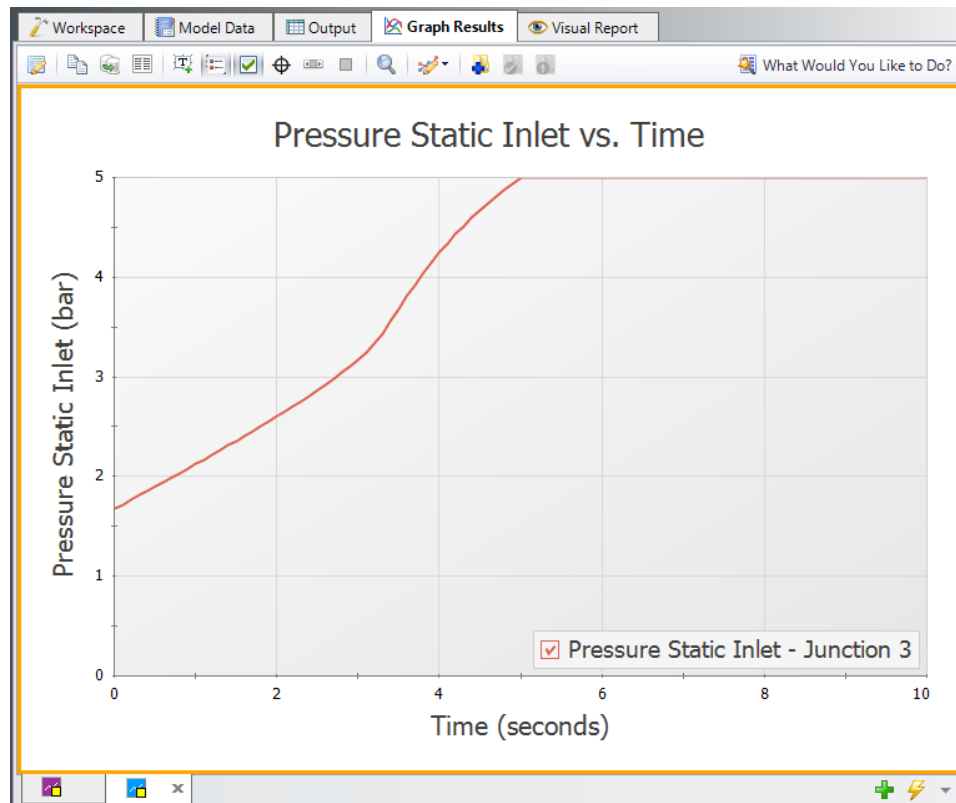


Figure 4: Transient graphs show how parameters vary with time

XTS Animating Results

Figure 1 shows the [Graph Results window](#) when using animation. As shown, additional controls become visible to allow the user to adjust animation. At the left is the Play button, which starts the animation. Next to it are the Pause and Stop buttons. If the Pause button is pushed, the animation stops at that time step. One can print the graph or copy the data if desired. The time slider controls (to the right of the Stop button) can also be used to set the time forward or backwards. The Stop button stops the transient and resets the time to zero.

The current time (in seconds) and time step are shown to the right of the time controls. At the far right are the speed controls. Slowing the animation involves adding time delays to the data display. Increasing the speed involves skipping time steps. The default speed, right in the middle, has no delays or skips.

If plotting a single path using the list of pipes in Graph Control (Figure 2), one can cross-plot on the animation design alerts (in Profile Along a Flow Path) or steady-state results (in EGL, HGL and Elevation Profile).

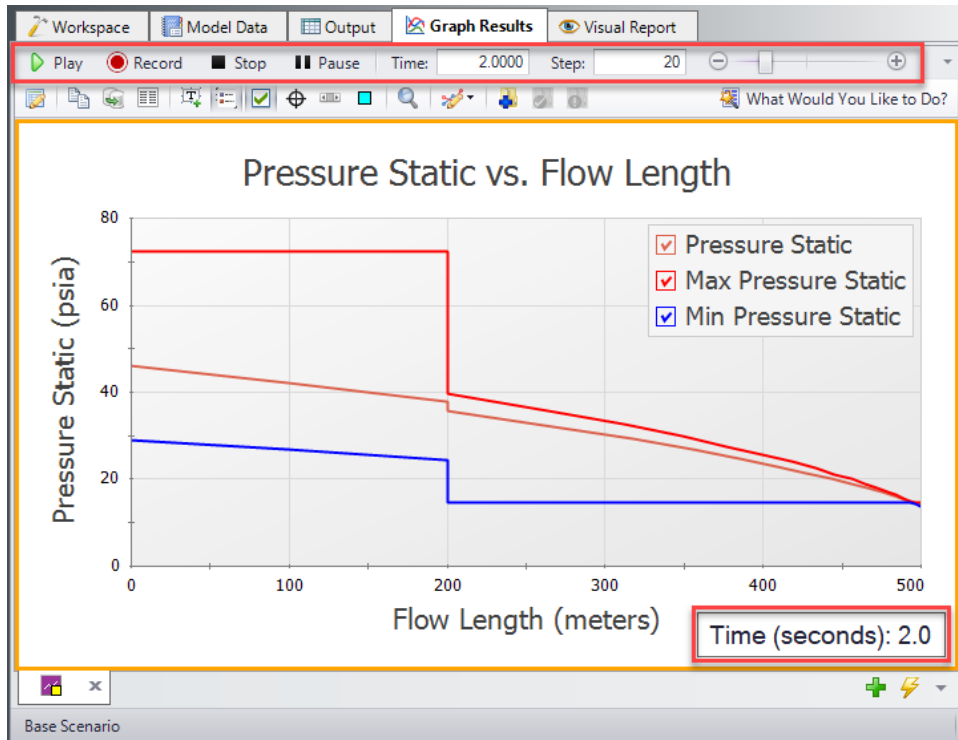


Figure 1: The animation options display additional controls

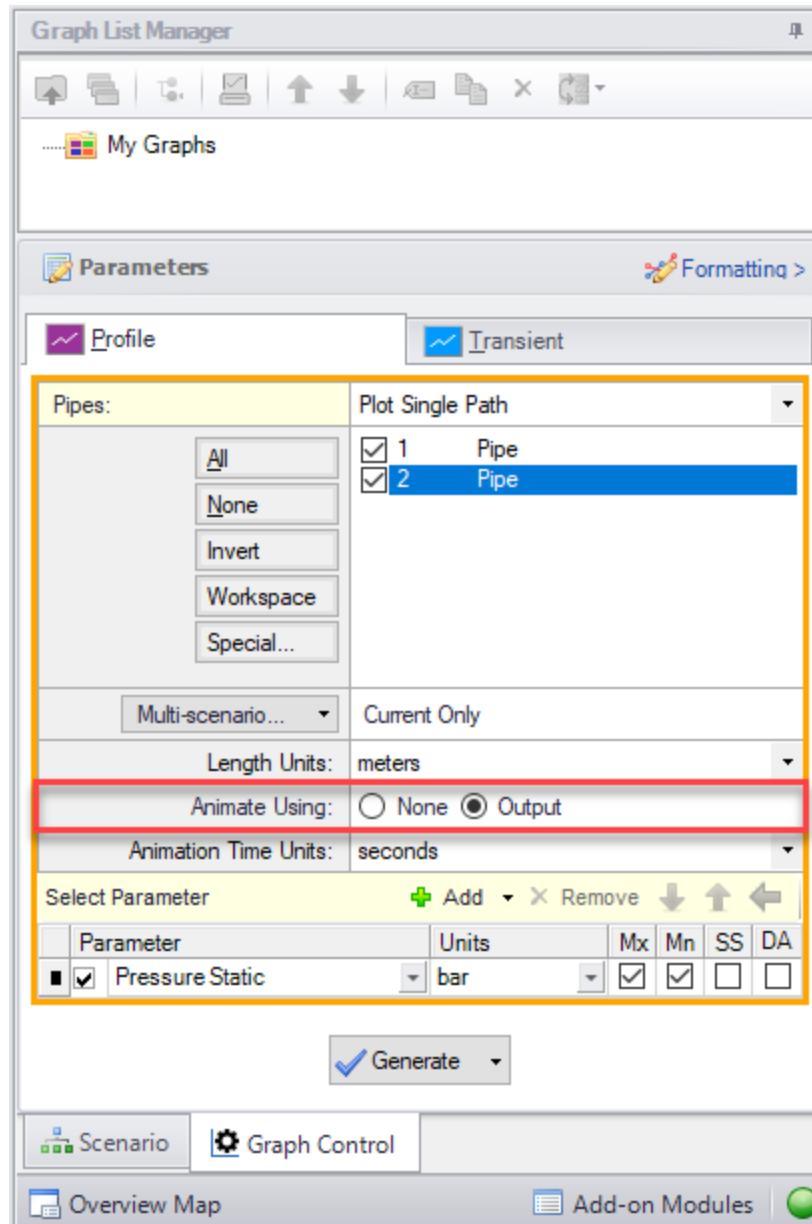


Figure 2: Enabling Animation in Graph Control

XTS Quick Graph

(XTS Module Only) [Transient output data](#) displayed in the [Output window](#) can be graphed by right-clicking on the data in the table and selecting Quick Graph (see Figures 1 and 2).

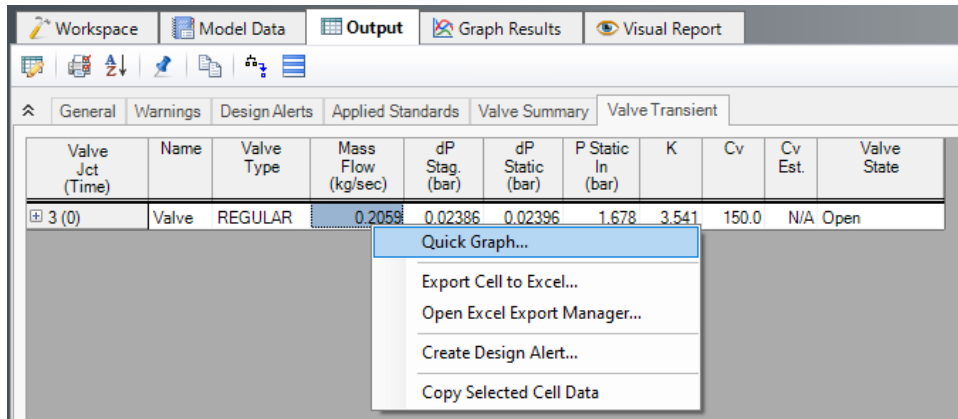


Figure 1: Quick Graph on the Output window is selected by clicking the with the right mouse button in a column of transient data

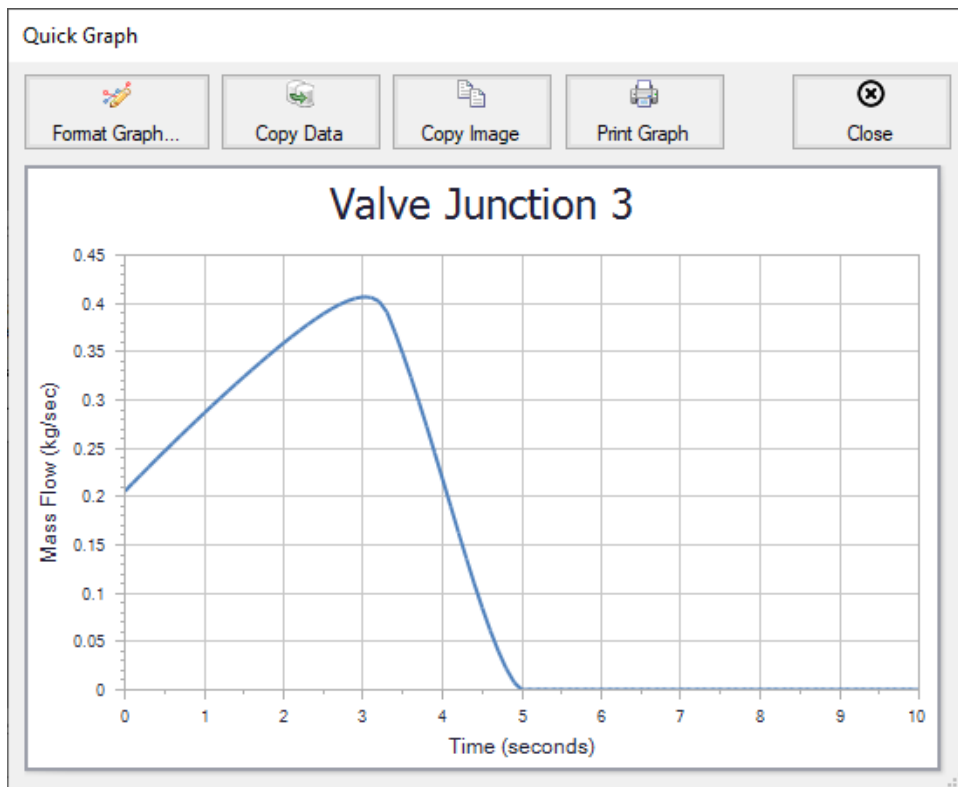


Figure 2: Quick Graph window is opened from Output window as shown in Figure 1

Related Topics

XTS in Output.....601

Quick Graph Formatting Window.....378

Libraries

Powerful customization capabilities are provided through the ability to create libraries which can store custom fluid properties, pipe materials, and all junction types except [Dead End](#) and [Tee/Wye](#) junctions. 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** - Arrow#.dat (where # is the corresponding version number).
- **Location** - Arrow installation folder, default is C:\AFT Products\AFT Arrow #.
- **Limitations** - Cannot be transferred to other users, local use only.

Local User Library

- **Purpose** - Stores user-specified data for most component types.
- **Filename** - Aro_User#.DAT (where # is the corresponding version number).
- **Location** - C:\Users\username\AppData\Roaming\Applied Flow Technology\AFT Arrow\#.
- **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.

Related Topics

Library Manager.....	614
Library Browser.....	614
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636

Fluid Library.....	619
Pipe Material Library.....	620
Pipe Material Library References.....	703
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624

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

Related Topics

Library Browser.....	614
Fluid Library.....	619
Pipe Material Library.....	620
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Libraries.....	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

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.

Types of Libraries

There are three types of external libraries in AFT Arrow:

- [Engineering Library](#) - Contain all component types except cost information
- [Energy Cost Library](#) - Contains energy cost data associated with components in existing engineering libraries
- [Cost Library](#) - Contains material, installation, and/or maintenance cost data associated with components in existing engineering libraries

All library types can be reviewed, added to the model, and connected to the model from the Library Browser. For engineering-type libraries additional actions are available to [edit libraries](#).

Adding and Removing Existing Libraries

To work with a library you must first make it available to AFT Arrow. Referring to Figure 1, clicking on the Add Existing Library or Add Existing Cost 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 ARROW [LOCAL USER LIBRARY](#) may not be removed and will always appear in the list of available libraries. External shared libraries will also always appear in the available library list.

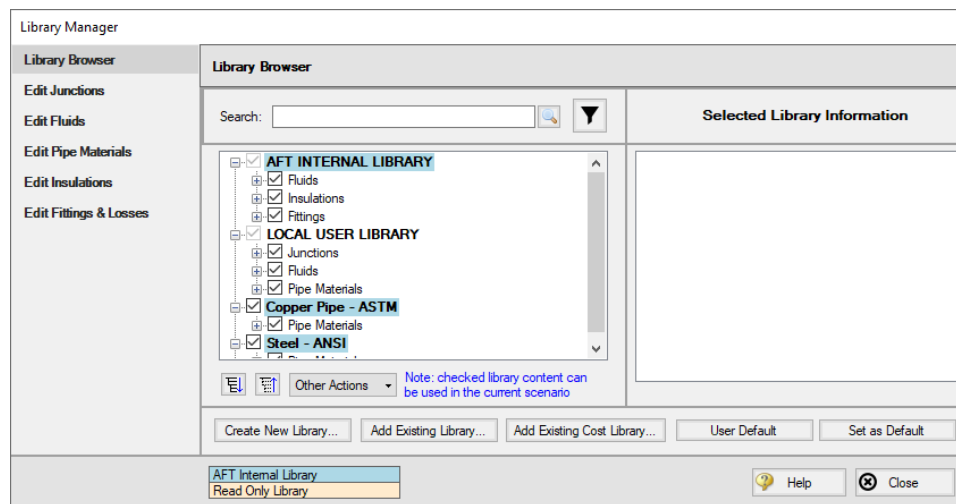


Figure 1: Library Browser in the Library Manager

Connecting and Disconnecting Libraries

In order to use the components from a library in the model the library must be **connected**. A library is connected to the model when it is shown with a check mark in the Library Browser. All libraries are automatically connected when they are first added to the Library Manager.

Sections or components in a library can also be connected/disconnected on an individual basis if the library as a whole is connected.

It may be desired to disconnect a library if you do not want the components from that library to be available for use in the current model, but you do want the library to be available for use in other models.

Clicking on Set As Default will establish the list of connected libraries as the default list, while clicking on User Default will set the connections to the default list.

AFT Internal Pipe Material libraries can be connected or disconnected as a group from the Other Actions button under the available libraries list.

Editing Library Contents

Existing content can be moved to/from, copied to/from, or deleted from available libraries in the Library Browser.

To edit an item right-click on the item in the Library Browser, then click the desired action from the right-click menu. Note that moving, copying, and deleting can be done to a library as a whole, or can be done for just a section or single component in a library. Individual components can also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.

Note: Editing Cost libraries must be done separately in the [Cost Library](#) or [Energy Cost Library](#) windows, accessed from the Library Menu.

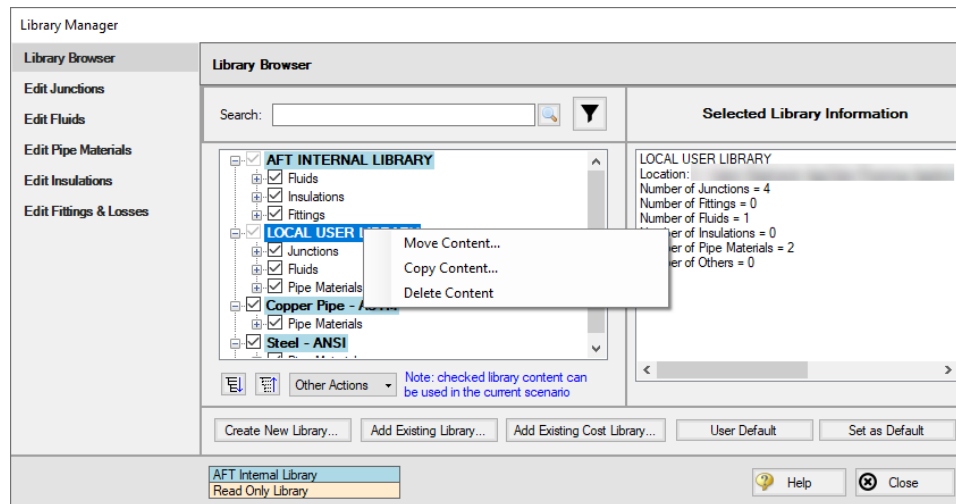


Figure 2: Library contents can be transferred or deleted from the right-click menu

Creating External Libraries

By default library items are stored in the AFT Default Internal Library or Local User Library, both of which cannot be shared with other users. Library items can be moved or copied to an external library which can be shared with multiple users.

To create an external library click the Create New Library button underneath the available libraries list. You will be prompted to provide a name for the file, and a library description (Figure 3). The provided description will be displayed as the library's name in the Library Manager.

To add content to an external library copy or move content from an existing library as described in the Editing Library Contents section. Junction components can also be directly added to external libraries as is described in the [Junction Libraries](#) topic.

For more details on creating and sharing external libraries see [Creating an Enterprise-wide Network Library](#).

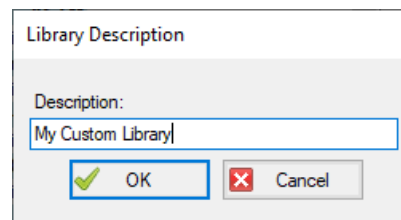


Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected

Related Topics

Library Manager.....	614
Fluid Library.....	619
Pipe Material Library.....	620
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Libraries.....	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

Junction Library

Adding Custom Junction Data to the Component Library

To simplify modeling and eliminate repetitive data entry, AFT Arrow allows you to save frequently-used junction data to a custom library. Once saved, the custom equipment data 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 pumps, 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 junction. Enter a descriptive name then click OK. If you open the junction again, the custom name will be selected in the Library Jct dropdown.

AFT Arrow accepts custom data for all junction types except [Dead End](#) and [Tee/Wye](#) junctions.

Editing the Component Library

You can delete or rename any existing library component in the Edit Junctions panel of the Library Manager, accessed from the Library menu. To edit an existing library entry, select the item in the list and click the Edit Data button. This opens the data into a Properties Window where the data can be modified.

Related Topics

Library Manager.....	614
Library Browser.....	614
Fluid Library.....	619
Pipe Material Library.....	620
Insulation Library.....	623
Fittings & Losses Library.....	624

Libraries	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

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 Arrow 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

Library Manager.....	614
Library Browser.....	614
Pipe Material Library.....	620
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Libraries.....	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

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 Arrow 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, 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.

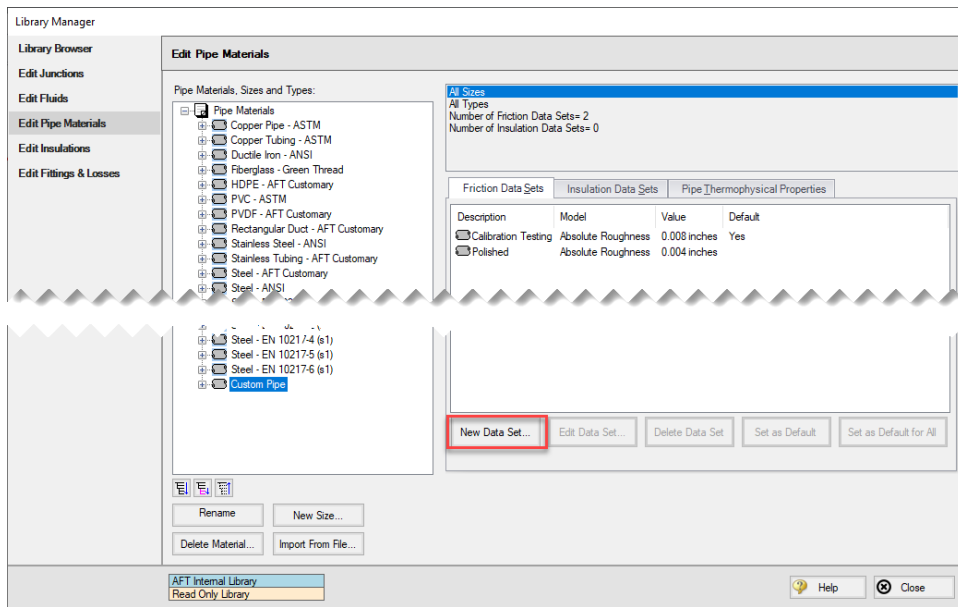


Figure 1: Friction Data Set tab on the Edit Pipe Materials panel

Insulation Data Sets

Insulation data sets are similar to friction data sets. An insulation data set can be associated with the entire pipe material, or with specific sizes or types. Insulation data sets are based on insulation data that has been specified in the [Insulation Library](#). An insulation type must be defined first in the Insulation Library before it can be used for an Insulation Data Set.

To create an insulation data set select the pipe material, schedule, or type that will be associated with the data set in the list on the left, then click the Insulation Data Set tab on the right and select the New Data Set button. Modify the Insulation Data Set description to provide a name, then choose an Insulation Library from the drop-down and define an Insulation thickness value. Once an insulation Data Set is defined it can be selected for use in the [Pipe Properties window Heat Transfer tab](#).

As an example, an insulation data set associated with Steel (i.e., the top level) might be based on insulation data in the Insulation Library called "Denver Heating Company K-Series". You might assign it a thickness of 1 inch and name it "Colorado Standard". Once entered, all steel pipe sizes would have a Colorado Standard insulation data set based on Denver Heating Company K-Series insulation that is one inch thick.

In practice, different thickness would likely be associated with different pipe sizes. To accomplish this, you could create insulation data sets for pipe one inch or less, call them "Colorado Standard", and make them ¼ inch thick. Pipes 1-4 inches could also have "Colorado Standard" sets, but would use ½ inch insulation, and pipes larger than 4 inch could have "Colorado Standard" and be specified as 1 inch thick. Then in the Pipe Properties Window you can choose "Colorado Standard" insulation, and the appropriate thickness for the size you have chosen would be used.

Finally, among multiple insulation data sets you can specify a default data set.

Pipe Thermophysical Properties

The Edit Pipe Materials panel allows you to define thermophysical properties for the pipe. The density of the pipe can be entered as a constant, and the thermal conductivity as a function of temperature. The thermal conductivity is used in [heat transfer calculations for pipes](#).

To add thermophysical properties select a Pipe Material to associate the data set with, then click Specify Properties from the Pipe Thermophysical Properties tab on the right. The Pipe Physical Properties window will be shown where the thermal conductivity data can be entered on the Pipe Material Property Data tab, and the density can be entered on the Density Data tab. A density calculator is provided to calculate the density using the pipe weight and dimensions.

Related Topics

Library Manager.....	614
Library Browser.....	614
Fluid Library.....	619
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Libraries.....	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

Insulation Library

The Edit Insulations panel is accessed from the [Library menu](#). This panel allows you to view the insulation data provided with AFT Arrow, 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 Topics

Library Manager.....	614
Library Browser.....	614
Fluid Library.....	619
Pipe Material Library.....	620

Junction Library..... 618
 Fittings & Losses Library..... 624
 Libraries..... 613
 Network Library Overview..... 629
 Library Verification..... 634
 Library Locations in General Preferences..... 636
 Pipe Material Library References..... 703

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 Arrow, 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 Browser](#)).

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, Arrow 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	
3								

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		
r/D=12				
r/D=14				
r/D=16				
r/D=20				
Valve	Angle	Type 1	Angle (degrees) Open Percentage L/D Pipe Size (nominal) None	K (constant) k / fT (Crane)
		Type 2		
		Direct flow		
	General			
	Ball	None		
	Butterfly	Plane disk - AR=0.92		
Plane disk				
t/D=0.1				
t/D=0.15				
t/D=0.2				

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

		W/D=.07 No Seat Width		
		Through flow Branch flow		
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		
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. Venturi 3 deg. Dall Tube	Area Ratio None	K (constant) k / fT (Crane)
	Honeycomb cell	L/D=2 L/D=4 L/D=6 L/D=8		

		L/D=10 L/D=12		
	Screen	Perforated Plate Circular wire Silk thread Round wire or netting		
	Tee	Through flow (dead branch) Branch flow (dead run)		
Flange	Steel	150 lb. 300 lb.	None	N/A

Related Topics

Fittings & Losses.....	194
Library Manager.....	614
Library Browser.....	614
Fluid Library.....	619
Pipe Material Library.....	620
Junction Library.....	618
Insulation Library.....	623
Libraries.....	613
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636
Pipe Material Library References.....	703

Network Library Overview

Custom libraries contain custom data generated by the user, including fluid properties, pipe materials, and data for junctions types such as compressors and valves.

All custom data created by the user is kept on the local computer in a file called ARO_USER9.DAT in the local user directory. This data is not shared and is always available to the local user for access or for editing.

In addition to the local user library, you can create multiple network or local custom libraries. The [Library Manager](#) window on the Library menu simplifies this process. See [Creating an Enterprise-wide Network Library System](#) for details on how to do this.

Related Topics

Libraries.....	613
Library Manager.....	614
Library Browser.....	614
Library Verification.....	634
Library Locations in General Preferences.....	636
Fluid Library.....	619
Pipe Material Library.....	620
Pipe Material Library References.....	703
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Benefits of Shared Libraries.....	630
Creating an Enterprise-Wide Network Library System.....	631

Benefits of Shared Libraries

When the Arrow 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 Arrow itself, so it is guarded data.

Once the data is placed on a network, users can connect to the library in the [Library Browser](#). 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 "Colorado Springs 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 transfers 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.

Related Topics

Network Library Overview.....	629
Creating an Enterprise-Wide Network Library System.....	631

Creating an Enterprise-Wide Network Library System

Overview

Users of AFT Arrow 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 Arrow by creating a file named DATABASE.LIB in the ARROW 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 Arrow uses to find these libraries. An External Library is any library file that is created by the user and listed in the ARO_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 Arrow 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 Arrow 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
Arrow Internal Libraries		
ARROW#.DAT	AFT Default Internal Library	AFT Arrow # folder, where # is the version number
ARO_USER#.DAT	AFT Arrow 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 (ARO_DBUSER.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in ARO_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. compressors, 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](#) (Aro_User#.DAT, where # is the corresponding version number) 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 Arrow 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 Arrow. 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 Arrow directory on the network. However, since Arrow 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 ARO_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 Arrow folder (e.g. AFT Products\AFT Arrow\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 Arrow. 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 Arrow 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 (ARO_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 Arrow will write the full path of the library to the External Library listing (ARO_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 Arrow 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 Arrow\Libraries\filename.dat). You can automatically have AFT Arrow 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 Arrow folder where the executable is located.

When the pipe materials library was added it became an External Library listed in ARO_DBUSER.LIB. Now we need to move this listing from the local machine to the AFT Arrow 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 Arrow 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 Arrow, 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 Database Manager used to provide the option to include formats/preference settings such as the Output Control layouts into shared libraries. These type of files can still be shared, but are no longer shared through the Library Manager. Instead these type of format files must be loaded directly through their control windows, such as loading the Output Control format being loaded in the Output Control window.

Related Topics

Network Library Overview.....	629
Benefits of Shared Libraries.....	630

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 was originally connected and connect to that library again when it is made available to AFT Arrow. If, however, you try to open the compressor/fan Properties window, AFT Arrow 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 an external library, but for some reason the data is not the same. For example, you may have a fan curve for the Rocky Mountain Fan Company's Model 100A in your model, but it does not match the fan 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 fan in the model might have a different fan curve than the one in the library for one of these reasons:

- When the model was built, the fan was connected to the Rocky Mountain Fan Company (i.e., RMFC) Model 100A. But since that time, the 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 fans 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 fan in your model, and to inquire into the cause of the inaccurate data within your organization.

Related Topics

Libraries.....	613
Library Manager.....	614
Library Browser.....	614
Network Library Overview.....	629
Library Locations in General Preferences.....	636
Fluid Library.....	619
Pipe Material Library.....	620
Pipe Material Library References.....	703
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624

Library Locations in General Preferences

When performing an engineering analysis with or even without cost estimating, there may be many engineering and [cost libraries](#) that underlie the model. These libraries need to be connected in the [Library Manager](#) 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 Arrow tries to open a model that requires certain libraries, and those libraries have not been connected already, AFT Arrow looks for them in the locations specified in the User Options. In addition, AFT Arrow looks in the model's folder. If AFT Arrow 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.

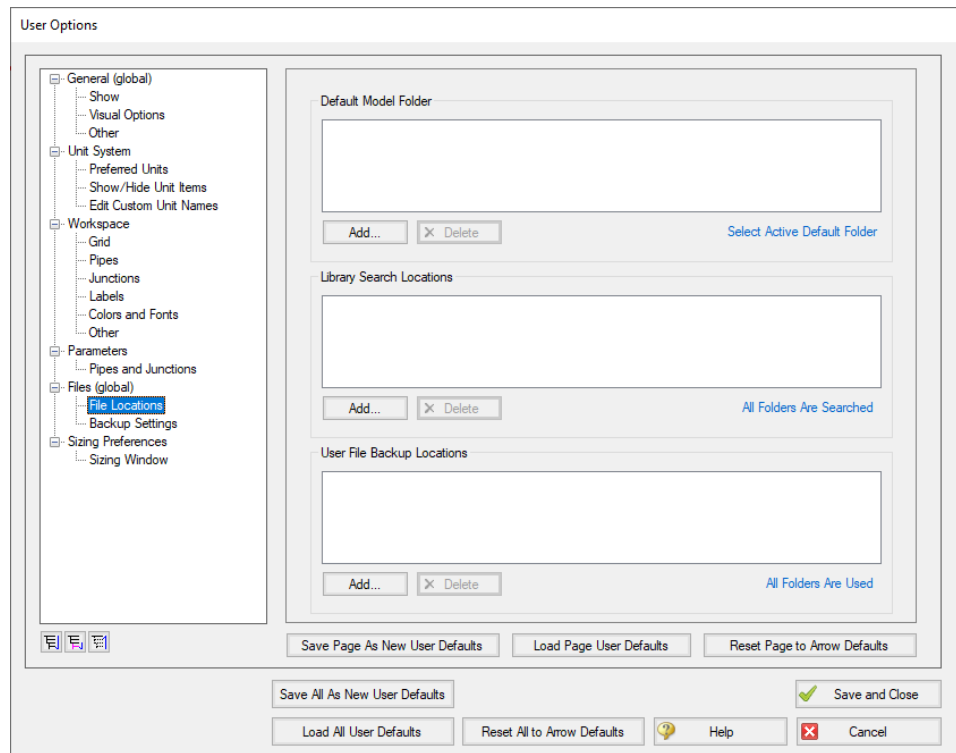


Figure 1: Library Search Locations can be defined in the Tools/User Options window

Related Topics

Libraries.....	613
Library Manager.....	614
Library Browser.....	614
Network Library Overview.....	629
Library Verification.....	634
Fluid Library.....	619
Pipe Material Library.....	620
Pipe Material Library References.....	703
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624

Theory and References

This section focuses on the fundamental theories that are used in AFT Arrows' solution methodology. Additionally, this chapter gives information on the source material for the theories used. Topics include discussions on the theories that comprise the network solution, detailed discussions of compressible flow theory and its implications, and information on references used in the development of AFT Arrow.

Content

Nomenclature	639
Review of Compressible Flow Theory	642
Compressible Flow Theory in Single Pipes	657
Sonic Choking	665
Role of Pressure Junctions - Detailed Discussion (Long)	674
Chempak Technical Information - Detailed Discussion	686
Stagnation Properties	698
Sound Power Level	699
References	701
Pipe Material Library References	703

Nomenclature

a	sonic speed
A	cross-sectional flow area of a pipe
C_d	discharge coefficient
c_p	specific heat, constant pressure
c_v	specific heat, constant volume
D	diameter of a pipe
e	internal energy

f	friction factor
F	Force
F_f	Parameter in Equation 18
F_g	Parameter in Equation 18
F_{T_o}	Parameter in Equation 18
F_g	Parameter in Equation 18
g	gravitational constant
h	internal convection coefficient
h	enthalpy, static
h_o	enthalpy, stagnation
k	thermal conductivity
K	loss factor
L	length of a pipe
\dot{m}	mass flow rate
M	Mach number
n	constant
Nu	Nusselt number
P	pressure, static
P_h	heater perimeter
P_o	pressure, stagnation
P_w	wetted perimeter
Pr	Prandtl number
q	heat rate to a pipe
Q	volumetric flow rate
q''	heat flux
r	radius
r	relaxation
R	gas constant
Re	Reynolds number
s	fan speed

s	entropy
T	temperature, static
T_o	temperature, stagnation
U	overall heat transfer coefficient
v	specific volume
V	volume
V	velocity
w	work
x	distance along pipe centerline
z	elevation
Z	compressibility factor
a, m, q	angle
β	diameter ratio
ε	roughness
γ	specific heat ratio
μ	dynamic viscosity
ρ	density
τ	shear stress
ω	rotational velocity

Subscripts

1	location 1 in pipe
2	location 2 in pipe
i	junction at which solution is sought
j	junctions with pipes connecting to junction i
o	stagnation
∞	infinity, far away, ambient

Related Topics

Theory and References.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703

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 \(1993\)](#), [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.

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 Arrow, 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 see significant compressible flow behavior. In some applications the gas velocities are low enough or the pressure drops are small enough that com-

pressible effects are insignificant. In such situations, systems can be modeled with traditional incompressible techniques. AFT Arrow is capable of modeling any gas flows.

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

$$P = R_g \rho T \tag{63}$$

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.

$$P = Z R \rho T \tag{64}$$

There are many methods of determining Z . These methods are found in textbooks and handbooks, each having strengths and weaknesses.

First Law and Isentropic Relationships

From the First Law of Thermodynamics we know that

$$\delta q + \delta w = \delta e \tag{65}$$

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

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

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)

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

Equation 86 is the classic equation for speed of sound, and since the specific heat ratio varies only slightly, the speed of sound effectively depends only on temperature.

For real gases, Equation 86 requires using the real gas equation (Equation 64) to obtain

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

As an example of sonic velocity, consider air at 70° F (530° R). The specific heat ratio for air is about 1.4, and assuming ideal gas ($Z = 1$) then

$$a_{air} = \sqrt{1.4 \left(1716.3 \frac{ft^2}{s^2 \circ R} \right) 530^\circ R} = \frac{1128 ft}{s} = \frac{344 m}{s}$$

The relationship between the gas velocity and sonic speed is known as the Mach number.

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

The Mach number is a dimensionless quantity that is the most important parameter in compressible fluid flow. Relating all parameters to the Mach number greatly simplifies compressible flow calculations.

In general, the sonic gas velocity is the maximum that can be attained in a system, thus the largest Mach number in a pipe system is 1. This is known as sonic flow. Flows less than Mach 1 are referred to as subsonic flows. It is in fact possible to attain Mach numbers greater than 1 by using a specially designed convergent-divergent nozzle. Flows greater than Mach 1 are referred to as supersonic flows. There are special applications for supersonic flows in internal flow systems, however these applications are not generally of interest to the piping engineer and thus no attempt is made to address supersonic flows in AFT Arrow. AFT Arrow 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:

$$q = \dot{m} d\left(h + \frac{V^2}{2} + g z\right) \quad (89)$$

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. 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

$$\frac{T_o}{T} = 1 + \frac{\gamma - 1}{2\gamma} \frac{V^2}{RT} \quad (92)$$

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.

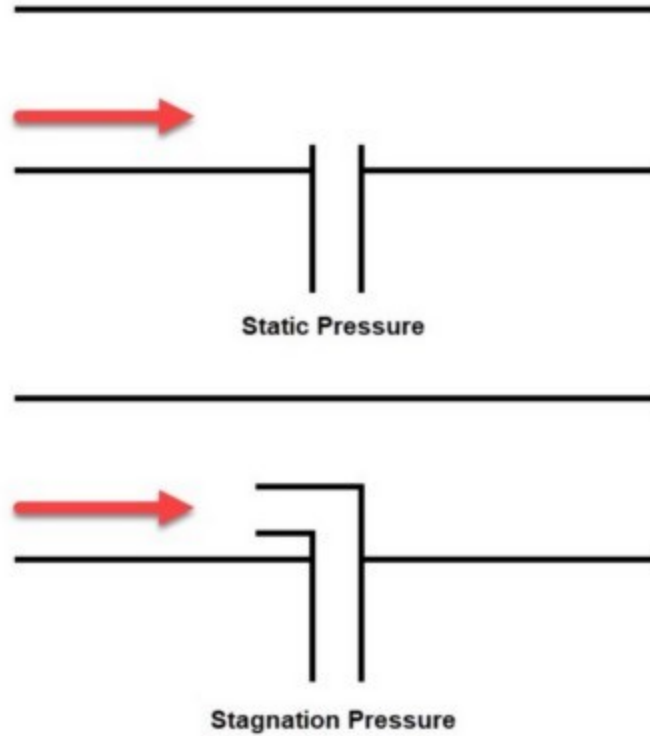


Figure 1: Comparison of static and stagnation pressure measurement at a pitot tube

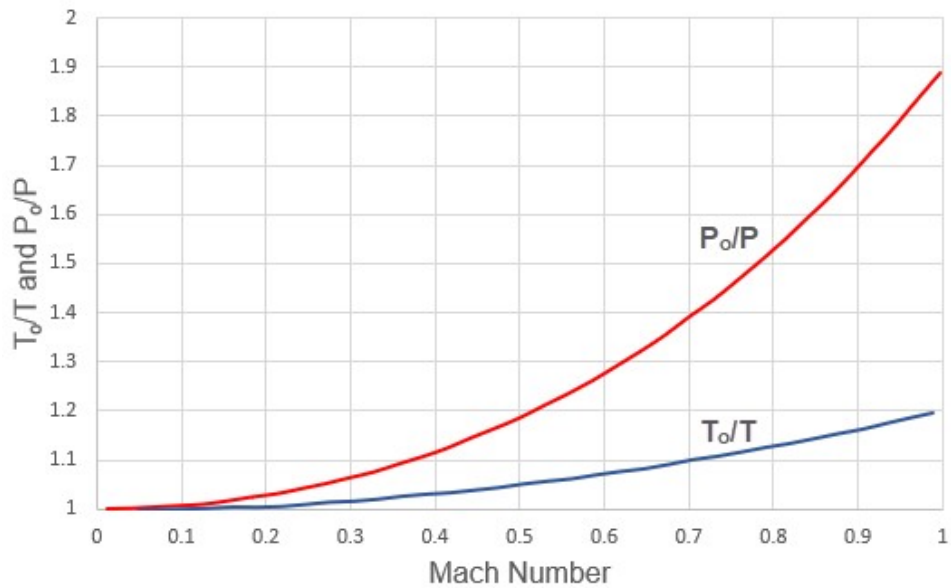


Figure 2: Variation of stagnation to static temperature ratio and stagnation to static pressure ratio with Mach number. Gamma = 1.4.

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 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,

$$q = \dot{m} d \left(h + \frac{V^2}{2} + g z \right) \quad (89)$$

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.

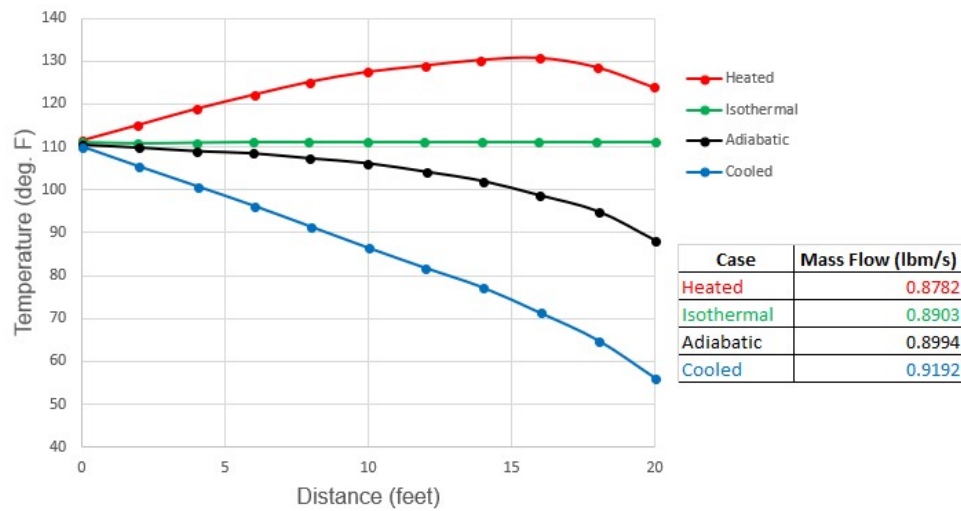


Figure 3: Comparison of temperature profiles and flow rate predictions for adiabatic, isothermal, heated and cooled air flow in a 20 ft. long 1 inch steel pipe. Inlet conditions are 100 psia, 111 °F and outlet pressure is 60 psia. Cooled flow has ambient temperature of 30 °F. Heated flow has ambient 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

Theory and References.....	639
Nomenclature.....	639
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703

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. AFT Arrow 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.

Related Topics

Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Network Implementation.....	662
Verifying Network Solutions.....	663
Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703

Extension of Single-Pipe Methods to Networks

Methods to solve the single pipe equations are available in the literature ([Anderson \(1984\)](#), [Saad \(1993\)](#), [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 Arrow 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 Arrow's 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 Arrow 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 Arrow's solution methods are not described.

Related Topics

Compressible Flow Theory in Single Pipes.....	657
---	-----

Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Network Implementation.....	662
Verifying Network Solutions.....	663

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 :

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

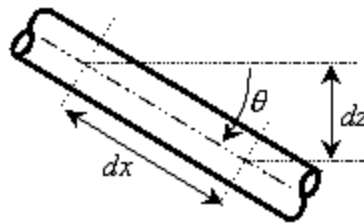


Figure 1: Relationship between distance along pipe (dx) and vertical elevation change (dz)

The fourth term on the left of Equation 2 accounts for pressure changes due to body forces. AFT Arrow models body forces due to [gravity and due to rotation](#). If 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). This can be changed in the [Environmental Properties Panel](#) to any value you wish, which allows for modeling aerospace vehicles or non-terrestrial systems.

If a [rotating system](#) is modeled, the rotations per minute is input in the [Environmental Properties Panel](#). You cannot model both gravitational and rotational accelerations at the same time. In practice, gas systems that rotate (such as turbomachinery) will be compact enough that elevation affects on pressure will be negligible.

In rotating systems, the junction elevations become distances from the rotation axis. The acceleration due to rotation becomes:

$$g_r = z \omega^2 \tag{4}$$

where z is the junction elevation (distance from the axis) and w is the rotational velocity. This term is interchangeable with g in Equation 2, and henceforth g will be used with the understanding that it can refer either to gravitational or rotational acceleration as in Equation 4.

Also note that in rotating systems, the g level will increase as the distance from the centerline is increased. For each computing section, AFT Arrow uses the acceleration based on the arithmetic average of z inlet and z outlet.

From the equation of state for a real gas:

$$P = Z \rho R T \tag{5}$$

Related Topics

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Network Implementation.....	662
Verifying Network Solutions.....	663

Continuity Equation at a Branch

By applying the continuity equation at branch i , we loop over all connecting pipes and can say that:

$$\sum_{j=1}^{n_{pipes}} \dot{m}_{ij} = 0 \tag{46}$$

where \dot{m}_{ij} is the flow rate in the pipe connecting junctions i and j .

Related Topics

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Network Implementation.....	662
Verifying Network Solutions.....	663
Flow at Junction X may not be balanced.....	716

Energy Equation at a Branch

By applying the energy equation at a branch i , we loop over all connecting pipes and can say that:

$$\sum_{j=1}^{n_{pipes}} \dot{m}_{ij} \left(h_{ij} + \frac{V_{ij}^2}{2} \right) = 0$$

or, in terms of stagnation enthalpy,

$$\sum_{j=1}^{n_{pipes}} \dot{m}_{ij} h_{0,ij} = 0 \quad (47)$$

where $h_{0,ij}$ is the stagnation enthalpy at the connection point of the 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

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Concentration Equation at a Branch.....	662
Network Implementation.....	662
Verifying Network Solutions.....	663

Energy may not be balanced at Junction X.....	715
---	-----

Concentration Equation at a Branch

The concentration equation is a special version of the continuity equations. It is needed only when gas streams of different composition flow into the branch (i.e., a mixing calculation is occurring).

By applying the continuity equation at branch i , we loop over all connecting pipes for each molecular species c and can say that:

$$\sum_{c=1}^{n_{gases}} \sum_{j=1}^{n_{pipes}} \dot{m}_{c,ij} = 0 \quad (48)$$

Related Topics

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Network Implementation.....	662
Verifying Network Solutions.....	663

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 must be equal. Additionally, the mass flow, energy flow, and species concentrations must 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.

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

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.

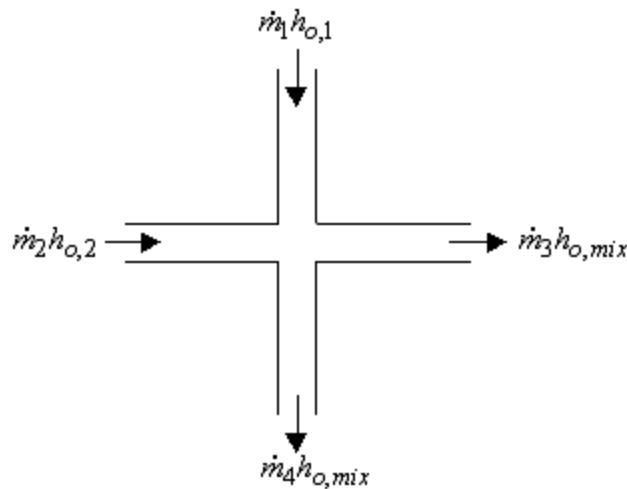


Figure 1: Flow weighted mixing and energy balance

Related Topics

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Verifying Network Solutions.....	663

Verifying Network Solutions

AFT Arrow uses a modified Newton-Raphson matrix method to obtain a system level balance of mass, energy and concentration. When the Solver is finished, the [continuity](#), [energy](#), and [concentration](#) equations are satisfied at all branching sections. Once solved, AFT Arrow goes back and sums up all mass flows, energy and concentrations at each branch to verify that these equations are satisfied.

After the Solver converges, AFT Arrow loops over each junction and adds up the inflow and out-flow of mass, energy and concentrations 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 Arrow 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

Related Topics

Compressible Flow Theory in Single Pipes.....	657
Extension of Single-Pipe Methods to Networks.....	658
Fundamental Equations.....	659
Continuity Equation at a Branch.....	660
Energy Equation at a Branch.....	661
Concentration Equation at a Branch.....	662
Network Implementation.....	662

Energy may not be balanced at Junction X.....	715
Flow at Junction X may not be balanced.....	716

Sonic Choking

Sonic choking occurs in a pipe system when at some location the gas attempts to accelerate to a velocity faster than the local sonic velocity. 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 is the dominant feature of the system behavior.

Predicting the flow rates under such sonic choking conditions can be very difficult depending on where the choking occurs. AFT Arrow is designed to identify sonic choking and predict the system behavior under such conditions.

Related Topics

Valve.....	299
Valve Modeling Closures with K Factors.....	302
Cv/Kv and Xt Loss Model.....	235
Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703
Subsonic vs. Sonic Pressure Losses.....	665
Sonic Choking Detailed Description.....	666
Multiple Choking Points in Single Flow Path.....	671

Related Blogs

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

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, discharge coefficient for

subsonic flow, or one of the other loss models discussed in the [Junctions](#) section.

- **Sonic** losses result when [sonic choking](#) occurs. Sonic losses are dependent on the minimum effective flow area and are unrelated to the K-factor. This is the C_dA for sonic choking.

The two loss mechanisms cannot occur simultaneously in nature or in AFT Arrow. If one mechanism occurs during the solution, the input data for the other mechanism is ignored by AFT Arrow.

It is also important to note that sonic choking at a restriction cannot be calculated if the C_dA for Sonic Choking is not entered. Thus, Arrow will always use subsonic loss calculations if no C_dA for Sonic Choking has been entered, even if the actual system would experience choked flow at that restriction.

Note: The C_dA for sonic choking may be different from the subsonic C_dA loss model option in Arrow. 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

Sonic Choking.....	665
Sonic Choking Detailed Description.....	666
Multiple Choking Points in Single Flow Path.....	671

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. 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.

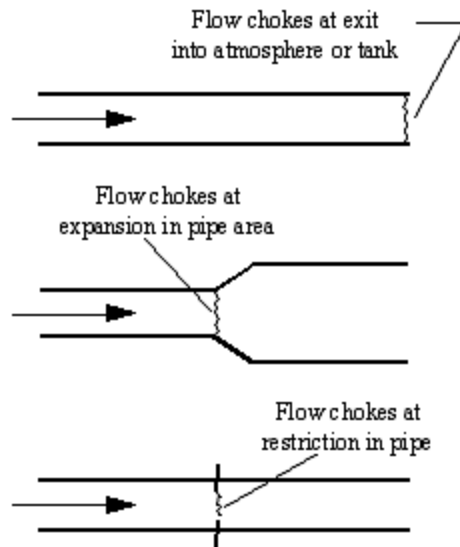


Figure 1: Three configurations which can result in sonic choking. From top to bottom, endpoint choking, expansion choking, and restriction choking.

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.

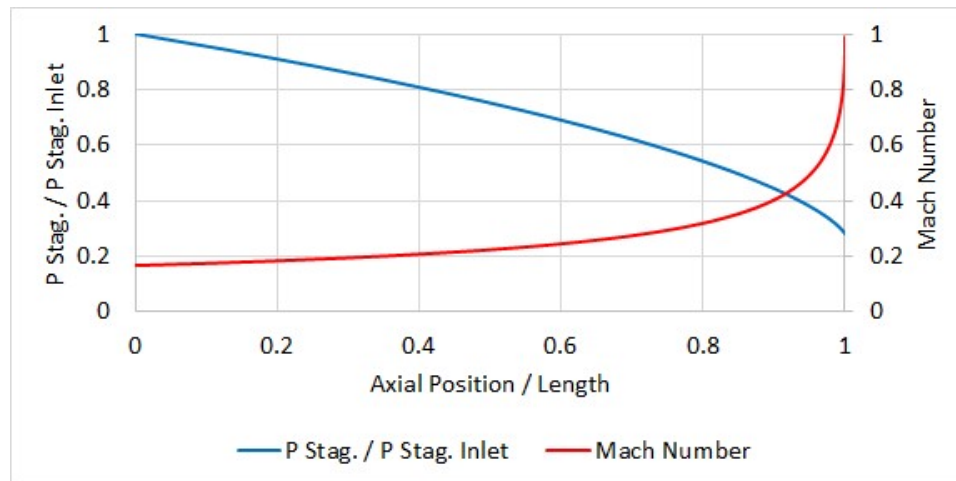


Figure 2: Example stagnation pressure and Mach number profiles for endpoint choking

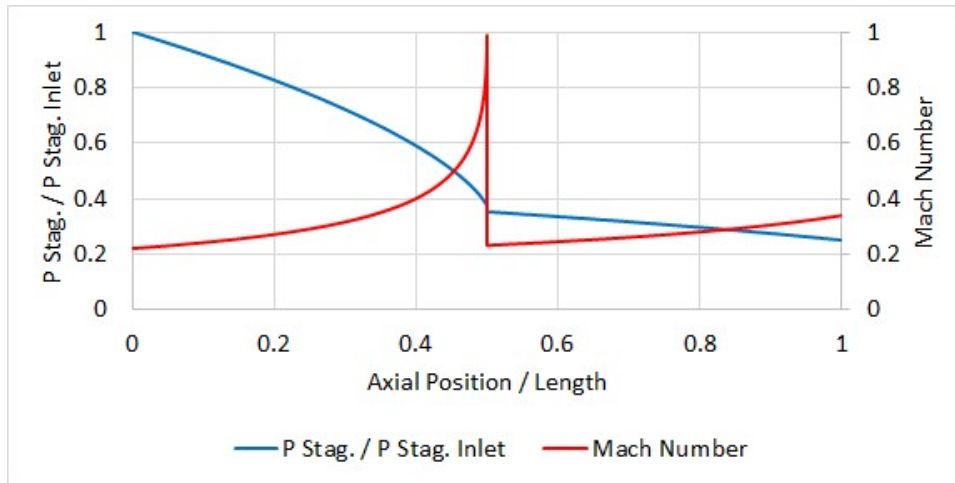


Figure 3: Example stagnation pressure and Mach number profiles for expansion choking

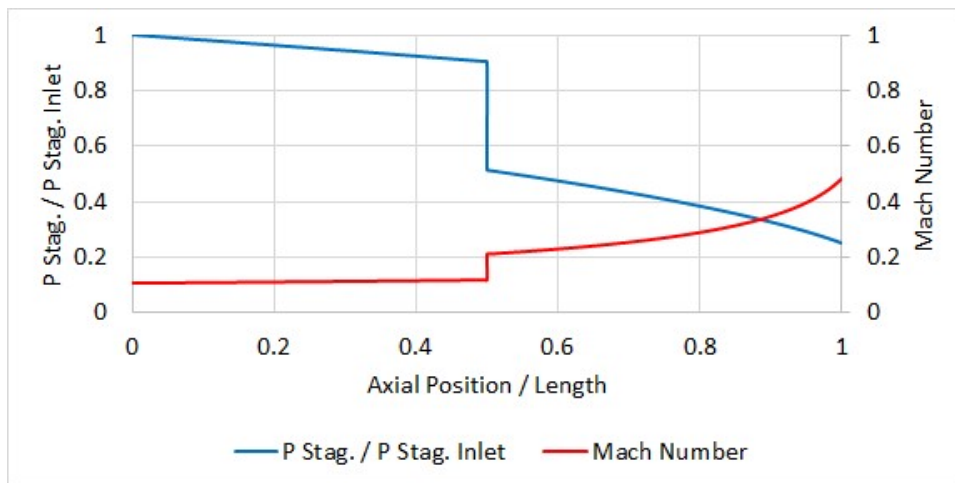


Figure 4: Example stagnation pressure and Mach number profiles for restriction choking. Note that the Mach Number reaches 1 within the restriction at 0.5.

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](#), pp. 97):

$$\dot{m} = A \frac{P_0}{\sqrt{T_0}} f(M) \quad (50)$$

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 conditions, P_o , and T_o , are not known at the point of choking. For instance, to apply these equations to end-point choking (Figure 1, top), one must calculate the stagnation pressure and temperature at the end of the pipe before the choke point. The problem is that the stagnation pressure and temperature before the choke point depend on the flow rate, which is what we are trying to calculate.

The only way to accurately calculate such a problem is to do the following:

1. Assume a flow rate
2. March down the pipe
3. If the Mach number reaches unity before the end of the pipe, lower the assumed flow rate and repeat steps 1-3 until the Mach number reaches sonic right at the pipe endpoint.

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 Arrow 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.

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.

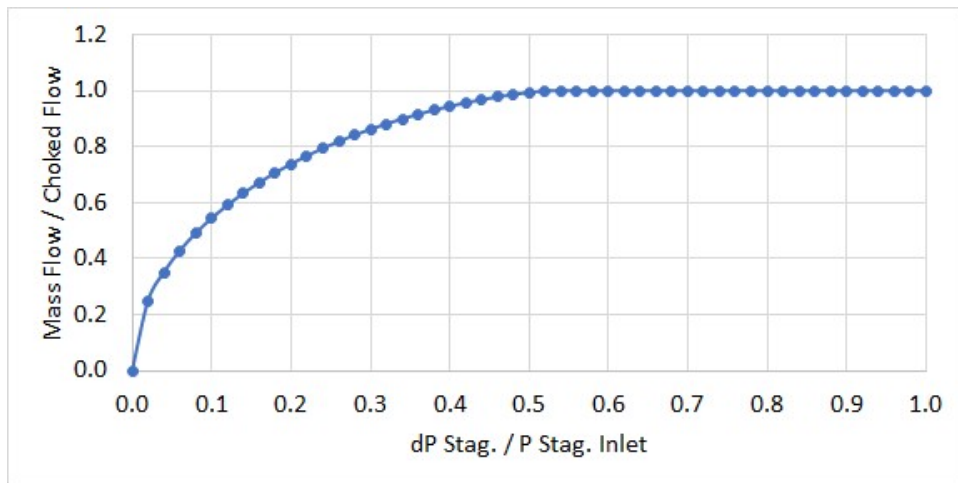


Figure 5: Example of dimensionless flow rate vs. dimensionless pressure drop. Sonic choking occurs just past the dimensionless pressure drop of ~0.5, after which the flow can no longer increase

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.

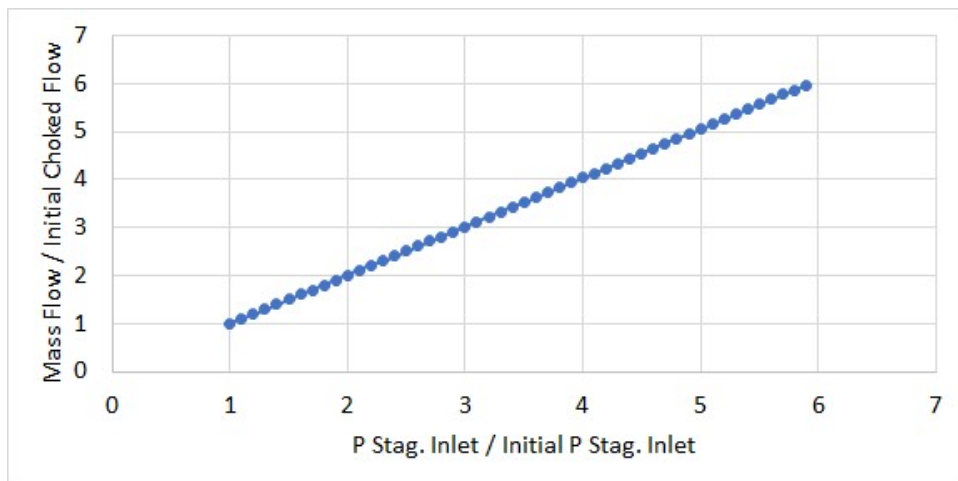


Figure 6: Example of increasing choked flow rate with supply pressure

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 and apply the governing equations to the remaining pipes. This boundary condition is facilitated by knowing the stagnation enthalpy change across the choke point is constant.

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

Sonic Choking.....	665
Subsonic vs. Sonic Pressure Losses.....	665
Multiple Choking Points in Single Flow Path.....	671

Multiple Choking Points in Single Flow Path

It should be noted that sonic choking can exist in any multiple of the above geometries and at multiple locations in the same pipe system. AFT Arrow's solution method incorporates this reality. The model shown in Figure 1 is that of a "triple choke steam flow", where sonic choking occurs in three places, as indicated in Figure 1. The pressure drop in the system is as shown in Figure 2. The Mach number as it progresses down the system is shown in Figure 3.

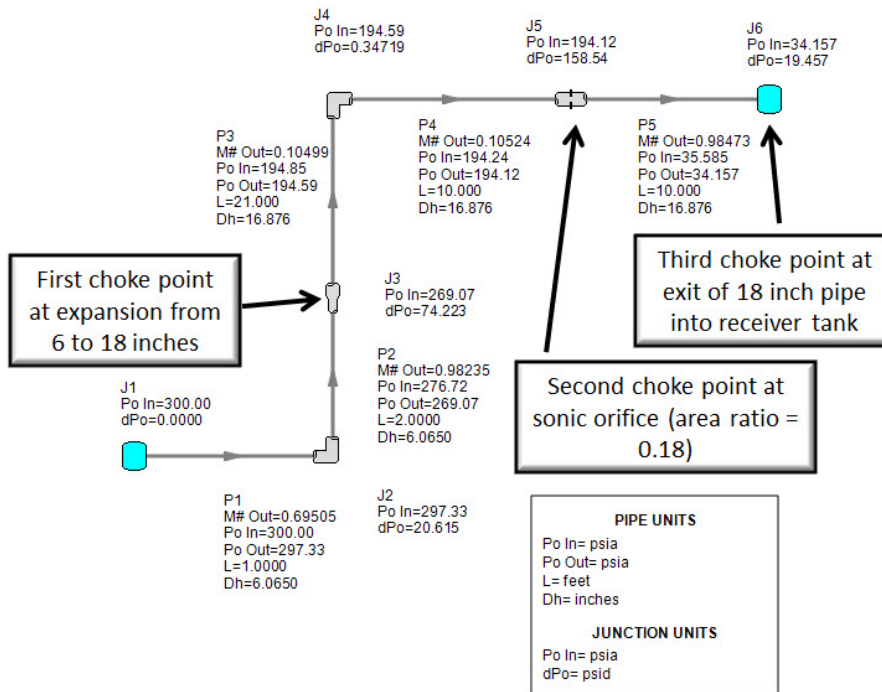


Figure 1: Steam flow in a pipe flow path with three choking points. Referred to as a Triple Choke Model. Solved using Mach March Method. [Walters and Olsen \(1997\)](#).

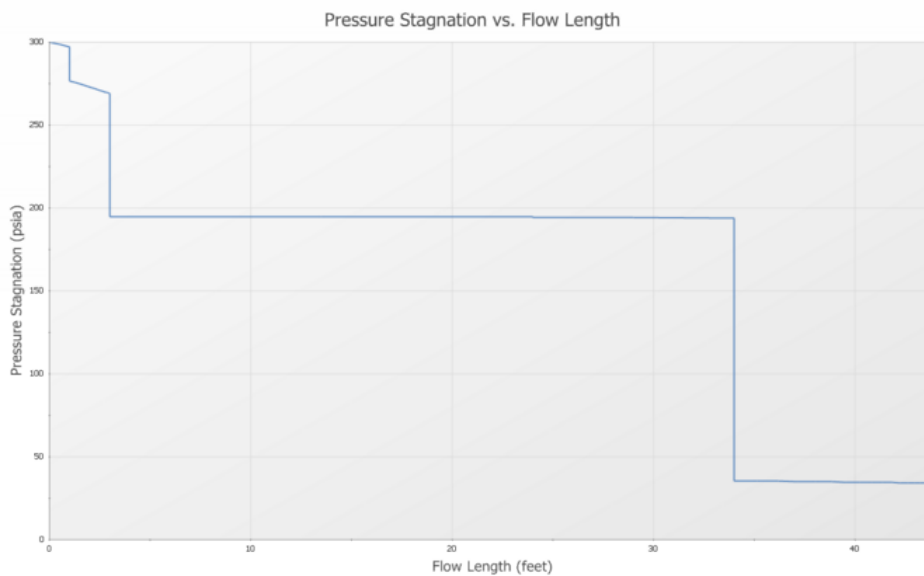


Figure 2: Stagnation pressure drop in Triple Choke Steam Flow model shown in Figure 1. [Walters and Olsen \(1997\)](#).

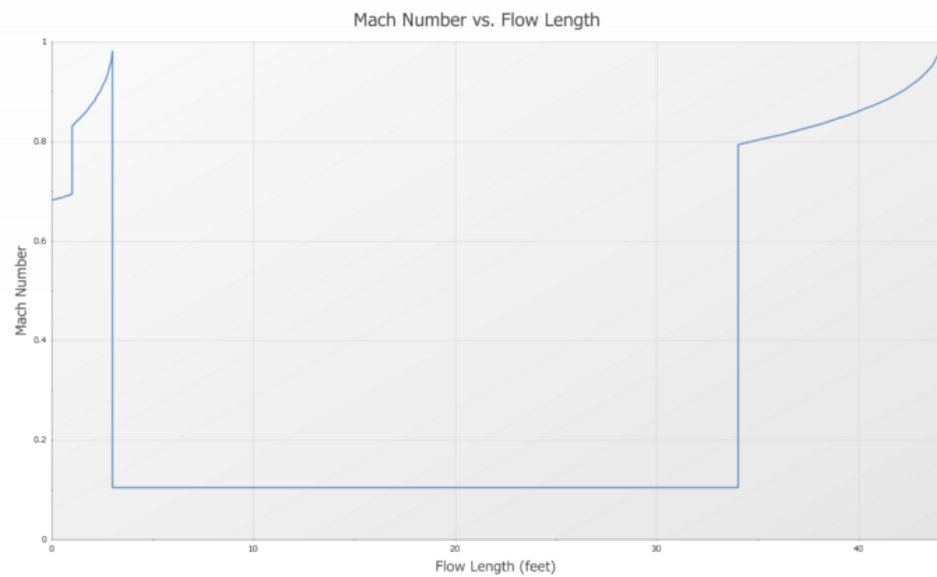


Figure 3: Mach number profile in Triple Choke Steam Flow model shown in Figure 1

Sonic choking in multiple locations in a flow path can be understood as follows. Recall that the sonic flow rate will have a unique value, given upstream conditions and the piping losses and thermal environment. No matter what happens at the exit boundary condition, the flow rate will not increase or change in any way. However, by changing the upstream conditions, the sonic flow rate can be changed. This is because the conditions are being changed on the upstream side of the choke point and can thus influence the fluid behavior.

To understand multiple sonic choking, consider the system in Figure 1. By iteration on flow rate it is determined that sonic choking will exist at the pipe expansion at junction J3. This fixes the flow rate for the entire system. This flow rate can then be entered as an upstream boundary condition on the remaining pipes. In this way the pressure can be obtained downstream of the choke point by iteration. After entering the flow rate as an upstream pseudo-boundary condition, it may turn out that sonic choking occurs again, such as it would for any pipe system with an upstream assigned flow rate. In this system the flow again chokes at the restricted area of an orifice at J5. Again, the known choked flow rate is used as a pseudo-boundary condition at the entrance to the pipe downstream of the orifice. Solving this pseudo system results in a third choking condition at the exit of pipe P5 as it connects to atmospheric pressure at J6. Choking also occurs as the flow exits the system.

The pseudo system that represents the triple choke system can be depicted as in Figure 4.

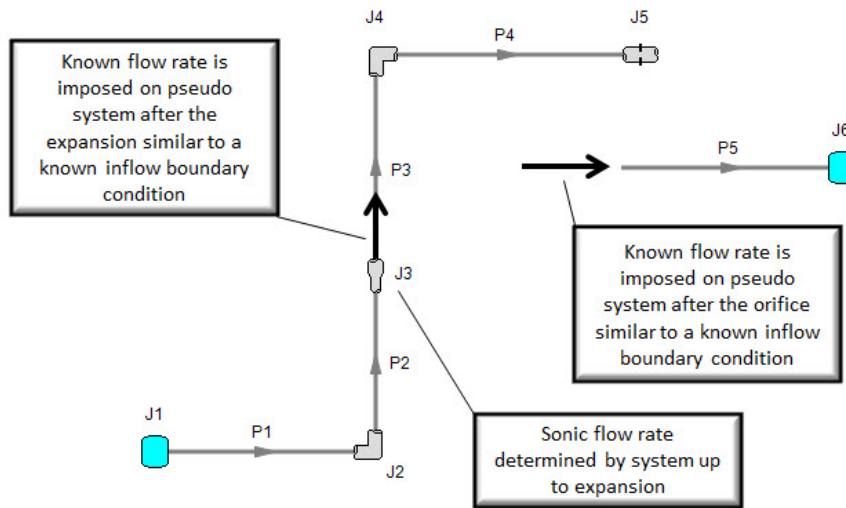


Figure 4: Triple Choke Steam Flow model pseudo system for determining pressure drops at choke points. Based on model shown in Figure 1. Walters and Olsen (1997).

Related Topics

Sonic Choking.....	665
Subsonic vs. Sonic Pressure Losses.....	665
Sonic Choking Detailed Description.....	666

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 Arrow, they do not need to think at the same depth as in the hand calculation. So when AFT Arrow 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 Arrow terms, the system has two assigned flow junctions. Heat transfer has no affect on the point being made here and is therefore ignored in all examples.



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 Arrow and if you try and run this model, AFT Arrow will inform you that it cannot run it because of the lack of a reference pressure. In AFT Arrow there are several junctions that can act as a reference pressure: the reservoir, assigned pressure, spray discharge, valve (with the exit valve option), orifice (with the exit orifice option), and exit type relief valves when open.

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 Arrow 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 each 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.

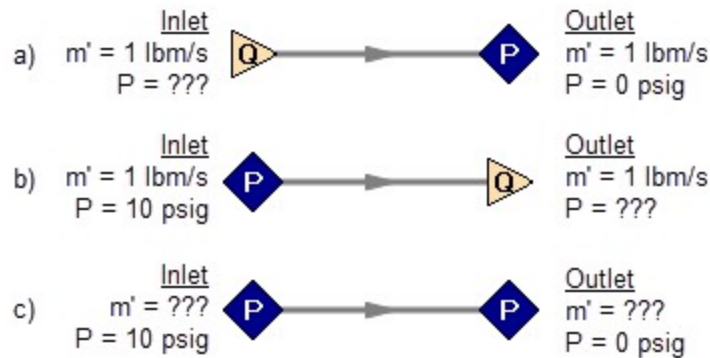


Figure 2a-c: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.

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.

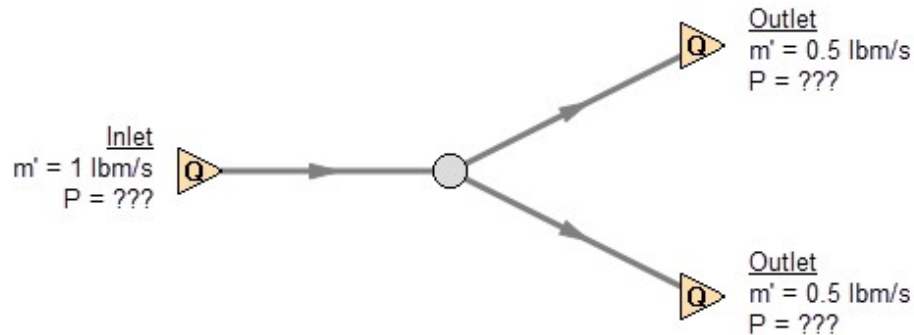


Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.

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 1 lbm/s, and the downstream flow is demanding 1 lbm/s. 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 Arrow, you will get the message shown in Figure 5. AFT Arrow identifies the part(s) of the model where a known pressure is needed.

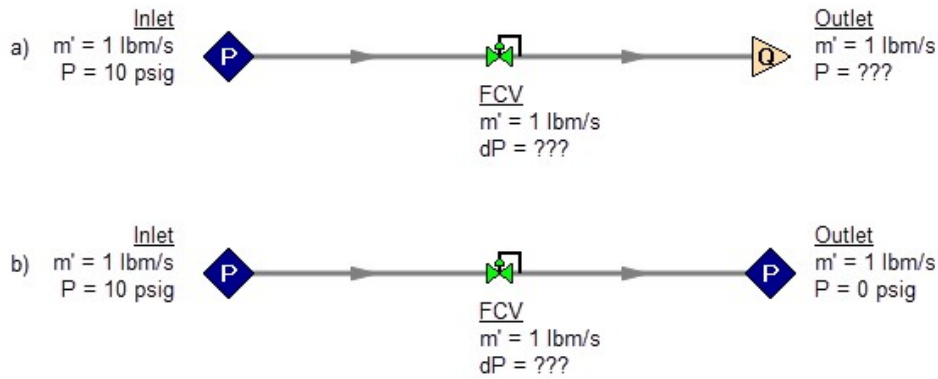


Figure 4a-b: The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists.

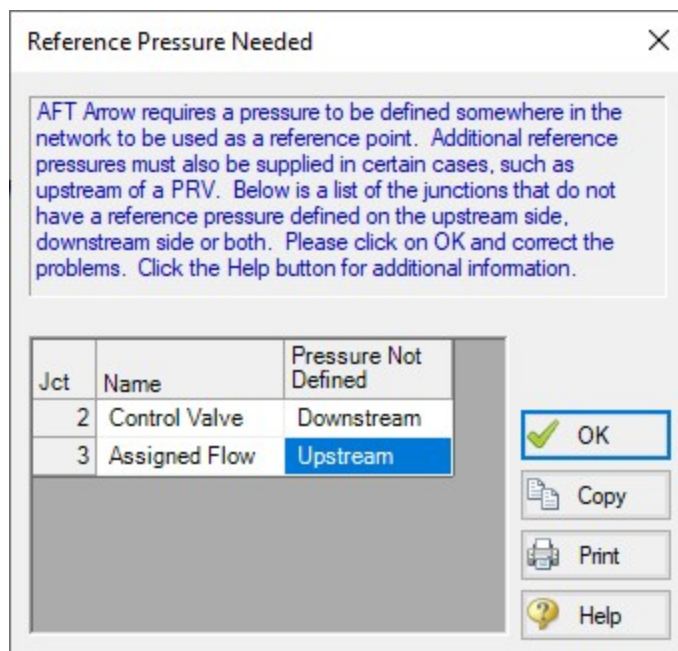


Figure 5: AFT Arrow message when you try to run the model shown in Figure 4a

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.

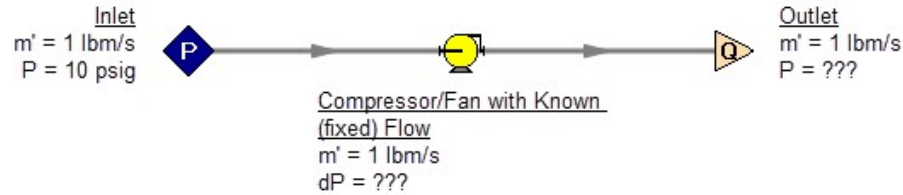


Figure 6: A compressor/fan modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists

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.

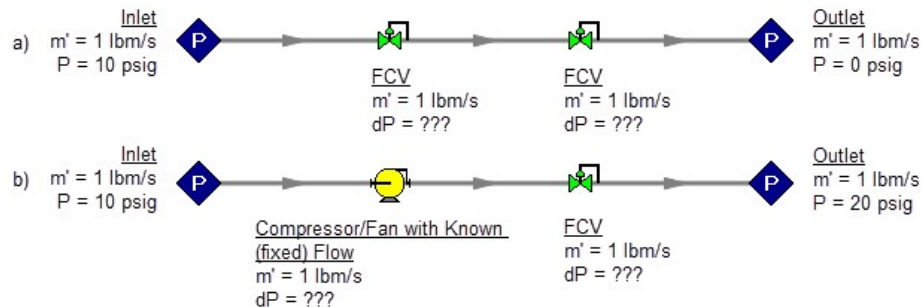


Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a compressor/fan modeled as a fixed flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions 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.

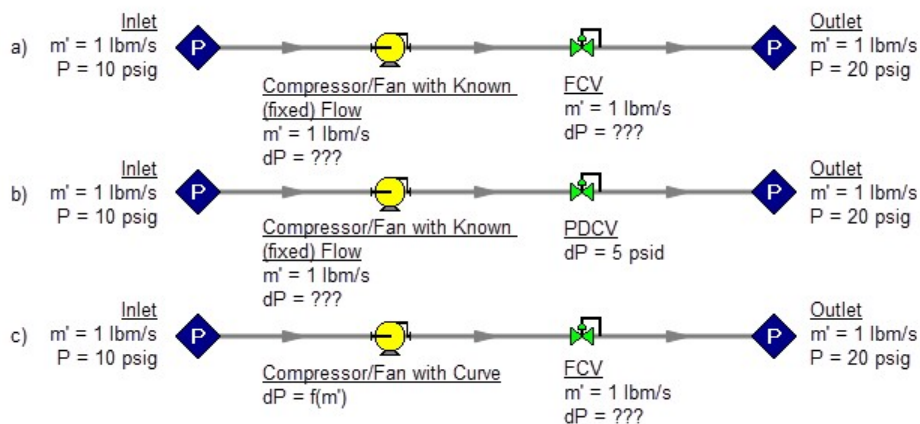


Figure 8a-c: The model at the top is the same as Figure 7b and does not have a unique solution. To size the compressor/fan, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the compressor/fan is sized, a 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.

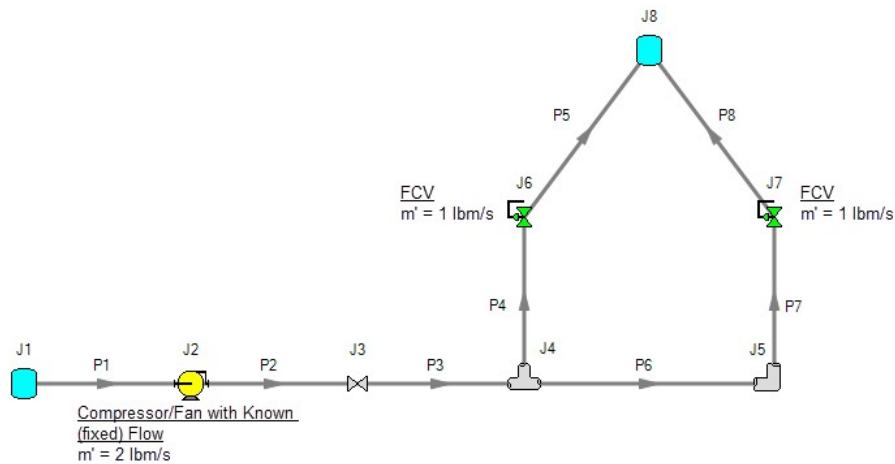


Figure 9: A compressor/fan with a fixed flow in series with two parallel FCV's. The model in its current form does not have a unique solution. To size the compressor/fan, first change the FCV at J7 to a PDCV.

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	2.000	59.83	51.69	160.1

Figure 10a-b: Results from changing J7 in Figure 9 to pressure drop control valve

AFT Arrow 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 PRV's and PSV's have in common with FCV's 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.

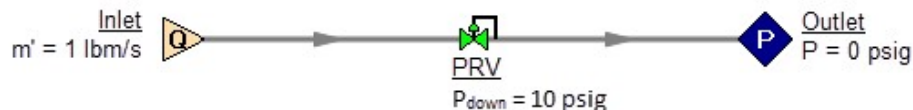


Figure 11: If a PRV is downstream of a known flow the PRV pressure drop cannot be calculated. This model does not have a unique solution.

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.

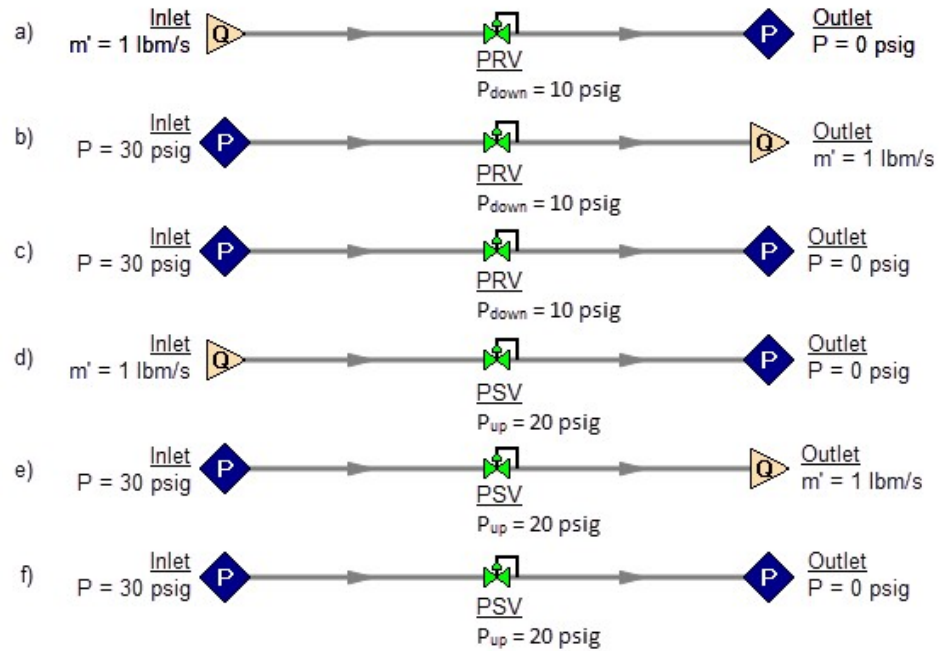


Figure 12a-f: Six possibilities for PRV and PSV configurations. Cases A and E do not have unique solutions. See Table 2 for comments.

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		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 a fixed 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 FCV's.

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 pipe or 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.

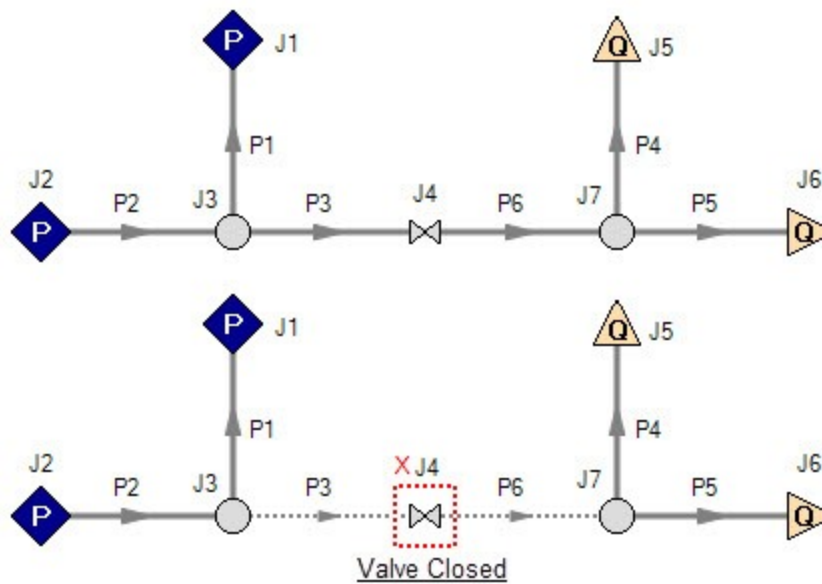


Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.

Summary

AFT Arrow models always require one reference pressure, and may require more if the engineer uses control valves, compressors or fans with fixed flows, or closed pipes or junctions. The actual number of reference pressures needed depends on the model, but AFT Arrow 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 Topics

Role of Pressure Junctions and How They Work.....	65
Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701

Pipe Material Library References.....703

Related Blogs

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

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. 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.

8.2.5 Freezing Points

Where possible, quoted freezing points are experimental. No accurate method of estimation of compound freezing point is available. In the absence of experimental data, a rough estimate was derived from the Joback method.

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,

$$V_b = a V_{cn}$$

$$a = 0.285$$

$$n = 1.048$$

8.3 Liquid Specific Volume

Liquid specific volume rises slowly and approximately linearly with rise in temperature to about $T_r = 0.8$. At higher temperatures, the values rise more rapidly to the critical point.

Experimental data or correlations derived from experimental data are available for most compounds.

8.3.1 Hankinson-Brobst-Thompson Equation

The saturated specific volume is given by,

$$V_s/V^* = V_r(0)(1 - wsrkV_r(1))$$

$$V_r(0) = \text{Sum}\{a_n(1 - T_{rn}/3)\} \quad 0.25 < T_r < 0.95$$

$$V_r(1) = \text{Sum}\{b_n T_{rn}/(T_r - 1.00001)\} \quad 0.25 < T_r < 1.0$$

$$a_0 = 1$$

$$a_1 = -1.52816$$

$$a_2 = 1.43907$$

$$a_3 = -0.81446$$

$$a_4 = 0.190454$$

$$b_0 = -0.296123$$

$$b_1 = 0.386914$$

$$b_2 = -0.0427258$$

$$b_3 = -.0480645$$

V^* , $wsrk$ and T_c are tabulated property constants. The user is referred to Hankinson, Thompson and to Reid et al (1987). Errors are typically about 1% with most being less than 2%.

8.3.2 Rackett Equation

If a reference volume (V_{ref} at T_{ref}) is available then

$$Z_{ra} = (P_c V_{ref} / RT_c)^n$$

$$n = 1/(1 + (1 - T_{ref} / T_c)^m)$$

$m = 2/7$ or other empirical constant

The saturated specific volume is given by,

$$V_s = V_{ref} Z_{ra}$$

with

$$x = -(1 - T_{ref}/T_c)^m + (1 - T/T_c)^m$$

In most cases, an experimental value of reference density was available. Where such a value was not available, values were derived from the group contribution method of Le Bas or derived from the critical volume using the Tyn & Calus relation. Tests by Madison Technical Software on over 80 liquids showed that these two methods were significantly more accurate than the Spencer and Danner method for Z_{ra} . The reader is referred to Reid et al for further details on these methods. With one or more experimental points, the Rackett equation gives errors of about 1% with most values less than 3%. If the reference volumes are estimated, typical errors are 3%.

8.3.4 The Effect of Pressure on Liquid Specific Volume

The effect of pressure on liquid specific volume is calculated when

$$P > P_s + 0.1 P_c$$

The correction is derived from the equation of state as follows,

$$V_L = V_{Ls} - V_{Ls,es} + V_{L,es}$$

V_L = specific volume at T and P

V_{Ls} = V_L at T and P_s from methods of this section

$V_{Ls,es}$ = V_L at T and P_s from the equation of state

$V_{L,es}$ = V_L at T and P from the equation of state

Liquid specific heat can in principle be derived from the equation of state but in practise, direct analytical or group contributions are preferred where experimental data are not available.

8.4 Liquid Specific Heat

8.4.1 Rowlinson-Bondi Method

$$(C_{pL} - C_{po})/R = 1.45 + 0.45/X + 0.25w(17.11 + 25.2X0.333/Tr + 1.742/X)$$

$$X = 1 - Tr$$

w = acentric factor

This method is generally applicable to the range from T_f to values approaching T_c . Note that C_{pL} approaches infinity as T approaches T_c .

Errors are generally less than 5% except in the case of hydrogen-bonding polar compounds (e.g. alcohols) at low temperatures. For these compounds, the Missenard group contribution method is preferred.

8.4.2 Missenard Method

The Missenard group contribution method yields values of coefficients in

$$C_{pL} = a + bT + cT^2$$

The accuracy is usually better than 5%. The method cannot deal with double bonds and is not applicable for $T_r > 0.75$

$$a = \text{Sum}\{a_n\}$$

$$b = \text{Sum}\{b_n\}$$

$$c = \text{Sum}\{c_n\}$$

The group contributions are available in Missenard. See also Reid et al.

8.4.3 The Effect of Pressure on Liquid Specific Heat

As noted above, the equations of state can be employed to estimate liquid specific heat, but the methods presented in 8.4.1 and 8.4.2 are more reliable. The equations of state however can be used to estimate the effect of pressure on liquid specific heat.

$C_{pLs} = C_{pL}$ at T_s and P_s determined by the methods of this section.

C_{po} = ideal gas specific heat at T_s

$C_{pL} = C_{pL}$ at T_s and $P > P_s$

The equations of state give estimates of

$$D_s = (C_{pLs} - C_{po})_{\text{es at } T_s \text{ and } P_s}$$

$$D = (C_{pL} - C_{po})_{\text{es at } T_s \text{ and } P}$$

The corrected value of the liquid specific heat is

$$C_{pL} = C_{pLs} + D - D_s$$

The correction is not applied when T is close to T_c

8.5 Liquid Viscosity

Liquid viscosity typically varies in magnitude by a factor of 100 or more between the freezing and critical temperatures. No generalized method is available to estimate or represent liquid viscosity adequately over the entire temperature range. Corresponding states methods are applicable above $T_r = 0.76$. From the freezing point to the boiling point, the influence of structure is strong.

8.5.1 Method of Van Velzen

The method of Van Velzen et al is a group contribution method of some complexity and range of applicability. It is the most frequently used group contribution method. The accuracy of the estimation averages about 10% and most estimates are better than 20%. Some of the limitations of the method are:

- Larger errors found with the first members of a homologous series
- Only normal and iso substitutions on alkyl chains can be treated
- Heterocyclic compounds cannot be treated
- Application only in the range T_f to T_b

The method is complex and the reader is directed to the original references for full details.

8.5.2 Method of Morris

The method of Morris is a group contribution method. This method is useful as a comparison and substitute for the Van Velzen method in cases where the Van Velzen method is not

applicable. The accuracy of estimation is of the same order as Van Velzen. The limitations of the method are,

- The method is less detailed than the Van Velzen method
- Applicable only in the range T_f to T_b
- No explicit treatment for heterocyclics or esters (apart from acetates).

The Morris method takes the following form

$$\ln(v/v^*) = 2.3026 J(1/T_r - 1)$$

$$J = (0.577 + \text{Sum}(D_i))0.5$$

The values of v^* are given for various categories of compounds. The constants v^* and the group contributions D are given in Morris.

8.5.3 Method of Letsou and Stiel

This is a corresponding states method with applicability over $0.76 < T_r < 1$. The method also predicts the viscosity at the critical point ($T_r = 1$). The accuracy is normally better than 5% up to $T_r = 0.92$ with higher errors encountered as the critical point is approached. Overall this is an excellent estimation method for high-temperature liquid viscosity. The only serious limitation is the restricted range of applicability.

The form of the relation is

$$v = (f_0 + w.f_1)/A$$

with

$$w = \text{acentric factor}$$

$$f_0 = a_0 + b_0 T_r + c_0 T_r^2$$

$$f_1 = a_1 + b_1 T_r + c_1 T_r^2$$

$$A = 0.176 \times 10^6 T_c^{0.1667} / M^{0.5} P_c^{0.667}$$

$$a_0 = 2.648 \quad a_1 = 7.425$$

$$b_0 = -3.725 \quad b_1 = -13.39$$

$$c_0 = 1.309 \quad c_1 = 5.933$$

In the above relations P_c is in bar and the viscosity is in units of Pa-sec.

8.5.4 Method of Przedziecki & Sridhar

In this method, the viscosity is related to changes in the specific volume.

$$v = V_0/E(V - V_0) \text{ centipoise}$$

$$V = \text{liquid molar volume in cc/mol}$$

$$E = -1.12 + V_c/D$$

$$D = 12.94 + 0.1 M - 0.23 P_c + 0.0424 T_f - 11.58 T_f/T_c$$

$$V_0 = 0.0085 w T_c - 2.02 + V_f / \{0.342(T_f / T_c) + 0.894\}$$

with

$$T_c = \text{critical temperature, K}$$

$$P_c = \text{critical pressure, bar}$$

V_c = critical volume, cc/mol

M = molecular weight

T_f = freezing point, K

w = acentric factor

V_f = specific volume at T_f

The authors recommend that the volumes be estimated from the Gunn and Yamada equation. The reader is referred to Reid for a discussion on this method. The method is less accurate below reduced temperatures of about 0.55. Errors vary widely but will normally be less than 20% for T_r greater than 0.55.

This method is used in CHEMPAK only where necessary. An error analysis by Reid et al indicates a higher level of error associated with this method than with the Van Velzen method for instance.

8.5.5 Interpolation and Extrapolation

Two regions are typically covered well by available experimental data, experimental correlations and by the above relations:

$273 < T < 0.6 T_c$: this region is normally covered by published data or by one of the methods 8.5.1, 8.5.2, 8.5.4

$0.76 T_c < T < T_c$: this region is well covered by the method of Letsou and Stiel (section 8.5.3)

This leaves two regions which are often not covered by the above methods

$T_f < T < 273$: this region may be covered by extrapolation using $\ln(v)$ versus $1/T$ extrapolation. The error due to the extrapolation in practise will not normally exceed 10% with a possible 20% error in the immediate vicinity of the freezing point.

$0.6 T_c < T < 0.76 T_c$: this region may be covered by interpolation between the $273 < T < 0.6 T_c$ region and the $0.76 T_c < T < T_c$ region using $\ln(v)$ versus $1/T$ interpolation. The errors due to interpolation in this case rarely exceed 5%.

8.5.6 The Effect of Pressure on Liquid Viscosity

The method of Lucas is applied:

$$v_L/v_{sL} = (1 + B.FA)/(1 + w.C.F)$$

v_L = viscosity at pressure P

v_{sL} = viscosity at saturation pressure P_s

$$F = (P - P_s)/P_c$$

w = acentric factor

$$A = 0.9991 - 0.0004674/(1.0523/Tr - 0.03877 - 1.0513)$$

$$B = 0.3257/(1.0039 - Tr) - 0.2906 - 0.2086$$

$$C = -0.07921 + 2.1616 Tr - 13.404 Tr^2 + 44.1706 Tr^3 - 84.8291 Tr^4 + 96.1209 Tr^5 - 59.8127 Tr^6 + 15.6719 Tr^7$$

8.6 Liquid Thermal Conductivity

8.6.1 Method of Latini et al

For specified categories of compounds, the method of Latini et al gives correlations for liquid conductivity for the range $Tr = 0.3$ to 0.8

The correlations are in the form

$$k = A(1 - Tr)^{0.38}/Tr^{0.167}$$

$$A = A_0 T_b^n M^m T_c^p$$

Category	A0	n	m	p
Alkanes	0.0035	1.2	-0.5	-0.167
Alkenes	0.0361	1.2	-1.0	-0.167
Cycloalkanes	0.0310	1.2	-1.0	-0.167
Aromatics	0.0346	1.2	-1.0	-0.167
Alcohols/Phenols	0.00339	1.2	-0.5	-0.167
Acids	0.00319	1.2	-0.5	-0.167
Ketones	0.00383	1.2	-1.0	-0.167
Esters	0.0415	1.2	-1.0	-0.167
Ethers	0.0385	1.2	-1.0	-0.167
Halides	0.494	0.0	-0.5	0.167
R20,R21,R22,R23	0.562	0.0	-0.5	0.167

Errors may be large for Diols and Glycols. The Acids equation is not applicable to Formic acid. The reader is referred to Reid for a discussion of the method.

8.6.3 Method of Sato-Riedel

This method gives the following relation:

$$k = (1.11/M^{0.5})f(Tr)/f(Tbr)$$

with

$$f(X) = 3 + 20(1 - X)^{0.667}$$

This method gives poor results for low molecular weight or branched hydrocarbons. Errors otherwise are likely to be less than 15%. The method should not be applied for $Tr > 0.8$

8.6.4 Method of Ely and Hanley

The method of Ely and Hanley has application to the high-temperature liquid region ($Tr > 0.8$). There are few data available for high temperature liquid conductivities. The method of Ely and Hanley is probably the best method available. Error estimates are unknown.

This method is used in CHEMPAK for $Tr > 0.8$ with caution. It appears to give reasonable results for non-polar compounds. Errors with polar compounds can be large.

8.6.5 The Effect of Pressure on Liquid Conductivity

The procedure derived from Missenard as presented in Danner and Daubert is employed:

$$k/k_s = 0.98 + 0.0079 Pr Tr^{1.4} + 0.63 Tr^{1.2} Pr / (30 + Pr)$$

k = conductivity at P

k_s = conductivity at P_s

8.8 Vapor Pressure

The vapor pressure is expressed in its reduced form

$$P_{vr} = P_v/P_c$$

Reduced vapor pressure varies from very low values at freezing point to unity at the critical point.

8.8.1 Published Correlations

The experimental correlations are commonly given in the following formats:

Wagner Equation

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

with

$$X = 1 - T_r$$

FKT Equation

$$\ln(P_v) = a + b/T + c \ln(T) + dP_v/T^2$$

Antoine Equation

$$\ln(P_v) = a + b/(T + c)$$

8.8.2 Gomez-Thodos Vapor Pressure Equation

Gomez-Nieto and Thodos give the following equation:

$$\ln(P_{vr}) = B(1/T_{rm} - 1) + G(T_r^7 - 1)$$

$$G = aH + bB$$

$$a = (1 - 1/T_{br})/(T_{br}^7 - 1)$$

$$b = (1 - 1/T_{brm})/(T_{br}^7 - 1)$$

$$H = T_{br} \ln(P_c/P_b)/(1 - T_{br})$$

For non-polar compounds,

$$B = -4.267 - 221.79/(H^2.5 \exp(0.038 H^2.5)) + 3.8126/\exp(2272.33/H^3) + D$$

$$m = 0.78425 \exp(0.089315 H) - 8.5217/\exp(0.74826 H)$$

$$D = 0$$

except $D = 0.41815$ for He, 0.19904 for 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),

$$m = 0.0052 M_0.29 T_c^{0.72}$$

$$G = (2.464.M) \exp(0.0000098 MTc)$$

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

The advantages of this method are,

- fit guaranteed at $T = T_b$ and $T = T_c$
- good performance with polar compounds
- good performance over $T_r = 0.5$ to 1

In addition, tests carried out by Madison Technical Software show the clear superiority of this method especially at low temperatures over the Lee-Kesler method.

8.8.3 Lee-Kesler Vapor Pressure Equation

Lee and Kesler give the following vapor pressure equation:

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

w = acentric factor

$$f(0) = 5.92714 - 6.09648/T_r - 1.28862 \ln(T_r) + 0.169347 T_r^6$$

$$f(1) = 15.2518 - 15.6875/T_r - 13.4721 \ln(T_r) + 0.43577 T_r^6$$

The characteristics of this equation are,

- guaranteed fit at $T_r = 1$ and 0.7
- accurate for non-polar compounds

This equation is used in the Lee-Kesler and Wu & Stiel equations of state.

8.8.4 Interpolation and Extrapolation

In many cases an accurate empirical equation is known which does not extend to the critical point or to the freezing point. The approach taken here is to fit the Wagner equation by least squares to the empirical equation and use the Wagner equation to extrapolate to the freezing point and to the critical point.

Extrapolation by this method to the critical method is a very accurate procedure. Extrapolation to the freezing point is less accurate but it does provide reasonable values.

In CHEMPAK, the vapor pressure correlations set out in this section are used to provide the basic data. Empirical relations are used wherever possible.

8.14 Notation

C Specific Heat

e Expansion Coefficient

h Enthalpy

log Logarithm to base 10

ln Natural Logarithm

m Dipole Moment

M Molecular Weight

P Pressure

R Gas Constant
r Riedel Parameter
s Entropy
T Temperature
v Viscosity
V Specific Volume
w Acentric Factor
x Mole Fraction
Y Wu & Stiel Polarity Factor
Z Compressibility

Subscripts

b Boiling
c Critical
es Equation of State
f Freezing
ig Ideal Gas
L Liquid
m Mixture
0 Low Pressure
p Constant Pressure
ra Rackett
ref Reference
r Reduced
s Saturated
v Vapor
v Constant Volume

Superscripts

(o) Simple Fluid
(r) Reference Fluid
(p) Polar Fluid

8.15 References

API Technical Data Book - Petroleum Refining, 4 Vols, API, Washington DC, 1988
CRC Handbook of Chemistry and Physics, CRC Press Boca Raton 1991

- Danner R P and Daubert T E , Data Prediction Manual, Design Institute for Physical Property Data , AIChE, NY 1983
- Daubert T E and Danner R P, Physical and Thermodynamic Properties of Pure Chemicals, Data Compilation, AIChE, Hemisphere NY 1989
- Engineering Sciences Data Units (ESDU), (9 Vols Data Compilation), London, England
- Gomez-Nieto M and Thodos G, Ind Eng Chem Fundam, Vol 17, 45, 1978, Can J Chem Eng, Vol 55, 445, 1977, Ind Eng Chem Fundam, Vol 16, 254, 1977
- Hankinson and Thomson, AIChEJ, Vol 25, 653, 1979
- International Critical Tables, National Research Council, 7 Vols, McGraw-Hill, NY 1926
- Joback K G, SM Thesis, MIT, June 1984
- Keenan et al, Steam Tables, John Wiley, NY 1978
- Knapp et al, Chem Data Ser, Vol VI, DECHEMA 1982
- Kreglewski and Kay, J Phys Chem, Vol 73, 3359, 1969
- Letsou A and Stiel L I, AIChEJ, Vol 19, 409, 1973
- Le Bas G, Molecular Volumes of Liquid Chemical Compounds, Longmans Green, NY 1915
- Lee B I and Kesler M G, AIChEJ, Vol 21, 510, 1975
- Li C C, AIChEJ, Vol 22, 927, 1976
- Lucas K, Chem Ing Tech, Vol 53, 959, 1981
- Missenard F A, Rev Gen Thermodyn, Vol 101, 649, 1970
- Morris P S, MS Thesis, Polytech Inst Brooklyn, NY 1964
- Perry et al , Chemical Engineer's Handbook (various editions), McGraw Hill, NY
- Plocker U J et al, Ind Eng Chem Proc Des Dev, Vol 17, 324, 1978
- Prausnitz and Gunn, AIChEJ, Vol 4, 430 and 494, 1958
- Reid R C et al, Properties of Liquids and Gases, 3rd Ed, McGraw Hill, NY 1977, 4th Ed, McGraw Hill, NY 1987
- Schick and Prausnitz, AIChEJ, Vol 14, 673, 1968
- Spencer and Danner, J Chem Eng Data, Vol 17, 236, 1972
- Spencer, Daubert and Danner, AIChEJ, Vol 19, 522, 1973
- Stiel and Thodos, AIChEJ, Vol 8, 229, 1962
- Teja A S et al, Chem Eng Sci, Vol 33, 609, 1978, AIChEJ, Vol 26, 337 & 341, 1980, Chem Eng Sci, Vol 36, 7, 1981, Ind Eng Chem Fundam, Vol 20, 77, 1981, Chem Eng Sci, Vol 37, 790, 1982, J Chem Eng Data, Vol 28, 83, 1983, Ind Eng Chem Proc Des Dev, Vol 22, 672, 1983
- Thomson, Brobst, Hankinson, AIChEJ, Vol 28, 671, 1982
- Tyn M T and Calus W F, Processing, 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

Related Topics

Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703

Stagnation Properties

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

Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Sound Power Level.....	699
References.....	701
Pipe Material Library References.....	703

Sound Power Level

The Energy Institute identifies flow through sharp pressure drops as a primary cause of high frequency acoustic excitation in gas systems which increases the likelihood of equipment failure. This excitation produces high levels of noise but also high frequency vibrations in pipe walls that can lead to rapid fatigue and ultimately failure. Such points of failure are known to occur most often at welded discontinuities near the vibration source. Because this form of excitation and the resulting potential for failure may be especially intense within blowdown and relief subsystems, which are crucial to the overall system safety, it is important to predict the intensity of the excitation when designing the system.

This excitation occurs within pressure reducing devices, such as valves and orifices. The equation for calculating this high frequency acoustic excitation, also known as the *Sound Power Level*, is as follows ([Energy Institute 2008](#), pg 60):

$$PWL_{source} = 10 \log_{10} \left[\left(\frac{P_1 - P_2}{P_1} \right)^{3.6} W^2 \left(\frac{T}{MW} \right)^{1.2} \right] + 126.1 + SFF$$

$$SFF = \left\{ \begin{array}{ll} 6 \text{ dB} & \text{if } M_1 = 1 \\ 0 \text{ dB} & \text{if } M_1 < 1 \end{array} \right\}$$

where PWL is expressed in terms of decibels, W is the mass flowrate in kg/s, P_1 and P_2 are inlet and outlet pressures, respectively, in units of kPa, T is the temperature at the junction inlet in Kelvin, and MW is the molecular weight in kg/kmol.

The SFF term is a correction factor for sonically choked junctions, as the sonic boundary generates additional noise and vibration.

Note: This calculation is for the Sound Power Level at the source and does not account for the location of the observer, the presence of sound insulation, or diffuser mufflers within the junction. For a full system noise analysis, other sources of noise such as compressors and turbulence in pipe flow should also be considered.

Sound Power Level can be calculated for the following junction types:

- Valve (Inline)
- Orifice (Inline)
- Bend
- Area Change
- Check Valve
- Control Valve
- Venturi
- Screen
- Relief Valve (Inline)
- General Component

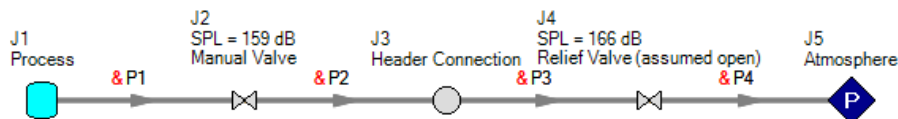


Figure 1: Calculated Sound Levels Displayed With Junction Names

Warnings and Design Alerts

A [Design Alert](#) can be set up for other thresholds if desired, but by default Arrow will display a [warning when Sound Power Levels exceed 155 dB](#).

Note: This warning will only be shown when Sound Power Level is selected as an output parameter in [Output Control](#).

The 155 dB threshold is important for its role in calculating Likelihood of Failure (LOF) due to high frequency acoustic excitation when using the method defined by the Energy Institute. When the PWL is greater than 155 dB, a user will need to determine whether the junction in question is

a valve with low noise trim fitted. If so, the *PWL* will need to be corrected accordingly. When *PWL* at the source is greater than 155 dB, the LOF analysis proceeds with calculations of *PWL* at nearby welded discontinuities and mitigation efforts or changes to the process design may be in order.

Related Topics

Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
References.....	701
Pipe Material Library References.....	703
The following junctions have a predicted Sound Power Level greater than 155 dB.....	723

References

1. ANSI/API 526 7th Edition - Flanged Steel Pressure-relief Valves, 2017, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA.
2. 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
3. Anderson, J.D., Jr., *Modern Compressible Flow: With Historical Perspective*, McGraw-Hill, New York, NY, 1982.
4. 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.
5. Bejan, A., *Advanced Engineering Thermodynamics*, John Wiley & Sons, New York, NY, 1988.
6. Çengel Y., *Heat and Mass Transfer, A Practical Approach*, 3rd Edition, McGraw-Hill, New York, NY, 2007.
7. Crane Co., *Flow of Fluids Through Valves, Fittings, and Pipe*, Technical Paper No. 410, Crane Co., Joliet, IL, 1988.
8. Eckert, E.R.G., and Drake, R., *Analysis of Heat and Mass Transfer*, McGraw-Hill, New York, NY, 1972.
9. Idelchik, I. E., *Handbook of Hydraulic Resistance*, 4th edition, Begell House, Redding, CT, 2007.
10. Incropera, F.P. et. al, *Fundamentals of Heat and Mass Transfer*, John Wiley & Sons, NJ 2007.
11. Ingersoll-Dresser Pumps, *Cameron Hydraulic Data*, Liberty Corner, NJ, 1995.

12. Ito, H., "Friction factors for pipe flow," *Journal of Basic Engineering*, Vol. 81, pp. 123-126, 1959.
13. Jeppson, R.W., *Analysis of Flow in Pipe Networks*, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.
14. Keenan, J.H., *Thermodynamics*, John Wiley & Sons, Inc., New York, NY, 1941.
15. Miller, D. S., *Internal Flow Systems*, 2nd edition, Gulf Publishing Company, Houston, TX, 1990.
16. Nayyar, M. L., *Piping Handbook*, 7th edition, McGraw-Hill, New York, NY, 2000.
17. Perry, R. H., Green, D. W., *Perry's Chemical Engineers' Handbook*, 7th Edition, McGraw-Hill, New York, NY, 1997.
18. Saad, M.A., *Compressible Fluid Flow*, 2nd Edition, Prentice-Hall, Englewood Cliffs, NJ, 1993.
19. Shapiro, A.H., *The Dynamics and Thermodynamics of Compressible Fluid Flow*, 2 vols., Ronald, New York, NY, 1953.
20. Smith, J.M., van Ness, H.C., and Abbott, M.M., *Introduction to Chemical Engineering Thermodynamics*, 6th edition, McGraw-Hill, New York, NY, 2001.
21. Val-Matic Valve and Manufacturing Corp., "Surge Control in Pumping Systems," Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.
22. Walters, T.W., "Gas-Flow Calculations: Don't Choke", *Chemical Engineering*, Chemical Week Associates, Jan. 2000, pp. 70-76.
23. 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.
24. 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.
25. Energy Institute, *Guidelines for the Avoidance of Vibration Induced Fatigue Failure in Process Pipework*, 2nd Edition, Energy Institute, London, UK, 2008.

Related Topics

Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
Pipe Material Library References.....	703

Pipe Material Library References

Prior to AFT Arrow 6, pipe material properties were compiled from a variety of unofficial sources. Many of the original sources are now obsolete. Beginning in AFT Arrow 6, pipe material properties are taken directly from national/international standards such as: ASTM, ANSI, AWWA, and ASME. Several new material standards are being 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 AFT Arrow 5 is still available, but is now called "AFT Customary".

Maintaining several independent pipe libraries facilitates including or excluding specific standards. The [Library Browser](#) 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
 - Additional References = ASTM D2996 - 17
- **HDPE - AFT Customary -**
 - Source = Applied Flow Technology
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayar, 7th ed, Table A3.2
- **PVC - ASTM -**

- Source = ASTM D1785-06
- Additional References = Thermal data from: Materials Science and Engineering, Callister, Table C.4
- **PVDF - AFT Customary -**
 - Source = George Fischer Engineering handbook page 425
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Rectangular Duct - AFT Customary -**
 - Source = Applied Flow Technology
- **Stainless Steel - ANSI -**
 - Source = ASME-B36_19M_2004
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conduct References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Type 304
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ration Referenecs = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for Type 304 @ 20oC
- **Stainless Tubing - AFT Customary -**
 - Source = Applied Flow Technology
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Steel - AFT Customary -**
 - Source = Applied Flow Technology
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- **Steel - ANSI -**
 - Source = ASME B36.10-2004
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1

- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
- Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - DIN 2391-1 -**
 - Source = DIN 2391-1-1994
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for 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 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 10217-1 (s1) -**
 - Source = BS EN 10217-1-2002
 - 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 10217-2 (s1) -**
 - Source = BS EN 10217-2-2002
 - 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 10217-4 (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 Comerial 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 Comerial 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

Related Topics

Libraries.....	613
Library Manager.....	614
Library Browser.....	614
Network Library Overview.....	629
Library Verification.....	634
Library Locations in General Preferences.....	636

Fluid Library.....	619
Pipe Material Library.....	620
Junction Library.....	618
Insulation Library.....	623
Fittings & Losses Library.....	624
Theory and References.....	639
Nomenclature.....	639
Review of Compressible Flow Theory.....	642
Compressible Flow Theory in Single Pipes.....	657
Sonic Choking.....	665
Role of Pressure Junctions - Detailed Discussion (Long).....	674
Chempak Technical Information - Detailed Discussion.....	686
Stagnation Properties.....	698
Sound Power Level.....	699
References.....	701

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

Exporting to Excel.....	422
GSC in Output.....	493
Cost Report.....	485
Troubleshooting.....	729

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

FCV/PRV/PSV Could Not Control Pressure - Pressure Was Added to Achieve Control Flow/Pressure...

A [Critical Warning message](#) in AFT Arrow

Full message is:

FCV/PRV/PSV could not control pressure/flow at Junction X. Pressure was added to achieve control flow. See Valve Summary for more information.

A [flow or pressure control valve](#) could not control pressure or flow and the setting is to Always Control (Never Fail). In order to achieve the control flow or pressure, the valve must add pressure and act like a compressor/fan.

The Valve Summary in the [Output window](#) gives more information.

This warning is useful for resolving operational problems in your system.

Global energy balance disabled by user in Advanced Parameters panel

A [Critical Warning message](#) in AFT Arrow

The displayed results do not represent a valid converged solution because the Energy Balance option was disabled by the user on the [Advanced Parameters panel](#) in the [Analysis Setup window](#).

Related Topics

Advanced Parameters Panel.....	155
Global mass balance disabled by user in Advanced Parameters panel.....	710

Global mass balance disabled by user in Advanced Parameters panel

A [Critical Warning message](#) in AFT Arrow

The displayed results do not represent a valid converged solution because the Mass Balance option was disabled by the user on the [Advanced Parameters panel](#) in the [Analysis Setup window](#).

Related Topics

Advanced Parameters Panel.....	155
Global energy balance disabled by user in Advanced Parameters panel.....	710

Junction X Had Reverse Flow - Compressor/Fan Pressure Could Not Be Predicted

A [Critical Warning message](#) in AFT Arrow

This message displays for [centrifugal compressors/fans](#) when the flowrate goes backwards through the compressor/fan. The pressure rise under such conditions is unknown, and the zero-flow value is used.

Junction X uses normal or standard flow units like Y in a flow path with dynamic gas mixing...

A [Critical Warning message](#) in AFT Arrow

Full message is:

Junction X uses normal or standard flow units like Y in a flow path with dynamic gas mixing. Flows cannot be calculated accurately unless GSC module is used.

When standard flowrates (like scfm) or normal flowrates (like Nm³/hr) are used as flow demands and the model has dynamic gas mixing, the "standard" or "normal" density cannot be obtained because there is no certain gas mixture at the required flow rate. Without a standard/normal density then the flowrate cannot be converted into a true mass flowrate.

The user should either change the model to use known pressures (like [Assigned Pressures](#) or [Tanks](#)) or use the [GSC module](#) to obtain the desired flowrate.

Pipe Sequences of zero friction exist, flow rate cannot be calculated

A [Critical Warning message](#) in AFT Arrow

This message is displayed if all of the pipes in a flow path are defined as frictionless by using the zero-length connector option, or the "frictionless" friction model.

At least one pipe in every flow path needs to include frictional losses in order to perform flow calculations.

Required flow to supply the pressure control valve at junction X is above sonic flow and cannot be supplied

A [Critical Warning message](#) in AFT Arrow

You have specified a pressure at a [pressure control valve](#) which requires a flow which is physically impossible because of [sonic choking](#).

Related Topics

Sonic Choking.....665

Setting relaxation lower than relative pressure/flow tolerance gives unreliable results

A [Critical Warning message](#) in AFT Arrow

Because of the nature of relaxation and relative solution tolerance, setting the relaxation to a value close to or lower than relative tolerance can lead to false convergence and invalid results. AFT Arrow alerts the user to this when inputting data in the [Tolerance 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 Arrow

You have specified an outflow which is physically impossible because of [sonic choking](#). All other results in your model except for the sequence of pipes leading up to the offending [Assigned Flow junction](#) will have an acceptable solution.

This message can also occur if you are using [Flow Control Valves](#) or [Compressor/Fans](#) modeled as a fixed flowrate.

Related Topics

Sonic Choking.....665

These results do not represent a converged solution

A [Critical Warning message](#) in AFT Arrow

This message alerts you to the fact that you chose to have results displayed for an unconverged solution. You should only use the displayed results for purposes of determining the reasons for nonconvergence. Under no circumstances should you actually use any of the results for engineering purposes. See the [Troubleshooting](#) topic or email your model to [AFT Support](#) for assistance.

Related Topics

Troubleshooting..... 729
Diagnostics Window..... 730
Use the Model Data Window..... 731
Transfer Results to Initial Guesses..... 734
Poor Compressor/Fan Curve Fits..... 735
Troubleshooting Resistance Curves..... 736
Use the Output Window Sort Feature..... 737

Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
Identifying and Restoring Corrupted Model Files.....	747

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

Chempak temperature and/or pressure is out of range for the following

A [Warning message](#) in AFT Arrow

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 indicates that the results are probably not valid. It also may indicate an input error in the model because of the extreme temperature or pressure state.

Compressor X is operating at a corrected flow greater/lower than the entered data

A [Warning message](#) in AFT Arrow

There are two variations of this Warning:

- *Compressor X (Junction Name) is operating at a corrected flow **greater than** the entered data.*
- *Compressor X (Junction Name) is operating at a corrected flow **lower than** the entered data.*

The listed compressor junction is using a [Compressor Map](#), and the final solution had a corrected mass flowrate that was higher than the Stall Line or lower than the Surge Line. Therefore, the operating point is outside the range of the Compressor Map, and the solution may have some uncertainty or error.

This Warning is similar to the [Warning about corrected speed](#). This Warning (about flow) describes the left/right boundaries of the Compressor Map.

Compressor X is operating at a corrected speed greater/lower than the entered data

A [Warning message](#) in AFT Arrow

There are two variations of this Warning:

- *Compressor X (Junction Name) is operating at a corrected speed **greater than the entered data**.*
- *Compressor X (Junction Name) is operating at a corrected speed **lower than the entered data**.*

The listed compressor junction is using a [Compressor Map](#), and the final solution had a corrected speed that was higher than the data entered for the Compressor Map. Therefore, the operating point is outside the range of the Compressor Map, and the solution may have some uncertainty or error.

This Warning is similar to the [Warning about corrected flow](#). This Warning (about speed) describes the top/bottom boundaries of the Compressor Map.

Compressor X is operating at a non-dimensional flow greater/lower than the entered data

A [Warning message](#) in AFT Arrow

There are two variations of this Warning:

- *Compressor X (Junction Name) is operating at a non-dimensional flow **greater than the entered data**.*
- *Compressor X (Junction Name) is operating at a non-dimensional flow **lower than the entered data**.*

The listed compressor junction is using a [Compressor Map](#), and the final solution had a non-dimensional mass flowrate that was higher than the Stall Line or lower than the Surge Line. Therefore, the operating point is outside the range of the Compressor Map, and the solution may have some uncertainty or error.

This Warning is similar to the [Warning about non-dimensional speed](#). This Warning (about flow) describes the left/right boundaries of the Compressor Map.

Compressor X is operating at a non-dimensional speed greater/lower than the entered data

A [Warning message](#) in AFT Arrow

There are two variations of this Warning:

- *Compressor X (Junction Name) is operating at a non-dimensional speed **greater than** the entered data.*
- *Compressor X (Junction Name) is operating at a non-dimensional speed **lower than** the entered data.*

The listed compressor junction is using a [Compressor Map](#), and the final solution had a non-dimensional speed that was higher than the data entered for the Compressor Map. Therefore, the operating point is outside the range of the Compressor Map, and the solution may have some uncertainty or error.

This Warning is similar to the [Warning about non-dimensional flow](#). This Warning (about speed) describes the top/bottom boundaries of the Compressor Map.

Concentration at Junction X may not be balanced

A [Warning message](#) in AFT Arrow

This warning alerts you to the fact that the converged results may be incorrect. Once a converged solution is obtained, AFT Arrow checks all junctions to ensure a concentration balance exists. It compares the imbalance to the flow [tolerance](#) you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the concentration out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to reduce your tolerances further and rerun the model.

If the magnitude of the concentration out of balance is large, then you have a converged solution that is incorrect. This can occur in rare circumstances. When this happens, then it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and perform hand calculations on the concentration balance at the affected junctions to verify whether an acceptable solution exists.

Control valve open percentage outside valid range of 0-100%

A [Warning message](#) in AFT Arrow

A [control valve](#) which includes open percentage data is operating outside the range of 0-100%.

Energy may not be balanced at Junction X

A [Warning message](#) in AFT Arrow

This warning alerts you to the fact that the converged energy balance results may be incorrect. Once a converged solution is obtained, AFT Arrow checks all junctions to ensure an energy balance exists. It compares the imbalance to the mass flow weighted enthalpy [tolerance](#) you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to reduce your tolerances further and rerun the model.

If the magnitude of the energy out of balance is large, then you have a converged solution that may be incorrect. This can occur in rare circumstances. When this happens, then it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and 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 you specify, 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 at Junction X could not control flow

A [Warning message](#) in AFT Arrow

The [Flow Control Valve](#) was not able to control the flow, and backflow may have occurred.

Flow at Junction X may not be balanced

A [Warning message](#) in AFT Arrow

This warning alerts you to the fact that the converged results may be incorrect. Once a converged solution is obtained, AFT Arrow checks all junctions to ensure a flow balance exists. It compares the imbalance to the flow [tolerance](#) you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to reduce your tolerances further and rerun the model.

If the magnitude of the flow out of balance is large, then you have a converged solution that is incorrect. This can occur in rare circumstances. When this happens, then it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and perform hand calculations on the flow 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 Arrow

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, 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

GSC Goal may not be valid

A [Warning message](#) in AFT Arrow

Full message is:

GSC Goal may not be valid. User goal = X. Final goal result = Y

This is a warning message that appears when the [GSC module](#) search was not able to meet a [goal](#). See [Goal Seek and Control Troubleshooting](#).

Inlet and outlet pipes at Junction X do not have the same stagnation enthalpy

A [Warning message](#) in AFT Arrow

The stagnation enthalpy going into a junction should equal the stagnation enthalpy out if there are no flow splits or heat input. This warning tells you that the values differ. Something is wrong somewhere and results are likely incorrect.

Junction X Compressor/Fan Added Negative Pressure

A [Warning message](#) in AFT Arrow

Full Message text is:

Junction X Compressor/Fan added negative pressure.

A compressor/fan junction with a fixed flow rate had a pressure loss across the [compressor/fan junction](#). This warning is meant to alert the user that the compressor/fan is not adding pressure as they might be expecting.

Junction X Exceeded Runout Flow Rate...

A [Warning message](#) in AFT Arrow

[Centrifugal compressor/fan curves](#) typically have a point at which they no longer operate on the curve. This may be called "end of curve" or "runout". Data defining runout for a particular compressor/fan is given by the manufacturer. Such data has been defined for this compressor/fan in the [configuration window](#), and the system is operating past the runout flow rate. Thus the results are invalid, because the compressor/fan cannot actually operate at that point.

Maximum system pressure exceeded enthalpy table's maximum pressure of X

A [Warning message](#) in AFT Arrow

Full message 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 library is being used with the Table Enthalpy model, and that the maximum 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 model exceeds the maximum pressure data is available for in the AFT Standard fluid library, 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 library such as NIST Refprop or Chempak with a higher maximum pressure.

Physical flow area is less than the sonic area

A [Warning message](#) in AFT Arrow

Full message 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 Arrow does not 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 Arrow needs to know the C_dA .

When this warning is displayed, AFT Arrow 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 Arrow 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.

Related Topics

Sonic Choking.....665

Pipe/Junction X property calculation exceeded maximum local iteration limit

A [Warning message](#) in AFT Arrow

The final network solution was not within tolerance for a local iteration. The local iteration limit set in the [Iterations 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 Arrow attempts the network solution again up to five times in a row. If the out of tolerance situation remains, AFT Arrow 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 Arrow 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, 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 Analysis Setup window 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 Arrow

The final network solution was not within tolerance for a local iteration. This was determined by AFT Arrow’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 Arrow attempts the network solution again up to five times in a row. If the out of tolerance situation remains, AFT Arrow 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 Arrow 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](#), but rather an internal “hard-wired” tolerance that has been set by the software developers and arrived at through experience. 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.

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 Analysis Setup window frequently solves the problem.

Another potential cause of this warning is if an orifice junction is causing [choked flow](#), but no [CdA for Sonic Choking](#) value is specified.

Pipe X temperature is equal to or greater/less than the max/min temperature limit of Y

A [Warning message](#) in AFT Arrow

There are two variations of this Warning:

- The following pipes had temperature **equal to or less than the minimum** temperature limit of X set in the Temperature Solver Limits panel under Steady Solution Control in the Analysis Setup window. Results are not reliable.
- The following pipes had temperature **equal to or greater than the maximum** temperature limit of X set in the Temperature Solver Limits panel under Steady Solution Control in the Analysis Setup window. Results are not reliable.

The user has specified limits in the [Pressure and Temperature Solver Limits panel](#) of the [Steady Solution Control](#) 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 at Junction X could not control pressure

A [Warning message](#) in AFT Arrow

The [Pressure Reducing Valve](#) was not able to control pressure, and backflow may have occurred.

PSV at Junction X could not control pressure

A [Warning message](#) in AFT Arrow

The [Pressure Sustaining Valve](#) was not able to control pressure, and backflow may have occurred.

Separator junction X is trying to remove Y but none is entering junction

A [Warning message](#) in AFT Arrow

A [Separator](#) junction is specified to remove a certain gas but none is flowing into the junction. The input data should be reviewed for the model.

Separator junction X is trying to remove Y of Z but only A is entering junction

A [Warning message](#) in AFT Arrow

A [Separator](#) junction is specified to remove a certain amount of gas but there is not enough flowing into the junction. The input data should be reviewed for the model.

Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range

A [Warning message](#) in AFT Arrow

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 following junctions have a predicted Sound Power Level greater than 155 dB

A [Warning message](#) in AFT Arrow

Note: This warning will only be shown when Sound Power Level is selected as an output parameter in [Output Control](#).

[Sound Power Level](#) is calculated in the manner prescribed by the [Energy Institute Guidelines for the Avoidance of Vibration Induced Fatigue Failure in Process Pipework](#).

The 155 dB threshold is important for its role in calculating Likelihood of Failure (LOF) due to high frequency acoustic excitation when using the method defined by the Energy Institute. When the *PWL* is greater than 155 dB, a user will need to determine whether the junction in question is a valve with low noise trim fitted. If so, the *PWL* will need to be corrected accordingly. When *PWL* at the source is greater than 155 dB, the LOF analysis proceeds with calculations of *PWL* at nearby welded discontinuities and mitigation efforts or changes to the process design may be in order.

Related Topics

Sound Power Level..... 699

Unique Solution Not Possible - Relief Valve Cycles

A [Warning message](#) in AFT Arrow

The [relief valve](#) cycles between its open and closed state. When closed, the pressure exceeds its cracking pressure and causes it to open. When open, the pressure drop resulting from flow causes the pressure to fall below the cracking pressure and so it closes. When the relief valve changes states, the steady-state Solver must re-run the simulation, but since the relief valve cycles between open and closed each time, a unique solution is not possible. Double check the inputs for the relief valve, the model boundary conditions, and other junction input data.

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

Check Valve closed at Junction X

A [Caution message](#) in AFT Arrow

This is a cautionary notice that a [check valve](#) closed, and the Solver will automatically re-run the model.

Flow through Junction X is negative. Loss factors may be in error.

A [Caution message](#) in AFT Arrow

This caution alerts you to flow in your junctions which is in the opposite direction from the connecting pipes reference directions. Any [loss information](#) you specified for junctions may be in error because it was defined assuming flow in the other direction.

Flow solution for Junction X is above/below the maximum/minimum value specified for the resistance curve. Flow rate: Y

A [Caution message](#) in AFT Arrow

There are two variations of this Caution:

- Flow solution for Junction X (Junction Type) is **above the maximum** value specified for the resistance curve. Flow Rate: Y.
- Flow solution for Junction X (Junction Type) is **below the minimum** value specified for the resistance curve. Flow Rate: Y.

This caution alerts you that the calculated flow through junction X is outside the range of the raw data that was entered in the [Resistance Curve](#). The Solver will use the [Curve Fit](#) of the entered data for all calculations, including pressure losses at flowrates that are extrapolated beyond the original data.

The user should check if the curve fit is still a good approximation of the data below the minimum data point, or above the maximum data point.

It is good practice to have the Resistance Curve pass through the point (0,0) since zero flow will result in zero pressure loss. For more information, see the help topic on [Troubleshooting Resistance Curves](#).

Junction X is downstream of a choking point

A [Caution message](#) in AFT Arrow

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](#), [Tank](#), [Spray Discharge](#)). 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 Arrow does not model the dynamics of how the flow stream exits into the ambient conditions.

Note No Scenarios have been set in the Multi-Scenario Output display

A [Caution message](#) in AFT Arrow

This caution is shown when multi-scenario output has been selected in [Output Control](#) but no [scenarios](#) have been selected. As a result, no output can be shown.

Pipe X length is shorter than the elevation change

A [Caution message](#) in AFT Arrow

This caution alerts you to a possible inconsistency in your input. When this warning occurs, you have defined a pipe with a length that is shorter than the elevation change between its connecting junctions. This inconsistency may be an input error or a result of a component which itself has an elevation change. AFT Arrow only accepts a single elevation per junction, so if the error is due to the latter possibility, pipe lengths or junction elevations should be adjusted accordingly.

Pipe X steady state Reynold's Number is less than 100. Minor Losses may be significantly lower than expected.

A [Caution message](#) in AFT Arrow

When Reynolds Numbers fall to extremely low values (i.e. less than 100) the fluid enters a flow regime in which the system losses behave in an unintuitive fashion. Based on relationships depicted by the moody diagram, as the Reynolds Number decreases when below 2000, the friction factor and resulting pressure loss increases. In practice, this relationship breaks down and system losses will begin to decrease somewhere between a Reynolds Number of 10 and 100. Due to limited academic knowledge on this subject, and since the Solver is not currently able to model this regime, a caution message is given.

Spray Discharge Junction X closed because inflow occurred and Special Condition - No Inflow was set

A [Caution message](#) in AFT Arrow

This is a cautionary notice that a Spray Discharge junction closed because the Solver determined that flow would enter the system, but the No Inflow [Special Condition](#) was set on the Optional tab of the [Spray Discharge Properties window](#). Therefore, the Spray behaved like a [Check Valve](#) and prevented the flow from entering the system.

The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace

A [Caution message](#) in AFT Arrow

Full Message Text

The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace:

Tee/Wye: J1, J2, J3, J4, J5, J6, J7, J8, J9, J10 and X more junctions. See the Model Diagnostics window for more information.

Explanation

The listed [Tee/Wye junctions](#) are using the [Detailed Loss Model](#) with the Arrangement Type set to "Auto", however, the automatic pipe orientation saved for the Tee/Wye Properties window is not consistent with the current orientation on the Workspace. For the K factors to be calculated correctly, the connections for pipes A, B, and C must be defined correctly. For example, on a 90 degree Tee, the branching flow pipe (C) in the Tee/Wye Properties window must match the intended branching pipe on the Workspace. If the branching pipe C was switched with the straight run pipe B, then the K factors would be switched. This can give incorrect results which may propagate through the model and compound with other Tee/Wye junctions since the Detailed Loss Model is dependent on flows and pressures.

A full list of the affected junctions can be found on the [Model Diagnostics window](#) (Tools menu > Model Diagnostics > Junctions > Tee/Wye > "Tee orientation inconsistent to workspace?").

Required Action

First, it is recommended to align the pipes on the Workspace in a way that resembles the true geometric orientation of the junction as depicted by either the [2D or Isometric grid](#). If there is a 90 degree angle, draw the pipes in a way that depicts this in the proper way. Furthermore, point the pipe [Reference Positive Flow Direction](#) towards the expected flow direction. The Auto

Arrangement Type will utilize the directions and angles between each pipe when attempting to automatically configure the pipe connections. Therefore, following these modeling practices can increase the success rate of properly identifying and applying the K factors to the correct pipes. The tips below can be used to help quickly apply these principles:

- The pipe direction can be switched by using the Reverse Pipe Direction icon on the Workspace Toolbar or using the F3 keyboard shortcut.
- Display the Workspace Grid and enable Snap to Grid from the Arrange menu.
- Rotate the Tee/Wye junction icon such that it is consistent with the pipes by right-clicking the icon and selecting either "Auto-rotate Icon" or "Customize Icon".

Next, open each Tee/Wye Properties window listed in the Caution message or in the Model Diagnostics window, configure and verify the proper pipe connections, click OK, and re-run the model. If needed, the Arrangement Type can be switched to "User" to guarantee that the junction retains the proper configuration.

If the losses due to the Tees/Wyes are negligible compared to the total system pressure drop, the user can consider switching to the Simple Loss Model which assumes the junction is lossless. This would be equivalent to morphing the junction into a [Branch](#).

Transient data did not extend to Stop Time at Junction X

A [Caution message](#) in AFT Arrow

([XTS Module Only](#))

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 Duration panel](#). 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.

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

A [Information message](#) in AFT Arrow

The flow at the listed Junctions is in a laminar regime. Further loss corrections may be necessary.

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

Diagnostics Window	730
Use the Model Data Window	731
Transfer Results to Initial Guesses	734
Poor Compressor/Fan Curve Fits	735
Troubleshooting Resistance Curves	736
Use the Output Window Sort Feature	737
Tee/Wye Junction Complexity	738
Lower the Flow Rate Relaxation	739
Try Absolute Tolerance	740
Make Initial Flow Rate Guesses for Pipes	741
Review the Iteration History	742
Change Boundary Conditions	742
Turn Off Parts of the Model	743
Break a Large Model Into Submodels	744
Change Solution Method	745
Try a Different Fluid, Equation of State or Accuracy Option	745
Using Lumped Adiabatic as a Starting Point	746

Related Blogs

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

Diagnosics Window

The Model Diagnostics window is a feature introduced in AFT Arrow 8. It 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 most 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 most common issues are included.

Data in the Model Diagnostics Window 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 the Flow Tolerances and the Environmental Properties for the model have been modified from the default, which is unusual. Model Diagnostics also indicates the junction J5 is using the detailed tee loss model, which can prevent convergence in some cases if not defined properly.

Data Source	Diagnostic Description	Result	Potential Impact	Diagnostic Details
Solution Control			Medium	
Tolerance	Unusual tolerance values?	False	None	
Tolerance	Relaxation on order of Tolerance?	False	None	
Tolerance	Pressure Tolerance modified?	False	None	
Tolerance	Flow Tolerance modified?	True	Medium	The Flow Tolerance is not Relative or the relative tolerance value has been modified.
Tolerance	Temperature/Enthalpy Tolerance modified?	False	None	
Tolerance	Concentration Tolerance modified?	False	None	
General	Relaxation Modified?	False	None	
General	Iteration Limits Modified?	False	None	
General	Matrix Method Modified?	False	None	
General	Pressure Limits Imposed?	False	None	
General	Temperature Limits Imposed?	False	None	
Advanced	Advanced Parameters Modified?	False	None	
Advanced	Heat Transfer Sectioning modified?	False	None	
System Properties			Low	
Viscosity Model	Non-Newtonian Viscosity Model?	False	None	
System Data	Environmental Properties modified?	True	Low	Ambient Pressure, Gravitational Acceleration, or Standard Conditions have been modified.
System Data	Transition Reynolds Numbers modified?	False	None	
System Data	Laminar and Non-Newtonian Corrections modified?	False	None	
User Options/General			None	
Pipes			Unknown	
Junctions			High	
Tee/Wye	Detailed Tees in use?	True	High	The following Tees are using the Detailed loss model.
J5 (Tee or Wye)				

Figure 1: Example of the Model Diagnostics window

Related Topics

Troubleshooting	729
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Use the Model Data Window

Besides its usefulness for documentation purposes, the [Model Data window](#) is a powerful tool for finding input errors in your model.

Look at the model in Figure 1. If you suspected an input error in one of the pipes, 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.

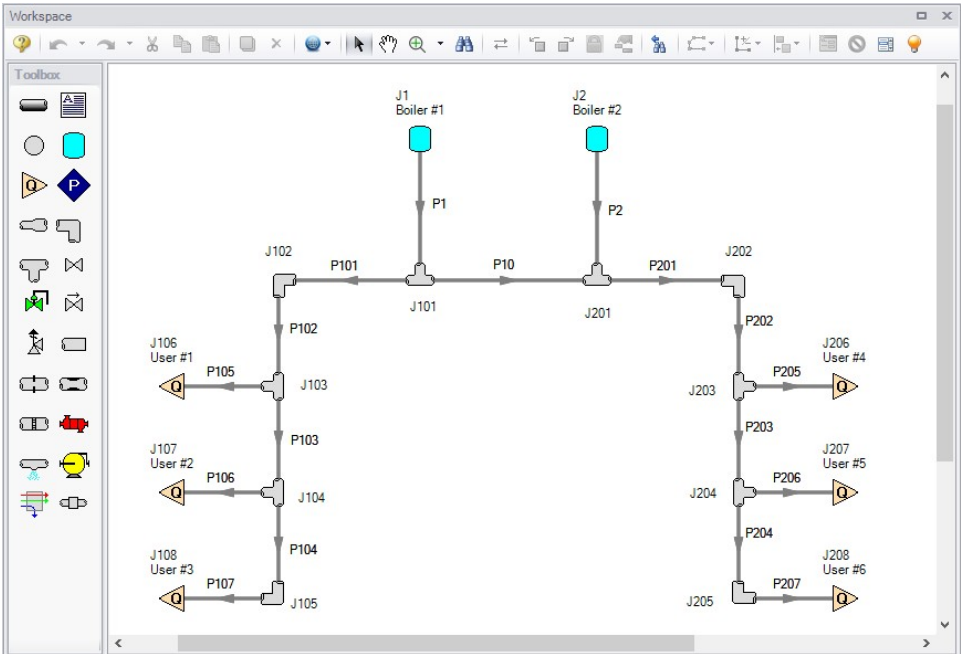


Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown)

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 (P102) has length data entered in miles rather than feet. This is not automatically an input error, as AFT Arrow accepts pipe input with mixed engineering units. However, the user often intends to use consistent units, so this probably is an error.

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744

Change Solution Method	745
Try a Different Fluid, Equation of State or Accuracy Option	745
Using Lumped Adiabatic as a Starting Point	746
These results do not represent a converged solution	712
Identifying and Restoring Corrupted Model Files	747

Related Blogs

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

Transfer Results to Initial Guesses

AFT Arrow's iterative [Solver](#) takes initial guesses (also called initial conditions) and applies the principles of mass, momentum and energy 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 answer is obtained, you can choose to take these results and use them as the initial guesses for a subsequent run. Using the converged results as initial guesses often gives faster convergence in subsequent runs because you are starting the [Solver](#) at or very close to the actual solution.

This feature is called Transfer Results to Initial Guess, and it is on the Output window [Edit menu](#) and [Toolbar](#).

Related Topics

Optional Input for Junctions	228
Optional Input for Pipes	214
Make Initial Flow Rate Guesses for Pipes	741
Troubleshooting	729
Diagnostics Window	730
Use the Model Data Window	731
Poor Compressor/Fan Curve Fits	735
Troubleshooting Resistance Curves	736
Use the Output Window Sort Feature	737
Tee/Wye Junction Complexity	738
Lower the Flow Rate Relaxation	739
Try Absolute Tolerance	740
Review the Iteration History	742
Change Boundary Conditions	742
Turn Off Parts of the Model	743
Break a Large Model Into Submodels	744
Change Solution Method	745
Try a Different Fluid, Equation of State or Accuracy Option	745
Using Lumped Adiabatic as a Starting Point	746

These results do not represent a converged solution..... 712
 Identifying and Restoring Corrupted Model Files..... 747

Poor Compressor/Fan Curve Fits

The [Compressor/Fan Properties window](#) allows curve fits of compressor/fan data up to fourth order. Depending on the number of data points entered, compressor/fan curve fits that look like the orange curve in Figure 1 are possible.

AFT Arrow interprets flow rates 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 Arrow will attempt to warn the user when a model containing a compressor/fan curve fit as seen in Figure 1 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 not operating to the right of the inflection point.

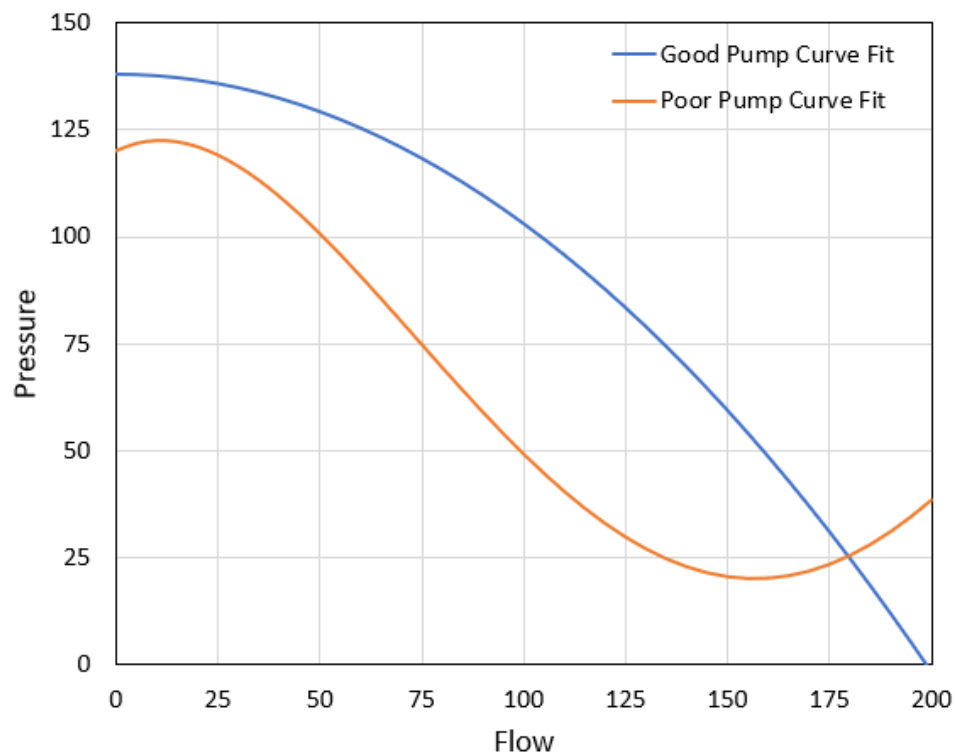


Figure 1: A good compressor performance curve fit in blue and a poor compressor performance curve fit in orange. All pump curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.

Related Topics

Compressors & Fans..... 253
 Compressor/Fan Configuration..... 263

Compressor/Fan Efficiency Data	267
Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

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.

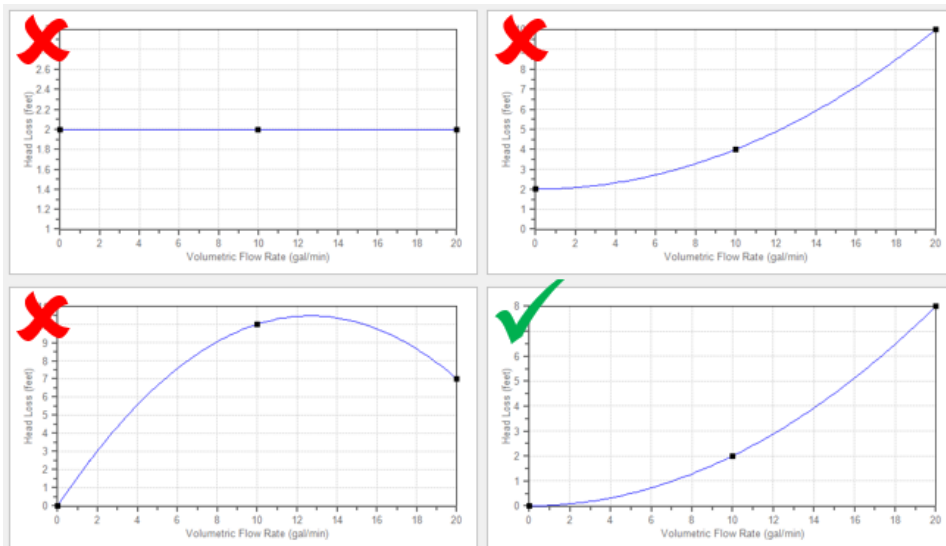


Figure 1: Example resistance curves showing correct and incorrect curve types

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.....	237
Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Use the Output Window Sort Feature

The [Sort feature](#) in the [Output Window](#) can be used 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.

Related Topics

Output Window.....	359
Output Control.....	364
Output Parameters.....	370
Applied Standards.....	371

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Tee/Wye Junction Complexity

[Tee/wye junctions](#) contain powerful and sophisticated models for determining the pressure losses at flow splits and converging flow streams. This power comes at the cost of additional model complexity. The complexity can lead to convergence problems in models with numerous tees. Here are a couple of recommendations.

Tee/Wye junctions have two loss options: Simple and Detailed. The Simple loss model ignores losses at the tee. The Detailed model uses the correlation summarized in the [Tee/Wye Loss](#) topic. The potential convergence problems occur when using the Detailed loss model.

The potential problem is not with the Detailed tee model itself. Rather, it is due to multiple tees coupling together in such a way that AFT Arrow cannot converge on all of the flow splits.

You can quickly and easily check if a convergence problem is being caused by tees. Using the [Global Junction Edit window](#), you can change all tees to the Simple loss model and rerun the model. If it now converges, the problem was due to the use of Detailed tee modeling.

Experience has shown that this problem can frequently be resolved by using [absolute tolerance criteria](#) in the Tolerance panel rather than relative tolerance.

You also have the option of just using the Simple (lossless) tee model for all tees. In many systems the loss due to tees/wyes is negligible, especially for systems which see significant loss from other components that dominate the system.

Related Topics

Tee/Wye.....	295
Tee/Wye Loss Model.....	296

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Related Blogs

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

[Give me some details about Detailed Tees!](#)

Lower the Flow Rate Relaxation

The [Iterations panel](#) allows you to change the [numerical relaxation](#) parameter for both flow rate and pressure. It is recommended that you use only 0.5 or 1.0 for pressure relaxation. But lower values of flow rate relaxation can be very helpful.

A good first step is to lower the flow rate relaxation to 0.05 or 0.1. By lowering the flow rate relaxation you improve AFT Arrow's ability to obtain convergence. The cost is slower run times. In some cases, accepting the slower run times that come with flow rate relaxation is the only means of obtaining convergence. You can lower the relaxation values as low as you want, but it is recommended it not be lowered below 0.01.

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736

Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Related Blogs

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

Try Absolute Tolerance

Most models converge more reliably with relative tolerances, which is the default in the [Tolerance panel](#). However, some models converge much better with [absolute tolerance](#). What this usually means is that 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.

If a model will not converge with relative tolerance, you can try changing to absolute tolerance. Even better, try the option to converge on either absolute or relative, which will offer the most flexibility.

Related Topics

Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744

Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Related Blogs

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

Make Initial Flow Rate Guesses for Pipes

If you do not provide any [initial guesses](#), AFT Arrow will make its own guesses. On occasion, the AFT Arrow guesses are far away from reality. For some models, providing a crude initial guess for all the pipe flow rates in the system can help the model converge.

The crude 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.

Related Topics

Optional Input for Junctions.....	228
Optional Input for Pipes.....	214
Transfer Results to Initial Guesses.....	734
Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Review the Iteration History

The [Iteration History window](#) can be opened from the [Solution Progress window](#). You can use this to see which pipes or junctions are farthest out of tolerance, which may direct your troubleshooting efforts toward these elements.

Related Topics

I.....	329
Iterations Panel.....	144
Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Change Boundary Conditions

Changing the boundary pressures or flow rates can help give insight into what is giving the model trouble.

If increasing the flowrate in a system resolves a convergence issue, it may indicate the convergence issues in the original system are due to narrow tolerances around low flowrates. In this case, using an [Absolute Tolerance](#) may be beneficial. If decreasing the flowrate resolves a convergence issue, it may indicate the original flowrate is above the [sonic flowrate](#). In this case, it may help to determine what the sonic flowrate is, and compare the desired flowrate against that value to determine if the desired value is physically realistic.

If increasing or decreasing all boundary pressures by a uniform amount resolves a convergence issue, it may indicate the convergence issues in the original system are due to difficulties finding valid fluid properties at extreme conditions. In this case, it may help to try using a different [fluid property library](#) or Equation of State.

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Turn Off Parts of the Model

Using the [Special Conditions](#) feature, you can turn off elements in your model and, in effect, shut down entire sections. Then you can focus on getting selected sections to run to convergence.

If the model converges when some sections are closed, this suggests that there is an issue with the inputs in the closed sections.

Related Topics

Special Conditions.....	187
Optional Input for Junctions.....	228
Optional Input for Pipes.....	214
Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739

Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Break a Large Model Into Submodels

AFT Arrow 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 into 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.

Related Topics

Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Change Solution Method

If a model does not converge, try changing the solution method in the [Solution Method panel](#). You also might consider changing the number of pipe segments or the [Segment Length](#) for the [Length March Increment](#).

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Try a Different Fluid, Equation of State or Accuracy Option

Convergence issues can come from Arrow failing to find valid fluid properties at a given pressure and temperature point. Options to resolve these issues include using a different [fluid property library](#), changing the [equation of state](#), or changing the [accuracy options](#) in the Fluid panel.

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738

Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

Using Lumped Adiabatic as a Starting Point

The [Lumped Adiabatic](#) solution method is a more general solution that ignores all heat transfer effects. The resulting solution is often found quickly, and while it has lower accuracy, it can provide a good set of initial guesses for the marching solutions.

An option in the [Solution Method panel](#) allows users to have AFT Arrow first solve the system using the Lumped Adiabatic method, and then use those results as the starting point for the marching solution. Using this option can result in a faster overall runtime, especially for a complex system.

Related Topics

Troubleshooting.....	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
These results do not represent a converged solution.....	712
Identifying and Restoring Corrupted Model Files.....	747

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 Arrow. 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.

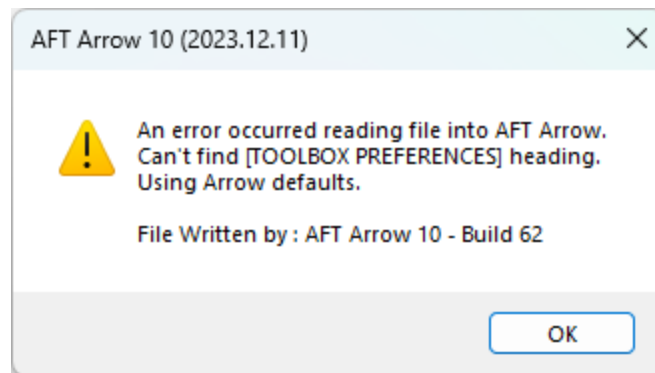


Figure 1: Error message indicating that some portion of the model file cannot be found

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).



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.

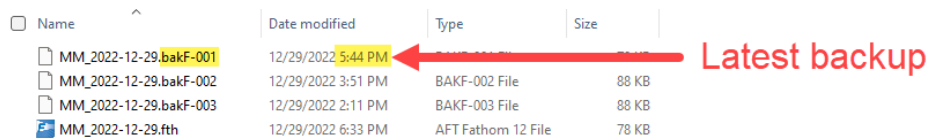
- AFT Fathom.**bakF-001**
- AFT Arrow.**bakA-001**
- AFT Impulse.**bakI-001**
- AFT xStream.**bakX-001**

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
 - 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

Figure 3: The newest backup file, identifiable by file extension numbering and date modified

Name	Date modified	Type	Size
<input checked="" type="checkbox"/> MM_2022-12-29 - Restored.fth	12/29/2022 5:44 PM	BAKF-001 File	78 KB
<input type="checkbox"/> MM_2022-12-29.bakF-001	12/29/2022 5:44 PM	BAKF-001 File	78 KB
<input type="checkbox"/> MM_2022-12-29.bakF-002	12/29/2022 3:51 PM	BAKF-002 File	88 KB
<input type="checkbox"/> MM_2022-12-29.bakF-003	12/29/2022 2:11 PM	BAKF-003 File	88 KB
<input checked="" type="checkbox"/> MM_2022-12-29.fth	12/29/2022 5:44 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.

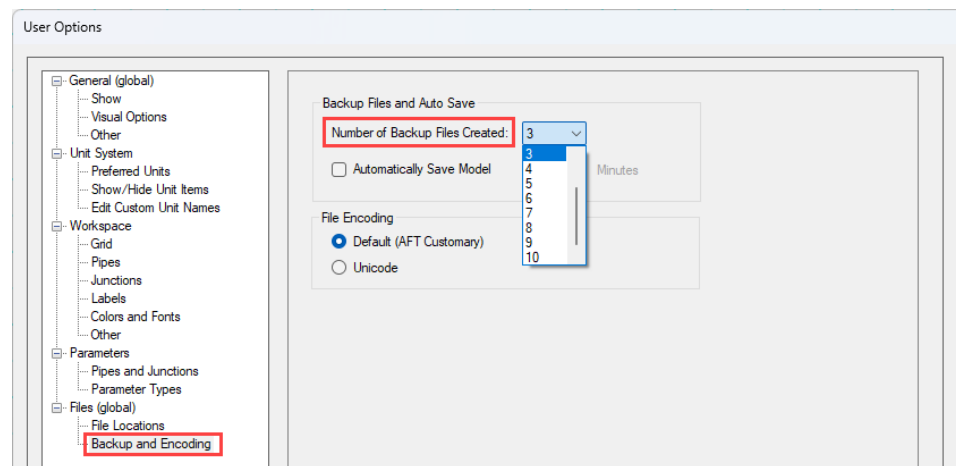


Figure 5: The number of backup files created and options to save automatically can be customized in User Options

Related Topics

Troubleshooting	729
Diagnostics Window.....	730
Use the Model Data Window.....	731
Transfer Results to Initial Guesses.....	734
Poor Compressor/Fan Curve Fits.....	735
Troubleshooting Resistance Curves.....	736
Use the Output Window Sort Feature.....	737
Tee/Wye Junction Complexity.....	738
Lower the Flow Rate Relaxation.....	739
Try Absolute Tolerance.....	740
Make Initial Flow Rate Guesses for Pipes.....	741
Review the Iteration History.....	742
Change Boundary Conditions.....	742
Turn Off Parts of the Model.....	743
Break a Large Model Into Submodels.....	744
Change Solution Method.....	745
Try a Different Fluid, Equation of State or Accuracy Option.....	745
Using Lumped Adiabatic as a Starting Point.....	746
These results do not represent a converged solution.....	712

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 Arrow.

Contents

Technical Support Options	751
Contacting AFT	752
Information on Software Installation	753
Chempak Database Installation	753
Output Files	755
Special File Control Features	756
Converting Models from Earlier Versions	757

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.

Related Topics

Technical Support.....	751
Contacting AFT.....	752
Information on Software Installation.....	753
Chempak Database Installation.....	753
Special File Control Features.....	756
Converting Models from Earlier Versions.....	757

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

Related Topics

Technical Support.....	751
Technical Support Options.....	751

Information on Software Installation	753
Chempak Database Installation	753
Special File Control Features	756
Converting Models from Earlier Versions	757

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>.

Related Topics

Technical Support	751
Technical Support Options	751
Contacting AFT	752
Chempak Database Installation	753
Special File Control Features	756
Converting Models from Earlier Versions	757

Chempak Database Installation

AFT Arrow does not include the [Chempak Fluid Library](#) by default. The Chempak Database is a separate product that must be downloaded and installed as an addition to AFT Arrow to access the Chempak Fluid Library in the software.

Chempak File Location

The Chempak Database is installed in **C:\AFT Products\Chempak Data** by default. If both AFT Arrow and the Chempak Database are installed to their default locations, no further configuration is necessary. However, if AFT Arrow or the Chempak Database are not installed in a default location, AFT Arrow will need to know where to look for the Chempak Database. This location is specified in the [control.aft file](#), located in the same folder as the Arrow.exe. If the files cannot be found the following message is displayed:

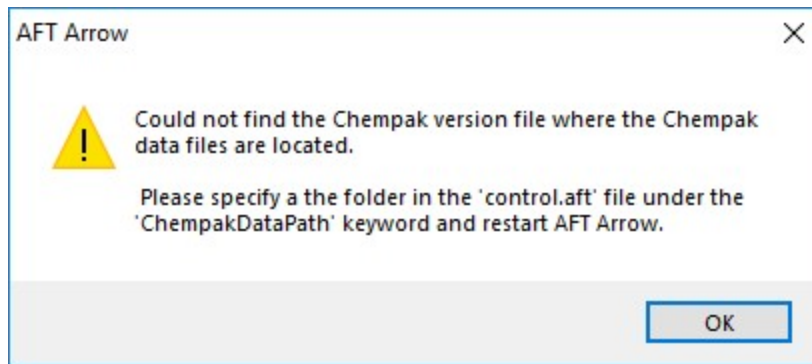


Figure 1: AFT Arrow cannot find the Chempak Database

Control.aft can be edited with a text editor, and by default looks like:

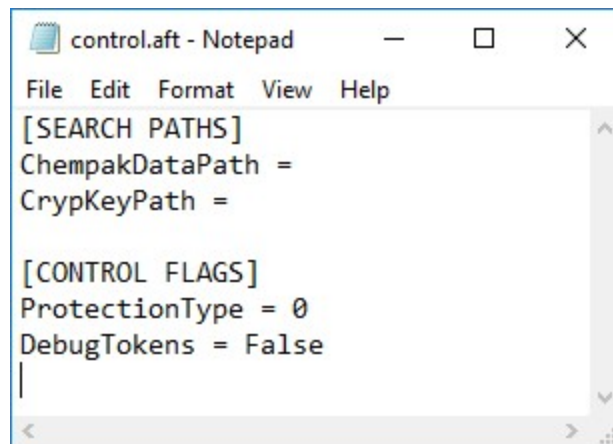


Figure 2: Default control.aft file

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

When upgrading to a new version of AFT Arrow, you may receive a message that the database is outdated.

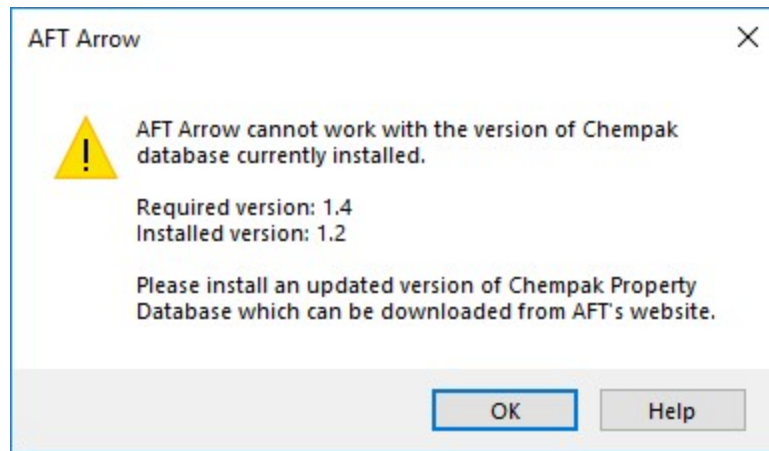


Figure 3: Outdated Chempak database

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.

Related Topics

Technical Support.....	751
Technical Support Options.....	751
Contacting AFT.....	752
Information on Software Installation.....	753
Special File Control Features.....	756
Converting Models from Earlier Versions.....	757

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.

In the [XTS Module](#), the file size will be heavily dictated by the simulation duration and time step specified on the [Simulation Duration panel](#) in Analysis Setup.

Special File Control Features

What is the Control.aft File?

Control.aft is read as AFT Arrow begins to load and instructs AFT Arrow 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 Arrow 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 Arrow will not find the important data.

Specifying Where Chempak is Located

Before AFT Arrow can use the [Chempak Fluid library](#) it needs to know where the data is located. The path is 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 Arrow to look in the C:\AFT Products\Chempak Data directory for the data.

Also see [Chempak Data Installation](#).

Related Topics

Technical Support.....	751
Technical Support Options.....	751
Contacting AFT.....	752
Information on Software Installation.....	753
Chempak Database Installation.....	753
Converting Models from Earlier Versions.....	757

Converting Models from Earlier Versions

Models from earlier versions of AFT Arrow 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 Add-on Modules 326
Activate Modules 311, 320
Add Component to Library 311
Additional (Minor) Losses 194
Additional Losses 194
AFT Standard fluid 82
Analysis Setup 76
Area Change 239
ASME Steam 82
Assigned Flow 241
Assigned Pressure 242

B

Basic Graph Tutorial 382
Bend 243
Blowers 253
Branch 247

C

Cautions 709
Checklist 180, 320
 Checklist 180
 Defining Pipes and Junctions 180
Chempak 756
Chempak Fluid 82

Color Map 338

Colors and Fonts 400

Compare Scenarios 310

Compressible Flow Theory 658-659, 662-663, 665

- Network Solutions 662
- Review of Compressible Flow Theory (Long) 642
- Sonic Choking 233, 666, 671

Compressor/Fan 253

Compressor/Fans 254, 260

- Heat of Compression 257

Control Valve 249

Control.aft 753, 756

Conventions 62, 230

Convergence Problems 731, 737-745

Converting models from earlier versions 757

Cost Application Manager 311

Cost Summary 311

Creating an Enterprise-Wide Network Library System 631

Creating and Managing Saved Groups 161

Critical Warnings 709

curve fit 264

Custom Libraries 613, 618-620, 630

Custom Units 311

D

Dead End 277

Design Alerts 310

Design Factor 228

Design Factors 214

desuperheater 247

E

eLicense Information 311

Exit Valves 300

F

Favorite Fittings & Losses 195

FCV 249

Fittings and Losses 194, 311

Flow Control Valve 249

Fluid Library 311

Fluid Property Viewers 311

Frictional Losses 216, 234

G

Gas Mixtures 662

General Component 278

Generate Disconnected Scenario 310

Goals 495

Graph Results 319

Grid 398

GSC Goals 495

H

Heat Exchanger 278

hydraulically smooth 217

I

Information Messages 709
Initial Flow Guess 214
Initial Guesses 228
Inspection 183
Installation 756
Insulation 199
Insulation Library 311
Interpolated X-Y Data 72, 256

J

Junction library 311
Junction Window Formats 224
Junctions 222, 248-249, 253, 279, 287, 296
 Area Change Properties Window 239
 Assigned Flow Properties Window 241
 Assigned Pressure Properties Window 242
 Bend Properties Window 243
 Branch Properties Window 247
 Check Valve Specifications Window 248
 Compressor/Fan Properties Window 253
 Control Valve Properties Window 249
 Dead End Properties Window 277
 General Component Properties Window 278
 Heat Exchanger Properties Window 278
 Orifice Properties Window 283

Screen Properties Window 290
Tank Properties Window 295
Tee/Wye Properties Window 295
Valve Properties Window 299
Venturi Properties Window 302

K

Keenan 286
Keep Network eLicense Checked Out 311

L

Labels 336, 399
Last Scenario 310
Libraries 613
Loss factor 230
Loss Model 293

M

Minimizing the Quick Access Panel 327
Mixtures 662
Model Data 319
Model Data Control 310
Model Data Print Content 310
Model Data Window 356
Model Diagnostics 311
Model Status Light 326
Modeling Losses 63, 229, 234, 665

N

Negative added pressure by pump at Junction X 718
Network Installation 756
Network Libraries 629
NIST REFPROP 82

O

Objects 177
Open Percentage Data 301
Orifice 283
Output 319
Output Control 310
Output File 755
Output Language 310
Output Messages 709
Output Print Content 310
Output Window 734
Overview Map 324

P

Panhandle equation 217
Parallel Pipes 214
PDCV 249
Pipe Line Size and Color 335
Pipe Material Library 311
Pipe/Junction property calculation exceeded maximum local iteration limit 719

Pipes 169, 189
 Pressure Drop in Pipes (Darcy-Weisbach Method) 216
Polynomial 72, 256
Pressure Drop 63, 194, 229, 234, 657, 665
Pressure Drop Control Valve 249
Pressure Reducing Valve 249
Pressure Sustaining Valve 249
Primary Windows 318
Properties Tab 322
PRV 249
PSV 249

Q

Quick Access Panel 332
Quick Graph Formatting 310

R

Raw Data Changes 353

S

Scale Tables 469
Scenario Comparison Tool 310
Scenario Manager 310, 321
Screen 290
Select Graph Parameters 310
Solution Progress with ANS 536
Sonic Choking 301
Special Conditions 214, 228

Stagnation Properties 73
Static Properties 73
Steady Solution Control group 111

T

Tank 295
Tee/Wye 295
Toolbox 173
Transient 586
Troubleshooting Models 731, 735-745
Turbomachinery 17, 149

U

User Library Fluid 82
User Options 311, 398

V

Valve 299
Venturi 302
Verification of Solutions 663
VFD 254

W

Warnings 709
Workspace 171, 318, 331
 Toolbox 173
Workspace Icon 229
Workspace Layers 331
 Active Layer 334

-
- From Workspace Selections 333
 - Workspace Layers Tab 324
 - Workspace Symbols 340, 401
 - Workspace Window 160, 171

