AFT Fathom™

Product Help Guide

AFT Fathom Version 13

Incompressible Pipe Flow Modeling

Published: February 10, 2025

AFT
Applied Flow Technology

Dynamic solutions for a fluid world ™

Contents

Overview of AFT Fathom

Welcome to AFT Fathom™ - a visual platform for modeling incompressible flow in pipe networks! See what's new in this version.

AFT Fathom 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 Fathom simplifies the pipe network modeling process.

AFT Fathom 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 re-running models are all accelerated because of AFT Fathom'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 Fathom increases your productivity in the modeling process.

What Can AFT Fathom Do?

AFT Fathom can be used to model a wide variety of incompressible flow systems, including:

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Pressure fed systems and gravity fed systems
- Pumped systems, including multiple pumps in parallel or in series
- Pumps with variable speed, controlled pressure, controlled flow, and viscosity corrections
- Systems with pressure and/or flow control valves
- Systems with valves closed and pumps turned off
- Heat Transfer and system energy balance
- Systems with variable density and viscosity
- Multiple design cases in a single model file (Scenario Manager)
- Non-Newtonian fluid behavior
- Cost calculation of pump energy usage

AFT Fathom provides many 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 Fathom'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.

Where to Start

The best place to start learning about AFT Fathom is to work through our example problems and video tutorials found in the Fathom Learning Resources.

Add-on Module Capabilities

AFT Fathom has several Add-on Modules that extend modeling capabilities. The modules can be used individually or together.

- The Extended Time Simulation (XTS) module allows the engineer to model time varying system behavior.
  - Pump starting and stopping over time
  - Valve opening and closing over time
  - Tank liquid level changes automatically calculated
- The Goal Seek & Control (GSC) module allows the engineer to perform multi-variable goal seeking and simulate control system functions.
- The Settling Slurry (SSL) module allows the engineer to perform the complex property and system interaction calculations associated with settling slurry flows.
  - Modeling of homogeneous and pseudo-homogeneous slurry mixtures
  - Slurry pump performance de-rating
- 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.

Who Can Use AFT Fathom

AFT Fathom assumes that the user possesses a good general knowledge of engineering pipe system hydraulics. Even the most advanced and easy-to-use software package cannot make up for a lack of fundamental knowledge on the part of the user. The level of knowledge assumed by AFT Fathom is consistent with that obtained in a typical engineering undergraduate course in fluid mechanics.

Related Topics

Engineering Assumptions

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

- Incompressible flow
- Steady-state conditions (the XTS Module allows transient modeling through steady-state snapshots)
- One-dimensional flow
- System is fluid-full
- Fluid properties are single phase
- No chemical reactions

Related Topics

New in Version 13

View the links below for key updates from past versions:

New in Version 12, New in Version 11, New in Version 10, New in Version 9

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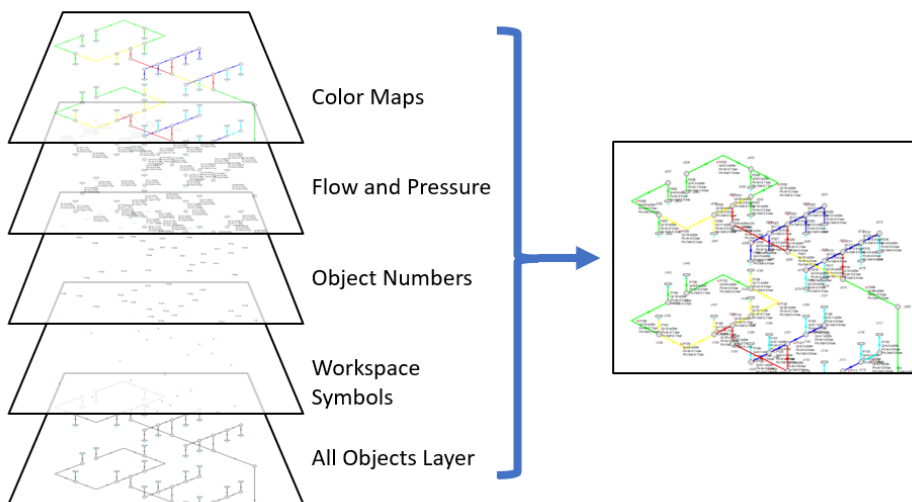
Figure 1: Workspace Layers can display input and output values superimposed over the model, show Color Maps, and control visibility of objects

4-Component Slurry Model

A new 4-Component Slurry Model for the SSL module, developed by leading industry experts, captures complex slurry regimes more accurately. The 4-Component Model introduces a division of slurry solids into four distinct groups, or components, based on the particle size distribution. Each component corresponds to a specific slurry behavior; Carrier Fluid, Pseudo-homogeneous, Heterogeneous, or Fully Stratified. Unlike simpler models that account for only one or two slurry regimes, the 4-Component Model greatly enhances modeling flexibility by automatically determining the effects contributed by each component regime. The model was developed by industry-leading experts and is respected for its accuracy in part because the individual components are based on well established sub models. The 4-Component Model offers a comprehensive and advanced approach to slurry modeling, enabling more accurate and detailed analysis of slurry behavior in various particle size distributions.

Carrier Fluid
particles <40 μm

Pseudo-homogeneous
40 μm < particles < 200 μm

Heterogeneous
200 μm < particles < 0.015D

Fully Stratified
0.015D < particles

The 4-Component Slurry model offers a comprehensive and advanced approach to slurry modeling, enabling more accurate and detailed analysis of slurry behavior in various particle size distributions.

The model was developed by industry-leading experts and is respected for its accuracy in part because the individual components are based on well established sub models.
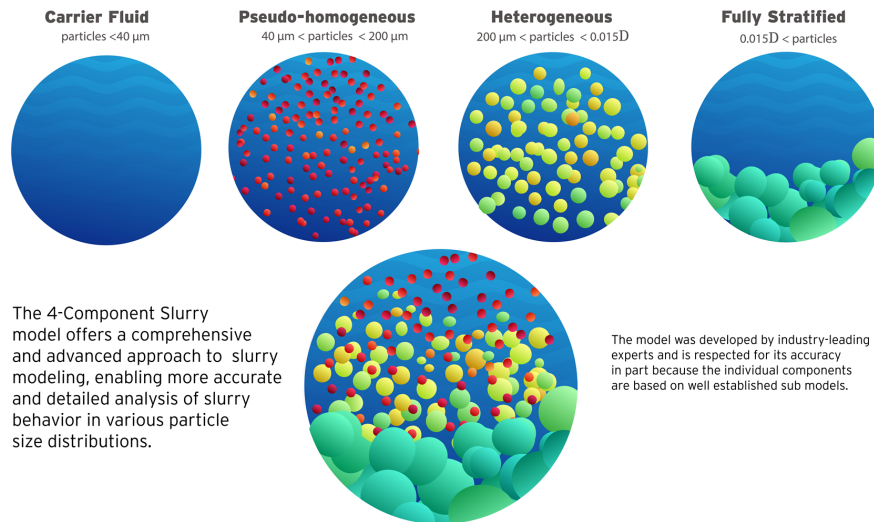
Figure 2: The 4-Component Slurry Model considers combined effects from four slurry regimes

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 3: 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.
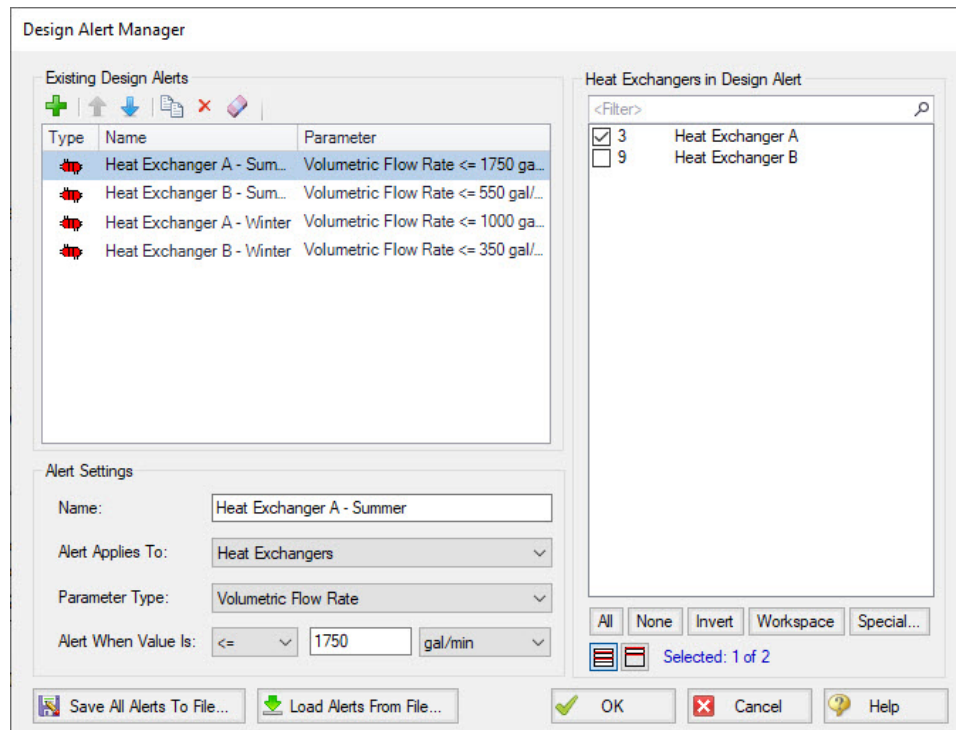
Figure 4: Steamlined Design Alerts Manager

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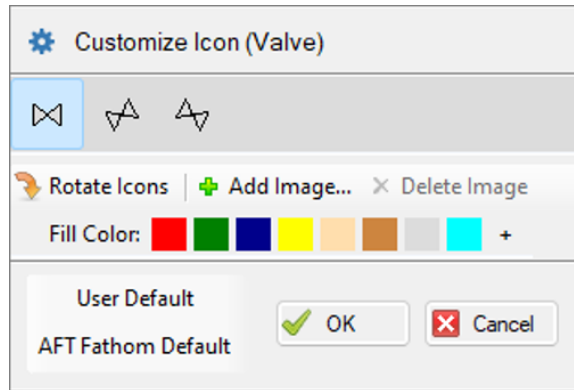
Figure 5: The Customize Icon window allows a custom image to be applied to any junction

- **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.
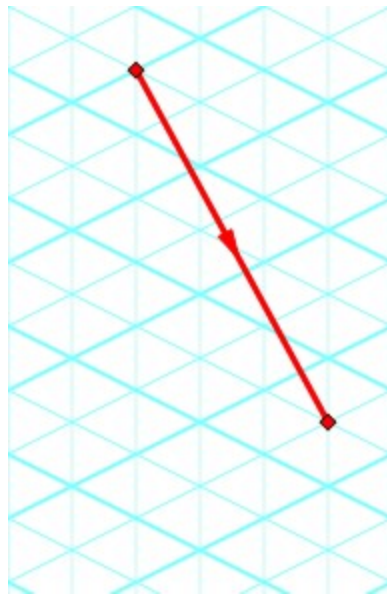
Figure 6: Isometric Freeform drawing mode does not force pipes to follow gridlines

- **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.
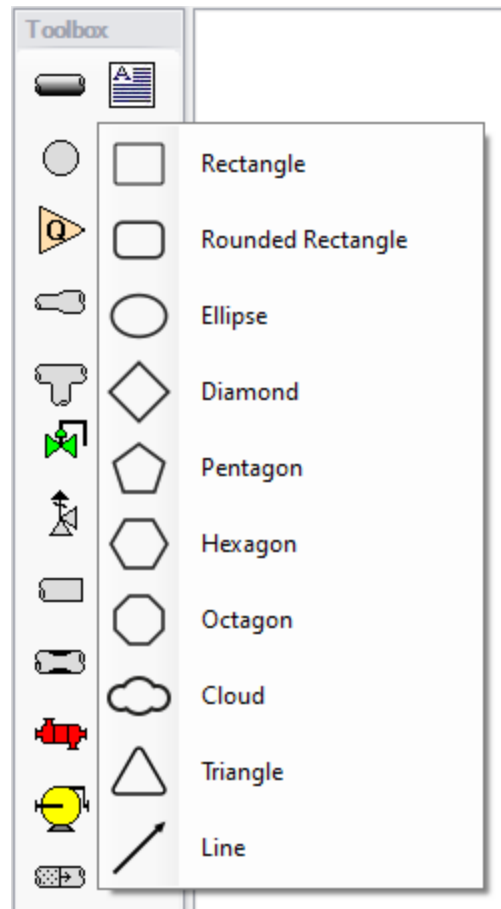
Figure 7: New Annotation options

- **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.
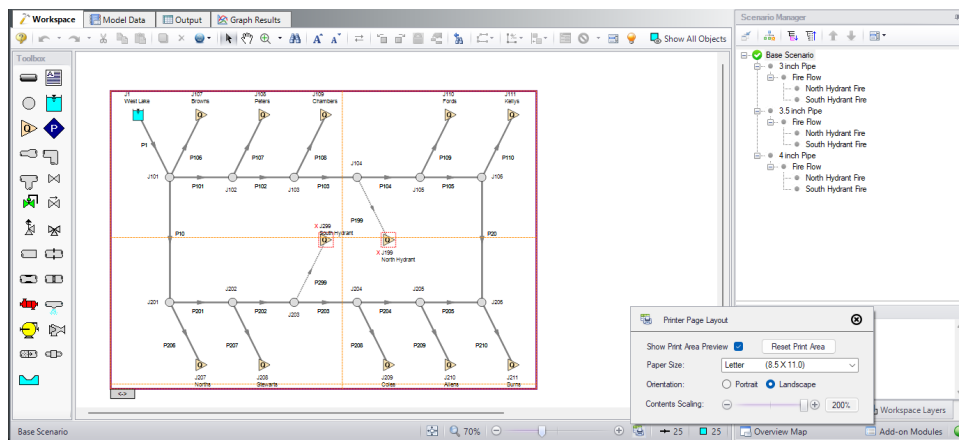
Figure 8: The Printer Page Layout feature allows the user to set the Workspace print area

- **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.
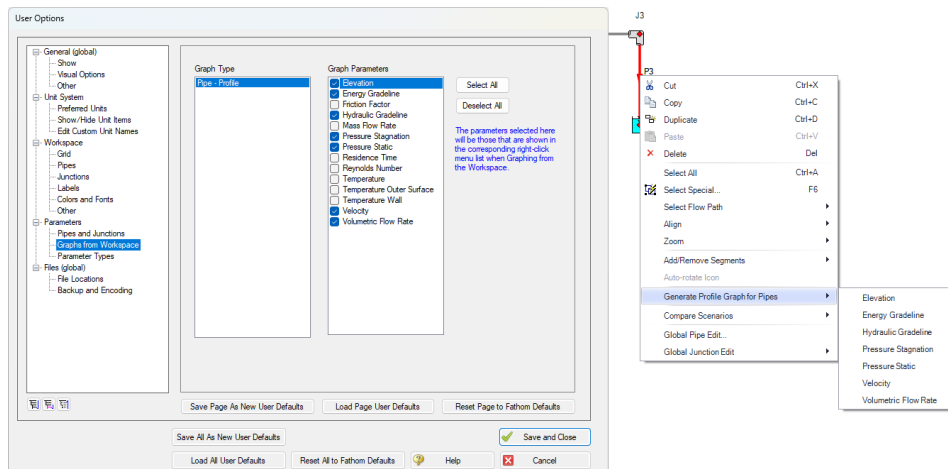
Figure 9: The right-click context menu for pipes on the Workspace can have the graphing parameter options modified

Miscellaneous Improvements

- **Modeling Additions and Improvements**
  - **Laminar Corrections** - Laminar corrections are now off by default, messages indicate if they were applied or are recommend, and output parameters indicate the correction strength.
  - **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.
  - **Distinguish User and AFT Fluids** - New category for user-defined fluids, making it easier to find custom entries.
- **Importing of PCF, EPANET, GIS, and CAESAR 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.
  - **CAESAR 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**
  - **Pump RPM** - Report pump speed in RPM, in addition to percent.
  - **Dimensionless Heat Transfer Parameters** - Grashof, Nusselt, Prandtl, and Rayleigh Numbers have been added as Output parameters.
  - **Deep Laminar Caution** - Simple loss models do not account for exceptionally low laminar flow, this new message provides a caution if you enter this range.
  - **POR/AOR in Pump Summary** - Show the values for Preferred and Allowed Operating Ranges in the Pump Summary.
  - **Event Messages in New Format** - Events are now shown in a cleaner, collapsible format similar to Warnings.

- **XTS Module Tank Overflow and Drain Warnings** - New messages to clearly indicate if a Tank has overflowed or drained during the transient.

Related Topics

Interface Improvements for Fathom 12

There were many improvements made to the interface in Fathom 12. 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 Fathom 11

- **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**

- **NFPA Configuration**: Enabling NFPA reports has moved from User Options to the **NFPA panel** under the Miscellaneous group. In addition selecting an NFPA group has moved from the Tools menu to the NFPA panel.

- **Extended Options**: Enabling loss model options such as equivalent length that was previously in the Extended Options window has been moved to the **Junction Loss Models**

panel under the Miscellaneous 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**

- **Slurry Data** (SSL Module Only): Settings have been moved to the **Fluid Properties 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
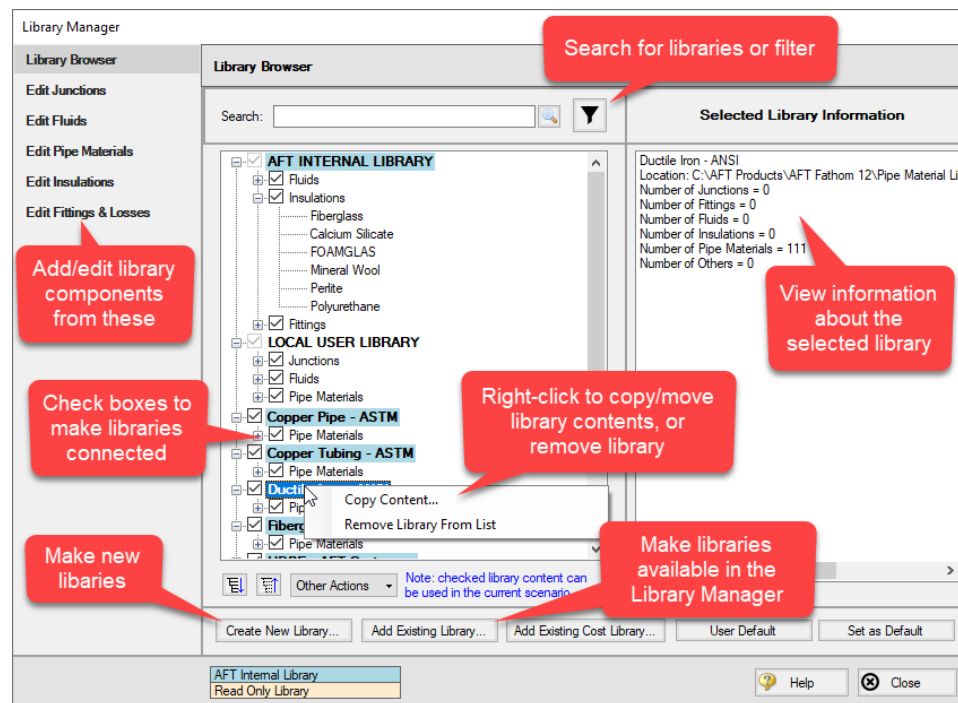
New in Version 12

Analysis Setup

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, select a viscosity model, enable heat transfer, 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 Fathom 12.
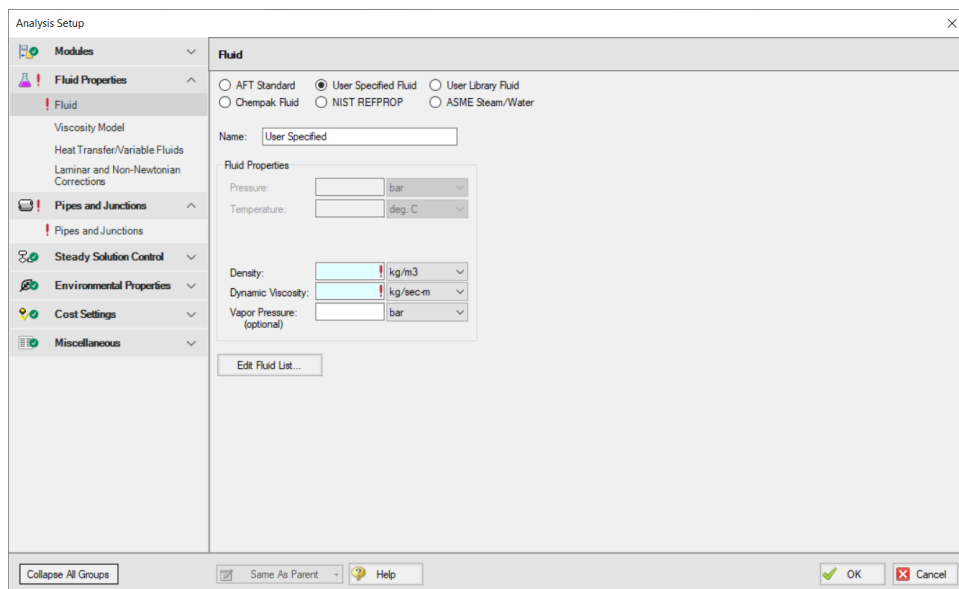
Libraries

The database feature interface was completely overhauled to improve workability. The new system is called the Library Manager and significantly improves the ease and workability for users to browse and connect input information, now called libraries, into their model. For a list of differences between the Database and Libraries interfaces see Interface Improvements for Fathom 12.
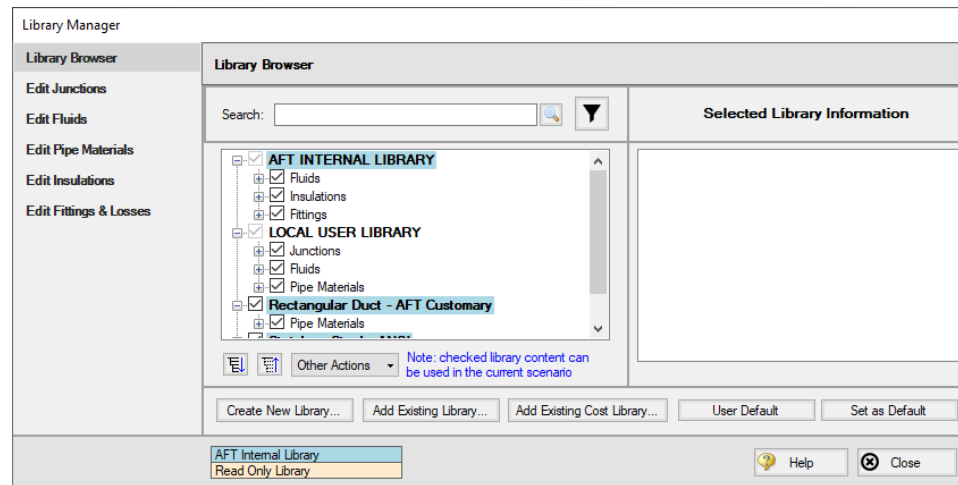
Multi-Scenario Graphing

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. See the Creating Multi-scenario Graphs tutorial.
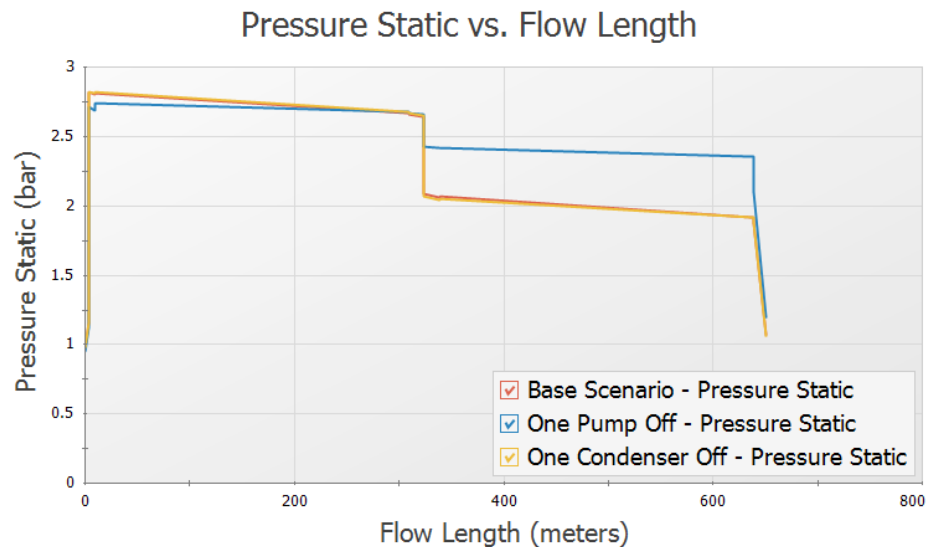
Improved Organization of the Warnings in the Output

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
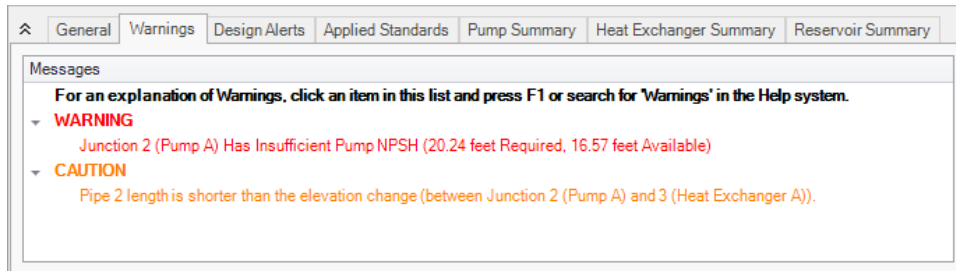
Miscellaneous Improvements

- **Pipes/Junctions**
    - Pipes can be defined as zero-length connectors to connect two junctions which are directly flanged together
    - Submerged pump can now be defined using surface pressure and pump depth
    - New Pump As Turbine (PAT) option on the Pump Properties window
    - Valve transients in the XTS module can be defined as open percent vs time
- **Workspace and Interface**
    - Pipes and Junctions can be set the Same as Parent directly from the Workspace
    - Custom Unit names can be defined in the Unit System section of the User Options window
    - Pipe/Junction notes can be searched in Select Special
    - Special Conditions can be directly changed from the toolbar for junctions that have multiple special condition options
- **Output and Reporting**
    - Show the number of warnings and design alerts in the batch run window
    - Heat Transfer output parameters added to Visual Report
- **Miscellaneous**
    - Herschel-Bulkley non-Newtonian viscosity model added for representing Power Law fluids which have an apparent yield stress
    - 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

New in Version 11

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

Multi-scenario Pump vs. System Curves

- Pump vs System curves from different scenarios can be plotted on the same graph within AFT Fathom
- Easily compare changes to pump operation in different cases
- For more information see Multi-Scenario Pump vs. System Curves Tutorial
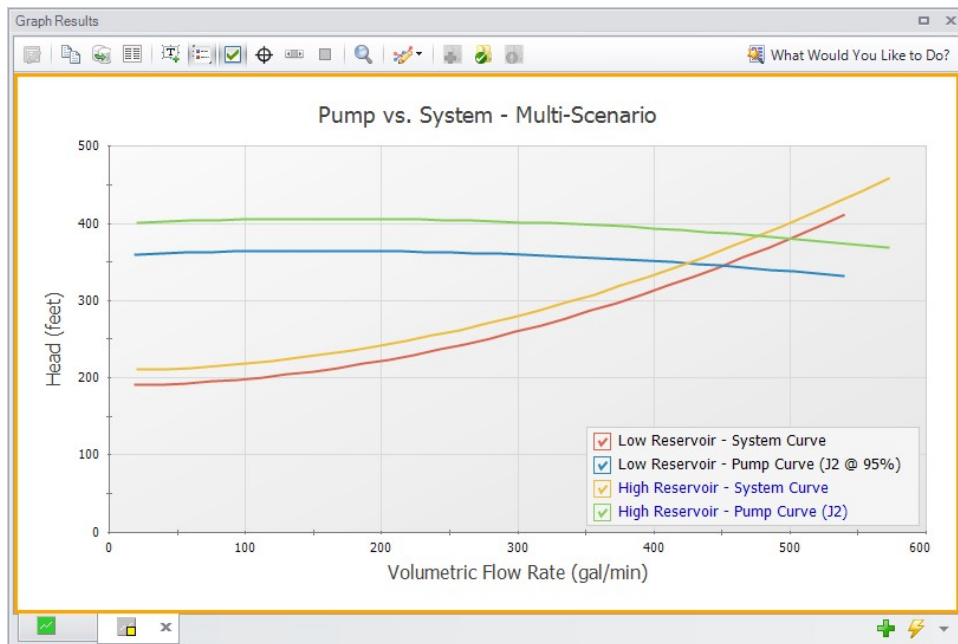
Junction- specific Parameters in Visual Report

- Output such as the percent of BEP for a pump or the Cv of a control valve can now be displayed in the Visual Report
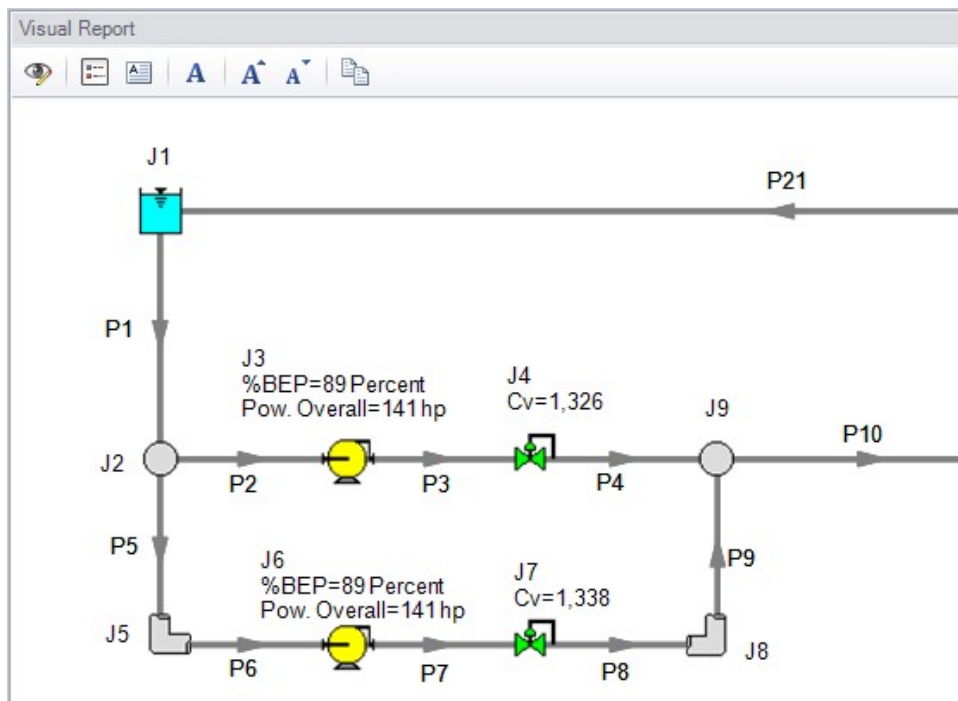
Notes in Output

- Notes can be added to any cells in the Output to save observations and comments directly with the model file

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
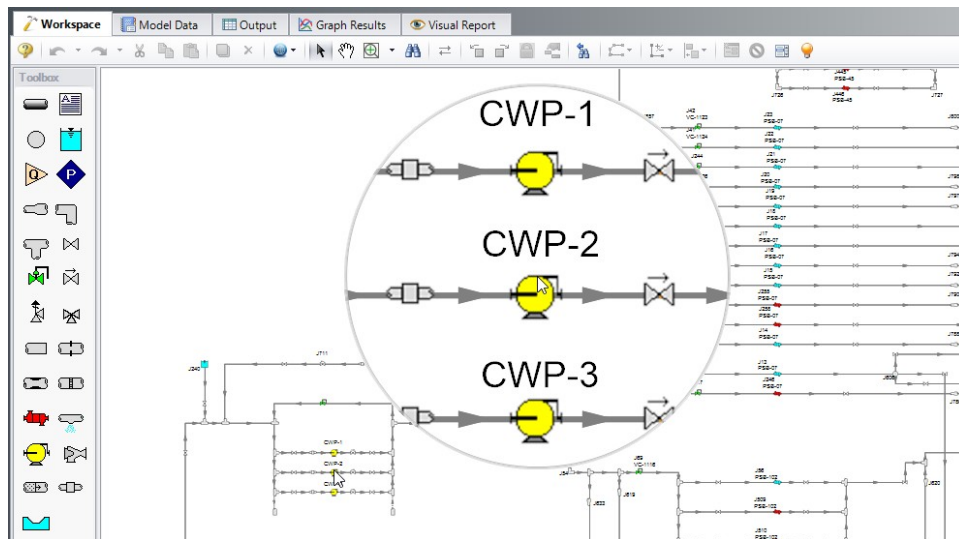
- **Automatic Junction Rotation** - Fathom will automatically rotate junctions to visually align with neighboring pipes to speed-up model formatting

- **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, and any elevations being imported as Intermediate Pipe Elevations

- **Split Pipes by Intermediate Elevations** - Pipes with intermediate elevations can be split with branches at intermediate elevation points (see Joining and Splitting Pipes)

- **Select Shortest Flow Path** - The shortest path between two selected junctions can be found using this feature in the Edit menu

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

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 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 Fathom 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 values are now included for bends with r/D between 0.5 and 1
  - 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

New in Version 10

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

	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	HEX heat rate set	Yes	Heat Exchanger	67	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	Yes	Heat Exchanger	66	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	Yes	Heat Exchanger	65	Heat Rate	Set equal to value	3700	Normal Heat
5	FCV setpoint set	Yes	Control Valve	71	Control Setting	Set equal to value	300	
6	FCV setpoint reduce by 20 gpm	Yes	Control Valve	72	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increase by 10%	Yes	Control Valve	73	Control Setting	Change by percent (+/-)	10	
8	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	
9								

AFT Transfer

- 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
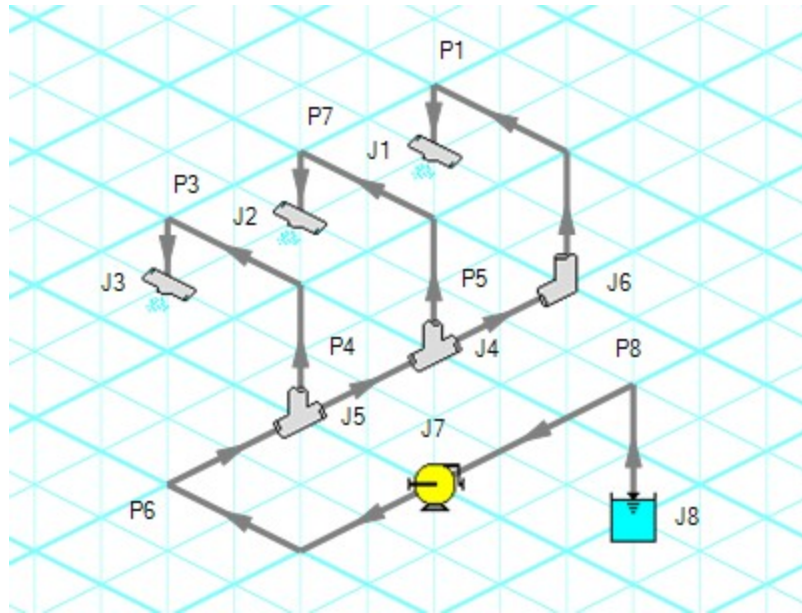
NIST REFPROP Fluid Database Now Included

- 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 on the Fluid panel in Analysis Setup
  - Use accurate and complete fluid information in your models without the need to locate and enter fluid properties manually

New Pump Features and Improved Pump Window

- New Interstage Bleed Pump Model
  - No need to manually calculate pump curves - enter pump curves directly from a pump's datasheet
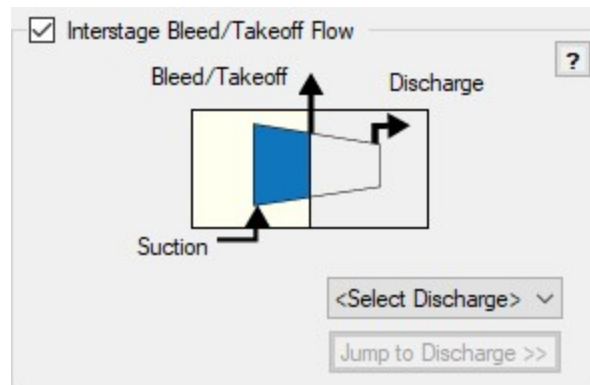
- New Positive Displacement Pump Model
  - Clarifies the specification of Positive Displacement Pumps
  - Include Acceleration Head Loss calculations
- Added Pump Curve and Sizing Analysis Types to Centrifugal Pump Model
  - Clarifies required data and model intention
  - For a detailed description see Pump Properties Window
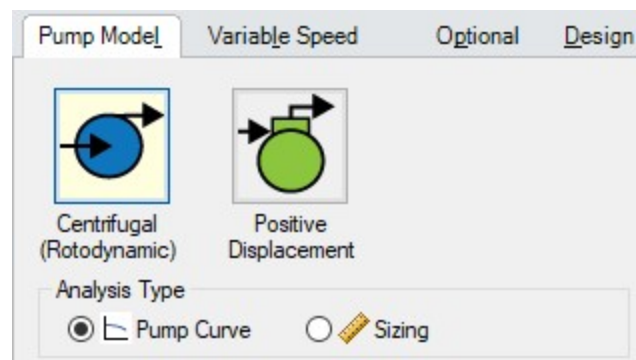
- Specify a centrifugal pump's Preferred Operating Region (POR) and Allowable Operating Region (AOR)
  - Quickly determine if a pump's behavior is acceptable
  - Automatically calculate a POR from *ANSI/HI 9.6.3-2017 - Rotodynamic Pumps - Guideline For Operating Regions*
  - For a detailed description see Pump Configuration > Configuration Data

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

New Multi-Condition Events for XTS Module

- • Create complex triggers for transient events in the eXtended Time Simulation (XTS) Module
  - • Only start a transient event when multiple conditions are met
  - • Utilize logical conditions - AND, OR, and NOT - to capture the effects of complex control systems
  - • For a detailed description see Multi-Condition Events
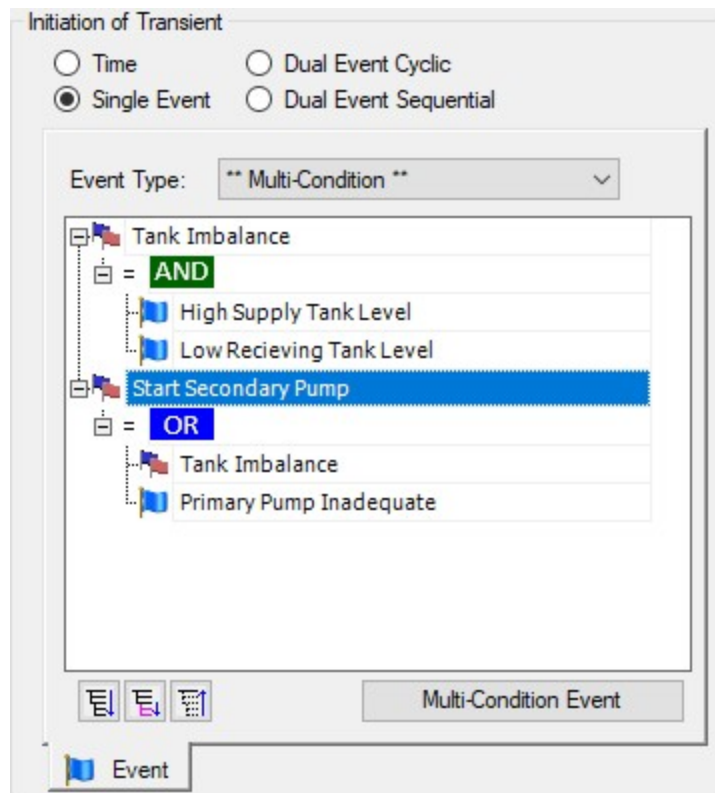
Improved Startup Window

- • 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, default pipe material, and NFPA settings

- 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 Neutal 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*
- Updates to Standards
    - Updated to *HI/ANSI 12.1-12.6-2016 Rotodynamic (Centrifugal) Slurry Pumps for Nomenclature, Definitions, Applications, and Operation* for use in Slurry De-rating
    - Updated to *HI/ANSI 9.6.7-2015 Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance* for use in Viscosity Corrections
    - Updated MIT Equation for Crude Oil to latest version specified in Pipeline Rules of Thumb for use as an additional pressure loss model

More Improvements

- **General**
    - Many new default Pipe Materials
    - Intermediate pipe elevations can now be entered as Length Along Pipe
    - Control Valve setting for Always Control (Never Fail) can be set to a default to off
    - Increased the loading speed for large models
    - Applying Design Alerts when using the XTS module now shows Design Alert Violations in transient Output tables, and all time steps that violate the Design Alert are highlighted
- **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
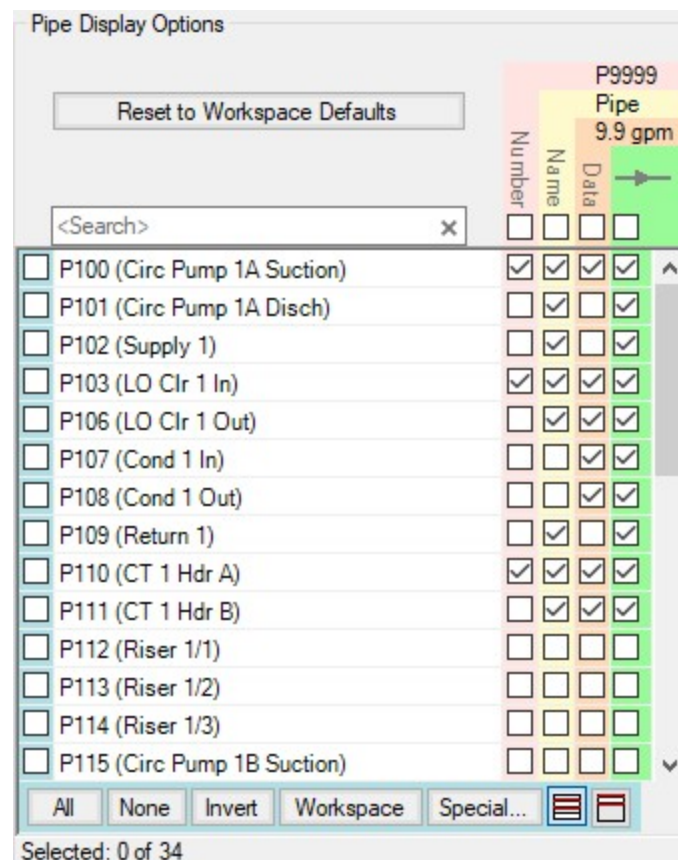
Related Topics

New in Version 9

General Interface

- Improved tab interface and enhanced Quick Access Panel functionality
- Improved search includes pipe and junction notes, names and numbers
- Output reports in German and Chinese languages
- Files are automatically locked when opened to prevent multiple users from accidentally opening the same file
- Global edit select all items within a section

Graph Results

- Multiple Graphs Sets can be added to Graph Groups for easy generation and display
- Multiple graph tabs on main Graph tab
- Multiple graphs tiled on same Graph Group 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
- Junction location shown on profile plots
- Annotations can be placed on graphs

Scenario Manager

- Scenario manager shows which scenarios have output
- Ability to create child scenarios without results saving duplication and load time

Workspace

- Scenario manager shows which scenarios have output
- Ability to create child scenarios without results saving duplication and load time

Output

- Results Diagrams which graphically show data for pipe heat transfer, pumps and heat exchangers
- Default colors for multi-scenario output.
- Support for NFPA output reporting

Visual Report

- Automatic Color Map generation

Pipes

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

Junctions

- Improved Relief Valve specification
- Improved clarity for data entry in the Reservoir/Tank Property window
- Submerged pumps
- When the speed is set for a pump, both the original and modified pump curves are shown on the Pump Property window
- New Weir junction
- Design Alerts can be applied to junctions

Solver

- 3-K method for laminar flow through fittings
- Equivalent Length option available for pipes, valves and bends

Modules

- GSC
  - Pipes now have GSC variables available such as roughness, design factor, insulation thickness, Reynolds number
  - New and improved searching method for multi-variable goal seeking – useful for model calibration
  - Variables and goals can be reordered
  - New goals and variables added

Related Topics

FAQ - Frequently Asked Questions

How has AFT Fathom Been Verified?

There are several aspects to AFT Fathom verification.

First, AFT Fathom has been compared to a wide variety of published results. Models, comparisons, and explanations for such published predictions are given in the Verification section of this site and the Verification folder installed with AFT Fathom.

Second, it can be verified that the AFT Fathom predictions agree with the fundamental equations for networks. See the Network Solution Methodology topic.

Finally, AFT Fathom predictions have been compared to test data and other analytical methods on numerous occasions and good agreement has been shown.

Can AFT Fathom Be Used to Model Gas Systems?

Yes, depending on the gas system (see Walters, 2000). For incompressible gas systems, AFT Fathom is perfectly appropriate. For compressible flow gas systems, a more sophisticated tool such as AFT Arrow should be considered.

Can AFT Fathom Model Two-Phase Flow?

No, AFT Fathom only models single phase flow and assumes all pipes are liquid full.

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

Heat transfer effects show up in several places in AFT Fathom. First, for pipes users can input pipe and insulation data thermal conductivity data and thickness, as well as external convection coefficients. The thermal conductivity data can be constant or temperature dependent. These heat transfer calculations and the associated physical properties are determined for each 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 Fathom performs the appropriate energy balance calculation across the heat exchanger. Third, heat transfer occurs in pumps due to inefficiencies.

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 Fathom Perform an Energy Balance Across Each Junction in the Model?

Yes, every junction except for heat exchangers and pumps are assumed to be adiabatic.

Does AFT Fathom Model Transient System Behavior?

Yes, in the optional XTS Add-on Module.

Can I Model a Non-Newtonian Fluid in AFT Fathom?

Yes, AFT Fathom offers several non-Newtonian fluid models including Power Law and Bingham Plastic.

Can AFT Fathom Model Fire Sprinkler Systems?

Yes. The Spray Discharge junction is perfectly appropriate for fire sprinklers. The fire sprinkler K values can be entered in directly into the Spray Discharge. In addition, Spray Discharge junctions can be turned on or off with a single mouse click. Evaluating multiple fire location scenarios is easily done, and use of the Scenario Manager allows all cases to be kept in the same model file.

Can AFT Fathom Model Ducting System Like Those in HVAC, Ventilation, and Dust Collection Systems?

Yes, AFT Fathom can model ducts. The rectangular geometry is modeled as a non-cylindrical pipe using a hydraulic diameter. Since ducting systems are usually incompressible, the solution methodology is appropriate.

Can I Keep All of My Design Cases Together in a Single File?

Yes, the Scenario Manager lets you create dependent design cases where changes are inherited by children.

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 Restricted Flow Area for cavitation calculations. No such ability exists for Fittings & Losses; thus, cavitation 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. In the case of Fittings & Losses, it assumed to act like a friction factor and be evenly distributed along the pipe. The Fittings & Losses 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 Reservoir Junction and an Assigned Pressure Junction?

There is an overlap in capability between the Reservoir junction and an Assigned Pressure junction and frequently they are interchangeable. Here are the differences. The Reservoir junction input pressure and temperature always corresponds to stagnation properties. In the Assigned Pressure junction, they can correspond to either static or stagnation properties. If static, only one pipe can be connected. If the stagnation option is used in the Assigned Pressure junction, it will behave identically to a Reservoir junction. Finally, the Reservoir junction allows pipes to connect at different elevations (i.e., depths), and also allows pipes to discharge above the liquid surface level.

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

AFT Fathom uses sophisticated models 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.

How Do I Size a Pump When There Are Not Any Flow Control Valves?

See the Pump Sizing topic.

How Do I Size a Pump When There Are Flow Control Valves?

See the Pump Sizing topic.

How Do I Model a Variable Speed Pump?

Enter the pump data as a polynomial in the Pump Properties Window and then set the pump speed directly. No entry is assumed to be 100% speed. If you enter something other than 100%, AFT Fathom uses the Affinity Laws to adjust the curve. If a control setpoint is entered, AFT Fathom will determine the speed necessary to meet that setpoint.

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

Enter the pump curve data in the Pump Properties Window and set the desired flow or pressure on the "Variable Speed" tab using the Controlled Pump option. AFT Fathom will calculate the pump speed required to produce the flow or pressure conditions specified and display the speed in the Pump Summary report.

How Do I Close a Valve, Pump, or Pipe?

Select the pipe or junction you want to close and choose Special Condition from the Edit menu. By default, AFT Fathom 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 Fathom 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 Fathom 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 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 window available in User Options.

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. The preferred units for a model can also be set in the Startup window.

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. See Global Pipe Edit and Global Junction Edit topics for more information.

How Do I Create a Liquid Mixture?

You can create mixtures with the NIST REFPROP or Chempak libraries (Chempak is an additional add-on). The AFT Standard fluid library does not support mixing. To create a mixture, go to the Fluid panel in Analysis Setup and choose either the Chempak or NIST REFPROP library, and then click the "Create New Mixture and Add…" button. Here you can specify the mixture components and percentages.

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 Scenarios or Models Right After Each Other?

Multiple scenarios or 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 Options menu. The second is pressing the F2 function key while in a Pipe and Junction Properties Window. The third is double-clicking in the blank space at the top of 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 tab, 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 on a row to automatically launch that pipe/junction's Properties Window.

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

Open the Output Control window from the Tools menu, change to the Show Pipes/Jcts tab, 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 straight between junctions. If you need to model a system other than a stationary earth-based system, the gravitational acceleration can be changed on the Environmental Properties panel of Analysis Setup.

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 Add/Remove Segments tool found on the Arrange Menu.

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

See the Libraries topic.

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

See the Network Libraries Overview topic.

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

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

Can I Customize the Junction Icons in AFT Fathom?

No, AFT Fathom icons are in a resource file that cannot be edited by the user.

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

There are no theoretical limits to model size, but there are a few practical limits. First, AFT Fathom accepts pipe and junction ID numbers up to 30,000. This limits the model size to 30,000 pipes and 30,000 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 AFT Fathom Model Reacting Flows?

No, AFT Fathom only models non-reacting flows.

Can I Install AFT Fathom Onto a Local or Wide Area Network?

Yes, AFT Fathom 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. See the Information on Software Installation topic.

Where Can I Find Reference Information About Pipe Materials?

See the Pipe Material Library References topic.

What Kind of Transients Can Fathom XTS Model?

A wide range, including liquid levels, valve open/close, pump speed changes, relief valve opening and more. Just as important, Fathom XTS can model time transients, event transients and inherent transients. Time transients are changes that are specified to occur based on the model time. Event transients, on the other hand, are triggered by an event that occurs in the system. Many kinds of events can be used, such as pressure, flow, velocity and liquid level. Inherent transients are changes that occur by the very nature of a system component. One example would be the liquid level change that occurs in a finite reservoir (or tank), while another would be a check valve opening and closing as a result of flow reversals.

Related Topics

Fathom Learning Resources

Online Resources

Additional resources can be found in the Learning Center on our website. Resources found there include:

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

Online Quick Start Video Tutorial

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

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

Beginner: Three Reservoir Problem

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

- Beginner: Three Reservoir Problem

Walk Through Examples

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

- Walk Though Examples

Module Walk Though Examples

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

- Module Walk Through Examples

Related Topics

Startup Window

The Startup Window is displayed when AFT Fathom 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 Fathom 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 ASME Water** - Selects ASME Water to be used as the working fluid of the model which requires the definition of Pressure and Temperature, and optionally can have Heat Transfer modeling enabled.
        - **Select A Recently Used Fluid** - Allows the selection of a recently used fluid from the dropdown menu and also requires Temperature.
    - **Modeling Preferences** - Displays various settings for the application, such as specifying the units to use, which grid to use, or whether to enable NFPA. 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.
        - **NFPA Settings** - Enable NFPA.
- **Resources**
    - **Current Version** - Open the AFT website to view the latest AFT Fathom 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 Fathom as quickly as possible. The resources include the Quick Start Guide PDF, Examples, and video tutorials.
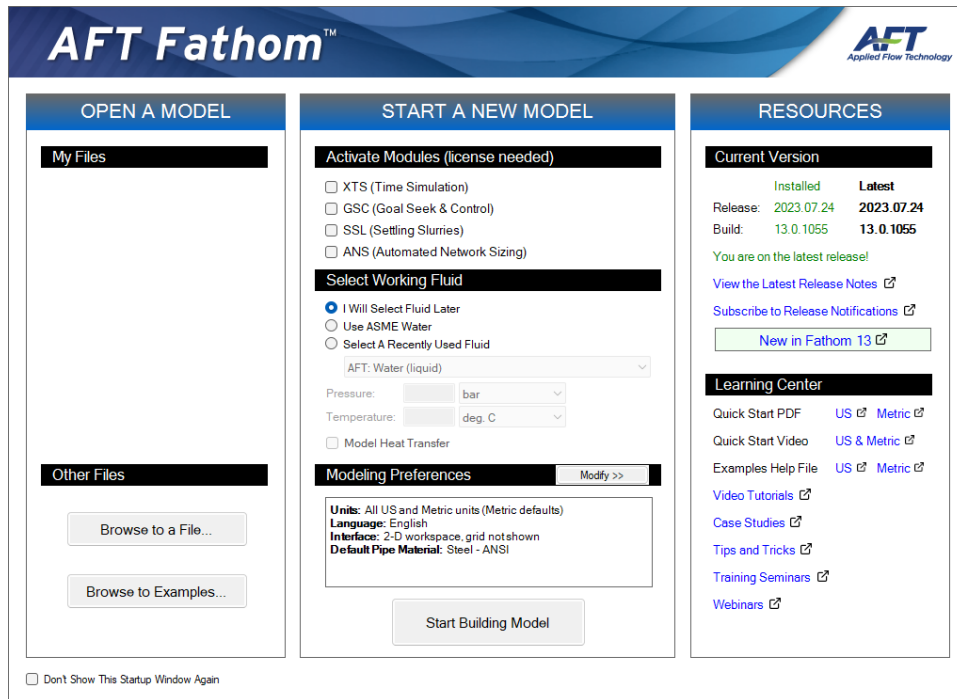
Figure 1: Startup Window in AFT Fathom
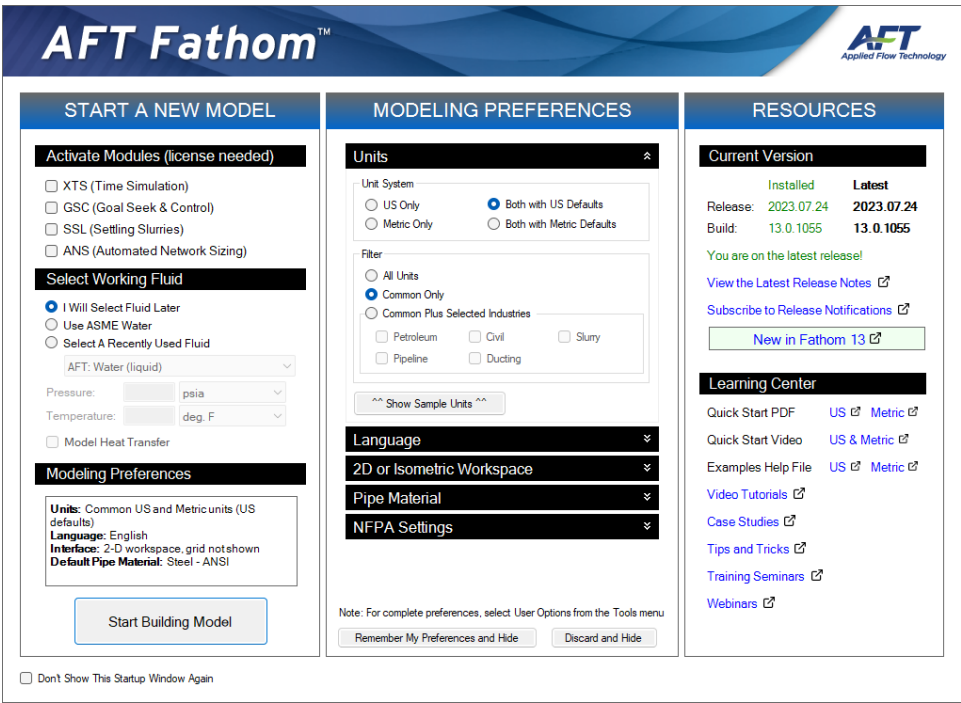
Figure 2: Startup Window showing Modeling Preferences

Related Topics

Fundamental Modeling Concepts

This section focuses on the critical concepts and theories that comprise the model building process in AFT Fathom. 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

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

Pipes are conduits for incompressible, one-dimensional fluid flow. The flow rate through the entire length of the pipe is always constant in steady-state simulations, but can vary if the XTS module is enabled. Because pipes have constant diameters, the fluid velocity is also 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, junctions also influence the flow or pressure behavior of the system. For example, a reservoir 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 fluid behavior.

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

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

In general, you do not need to specify the actual flow direction in a pipe, because AFT Fathom 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 pumps or valves. AFT Fathom uses the flow direction of the pipes to determine in which direction to increase or reduce pressure.

Related Topics

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 Fathom 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 pumps 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 fluid flow to come into or pass out of the system in a prescribed manner. The most obvious of these is the Reservoir junction. Reservoirs maintain a constant pressure at a node, and the flow adjusts according to the governing equations. While in principle the fluid is stored or removed from the reservoir, AFT Fathom considers the fluid as coming into or going out of the system. The stored reservoir fluid is not part of the system.

The convention for defining inward or outward flow for the system or any AFT Fathom 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

Features for Modeling Irrecoverable Losses

AFT Fathom works almost exclusively with the concept of the loss factor, also known as the 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 Cd (discharge coefficients) for orifices.

AFT Fathom has methods available for modeling irrecoverable losses with the Equivalent Length method with either user-defined equivalent lengths or NFPA 13 table data (which is the table on the Fittings & Losses tab on the Pipe Properties window). In order for the fittings and losses to be modeled as equivalent lengths, the toggle switch must be set to use Equivalent Lengths. Refer to the NFPA panel topic for more information on generating an NFPA Output Report.

Related Topics

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 Fathom 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 Fathom, you should follow these guidelines:

- Be aware of what AFT Fathom can and cannot do and the assumptions it makes.

- Learn and use the tools in AFT Fathom 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 Fathom 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 Fathom 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 Fathom 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

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 Fathom terms, the system has two assigned flow junctions.

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

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

The model in Figure 1 can be built with AFT Fathom and if you try and run this model, AFT Fathom will inform you that it cannot run it because of the lack of a reference pressure (Figure 2). In AFT Fathom there are several junctions that can act as a reference pressure: the reservoir, assigned pressure, valve (with the exit valve option), orifice (with the exit orifice option), spray discharge, relief valve junctions which are open to an external pressure, and pressure control valves.

When you try and run a model that has one or more sections completely bounded by known flows (or Flow Control Valves or Pumps 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.
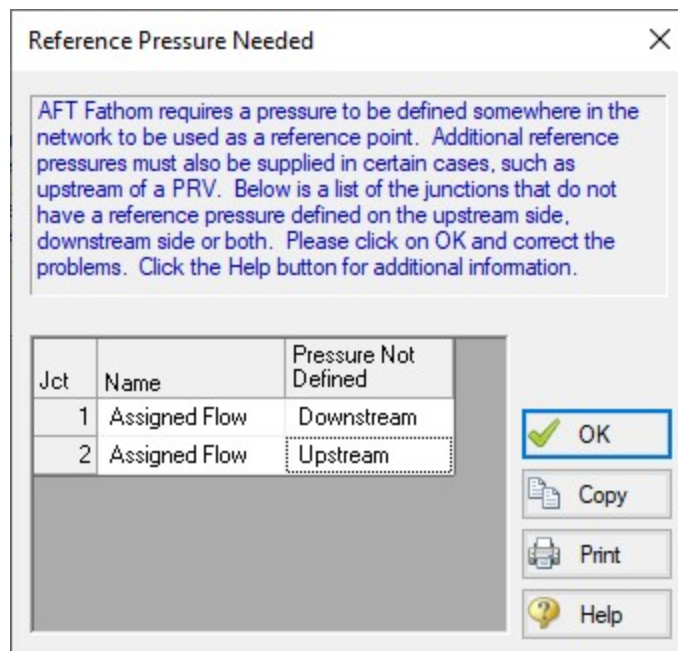
Figure 2: AFT Fathom message when you try to run a model without sufficient reference pressures

How Pressure Junctions Work

Pressure junctions (e.g., reservoirs or assigned pressures) in AFT Fathom 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. This section offers a physical example of how pressure junctions work.

Consider the physical pumping system shown in Figure 3. This system pumps water from Lake Ontario to Lake Huron.
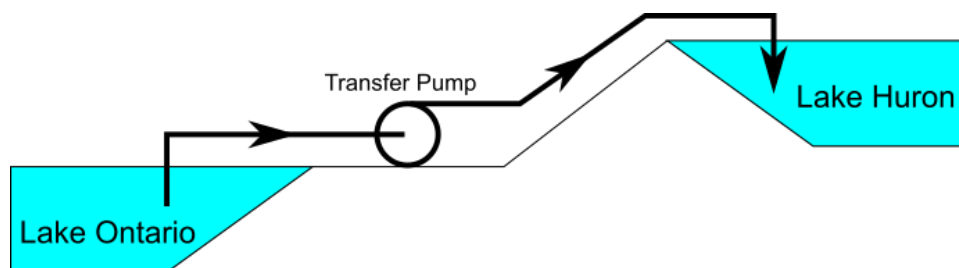
Figure 3: Physical pumping system to transfer water from shoreline supply tank (J1) to discharge tank (J2) at top of hill

An engineer is tasked with sizing the pump for this system. The engineer has an idea of the discharge head needed for the pump, and so builds an AFT Fathom 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.
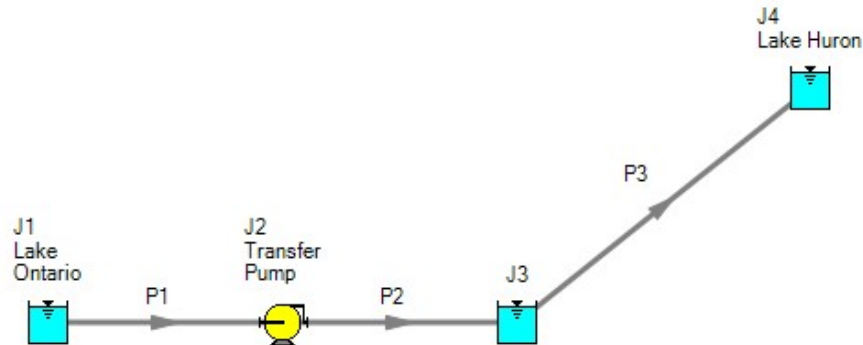
Figure 4: Model to size pump attempting to represent system in Figure 3. It does not accurately represent Figure 3
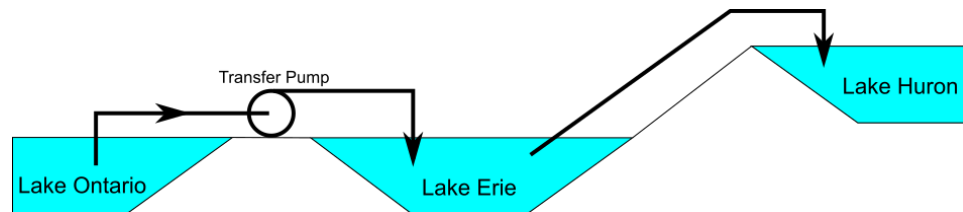
Figure 5: Physical system represented by model in Figure 4

Why? Because the reservoir junction (J3), which is an infinite source of fluid, has been located between the pump (J2) and Lake Huron (J4). The J3 reservoir isolates Lake Ontario and the pump (J1 and J2) from Lake Huron (J4). The J3 reservoir is very much like placing Lake Erie between the supply and discharge piping.

If the engineer changes the surface elevation in J3, it is like changing the surface elevation of Lake Erie. It will change the flow rate in the pipes, but no matter how high the water level of Lake Erie, it does not change the fact that the pumping system is isolated from Lake Huron. The flow rate in the piping from J4 to J3 is entirely dependent on the difference in surface elevations between J3 and J4, and is not influenced by the J2 pump in any way.

How then should the physical system in Figure 3 be modeled? It should be modeled as shown in Figure 6.
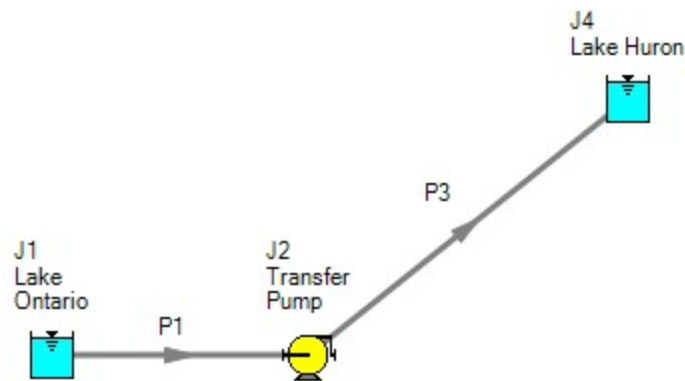
Figure 6: Proper model of physical system shown in Figure 3

If the goal is to size the pump at J2, the model in Figure 6 can be used with the pump modeled as a fixed flow pump.

Related Topics

Open vs. Closed Systems

Model a Closed Loop System

A closed circulating system can easily be modeled with AFT Fathom by locating a Reservoir 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 Reservoir 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 Reservoir 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 Reservoir junction as a reference pressure for a closed system, you should select the Balance Energy in Reservoir option, which will allow the reservoir 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 either a Reservoir or Assigned Pressure. Pressure type junctions are an infinite source of fluid and do not balance flow. How then can one be used to balance flow in a closed system?

To answer this question, it is worth considering how AFT Fathom views a closed system model. AFT Fathom 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 Fathom'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.
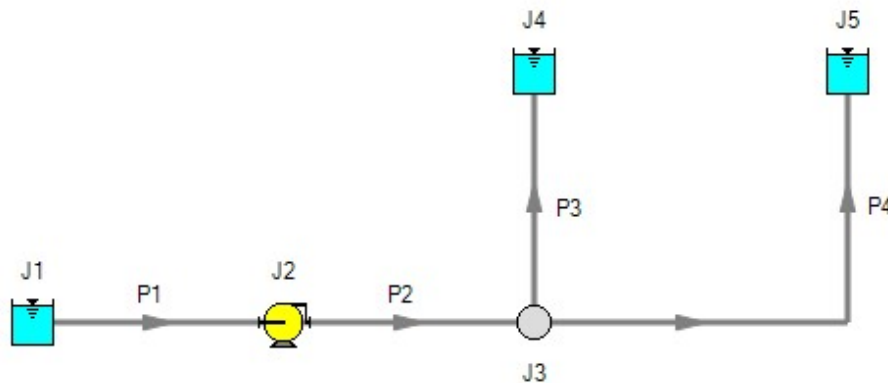
Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5

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., surface elevation and pressure) is used for J1, J11 and J12 in the second system, it will appear to AFT Fathom as an identical system to the first system. The reason is that AFT Fathom takes the first system and applies the J1 reservoir pressure as a boundary condition to pipes P4, P9 and P10. The second system uses three reservoirs to apply boundary conditions to P4, P9 and P10. But if the reservoirs all have the same elevation and pressure, 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 Fathom.

But how is the flow balanced at J1 in the first system? Looking at the first system, one sees that to obtain a system balance, whatever flows into P10 must come back through P4 and P9. Because there is overall system 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 Fathom is not applying a mass balance to J1 directly. It is merely the result of an overall system balance.
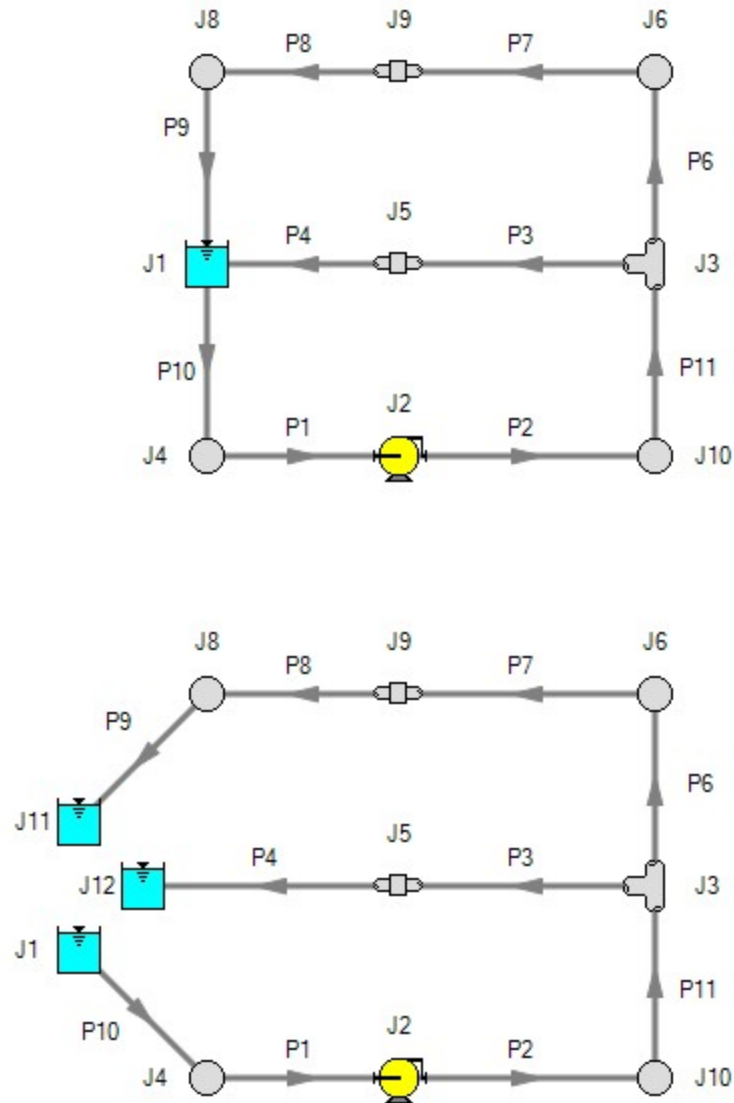
Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two system will appear identical to AFT Fathom.

Balancing Energy

Here the thermal aspects of open and closed systems are discussed.

Consider the system shown in Figure 3 .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 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 reservoir (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 added to the fluid from pumps.

The only way to obtain an overall system energy balance is for the J1 reservoir temperature to adjust to the mixture temperature of all inflowing pipes. This is the function of the "Balance Energy At Junction" feature in the reservoir and assigned pressure junctions. The junction temperature (input as maybe 100° F) is allowed to "float" and find its own equilibrium. Each iteration the floating reservoir 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 enthalpy) that will yield an energy balance at J1. This will be the temperature/enthalpy from Equation 3 in the Network Implementation topic.
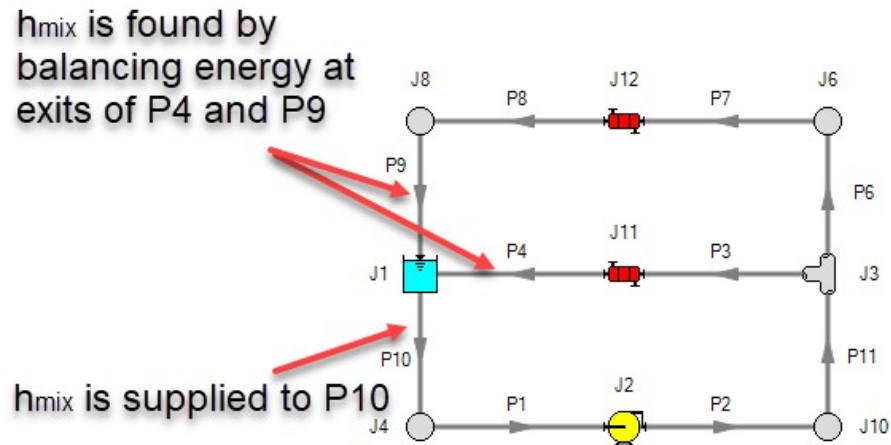
Figure 3: Example of balancing energy in a closed system

Related Topics

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:

- Pump 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:
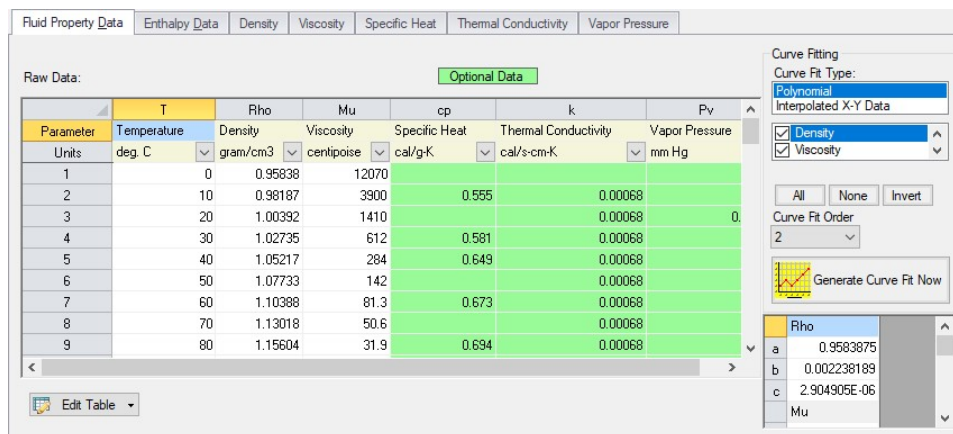
Figure 1: Custom Fluid Definition

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 above fluid has a rapidly decaying viscosity that does not follow a polynomial relationship. Fitting the data to the standard 2nd order polynomial results in the following:
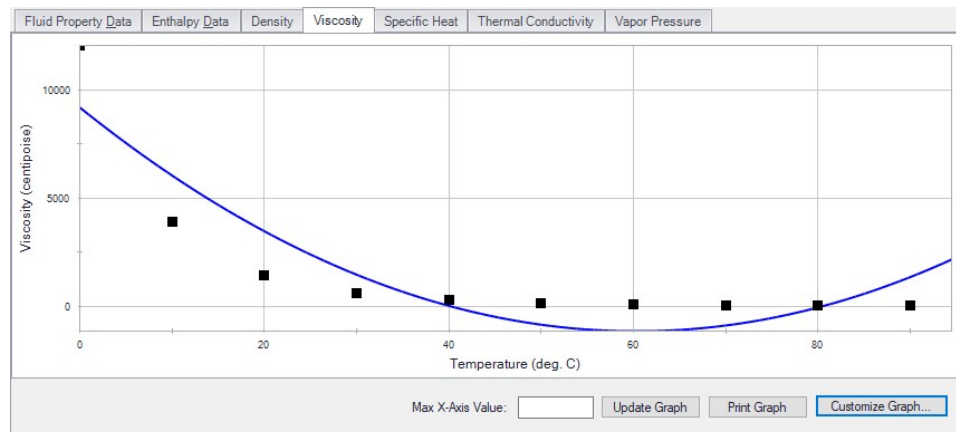
Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data

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

After data has been curve fit, the constants for the resulting curve are displayed in the lower-right of the associated window.

Related Topics

Stagnation vs. Static Pressure Boundaries

With two exceptions (to be discussed), all pressure-type boundary conditions in AFT Fathom are stagnation. This works very well for things such as storage tanks, cooling ponds, lakes, etc., 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 Fathom by use of a Reservoir junction.

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.

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 Fathom model is applied at the exit of the reservoir, not within the reservoir 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.
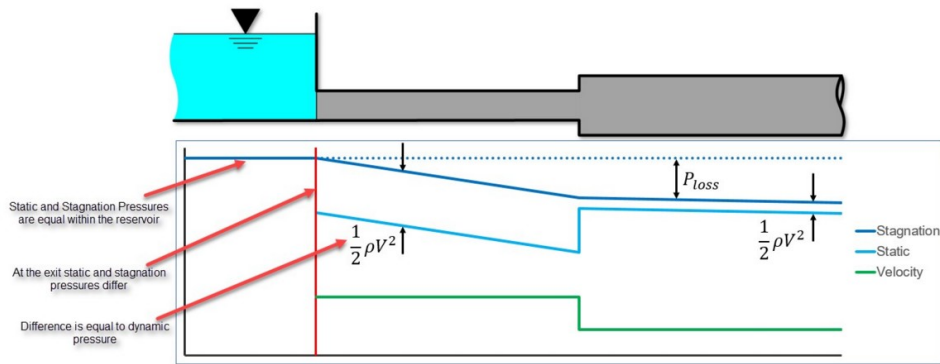
Figure 1: Pressure profile along horizontal pipe shown

The choice of either stagnation or static pressure as a boundary condition depends on whether there is a velocity associated with the pressure that is specified. If the pressure is in a vessel or large (lower velocity) header, then a stagnation pressure is appropriate. If the pressure is in a pipe somewhere in the middle of a system, static pressure is more suited.

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.

In AFT Fathom, 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 for 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?

When the user supplies a stagnation pressure boundary condition, AFT Fathom can use Equation 6 in the Network Solution Methodology topic. Here, no velocity information is required. A

stagnation pressure boundary can thus be supplied to multiple pipes that might have different velocities. That is why the same reservoir junction can connect with multiple pipes.

However, if the user specifies a static pressure as the boundary condition, to apply Equation 4 in the Network Solution Methodology topic, a unique velocity 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 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

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. Open Analysis Setup from either the Common Toolbar or the Analysis menu.

The controls in Analysis Setup are divided into different *groups* in the navigation pane on the left side 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 listed below:

- Modules Group
- Fluid Properties Group
- Pipes and Junctions Group
- Transient Control Group - (XTS Module only)
- 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
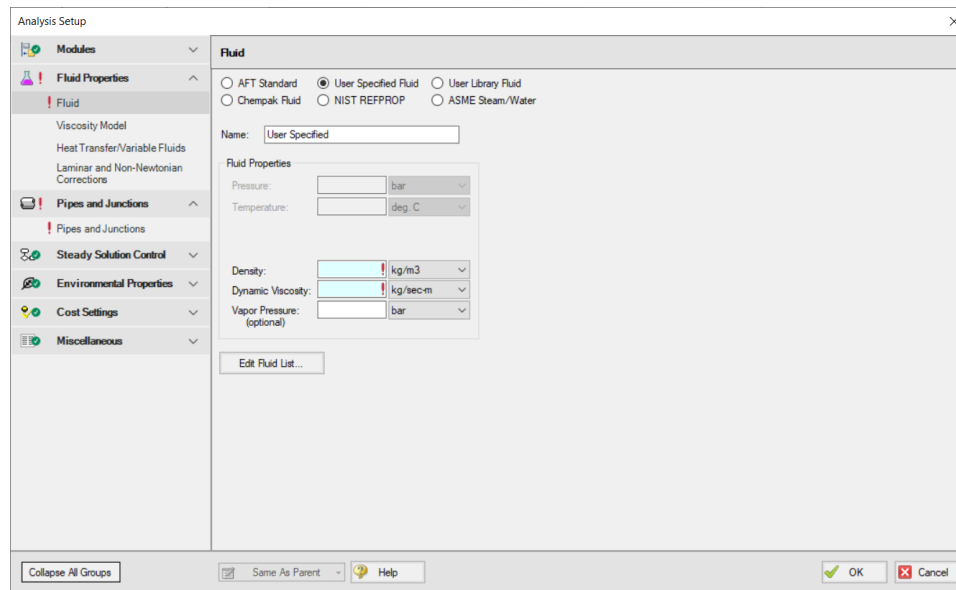
Figure 1: Default state of Analysis Setup in a new blank model

Modules Group

AFT Fathom has four optional add-on modules which extend its modeling capabilities into other areas. The modules allow multivariable goal seeking and control simulation, settling slurry modeling, 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 four AFT Fathom add-on modules. An additional license to the base AFT Fathom 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.
- SSL (Settling Slurry) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.
- 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

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, SSL, XTS, or ANS Add-on Modules.
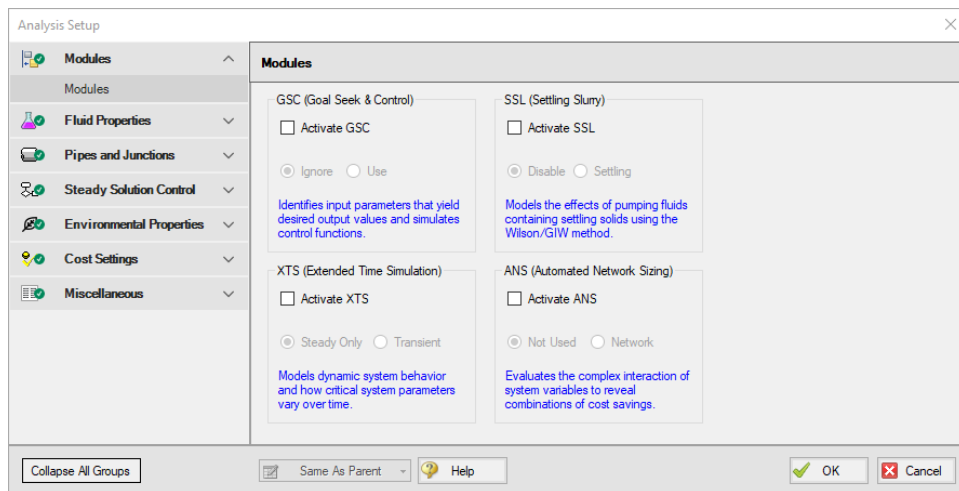
Figure 1: The default state of the Modules panel

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 Fathom application can have the GSC module activated.

Setting the GSC module to Ignore causes the GSC module to function like the base AFT Fathom 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 Fathom application, this data will be lost. Table 1 relates the differences between using AFT Fathom 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 Fathom 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
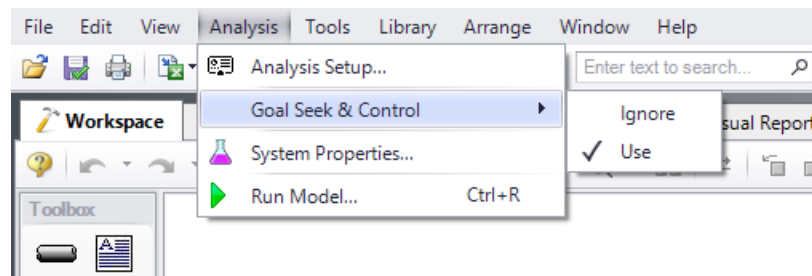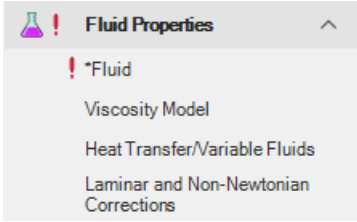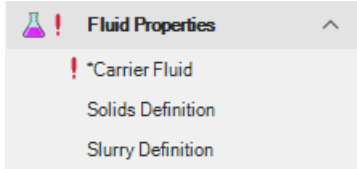
Figure 2: Select Use from the Goal Seek & Control option in the Analysis menu to perform goal seeking when running the model

SSL (Settling Slurry)

Activate the Settling Slurry module by checking the box next to Activate SSL. Once activated, enable it by selecting Settling or disable it by selecting Disable. When the SSL module is activated, the four standard items in the Fluid Properties group are replaced with the three SSL

module fluid properties items as shown in Table 2 below. These items are <u>Carrier Fluid</u>, <u>Solids Definition</u>, and <u>Slurry Definition</u>. Additionally, the Library menu will have a Solids Library option. These items are present while SSL is activated and enabled, but will be switched back if SSL is disabled.

Table 2: Analysis Setup items in Fluid Properties group with and without SSL active

Without SSL Active	With SSL Active
<img flask icon> ! **Fluid Properties** ⌃ ! *Fluid Viscosity Model Heat Transfer/Variable Fluids Laminar and Non-Newtonian Corrections	<img flask icon> ! **Fluid Properties** ⌃ ! *Carrier Fluid Solids Definition Slurry Definition

Activating the SSL module can be done before or after a model is built. If activated after, then additional input will be required for solids data. Pre-existing models built with standard AFT Fathom can be opened with SSL and settling slurry data added.

Settling slurry calculations can be turned off at any time by selecting Disable. The Disable mode causes the SSL module to function like standard AFT Fathom. If a model with SSL activated is opened in standard AFT Fathom without SSL activated, then the SSL data will be lost once the model is saved. Table 3 relates the differences between not using SSL and using it in Disable mode. The purpose of Disable mode is that it allows standard clear fluid calculations to be performed without deactivating SSL. Deactivating the SSL module will cause the loss of the slurry input data for the active scenario.

The SSL module can also be toggled between Settling and Disable from the Analysis menu in the Slurry option as shown in Figure 3. Use this feature to quickly toggle the SSL module between Settling and Disable without opening the Modules panel in Analysis Setup.

Table 3: AFT Fathom feature accessibility based on the SSL module activation and enablement

Features	SSL Not Activated	SSL Activated	
		SSL Disabled	**SSL Enabled**
SSL items in Fluid Properties group	Not Visible	Not Visible	Visible
Slurry in Analysis menu	Not Visible	Visible	Visible
Solids Library in Library menu	Not Visible	Visible	Visible
Flowrate specified in terms of slurry, solids or clear liquid in several junctions	Not Visible	Not Visible	Visible
Slurry data in Output	Not Visible	Not Visible	Visible
Initial Velocity Guess Ratio V/Vsm in Advanced Parameters panel	Not Visible	Not Visible	Visible
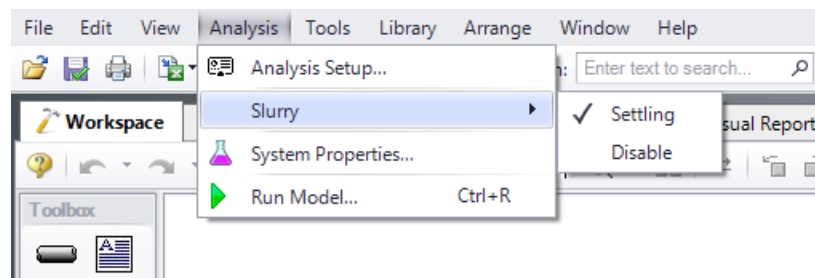
Figure 3: Select Settling from the Slurry option in the Analysis menu to perform settling slurry calculations when running the model

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 Fathom 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 Fathom. 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 Fathom, 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 Fathom 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
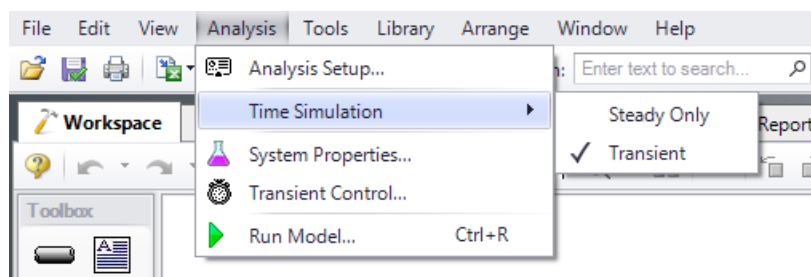
Figure 4: Transient mode is enabled from the Analysis menu

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 Fathom 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 Fathom application. However, all ANS data and settings in the model will still be retained. If the model is opened in the base AFT Fathom 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.
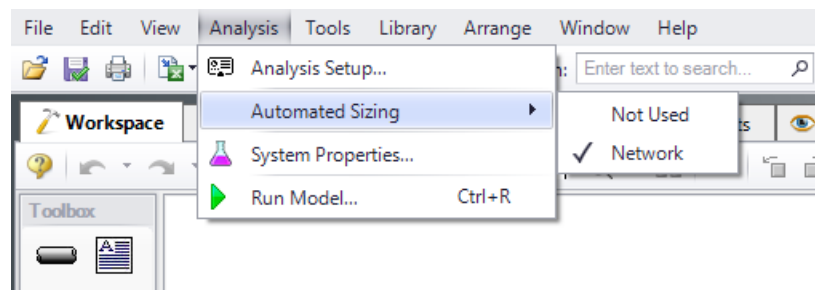
Figure 5: Enabling Automated Sizing from the Analysis menu

Related Topics

Fluid Properties Group

The Fluid Properties group in Analysis Setup gives you control over the fluid properties that influence your pipe system analysis.

While there are a four panels available to input the fluid properties, only one panel is required to run the model. The rest of the panels offer optional input.

Note: If the SSL module is activated the panels in the Fluid properties group will change to show the slurry panels. See SSL Module Fluid Properties for more information.

- Fluid panel - Required panel. Allows the user to define the fluid properties directly, or from one of the available fluid libraries.

- [Viscosity Model panel](#) - Optional panel. Allows user to define a non-Newtonian viscosity model.

- [Heat Transfer/Variable Fluids panel](#) - Optional panel. Allows user to enable one of the heat transfer property options.

- [Laminar and Non-Newtonian Corrections panel](#) - Optional panel. Allows the user to choose what correction factors to apply for laminar or non-Newtonian flow and to which objects the corrections should apply.

Related Topics

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 default selection of User Specified requires the manual definition of Density, Dynamic Viscosity, and Vapor Pressure (optional if cavitation detection is not required). 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. Many of these fluids have thermal property information that allows for the calculation of [heat transfer](#).

- **[ASME Steam/Water](#)** - Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation (1997).

- **[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 approximately 700 fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.

- **User Library Fluid** - If the desired working fluid is not available via the other sources, custom fluid properties can be entered and saved to the User Library for repeated use in the future. Fluids can be created using the Fluid Libraries menu.

Select a library by clicking the radio button next to the name of the library. After selecting a library, the Fluids Available in Library list is populated. One of the fluids can be selected and used with the Add To Model button.

Note: All fluids in Fathom are treated as incompressible as stated in the Engineering Assumptions. This includes fluids in the gas phase and fluids above the critical point.

AFT Standard Fluid Library

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. Many of these fluids have thermal property information that allows for the calculation of heat transfer.

Generally, fluids in the AFT Standard library are assumed to have properties that only vary with temperature which follows the incompressible engineering assumption of Fathom.

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

Vapor pressure is listed as optional since it is only needed to check if cavitation exists in the pipes/junctions.
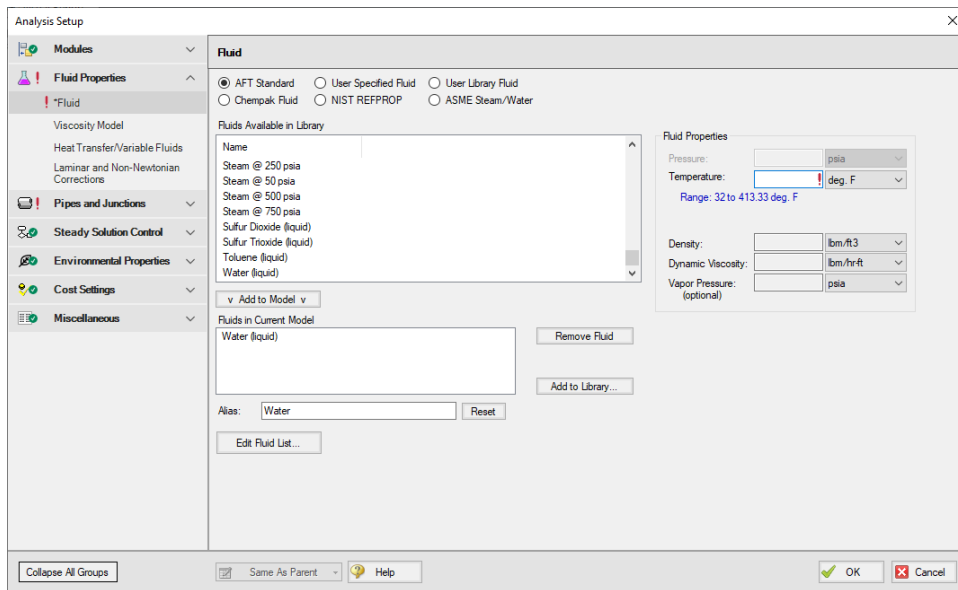
Figure 1: Fluid panel in Analysis Setup with the AFT Standard fluid library selected

ASME Steam/Water 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.

After selecting ASME Steam/Water, "ASME '97 Water" is automatically added to in the Fluids in Current Model area. Select the Fluid Phase and enter a Pressure and Temperature to calculate the properties as shown in Figure 2.

Detailed fluid property 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.
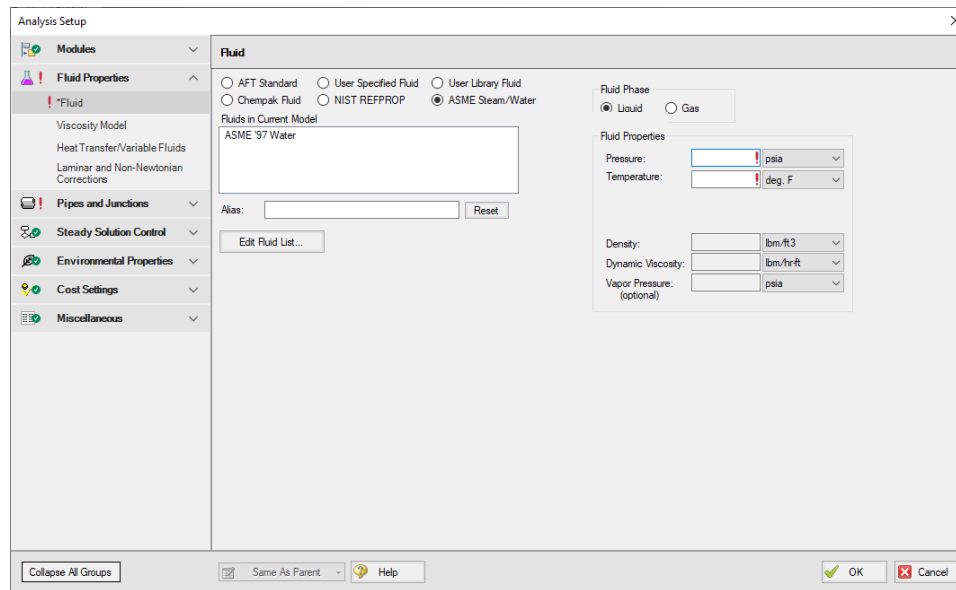
Figure 2: Fluid panel in Analysis Setup with the ASME Steam/Water fluid library selected

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. 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.

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model. Select the Fluid Phase and enter both a Pressure and Temperature in the Fluid Properties area on the right, as shown in Figure 3.

Fluids from the NIST REFPROP library support fluid mixtures, but no chemical reactions are modeled as listed in the Engineering Assumptions.
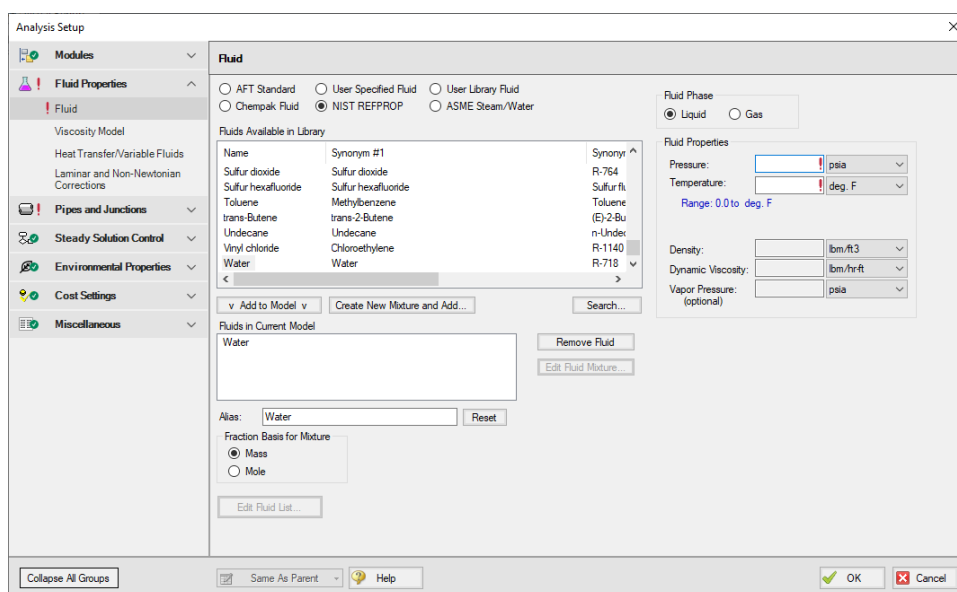
Figure 3: Fluid panel in Analysis Setup with the NIST REFPROP fluid library selected

Chempak Fluid Library

Optional add-on for purchase which contains data for approximately 700 fluids. Chempak is licensed from Madison Technical Software.

Fluids from the Chempak library support fluid mixtures, but no chemical reactions are modeled as listed in the Engineering Assumptions.

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model. Select the Fluid Phase and enter both a Pressure and Temperature in the Fluid Properties area on the right, as shown in Figure 4.

Information on the theory and calculation methods used in Chempak is given in the Chempak Technical Information topic.

Detailed fluid property 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 Database license is required to use the Chempak Viewer.

For more information on AFT Chempak Viewer see the Chempak Viewer Help File.
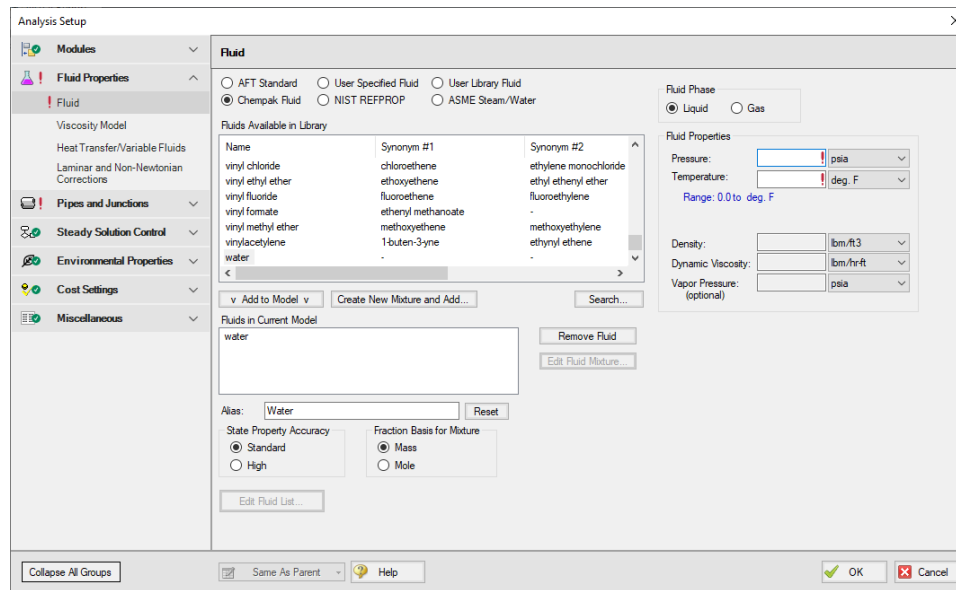
Figure 4: Fluid panel in Analysis Setup with the Chempak fluid library selected

State Property Accuracy Options

The State Property Accuracy options are applicable only to Chempak fluids and are located on the Fluid panel below the Fluids in Current Model area.

The State Property Accuracy influences the number of elements used in curve fits for calculating property data. There are two accuracy options: Standard and High. You will usually want to start with standard accuracy, which is selected by default, and use high accuracy for final runs, depending on the importance and run time for the model. The high accuracy option will increase run time by about a factor of two.

When performing interpolations for properties, a 4 point or 16 point interpolation scheme can be used. The 4 point scheme implemented in AFT Fathom 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.

Related Topics

Creating Liquid 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.

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

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

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

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

Note: The NIST REFPROP fluid library is not able to calculate the mixing parameters for most mixtures containing water, helium, hydrogen, neon, carbon monoxide, fluorine, methanol, or ethanol.

Related Topics

Transport Properties

The transport properties in AFT Fathom (specific heat, dynamic viscosity, and thermal conductivity) are assumed to be functions of temperature only. In general this is a good assumption, although systems at very high pressure usually start to show some pressure dependence. These properties are modeled as polynomial curve fits of temperature or interpolated.

Related Topics

Viscosity Model Panel

AFT Fathom can model systems with Newtonian or non-Newtonian fluids. A Newtonian fluid is one where the fluid viscosity is fixed and independent of fluid velocity, while a non-Newtonian fluid is one where the fluid viscosity is variable and dependent on fluid velocity. Most industrial fluids are Newtonian fluids, although there are important applications for non-Newtonian fluids. By default, AFT Fathom has the Newtonian viscosity model selected.

AFT Fathom offers several viscosity models:

- Newtonian (default)
- Paper Stock - Duffy
- Paper Stock - Brecht & Heller
- Power Law
- Bingham Plastic
- Herschel-Bulkley
- Homogeneous Scale-up

Modeling non-Newtonian fluids is a complex task. A myriad of theoretical models and test equipment exist, and each have various names unique to an industry or application. Each of the non-Newtonian viscosity models require rheological inputs to be defined. Those inputs are detailed on the pages for each model as well as an explanation as to what the viscosity model is meant for, how to use the model, and how the model calculates hydraulic behavior in a system.

Frictional Losses

Each non-Newtonian fluid model adjusts the frictional losses in the system from the Newtonian fluid model. The equations and correlations used to determine pipe frictional losses are detailed on each model's page. These methods to determine frictional losses replace the pipe friction model specified in the Pipe Properties window. For pipe Fittings and junction-specific losses, all

non-Newtonian models use the same method, as detailed in the Laminar and Non-Newtonian Corrections panel topic.

Related Topics

Duffy Method (Pulp and Paper Stock)

The Duffy method is a system developed by G.G. Duffy at the University of Auckland, New Zealand and formalized by the Technical Association of the Pulp and Paper Industry (TAPPI). The model uses correlations based on data obtained from laboratory flow loops and real-world pulp mill piping systems.

TAPPI states that the correlations behind this model do not necessarily provide the actual design value of friction loss. Instead, it provides a means to determine the minimum value of pipe friction loss an engineer can expect. Each system designer must determine and apply a safety factor as they deem necessary.

Fathom contains data on 21 different stock fluid selections that can be used with the Duffy method. The constants used for each stock fluid are displayed in the General Results section of the output. Additionally, the Vmax and Vwater terms, the dryness coefficient, and the Safety Factor can all be displayed in the Pipe Output.

Note: The Duffy method is only valid for pulp consistencies between 2% and 6%.

Duffy Method Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Duffy method is selected.
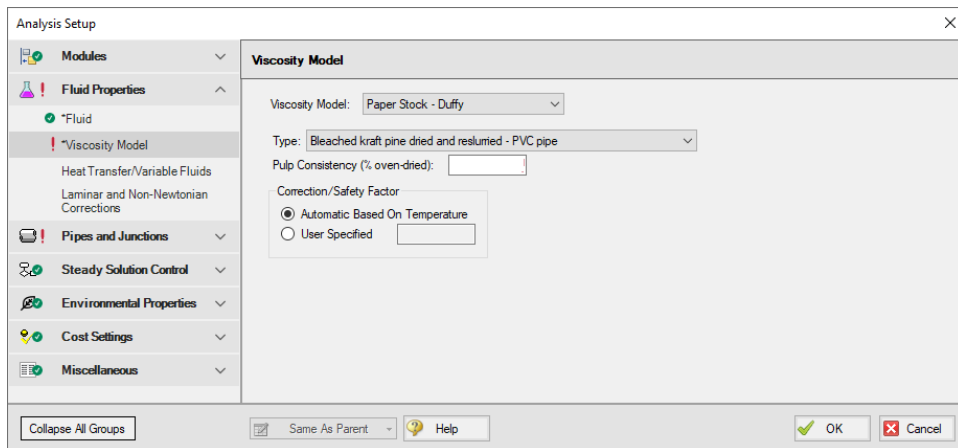
Figure 1: Viscosity Model panel for the Duffy Method

Users must select a type of pulp to model and define a pulp consistency. The user must also choose to correct the Duffy method parameters based on temperature automatically or via a user-specified correction. The constants used by Fathom are based on a pulp temperature of 35 deg. C. The Automatic Based on Temperature will use the temperature specified on the Fluid panel (or the temperature in each pipe in the case of Variable Fluid Properties) to determine the pulp temperature.

There are 21 pulp types to choose from:

- Bleached kraft pine dried and reslurried - PVC pipe
- Bleached kraft pine dried and reslurried - SS pipe
- Bleached straw - copper pipe
- Bleached sulfite - copper pipe
- Hardwood NSSC CSF = 620 - PVC pipe
- Kraft - copper pipe
- Kraft birch dried and reslurried - PVC pipe
- Long-fibered kraft dried and reslurried - PVC pipe
- Long-fibered kraft never dried CSF = 260 - PVC pipe
- Long-fibered kraft never dried CSF = 550 - PVC pipe
- Long-fibered kraft never dried CSF = 650 - PVC pipe
- Long-fibered kraft never dried CSF = 725 - PVC pipe
- Long-fibered kraft never dried CSF = 725 - SS pipe
- Newsprint broke CSF = 75 - PVC pipe
- Refiner groundwood CSF = 150 - PVC pipe
- Refiner groundwood (hardboard) - PVC pipe
- Refiner groundwood (insulating board) - PVC pipe
- Stone groundwood CSF = 114 - PVC pipe
- Unbeaten aspen sulfite never dried - SS pipe

- Unbleached straw - copper pipe
- Unbleached sulfite - copper pipe

Theory and Correlations

AFT Fathom models frictional losses for the Duffy method using correlations taken from the TAPPI Generalized Method for Determining the Pipe Friction Loss of Flowing Pulp Suspensions.

The maximum velocity (m/s) for the Duffy model is given by:

$$V_{max} = K'\ C^{\sigma}$$

The velocity at which pulp effects are negligible and the frictional losses of the pulp mixture are the same as the frictional losses of pure water:

$$V_{water} = 1.22\ C^{1.40}$$

In the case where $V < V_{max}$:

$$\frac{dH}{L} = F\,K\,V^{\alpha}\ C^{\beta}\ D^{\gamma}$$

Where:

- dH/L = head loss per length (m water/100 m pipe)
- C = pulp consistency (% dryness)
- D = inner pipe diameter (mm)
- F = correction factor for temperature or pipe roughness (dimensionless)
- K', K, σ, α, β, γ = constants depending on pulp type and pipe material (dimensionless)
- V = actual pipe velocity, when $V_{max} > V$ (m/s)
- $V = V_{max}$, when $V_{water} > V > V_{max}$

When pipe velocity is larger than V_{water}, standard pipe friction models are used, treating the pulp mixture as pure water.

Related Topics

Brecht & Heller Method (Pulp and Paper Stock)

The Brecht and Heller method is a system developed by Brecht and Heller at the University of Maine and formalized by the Technical Association of the Pulp and Paper Industry (TAPPI) and the Cameron Hydraulic Handbook. The model uses correlations based on data obtained from laboratory flow loops and real-world pulp mill piping systems.

This method is considered more conservative than the Duffy method, meaning the Brecht and Heller method will predict larger pressure losses in a system, all else constant.

Fathom contains data on eight different stock fluid selections that can be used with the Brecht and Heller method. The constants used for each stock fluid are displayed in the General Results section of the output. Additionally, the pseudo-Reynolds Number (Ré) and Friction Factor terms can be displayed in the Pipe Output.

Brecht and Heller Method Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Brecht & Heller method is selected.
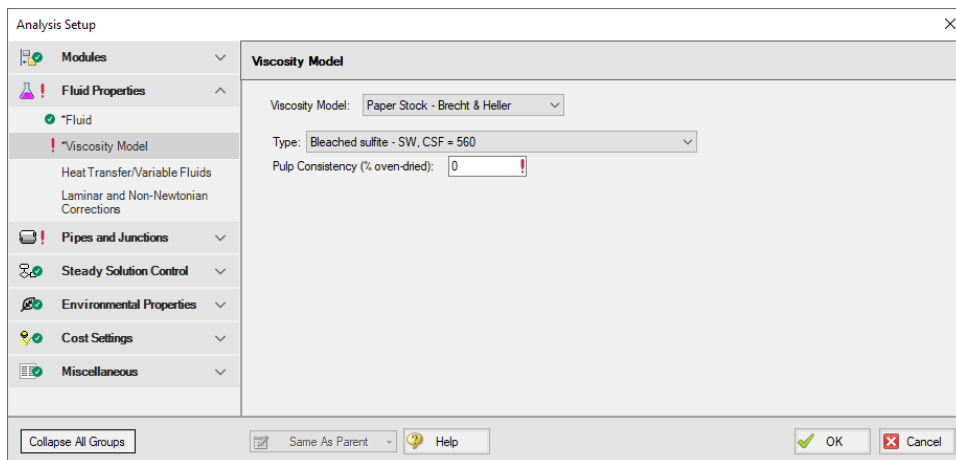
Figure 1: Viscosity Model panel for the Brecht & Heller Method

Users must select a type of pulp model and define a pulp consistency. There eight pulp types to choose from:

- Bleached sulfite - SW, CSF = 560
- Pre-steamed groundwood - SW, CSF = 200
- Reclaimed fiber
- Soda - HW
- Stone groundwood - SW, CSF = 70
- Unbleached kraft - SW, CSF 730

- Unbleached sulfite - SW, CSF = 640
- Unbleached, unrefined sulfite SW

Theory and Correlations

AFT Fathom models frictional losses for the Brecht and Heller method using correlations taken from the Cameron Hydraulic Handbook, pg 3-88.

The pseudo-Reynolds number for the paper stock (*Ré*) is:

$$Re' = \frac{\rho V D^{0.205}}{C^{1.157}}$$

The pipe friction factor is:

$$f = \frac{3.97}{Ré'^{1.636}}$$

The friction loss in feet along the pipe is:

$$dH = \frac{f V^2 L K}{D}$$

Where:

- D = inner diameter (feet)
- L = pipe length (feet)
- V = average velocity (feet/sec)
- ρ = stock fluid density (lbm/ft3)
- C = pulp consistency (% dryness)
- K = friction factor constant, specific to pulp type (dimensionless)

Related Topics

Power Law Model

The Power Law model for non-Newtonian fluids is one of the most common viscosity models used in hydraulic analysis. This model states that the shear stress required to shear a fluid at a given rate can be described by the following equation:

$$\tau = K \left(\frac{du}{dy} \right)^n$$

Where τ is the shear stress, du/dy is the shear rate (or velocity gradient), and K and n are the Power Law constants.

The Power Law model defines η, the apparent viscosity of the fluid as:

$$\eta = K \left(\frac{du}{dy} \right)^{n-1}$$

In the Power Law model, a value of $n=1$ corresponds to a Newtonian fluid, while $0 < n < 1$ is a Shear Thinning (pseudoplastic) fluid and $n > 1$ represents a Shear Thickening (dilatant) fluid.

To employ this model, users must have values for K and n, or have rheological test data.

Power Law Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Power Law model is selected.
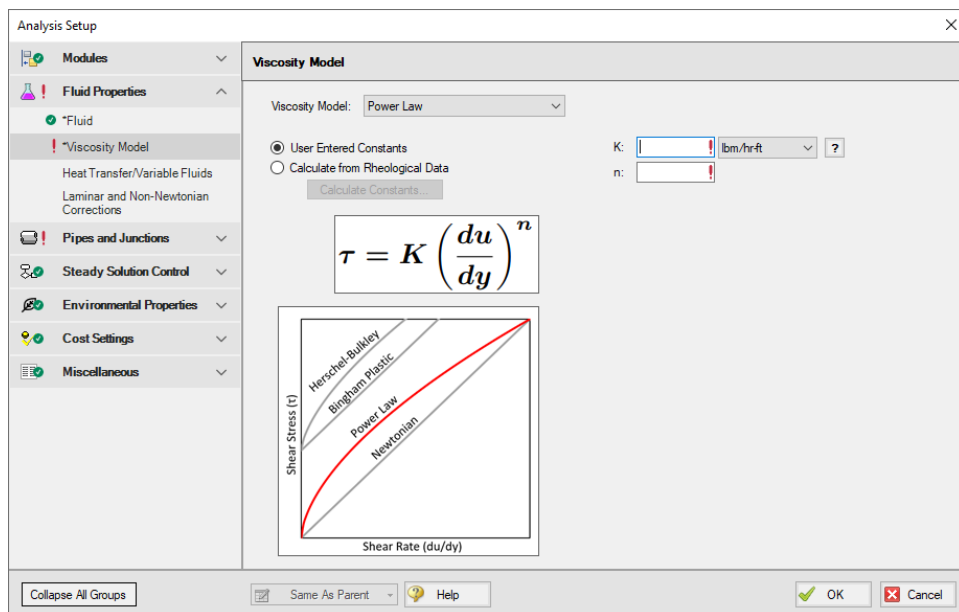
Figure 1: Viscosity Model panel for the Power Law model

Users must enter values for the Power Law constants, *K* and *n*. If the user has raw rheological test data instead, they can enter that directly using the Calculate from Rheological Data option.

Theory and Correlations

AFT Fathom models the frictional losses due to a Power Law fluid using correlations taken from Darby, pg. 166-167. Note that the original equations in Darby use the Fanning friction factor, while AFT software uses the Moody friction factor. The equations below have been modified from the original accordingly.

$$f = (1 - \alpha) f_L + \frac{\alpha}{\left(f_T^{-8} + F_{Tr}^{-8}\right)^{\frac{1}{8}}}$$

Where:

$$f_L = \frac{64}{Re_{pl}}$$

$$f_T = \frac{0.2728 \, n^{-0.5}}{Re_{pl}^{1/(1.87 + 2.39n)}}$$

$$f_{Tr} = 7.16 * 10^{-4} e^{-5.24n} Re_{pl}^{0.414 + 0.757n}$$

$$\alpha = \frac{1}{1 + 4^{-\Delta}}$$

$$\Delta = Re_{pl} - Re_{plc}$$

The Reynolds number for power law fluid (Re_{pl}) is defined as:

$$Re_{pl} = \frac{8 \rho V^{2-n} D^n}{K \left(\frac{2(3n+1)}{n}\right)^n}$$

The critical Reynolds number for power law fluid (Re_{plc}) where laminar flow ceases is defined as:

$$Re_{plc} = 2100 + 875 \, (1 - n)$$

The friction factor calculated using this correlation is then used to determine pressure loss following Equation 3 in the Network Solution Methodology topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Power Law constants.

Related Topics

Bingham Plastic Model

The Bingham Plastic model for non-Newtonian fluids is one of the most common viscosity models used in hydraulic analysis. This model states that the shear rate resulting from a given shear stress applied to the fluid can be described with the following piecewise equation:

$$\frac{du}{dy} = \begin{cases} 0, & \tau < \tau_o \\ \frac{(\tau - \tau_o)}{\mu_\infty}, & \tau \geq \tau_o \end{cases}$$

Where du/dy is the shear rate (or velocity gradient), τ is the applied shear stress, τ_o is the yield stress, and μ_∞ is the plastic viscosity or coefficient of rigidity.

The Bingham Plastic model describes η, the apparent viscosity of the fluid as:

$$\eta = \frac{\tau_o}{du/dy} + \mu_\infty$$

The Bingham Plastic model uses a constant value for plastic viscosity, meaning a value for μ_∞ of 0 would result in a Newtonian fluid. Some fluids may see Bingham Pseudoplastic behavior, where shear stress and shear rate do not have a linear relationship. These fluids can be modeled using the Herschel-Bulkley viscosity model.

To employ this model, users must have values for τ_o and μ_∞, or have rheological test data.

Bingham Plastic Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Bingham Plastic model is selected.
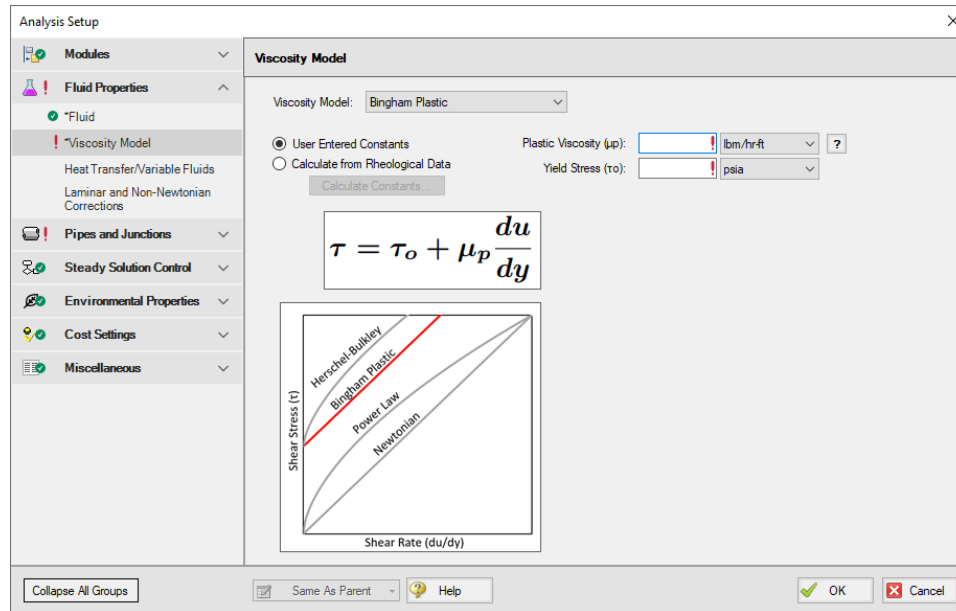
Figure 1: Viscosity Model panel for the Bingham Plastic model

Users must enter values for the Bingham Plastic constants, μ_∞ and τ_o. If the user has raw rheological test data instead, they can enter that directly using the Calculate from Rheological Data option instead.

Theory and Correlations

AFT Fathom models the frictional losses due to a Bingham Plastic fluid using correlations taken from Darby, pg. 168-169. Note that the original equations in Darby use the Fanning friction factor, while AFT software uses the Moody friction factor. The equations below have been modified from the original accordingly.

$$f = \left(f_L^m + f_T^m \right)^{\frac{1}{m}}$$

Where:

$$m = 1.7 + \frac{40,000}{Re}$$

$$f_L = \frac{64}{Re} \left(1 + \frac{He}{6\,Re} - \frac{64He^4}{3f_L^3\,Re^7} \right)$$

$$f_T = 4\left(\frac{10^a}{Re^{0.193}}\right)$$

$$a = -1.47\left(1 + 0.146e^{-2.9*10^{-5}He}\right)$$

The Reynolds number for a Bingham Plastic fluid is defined as:

$$Re = \frac{\rho V D}{\mu_\infty}$$

The Hedstrom number for a Bingham Plastic fluid is defined as:

$$He = \frac{\rho\,\tau_o D^2}{\mu_\infty^2}$$

The friction factor calculated using this correlation is then used to determine pressure loss following Equation 3 in the Network Solution Methodology topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Bingham Plastic constants.

Related Topics

Herschel-Bulkley Model

The Herschel-Bulkley model for non-Newtonian fluids states that the shear rate resulting from a given shear stress applied to the fluid can be described with the following equation:

$$\tau = \tau_o + K\left(\frac{du}{dy}\right)^n$$

Where *du/dy* is the shear rate (or velocity gradient), τ is the applied shear stress, τ_o is the yield stress, *K* is the consistency coefficient, and *n* is the flow behavior index.

The Herschel-Bulkley model describes describes η, the apparent viscosity of the fluid as:

$$\eta = \frac{\tau_o}{du/dy} + K\left(\frac{du}{dy}\right)^{n-1}$$

In the Herschel-Bulkley model a value for τ_o of 0 and a value for n of 1 would result in a Newtonian fluid. The Herschel-Bulkley model is equivalent to the Bingham Plastic model if $n = 1$, and is equivalent to the Power Law model if $\tau_o = 0$.

To employ this model, users must have values for τ_o, K and n, or have rheological data.

Herschel-Bulkley Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Herschel-Bulkley model is selected.
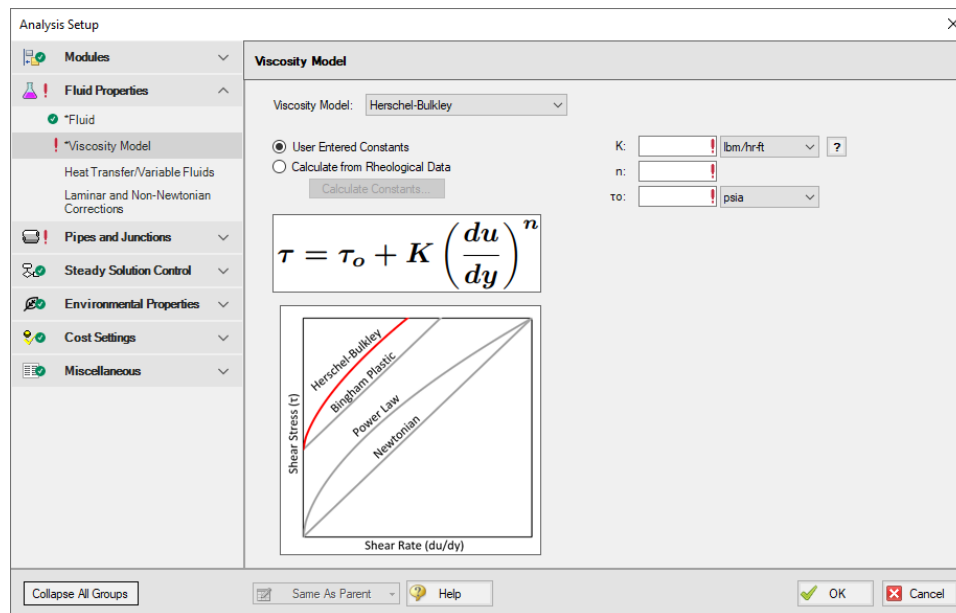
Figure 1: Viscosity Model panel for the Herschel-Bulkley model

Users must enter values for the Herschel-Bulkley constants, K, n, and τ_o. If the user has raw rheological test data instead, they can enter that directly using the Calculate from Rheological Data option.

Theory and Correlations

AFT Fathom models the frictional losses due to a Herschel-Bulkley fluid using correlations as are summarized in Garcia & Steffe (1986). Note that the original equations in the reference use the Fanning friction factor, while AFT software uses the Moody friction factor.

The **laminar flow** friction factor is defined as

$$f_{L,fanning} = \frac{16}{\psi Re}$$

with

$$\psi = (1 + 3n)^n (1 - \xi_o)^{1+n} \left[\frac{(1-\xi_o)^2}{(1+3n)} + \frac{2\xi_o(1-\xi_o)}{(1+2n)} + \frac{\xi_o^2}{(1+n)} \right]^n$$

and ξ_o, the dimensionless unsheared plug radius:

$$\xi_o = \frac{\tau_o}{\tau_w} = \frac{\tau_o}{0.5 f \rho v^2}$$

The **turbulent flow** friction factor can be calculated as follows

$$f_{T,fanning} = \left[0.45 - \frac{2.75}{n} + \frac{1.97}{n} ln(1 - \xi_o) + \frac{1.97}{n} ln \left(Re \left(\frac{1+3n}{4n} \right)^n f^{1-(n/2)} \right) \right]^{-2}$$

where

$$\xi_o = \frac{16(2\,He)^{n/(2-n)} \left(\dfrac{n}{1+3n} \right)^{2n/(2-n)}}{Re^{2/(2-n)}\,f}$$

Where *He* is the Hedstrom number for Herschel-Bulkley fluids and is defined as

$$He = \frac{D^2 \rho}{K} \left(\frac{\tau_o}{K} \right)^{\frac{2-n}{n}}$$

Re is the Reynolds number for Herschel-Bulkley fluids, and is equivalent to the Reynolds number for Power Law fluids:

$$Re_{pl} = \frac{8\,\rho\,V^{2-n}\,D^n}{K \left(\frac{2(3n+1)}{n} \right)^n}$$

The **critical Reynolds number**, which represents the transition from laminar to turbulent flow for Herschel Bulkley fluids is defined as

$$Re_c = \frac{6464\,n\,\psi_c^{(2/n)-1}(2+n)^{(2+n)/(1+n)}}{(1+3n)^2(1-\xi_{oc})^{(2/n)+1}}$$

Where n is the flow behavior index, Ψ_c is equivalent to Ψ with $\xi_o = \xi_{oc}$, and ξ_{oc} is defined implicitly as follows

$$He = \frac{3232(2+n)^{(2+n)/(1+n)}\xi_{oc}^{(2/n)-1}}{n(1-\xi_{oc})^{(2/n)+1}}$$

The friction factor calculated using this correlation is then used to determine pressure loss following Equation 3 in the Network Solution Methodology topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Herschel-Bulkley constants.

Curve Fit Methodology

Unlike the Power Law and Bingham Plastic models the Herschel-Bulkley method uses three parameters, so a simple linear regression cannot be used to curve fit all three constants.

Instead Fathom employs the method discussed in Mullineux (2008), which first solves a root-finding equation to determine n, then performs a non-linear regression of τ vs x^n to determine the constants K and τ_o.

The curve fit results using this method have been proven to be identical to the results of a non-linear least squares regression, which is one of the most common methods used to curve fit the Herschel-Bulkley parameters.

If a different curve fitting method is preferable to represent the fluid being modeled the User Entered Constants option should be selected instead.

Related Topics

Homogeneous Scale-up Method

The Homogeneous Scale-up method for non-Newtonian fluids is based on the idea that test data for velocity and pressure drop along pipes of a given diameter can be scaled up to model the velocity and pressure drop for a pipe of a large size.

This scale-up is similar to using the affinity laws to predict pump behavior. Instead of scaling pressure rise and volumetric flowrate for a pump, the Homogeneous Scale-up method scales the frictional gradient (j_m) and mean velocity (V_m) for a pipe.

To employ this method, users must have test data from the same Tube Flow Rheometer as is available when entering rheological data for the Power law, Bingham Plastic, and Herschel-Bulkley models.

Homogeneous Scale-up Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Homogeneous Scale-up method is selected. Click the Enter Scaled Data button to enter data.
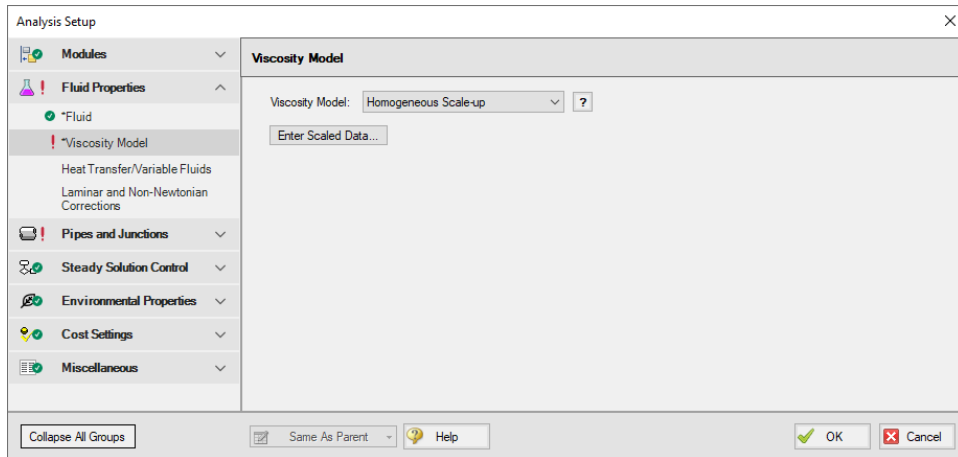
Figure 1: Viscosity Model panel for the Homogeneous Scale-up method

The Homogeneous Scale-up window is shown below in Figure 2. Here, the user must specify information about the pipe used to obtain the test data along with the test data itself. The flow data can be entered as mass flowrate, velocity, volumetric flowrate, or shear rate data as determined by a tube flow rheometer. The Head/Pressure data can be entered in terms of Head gradient, Pressure gradient, or shear stress data as determined by a tube flow rheometer.

Each data point must also be marked as laminar or turbulent flow.

Figure 2: Homogeneous Scale-up Data entry window

The user can also generate a graph of their data plotted against other viscosity models and larger scaled pipe sizes. This interface is shown in Figure 3 below. Note that generating a cross-plot for the Power Law or Bingham Plastic models does not mean Fathom will use those models in the system. The cross-plot is intended only to give a visual comparison of the data.
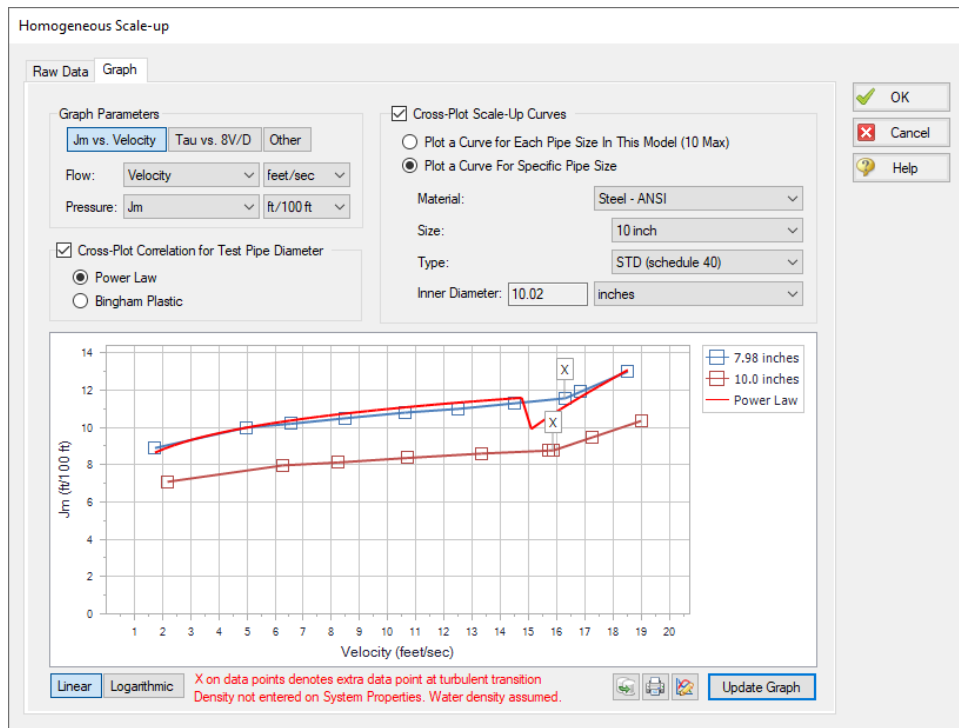
Figure 3: Graphed data for the Homogeneous Scale-up method

Theory and Correlations

The Homogeneous Scale-up methods are based on Chapter 3 of Wilson, et al, 2006, page 67.
Laminar data scales as follows:

$$(j_m)_2 = (j_m)_1 + \frac{D_1}{D_2}$$

$$(V_m)_2 = (V_m)_1 + \frac{D_2}{D_1}$$

Turbulent data scales as follows:

$$(j_m)_2 = (j_m)_1 + \frac{D_1}{D_2}$$

$$(V_m)_2 = (V_m)_1 + 2.5\,U\,\ln\left(\frac{D_2}{D_1}\right)$$

$$U = \sqrt{\frac{\tau}{\rho}}$$

Where:

- Data point 1 is the test data
- Data point 2 is the larger system
- τ is the shear stress at the pipe wall

The pressure drop calculated using these methods is then substituted into the network solution and iterated upon until a converged solution is available.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data the user has available.

Related Topics

Using Rheological Data

For the Power Law, Bingham Plastic and Herschel-Bulkley viscosity models, users are expected to define constants for each model. If the user already has these constants, they can be entered directly into the software. If the user only has the original rheological data, Fathom can calculate the viscosity model constants from that data.

The interface used to enter this data and the processes Fathom employs to manipulate it are similar for all of the viscosity models mentioned above. This page discusses rheological data only in the context of the Power Law model for simplicity. Unless specified otherwise, the content discussed here is applicable to all models.

Calculate from Rheological Data Interface

The Calculate from Rheological Data option can be accessed from the Viscosity Model panel in Analysis Setup. Click the Calculate from Rheological Data radio button and click the Calculate Constants button as shown below in Figure 1.
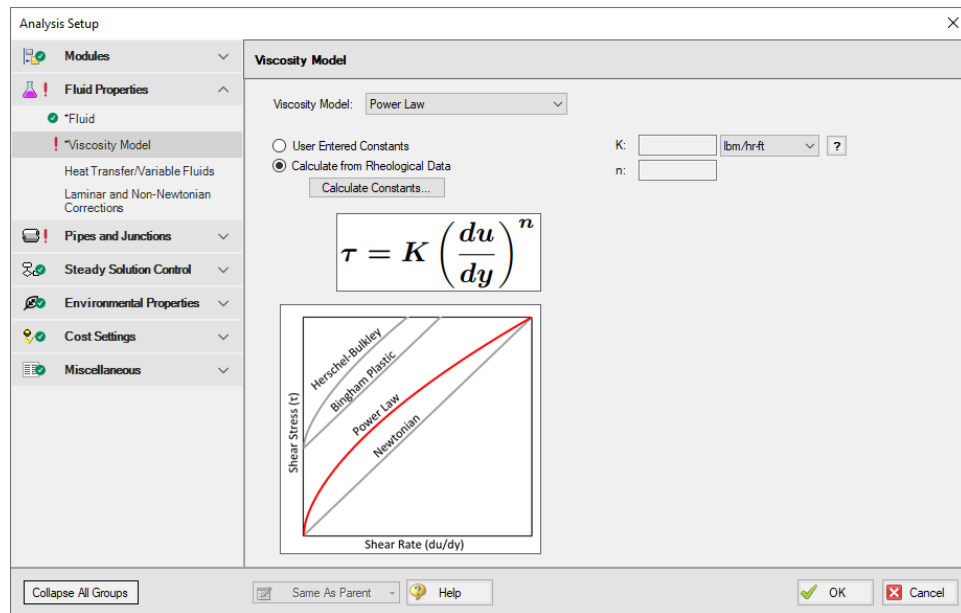
Figure 1: Power Law viscosity model using the Calculate from Rheological Data option

The Calculate Power Law Constants window will open to enter rheological data. This window is shown below in Figure 2. In this window, users are asked to enter Shear Stress (τ) vs. Shear Rate data in units of pressure and inverse time, respectively.

Two Raw Data Type options are presented above the table, Tube Flow Rheometer Data (8*V/D*) and Shear Rheometer Data (*du/dy*). These options refer to the experimental setup used to obtain the shear stress vs shear rate data entered in the table. Fathom version 11 and earlier assumed all entered data was the Tube Flow Rheometer Data (8V/D) type. Refer to the Types of Rheological Data section below for more information on which option to use.

Note: For Herschel-Bulkley fluids only the Shear Rheometer Data option is available at this time. This is because the method employed in Fathom to convert from Tube Flow Rheometer Data to Shear Rheometer Data has unknown accuracy for use with the Herschel-Bulkley model.

Figure 2: Rheological data input form

The user can click Generate Curve Fit Now to curve fit the data and obtain the viscosity model constants. Generating the curve fit will show the user a plot of their data along with the fitted constants, as shown below. Clicking OK will transfer those constants to the Viscosity Model panel and display the resultant Shear Stress vs Shear Rate curve fit graph.
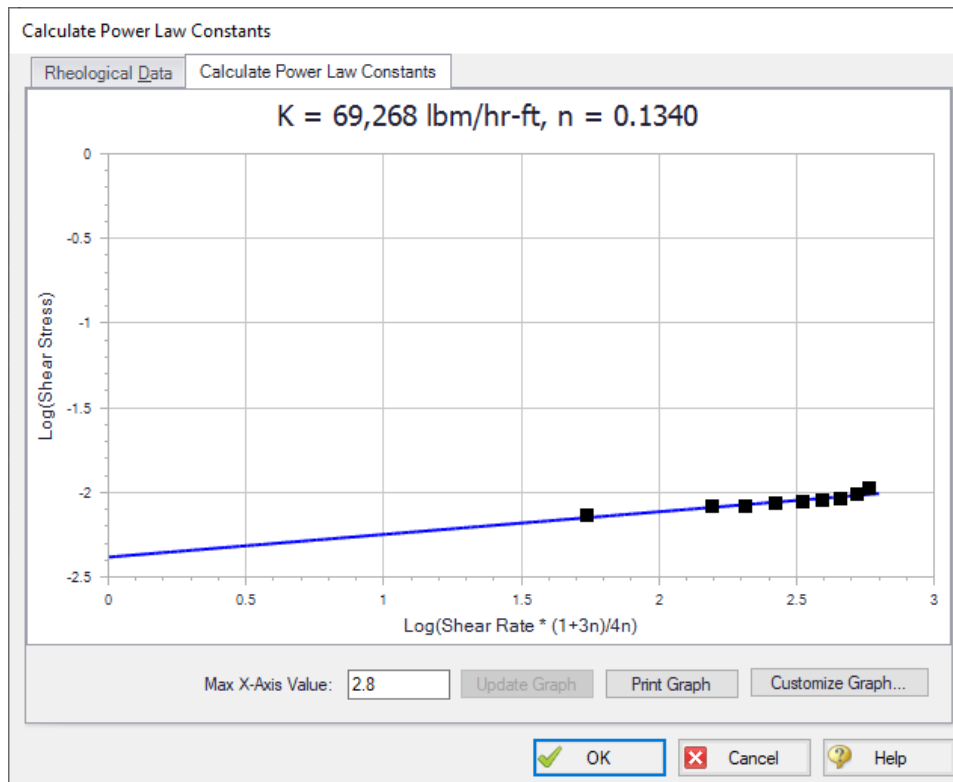
Figure 3: Curve fit rheological data and constants

Types of Rheological Data

Two major types of rheological test data exist. Many names and potential test devices exist for each type. For the purposes of this discussion, we must pick a single name for each type. One experimental setup is a tube flow rheometer while the other is a rotational rheometer.

Tube Flow Rheometer

A tube flow rheometer system consists of a pipe of some diameter and length connected to a high pressure source and a low pressure source. Varying the high pressure source drives flow through the pipe at different velocities. This experimental setup allows the engineer to obtain data on the pressure gradient (pressure drop per length) along the pipe for a given velocity. The engineer could also obtain this type of rheological data by directly measuring pressure and velocity along a pipe in an existing facility.

This data is then manipulated into shear stress and shear rate data, respectively. Specifically, the tube flow rheometer gives shear rate data where the shear rate is the bulk shear rate of the fluid. The measured velocity is the bulk velocity of the fluid, meaning only bulk shear rate can be calculated. Bulk shear rate can be calculated from bulk velocity using the equation 8V/D.

This type of rheological data may be more common in industries considering slurry applications or when the process facility is already in operation and data is obtained via direct measurements of the system operations.

Rotational Rheometer

A rotational rheometer system consists of a rotating device in contact with the fluid being studied. Common examples of this system include the cup and bob (couette) rheometer and the cone and plate rheometer. Varying the angular rate of the rotating device creates different shear rates in the fluid. This experimental setup allows the engineer to obtain data on the torque required to achieve a given angular rate of rotation.

This data is then manipulated into shear stress and shear rate data, respectively. Specifically, the rotational rheometer gives shear rate data where the shear rate is the wall shear rate of the fluid. The angular rate of rotation and equipment geometry are directly known, and thus the shear rate is calculated at the wall of the rotating device.

This type of rheological data is more common when data is obtained via small-scale, laboratory tests. If rheological data is presented with information about rate of rotation (RPM, revolutions per second, Hx, etc.), the data was likely obtained from a rotational rheometer.

Finally, modern rotational rheometers will commonly manipulate the directly measured torque and angular rate of rotation data for the engineer, returning the raw data along with shear stress or viscosity vs. shear rate data. Note that Fathom requires the user to define shear stress vs. shear rate data when entering rheological test data. If the user only has viscosity vs. shear rate data, they should curve fit the constants for the desired viscosity model manually.

Converting Between Types of Rheological Data

Although two types of rheological data exist, most viscosity models expect rotational rheometer data. Fathom employs the Rabinowitsch-Mooney technique to convert from tube flow rheometer data to rotational rheometer data when necessary.

The Rabinowitsch-Mooney technique can be applied to obtain rotational rheometer data from tube flow rheometer data as follows:

1. Plot the tube flow rheometer data on a log-log scale.
2. Determine the slope of the data. The slope is represented by *n'*.
3. Multiply the tube flow rheometer data by the value:

$$\frac{3n' + 1}{4n'}$$

Fathom now has rheological data in the form of rotational rheometer (rheogram) data, which can be used to directly fit the viscosity model constants.

Related Topics

Heat Transfer/Variable Fluids Panel

The Heat Transfer/Variable Fluids panel, shown in Figure 1, allows users to define how the fluid properties will vary. The fluid selected on the Fluid panel must have data entered for the heat transfer parameters, such as specific heat, for the Heat Transfer options to be available, as is discussed further in the Custom Fluids topic.

Note: Heat transfer options may be disabled for several reasons:
1. Only Newtonian fluids can be modeled when heat transfer is enabled.
2. User Specified Fluids only contain data for density and viscosity, and thus heat transfer can't be modeled.
3. User Library Fluids can only model heat transfer if they contain data for specific heat, thermal conductivity, and a reference temperature/enthalpy point.

- **Constant Fluid Properties** - All fluid properties are the same for every pipe and are as specified on the Fluid panel. Note that non-newtonian viscosity models will still cause the viscosity to vary if the constant fluid properties option is selected.

- **Variable Fluid Properties** - The values for density and viscosity on the Fluid panel are the default fluid properties. You can enter different fluid property values, if desired, for any pipe in the Pipe Properties window. This does not mean that fluid properties can vary on their own in the model - they are constant in each pipe and must still be directly specified by the user.

  This option may be useful to model different pipes carrying different fluids, or to model changes in temperature across a component. Fathom cannot perform dynamic mixing; however, fluid properties of pure components may be individually specified in upstream pipes, and the resulting mixture properties can be individually specified in downstream pipes.

- **Heat Transfer With Energy Balance (Single Fluid)** - All fluid properties are determined dynamically during the run based on the most recent solution for temperature and pressure - thus they are inherently variable. Only one fluid is used in the model. Additional inputs will be required in the pipes and junctions which are specified to model heat transfer. If a User Library Fluid is utilized in the model, it will need to have data for specific heat, thermal conductivity, and a reference temperature/enthalpy point to allow heat transfer.

- **Heat Transfer With Energy Balance (Multiple Fluid)** - All fluid properties are determined dynamically during the run based on the most recent solution for temperature and pressure - thus they are inherently variable. Piping loops with different fluids can be modeled using Fluid Groups to input multiple fluids into the model, such as to represent different sides of a heat exchanger. However, dynamic mixing of fluids is not supported. Additional inputs will be required in the pipes and junctions which are specified to model heat transfer.

Always Use Constant Density - Only applicable to *Variable Fluid Properties* and *Heat Transfer With Energy Balance (Single Fluid)*. This option forces the fluid density to remain constant throughout the model. This can be convenient when density variations due to temperature are minimal but viscosity changes are significant. Enabling this option has only minimal impact on results in these cases, and can greatly simplify and clarify the head and gradeline values which would otherwise change by some minimal amount.
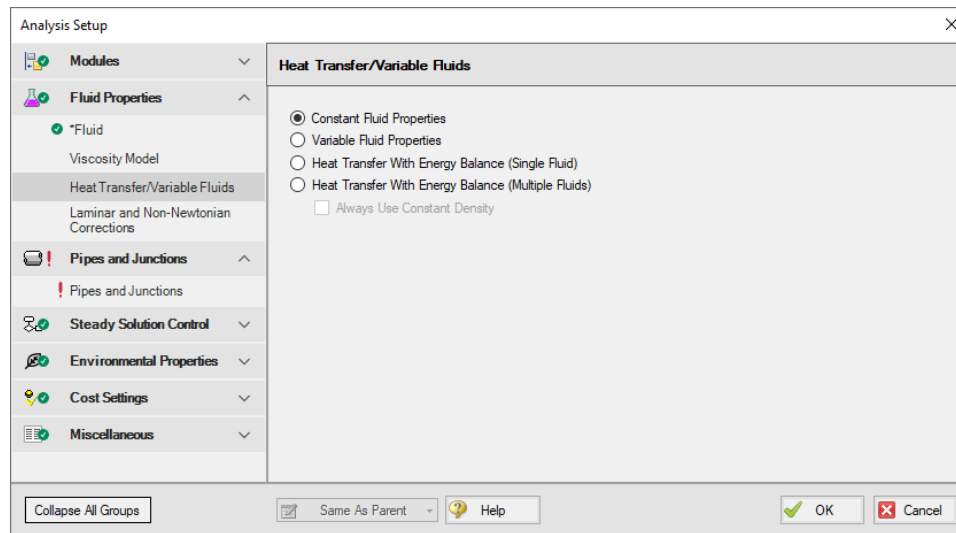
Figure 1: Heat Transfer/Variable Fluids panel

Related Topics

Modeling Heat Transfer

AFT Fathom can model heat transfer simultaneously with the hydraulic calculations. Since many pipe flow design calculations neglect heat transfer, this capability is turned off by default. Hence temperature and heat loss information is not required to specify a model.

Heat transfer modeling is enabled on the Heat Transfer/Variable Fluids panel by selecting one of the two Heat Transfer With Energy Balance options. Once selected, temperature input is required at boundary junctions such as reservoirs. By default pipes are adiabatic, but different heat transfer models can be selected such as Convective Heat Transfer and Constant Heat Flux. With Convective Heat Transfer pipes can be assigned insulation and convection coefficient data to calculate heat loss or gain.

In addition, heat exchangers provide input for heat transfer with eleven heat exchanger thermal models available. Thermal linking with other heat exchangers is also provided to model cases such as heat transfer between both sides of a heat exchanger.

Finally, heat addition at pumps can also be modeled.

As heat transfer takes place during the system hydraulic solution, physical properties (such as viscosity) are dynamically updated based on resulting temperatures and a system energy balance is performed as part of the simulation.

Related Topics

Fluid Groups

The purpose of fluid groups is to allow heat transfer calculations in different loops which have different fluids. When enabled, a new menu item called Fluids appears on the Edit menu Groups fly-out menu. Here is how this feature is enabled and used:

1. On the Fluid panel, select a base fluid
2. On the Heat Transfer/Variable Fluids panel, select Heat Transfer With Energy Balance (Multiple Fluids)
3. Select each pipe in the fluid loop on the Workspace and go to the Edit menu, Groups option, then Create to give the group a name. At this point the pipes and junctions are standard AFT Fathom groups.
4. Go to the Heat Transfer/Variable Fluids panel
5. Check the Use As Fluid Group box for the desired fluid group
6. Specify what fluid is to be used in each fluid group.
7. Click OK

Now when you run the model, each of the pipes and junctions in each group will take on the fluid properties assigned to that fluid group.

Note: If you are using a custom fluid in your model and make changes to it in the fluid library, you will need to save and re-open the model for the changes to be applied to any fluid in a Fluid Group.

Here are some guidelines for use:

- A pipe or junction can belong to one and only one fluid group
- Every pipe and junction must belong to a fluid group

Use with Heat Exchangers

Fluid groups allow heat exchangers to be modeled with thermal links to other heat exchangers carrying different fluids.

Thermal Links

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 specified in a heat exchanger properties window, the user must specify the heat transfer area and overall convection coefficient. AFT Fathom will force the two heat exchanger junctions to use the same value.

Related Topics

Laminar and Non-Newtonian Corrections Panel

AFT Fathom uses the methods and correlations discussed for each viscosity model only to model frictional losses for pipes. Frictional losses due to pipe fittings or junction losses are modeled using the methods discussed on this page regardless of viscosity model.

It is generally believed that the pressure drop through valves, fittings, etc derive from the physics of bulk fluid motion, wall separation, turbulence, etc and hence are not affected by Non-Newtonian fluid behavior in turbulent flow. Thus, pressure loss due to pipe fittings or junction-specific losses during turbulent flow are modeled using the same methods employed for Newtonian flow.

During laminar flow, pipe fittings and junction losses may not be similar for both Newtonian and Non-Newtonian fluids. AFT Fathom offers two options to model these losses during laminar flow. The correction options are:

- Use Adjusted Turbulent K Factor Method (ATKF) - derived from the Crane hydraulic handbook (Crane 1988).
    - Also Apply in Turbulent Region
    - Use Improved Method
- Use Non-Newtonian to Newtonian Pipe Friction Ratio Multiplier

Modifying Losses for Fittings and Junctions Interface

The options listed above to modify losses at fittings and junctions can be found on the Laminar and Non-Newtonian Corrections panel in the Fluids Properties group of the Analysis Setup, as shown in Figure 1 below.
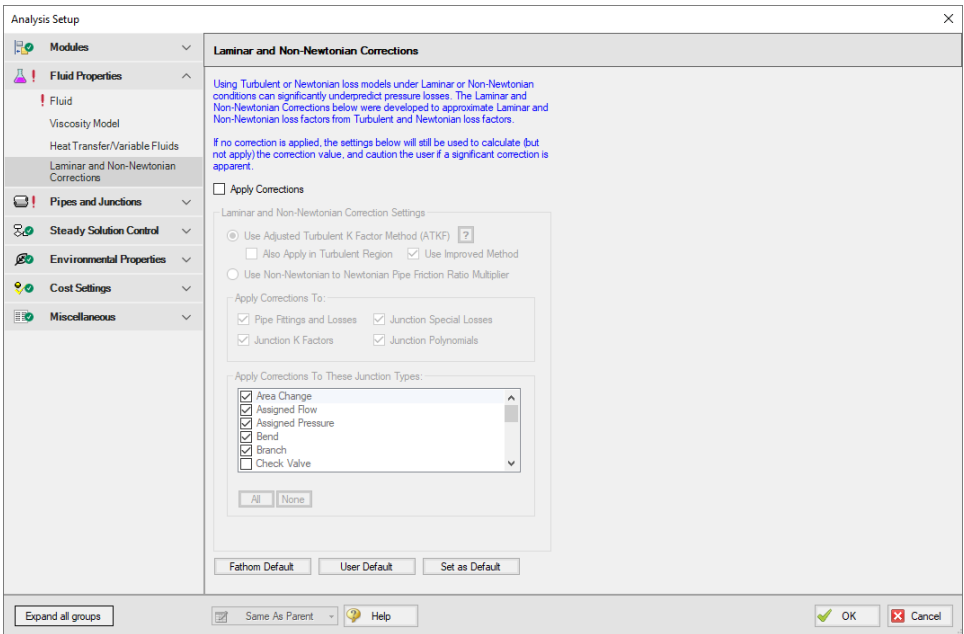
Figure 1: Options to modify fitting and junction losses on the Laminar and Non-Newtonian Corrections panel

These corrections are off by default and "Apply Corrections" must be selected to access them. Users can select which correction method is applied and can select which junction types the loss model will be applied to. Fathom Default can be selected at any time to reset the selected junction types.

Non-Newtonian to Newtonian Pipe Friction Ratio Multiplier

One approach to modeling losses associated with fittings and junction losses is to assume that the ratio of those losses in a Non-Newtonian and Newtonian fluid is equivalent to the ratio of pipe friction in a Non-Newtonian and Newtonian fluid. One can write:

$$K_{Non-Newtonian} = K_{Newtonian} \frac{f_{Non-Newtonian}}{f_{Newtonian}}$$

This approach uses the pipe friction at the adjacent upstream pipe and the uncorrected loss factor in a Newtonian fluid to model the losses associated with the fitting or junction for a Non-Newtonian fluid.

Related Topics

Adjusted Turbulent K Factor Method (ATKF)

Crane (1988) includes an equivalence between equivalent length methods and K-factor on the nomograph chart on page A-30. This nomograph is based on theory as discussed below.

The basic law of pressure drop for friction factor is

$$dP = f\frac{L}{D}\left(\frac{\rho V^2}{2}\right) \tag{1}$$

For K-factors the pressure drop is given by

$$dP = K\left(\frac{\rho V^2}{2}\right) \tag{2}$$

From equations 1 and 2 it can be seen that there is an equivalence between *f*L/D* and *K*.

The equivalent length method works by adding to the physical length as follows:

$$dP = f\frac{L + L_{eq}}{D}\left(\frac{\rho V^2}{2}\right) \tag{3}$$

Hence the pressure drop due to the methods can be related as follows:

$$K = f\frac{L_{eq}}{D} \tag{4}$$

An appropriate equivalent length can be determined from standard AFT Fathom K-factor data by looking at large Reynolds number and hence fully turbulent conditions

$$L_{eq,turb} = K_{turb}\frac{D}{f_{turb}} \tag{5}$$

And finally a general K-factor which is based on equivalent length can be found in the following:

$$K = K_{turb} \frac{f}{f_{turb}}$$

(6)

To apply the method, take the AFT Fathom K-factors (K_{turb}) and multiply them by the relevant pipe friction factor, *f*, at actual conditions (and Reynolds number) and divide them by the turbulent friction factor, f_{turb}, evaluated at very large Reynolds number.

Improved ATKF Method

For the Improved ATKF method, the turbulent friction factor in equation 6 is is replaced with the turbulent friction factor calculated for commercial steel at very large Reynolds numbers,which is about 0.023 for a roughness of 0.0018 in. This allows the adjusted K factor to be independent of pipe roughness in the laminar flow region so that it more closely mimics the expected behavior for the 3K method from Darby and the equivalent length method.

Comparison of Equation 6 to Published Data

A study was performed to evaluate how Equation 6 compares to published data (Wood 2012). A comparison for a 10 foot long, 2 inch in diameter pipe flowing water was performed with the following:

1. No fittings in the pipe
2. Fittings of 12 elbows of 1.5 *r/D* with a total *K* of 3.17 but no corrections (this is the way standard AFT Fathom works)
3. Darby 3-K method for 12 elbows (see Darby, 2001, pp.209)
4. Equivalent length of 12 elbows with L_{eq}/D = 16 (see Darby, 2001, pp.209)
5. Use Equivalent Length in Pipe Diameters (*L/D*) Derived From K-Factors, Eq. 6

Figure 1 shows the results on a log-log plot. One would not expect the first case to match the others as no fittings were included. Hence the head gradient was less than the other cases. However, Case 1 does illustrate how Case 2 works in the standard AFT Fathom. Case 2 agrees with Cases 3-5 at high Reynolds Number but agrees with Case 1 at low Reynolds number. This is a result of using turbulent, high Reynolds Number K-factors for low Reynolds Number applications.

Cases 3, 4 and 5 all agree well throughout the entire range of Reynolds Numbers. There is some small discrepancy at high Reynolds Number which is merely a result of various reference differences over the exact pressure loss through a 1.5 *r/D* elbow.

The conclusion from Figure 1 is that the Equation 6 method allows reliable adjustment of turbulent pressure loss data for fittings and junctions to laminar applications.
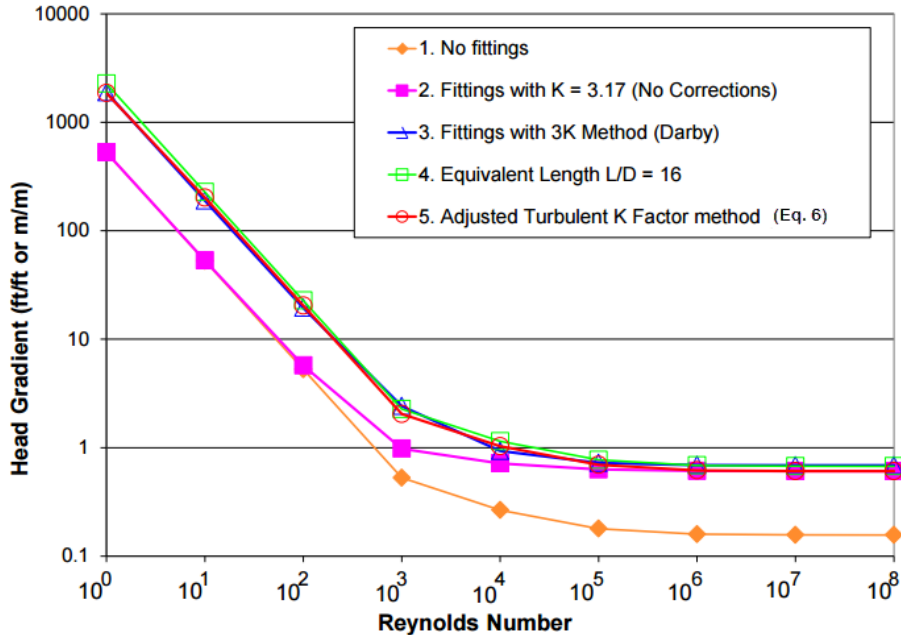
Figure 1: Log-log plot of dimensionless head gradient vs. Reynolds number for various fitting loss models

Related Topics

Improved ATKF Method

For commercial steel pipe and pipes with a similar roughness, it has been shown that the equivalent length, Darby 3-K, and Adjusted Turbulent K Factor (ATKF) methods will all produce similar results in the laminar flow region (see Adjusted Turbulent K Factor (ATKF) Method).

However, this is not the case when the pipe roughness varies significantly from that of commercial steel. At different roughness values the predicted head loss with the original ATKF method varies inversely with pipe roughness, so that smoother pipes will experience higher head loss in the laminar regime as is shown in Figure 1. The Darby 3-K method shows no dependence on pipe roughness in the laminar regime, as is shown in Figure 2. It would be expected that head loss in the laminar regime would behave as predicted by the 3-K method, based on the fact that the laminar friction factor is independent of pipe roughness.
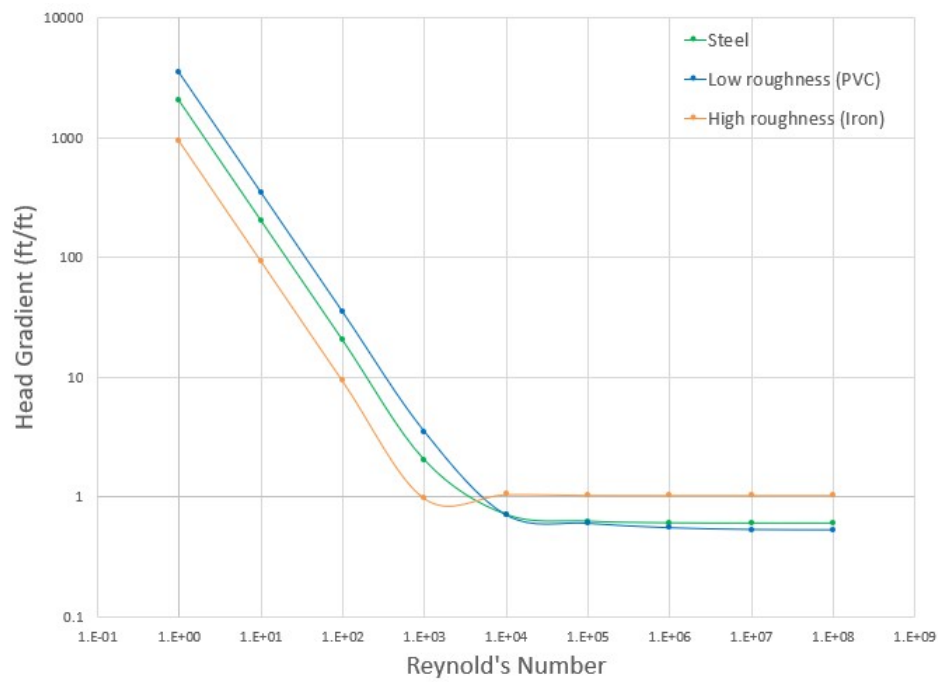
Figure 1: Head Gradient calculated with the original ATKF method at different roughness values, showing differences in the laminar region
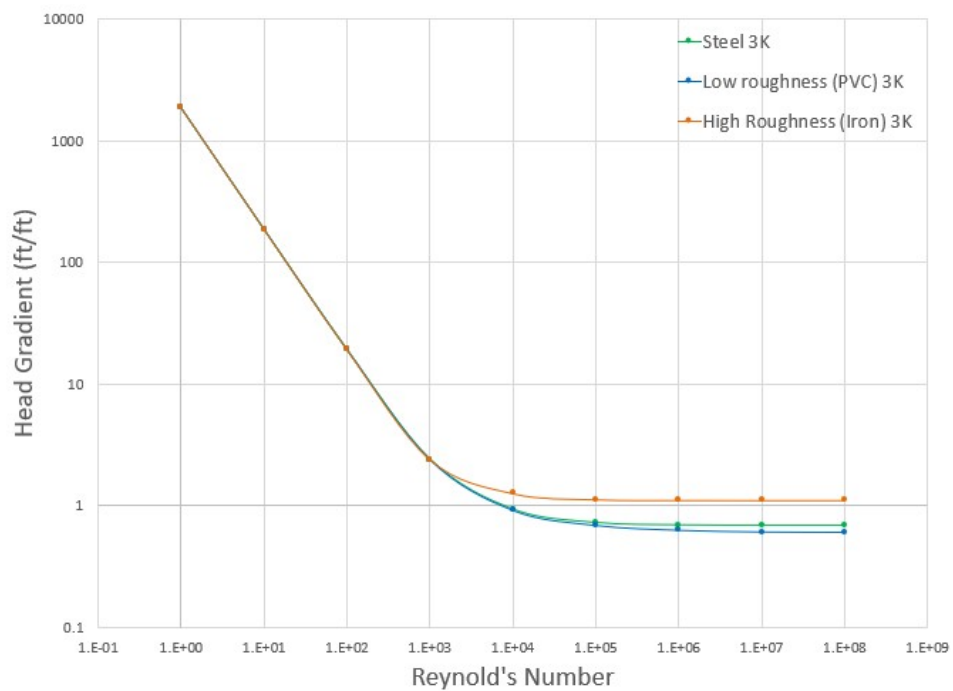
Figure 2: Head Gradient calculated with the 3-K method at different roughness values, showing no differences in the laminar region

To remove the dependence on pipe roughness in the ATKF method, the Improved ATKF method uses a fixed turbulent friction factor value calculated for commercial steel pipe (roughness 0.0018 in) at very large Reynolds numbers. The Improved ATKF method can be written as follows, replacing f_{turb} with the reference turbulent friction factor, $f_{turb,ref}$:

$$K = K_{turb} \frac{f}{f_{turb,ref}}$$

With the Improved ATKF factor it can be seen that the head loss is no longer dependent on pipe roughness (Figure 3).
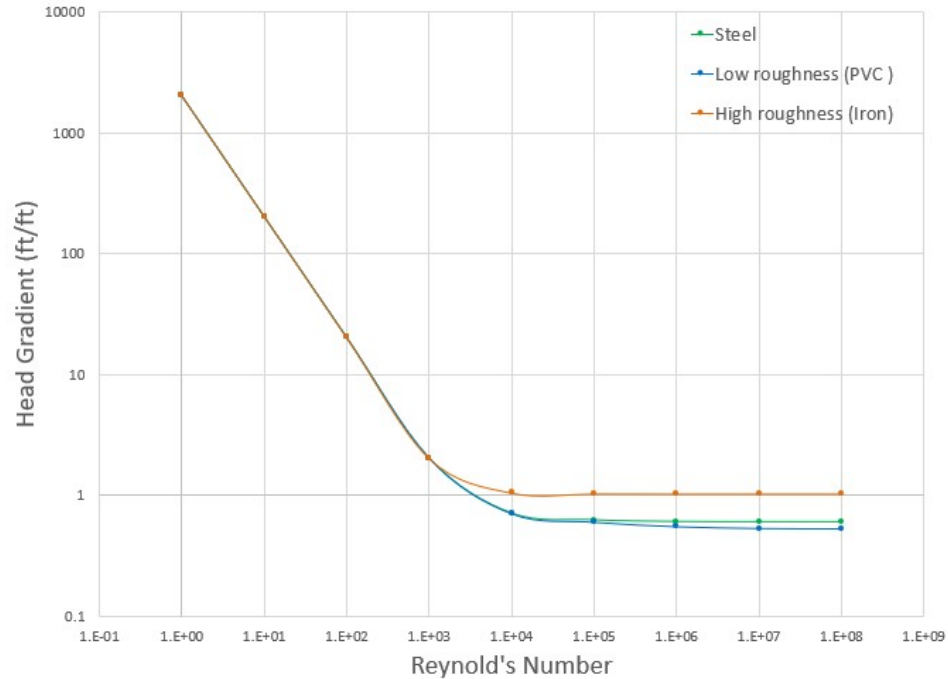
Figure 3: Head gradient calculated with the Improved ATKF method at different roughness values

Related Topics

Note about Laminar Corrections for Fittings

There is ambiguity in the literature about how to calculate pressure loss for fittings with laminar flow. Here is a quick summary of different views:

No Laminar Correction Required

Crane (1988) suggests no correction is required. On pages 2-8 through 2-11 Crane discusses this. Worked examples 4-7, 4-8 and 4-9 use this assumption.

This assumption is the same as method 2 in the ATKF comparison section and Figure 1 (in the aforementioned topic), and is consistent with how AFT Fathom has always performed laminar calculations. This assumption is available by merely not applying the laminar methods discussed in this chapter.

Laminar Correction is Required

Publications such as Darby (2001) discuss the necessity for laminar corrections for fitting K-factors. On pages 209-214 "2-K" and "3-K" methods are discussed that purport to cover the entire range of Reynolds number.

Based on Equation 6 and Crane's (1988) nomograph of page A-30 a laminar correction based on equivalent length methods can be applied. This gives similar pressure drop results as the 3-K method and is available through the Adjusted Turbulent K Factor method.

Low Reynolds Number Flow

In addition to the discussion above, there is a separate debate over whether flow at very low Reynolds numbers (less than 100) should be treated separately when determining if corrections are needed for fittings and losses. There are conflicting CFD studies and lab tests showing that corrections are needed, or are not needed. For additional details please see the below journal articles:

- **No corrections needed at low Re**
  - Liou, R. J., et.al, "Numerical Analysis of Steady Flow in a 90-deg Circular Section Bend", J. Eng. Mech., v. 114 (1988), p. 1871.
  - Berger, S. A., et.al, "Flow in Curved Pipes", Ann. Rev. Fluid Mech., v. 15 (1983), p. 461.
- **Corrections needed at low Re**
  - Edwards, M. F., et.al, "Head Losses in Pipe Fittings at Low Reynolds Numbers", Chem. Eng. Res. Des., v. 63, (Jan 1985), p. 43.
  - Kittredge, C. P. and Rowley, D. S., "Resistance Coefficients for Laminar and Tur-

bulent Flow Through One-Half-Inch Valves and Fittings", Trans. ASME, v. 79, (Nov. 1957), p. 1759.

Related Topics

SSL Module Fluid Properties Group

(SSL Module Only) When the Settling Slurry (SSL) Module is activated on the Modules panel, the items in the Fluid Properties Group change to allow slurry physical property calculation.

Engineers who are new to slurry calculations may not recognize some of the properties in the following panels. These properties all relate to the theory of slurry calculations, which is discussed in the Slurry Calculation Methods section.

- Carrier Fluid panel - Define the Carrier Fluid Properties
- Solids Definition panel - Define the Slurry Calculation Method and Solids Specifications such as M (Stratification Ratio Exponent), Terminal Velocity Parameter, density, d_{50}, and d_{85}
- Slurry Definition panel - Define the Slurry Model and Slurry Properties

Related Topics

Carrier Fluid Panel

(SSL Module Only) On the Carrier Fluid panel, users select and define the carrier fluid for the slurry. The available options are:

- Basic (Water)
- Advanced

Basic (Water) Carrier Fluid

The Basic (Water) option uses the Water (liquid) fluid from the AFT Standard fluid library as the carrier fluid and treats all pipes as having constant slurry properties (Figure 1).

Specify the water temperature for the slurry property calculation.
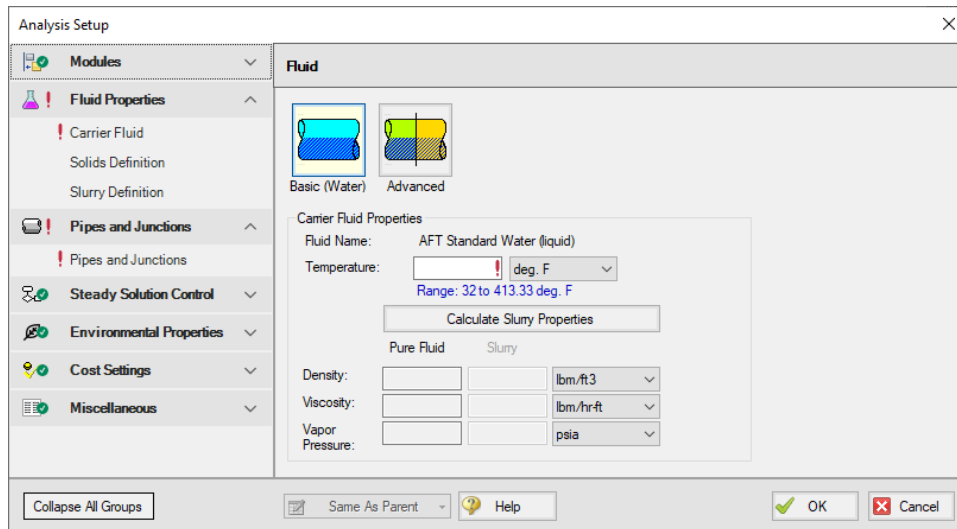
Figure 1: Carrier Fluid panel with the Basic (Water) carrier fluid option selected

Advanced Carrier Fluid

The Advanced option allows the user to specify any fluid as the carrier fluid, and will allow the slurry properties to vary among pipes.

The Carrier Fluid panel looks very similar to the Fluid panel when the SSL module is not activated. Specify the fluid, its temperature (and pressure in some cases), and fluid property variation on this panel. Users can access data from all fluid libraries. The User Specified Fluid option is disabled because it is not saved in a library. Custom fluids should be saved to a library and selected using the User Library Fluid option.
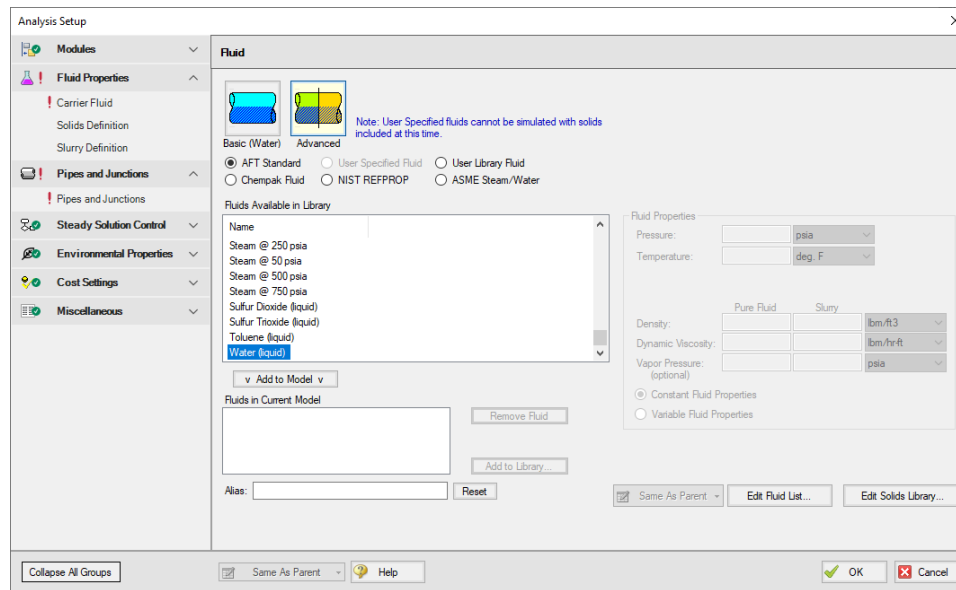
Figure 2: Carrier Fluid panel with the Advanced option selected

Related Topics

Solids Definition Panel

(SSL Module Only) On the Solids Definition panel, users define the Slurry Calculation Method and Solids Specifications. These inputs are the same for both the Basic (Water) and Advanced carrier fluid options.

Override options are provided for a number of parameters that allows users to modify data pulled from libraries should modeling requirements need this.
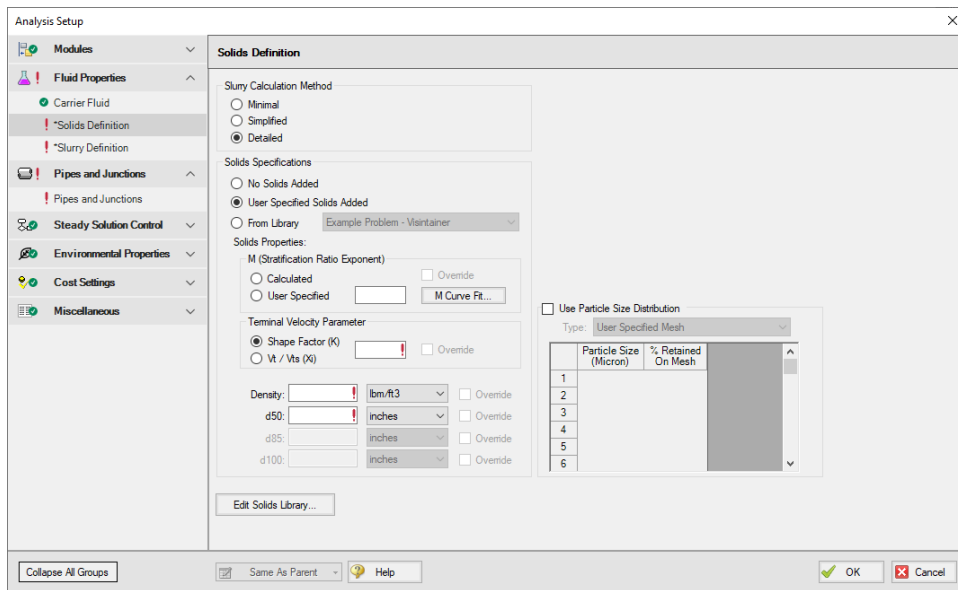
Figure 1: Solids Definition panel when Detailed Slurry Calculation Method and Calculated M are selected

Slurry Calculation Method

There are three calculation methods available to model slurries when using the Wilson, Addie, Clift slurry model. These are all variations of the Wilson, Addie, Clift slurry approach, and this selection is only applicable when Settling Slurry - Wilson, Addie, Clift is chosen as the active Slurry Model in the Slurry Definition panel. Each method requires different amounts of input data resulting in the use of different calculations. These calculations are described in the Slurry Calculation Methods section.

- Minimal Method - Used in the preliminary design phase where only rough data are available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data the calculations should only be used for early stage approximations because it often over-predicts losses. This is a conservative method that assumes a fully-stratified sliding bed.

- Simplified Method - Additional information is required and gives a better solution. This method requires the solids density, d50 particle size, a user-specified *M* exponent, and the solids concentration. This method assumes heterogeneous flow with suspended particles. This is a more accurate model than the Minimal method because it accounts for more slurry definition.

- Detailed Method - The default and provides the most rigorous solution methodology. The solids density, d50 particle size, terminal velocity parameter, and the solids concentration are required. Additionally, if the *M* exponent is to be calculated, the d85 particle size is also required. As with the Simplified Method, this method also assumes heterogeneous flow with suspended particles.

Solids Specifications

In the Solids Specification area, select from the following options for solids:

- No Solids Added - The model will run using the carrier fluid without any solids, no further input is required.

> **Note:** The No Solids Added option is only compatible with the Newtonian Slurry Model option on the Slurry Definition panel. The other Slurry Models imply that solids are present in the fluid, therefore, the Slurry Definition panel will be undefined until the Solids Specifications is changed to one of the options below.

- User Specified Solids Added - Define the solids using the Solid Properties fields.
- From Library - Use solids already defined in the Solids Library. Note no default solids library is provided by AFT, library items must be defined first in the Library Manager to be available here.

Solids Properties

By selecting User Specified Solids Added, the Solids Properties fields become available to define the solids. Depending on the Slurry Calculation Method selected at the top of the Solids Definition panel, certain fields are required:

- Minimal
  - Density
- Simplified
  - User Specified M (Stratification Ratio Exponent)
  - Density
  - d_{50}
- Detailed
  - M (Stratification Ratio Exponent)
  - Terminal Velocity Parameter
  - Density
  - d_{50}
  - d_{85} - if Calculated M (Stratification Ratio Exponent) is selected

The **M (Stratification Ratio Exponent)** is a parameter in the stratification ratio equation. The user can allow AFT Fathom to calculate M based on best available methods, or input M directly. The process of estimating M from recorded data is discussed elsewhere.

The **Terminal Velocity Parameter** must be entered when using the Detailed calculation method. This can be entered as either a Shape Factor (K) or the ratio of the particle's terminal velocity (V_t) to that of a sphere (V_{ts}), designated as V_t / V_{ts} (X_i). If a Shape Factor is entered, it is used in conjunction with the graph shown in Wilson, et al. 2006, on page 47, to obtain a value for Xi. Typical shape factors are given in Wilson, et al, 2006, on page 46 where K = 0.26 for sand.

In the lowest area, enter the required values for mean density of the solid particles, d_{50} (mass-median particle diameter) and d_{85} (diameter for which 85% by mass of the particles are finer).

4-Component Slurry Model

The 4-Component Slurry Model requires a d100 and Particle Size Distribution.

Related Topics

Slurry Definition Panel

(SSL Module Only) The Slurry Definition panel allows selection of the Slurry Model and related options such as the Slurry Calculation Method, as well as input of solids concentration relative to the carrier fluid and estimation of bulk slurry properties, as shown in Figure 1.

Figure 1: Slurry Definition panel when Detailed Slurry Calculation Method and User Specified Solids Added are selected

Slurry Calculation Method

There are three calculation methods available to model slurries when using the Wilson, Addie, Clift slurry model. These are all variations of the Wilson, Addie, Clift slurry approach, and this selection is only applicable when Settling Slurry - Wilson, Addie, Clift is chosen as the active Slurry Model in the Slurry Definition panel. Each method requires different amounts of input data resulting in the use of different calculations. These calculations are described in the Slurry Calculation Methods section.

- Minimal Method - Used in the preliminary design phase where only rough data are available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data the calculations should only be used for early stage approximations because it often over-predicts losses. This is a conservative method that assumes a fully-stratified sliding bed.

- Simplified Method - Additional information is required and gives a better solution. This method requires the solids density, d50 particle size, a user-specified *M* exponent, and the solids concentration. This method assumes heterogeneous flow with suspended particles. This is a more accurate model than the Minimal method because it accounts for more slurry definition.

- Detailed Method - The default and provides the most rigorous solution methodology. The solids density, d50 particle size, terminal velocity parameter, and the solids concentration are required. Additionally, if the *M* exponent is to be calculated, the d85 particle size is also required. As with the Simplified Method, this method also assumes heterogeneous flow with suspended particles.

Slurry Definition

The **Concentration Type** is used to specify a value in the **Amount Solids Added** field. The available options for Concentration Type are:

- Grams per Liter of Mixture
- Volume Fraction
- Mass Fraction
- Solids Ratio (M-solids/M-liquid)

There are a few **Slurry Model** options to choose from. These options determine the calculations used in the Solver.

- **Newtonian** - Functionally disables most of the SSL module calculations and preserves only the viscosity properties of the fluid in the model.

> **Note:** The No Solids Added option from the Solids Definition panel is only compatible with the Newtonian Slurry Model option. If you have already specified solids on the Solids Definition panel, you must select one of the other Slurry Models below.

- **Settling Slurry - Wilson, Addie, Clift** - Default option, applies slurry calculations from the Wilson, Addie, Clift Slurry Models.
- **Settling Slurry - 4-Component Model** - The 4-Component Slurry Model developed by GIW Industries Inc. - A more complex model that takes into consideration the contribution of different size particles in the slurry flow.
- **Wasp-Durand Heterogeneous Model** - The Wasp-Durand Slurry Model is a two-layer slurry approach with heterogeneous and homogenous fluid components.

The **Sliding Friction Coefficient (μs)** is the coefficient of mechanical sliding friction between the granular mass and the boundary - where the boundary is where the motion takes place between the mass and a rigid boundary, i.e. the pipe wall. As the Sliding Friction Coefficient increases, so does the friction and thereby the pressure drop through the pipe. Typical values range from 0.4 to 0.5. This field is only available when the Wilson, Addie, Clift Minimal Slurry Calculation Method or the 4-Component slurry model are selected.

The **Bed Concentration (C$_{vb}$)** is an optional parameter for the Wilson, Addie, Clift slurry methods and is only used to determine the actual settling velocity (V_s) shown in the output. These calculations are described in Wilson (1986) and are functions of the pipe diameter, particle size (d_{50}), particle density and relative concentration. If no value is entered for this parameter then a default of 0.60 is used. The maximum value of these settling velocities (V_s) is denoted V_{sm} and is the usual parameter of interest in designing a system to avoid operating with a stationary bed. This field is only available when the Simplified and Detailed Slurry Calculation Methods are specified at the top of the panel.

Deposition Velocity (Vsm)

The Deposition Velocity (V_{sm}) can be determined using one of the options below. This is useful when the calculation method yields a slightly different value than, say, test results.

- **Nomograph - Equation Limited** (Wilson, Addie, Clift only) - This option is the default and uses the nomograph from Wilson (1986) page 111. The nomograph is noted to be conservatively high by Wilson, so an equation that calculates the max Vsm is used to limit the value of Vsm.

  The results calculated from the nomograph will be checked against Equation 5.11 from Wilson (1986) as is shown below. If the value from the nomograph exceeds $V_{sm,max}$ calculated from Equation 5.11, then the $V_{sm,max}$ from Equation 5.11 will be used instead.

$$\frac{V_{sm,max}}{\sqrt{2gD(S_m - S_f)}} = \left(\frac{0.018}{f}\right)^{0.13}$$

Note: If the V_{sm} is calculated as a negative value using Equation 5.11 from Wilson then the V_{sm} will be displayed as 0 in the Output. If the V_{sm} is shown as 0, the inputs for the fluid and solid should be verified, and the user should consider calculating the V_{sm} using a separate method.

- **Equation Only** (Wilson, Addie, Clift only) - This option will cause the V_{sm} to be calculated solely using Equation 5.11.
- **User Specified** (Wilson, Addie, Clift only) - This option allows the user to directly specify V_{sm}.

   Note: As Wilson describes there are times where operating at a velocity lower than V_{sm} is economically attractive and is without a stationary bed. These cases include a system where the solid concentration at the desired operating point is very different than that at the V_{sm} conditions. Wilson points out that this is rarely the case, and so the use of V_s instead of V_{sm} should be done with caution.

- **Nomograph Only** (4-Component only) - This option will cause deposition velocities to be calculated from the Wilson, Addie, Clift nomograph, similar to the Nomograph - Equation Limited Wilson, Addie, Clift option without the limiting equation. The other notable difference is that two deposition velocities are calculated for the 4-Component method from two different diameters; $V_{sm,h}$ from d_{50h} and $V_{sm,s}$ from d_s.
- **Match Heterogeneous Calculation** (Wasp-Durand only) - The calculation method for Wasp-Durand deposition velocities is coupled to the selection of the Heterogeneous Gradient Calculation Method. The following methods are used, which are described in more detail in [Wasp-Durand Heterogeneous Slurry Model](#):
  - Wasp-Durand
  - Liu Dezhong
  - Fei Xiangjun

Wasp-Durand Options

When using the Wasp-Durand Slurry model, the Wasp-Durand Options section will be displayed within the Slurry Definition panel. Equations associated with these options may be found in the [Wasp-Durand Heterogeneous Slurry Model](#) section.

The **Sedimentation Velocity Model** determines which model is used to calculate sedimentation velocity, which is calculated per particle size group and is a particle settling velocity. This is the terminal particle freefall velocity that occurs when the drag, gravity, and buoyancy forces on a particle are balanced. There are two options available. Either option is appropriate for a wide range of flow states and particle sizes, including laminar, transitional, and turbulent flow, so the selection of model will depend on personal preference or industry standards:

- Liu Dezhong
- Ruby & Zanke (1977)

The **Heterogeneous Gradient Calculation Method** determines the set of equations to use for calculating the heterogeneous component of the slurry pressure gradient, along with the depos-

ition velocity, as these options are always coupled for Wasp-Durand. Three options are available:

- Wasp-Durand
- Liu Dezhong
- Fei Xiangjun

The **Apply Apparent Slurry Properties** option allows the viscosity of the homogeneous (vehicle) component to be affected by solids, or specified by the user manually. Without this option enabled, the clear fluid carrier fluid viscosity will be used. The following viscosity options are available when applying apparent slurry properties:

- **User Specified Viscosity** - The viscosity used for the homogeneous pressure gradient is specified by the user rather than being inherited from the carrier fluid or calculated from the carrier fluid and particle size distribution. This option treats the homogeneous component as Newtonian.
- **Apply Viscosity Model** - These options calculate the carrier fluid viscosity including the contribution of solids. Solids of all sizes will contribute to the homogenous viscosity, but finer particles contribute more significantly.
    - **Thomas (1965)** - This can be used to calculate a Newtonian viscosity value or Bingham stiffness factor for each pipe.
    - **Fei Xiangjun - Mineral** - This method calculates a Newtonian viscosity or Bingham stiffness factor for each pipe, and can also calculate a global yield stress value for the slurry.
    - **Fei Xiangjun - Coal** - This method is mathematically identical to the Fei Xiangjun - Mineral viscosity models, but the empirical values are tuned to coal slurry data rather than mineral slurries.

When active, the **Use Hanks' Bingham Plastic Friction Model** checkbox replaces the Newtonian homogeneous component friction calculations with a non-Newtonian Bingham plastic approximation. With this option, yield stress may be manually specified or calculated using the Fei Xiangjun methods. The Bingham stiffness factor is calculated iteratively for each pipe using the Fei Xiangjun or Thomas (1965) methods.

Related Topics

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 junc-

tions 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

Pipes and Junctions Panel

The Pipes and Junctions panel within Analysis Setup displays the status of pipes and junctions within the model.
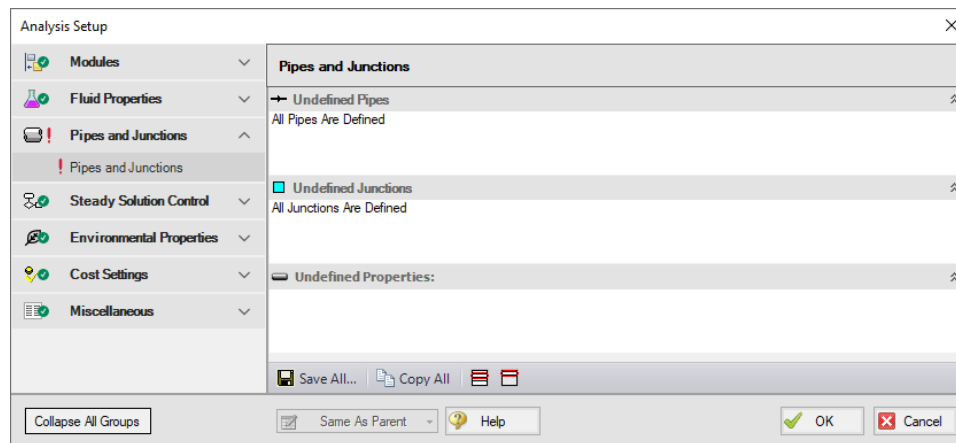
Figure 1: Default state of the Pipes and Junctions panel

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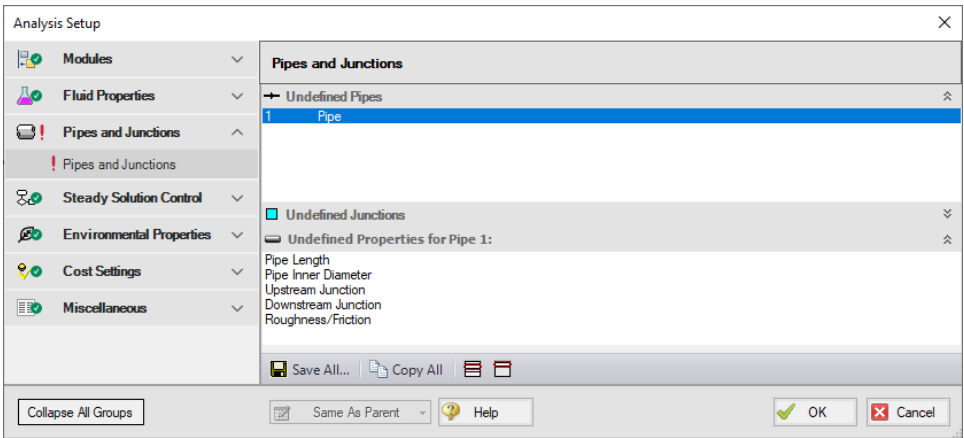
Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed

Related Topics

Transient Control Group

The Transient Control Group in Analysis Setup is where settings pertaining to a transient simulation using the XTS Module are managed. 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 on the Simulation Duration panel are defined. The model cannot run until the Transient Control Group has a green checkmark.

Related Topics

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.
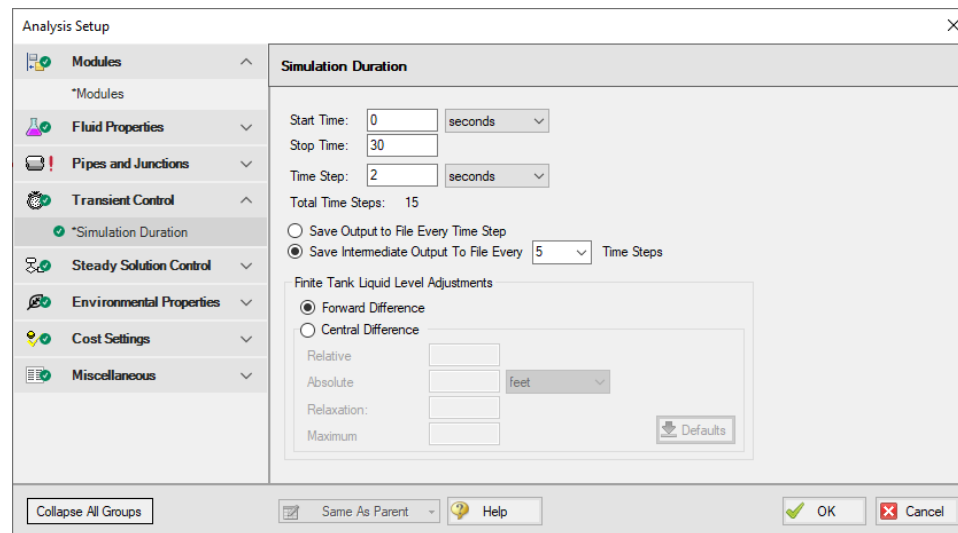
Figure 1: Simulation Duration panel fully defined with example values

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 transient simulation, the size of the time step, and the frequency of the data saved to the transient output file.

Also available is [Forward Difference and Central Difference](#) options for Finite Tank Liquid Level Adjustments. If finite tank reservoirs are not used in the model, then there is no difference between the two options. However, there is an impact when one or more [finite tanks](#) do exist in the model.

Related Topics

Finite Tank Liquid Level Adjustments

(XTS Module Only) The Simulation Duration panel allows one to set forward and central difference options. The forward and central difference options are only relevant when finite tanks are modeled. They relate to what liquid levels are used in the tank for each time step calculation. If the assumed time step is small enough, then the two methods yield virtually the same results. However, as the time step is increased, the central difference method is much more accurate. The drawback is that the central difference method is iterative and takes more computation time.

Forward Difference Method

The forward difference method solves the pipe system as if all tank levels remain at the same value as the beginning of the time step. Since the liquid level is known at the beginning of the time step, no time-based iterations are required. (The iterations for flow and pressure common to steady-state modeling still occur).

Once the solution is complete for the time step, the solved flowrates are used to update the tank liquid level for the next time step. Figure 1 depicts how this works.

Since in reality the tank level changes during the time step, precision is lost by using the liquid level at the beginning of the time step. As discussed earlier, making the time steps shorter helps alleviate this problem.
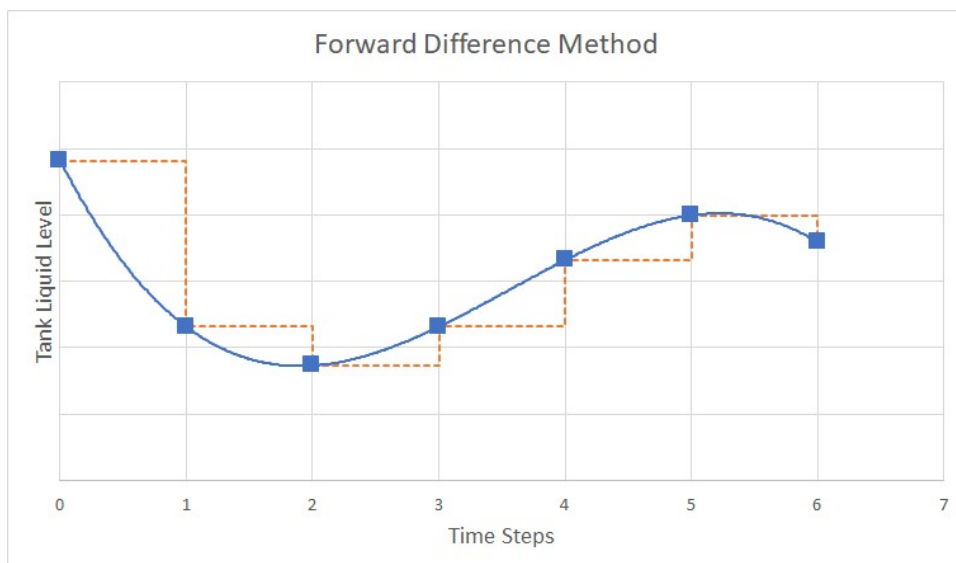
Figure 1: Forward difference method for time calculation assumes liquid level stays constant during each time step, allowing a direct calculation for each time

Central Difference Method

The central difference method solves the pipe system using tank liquid levels which represent the average over the time step (see Figure 2). Since the liquid level changes during the time

step, it is more accurate to use the average over the time step. However, the difficulty is that the average is taken between the liquid level at the beginning and end of the time step and, while the liquid level is known at the beginning, it is not known at the end. That is what we are trying to solve for. Hence the transient solution must assume a liquid level at the end of the time step and iterate on that value until convergence.
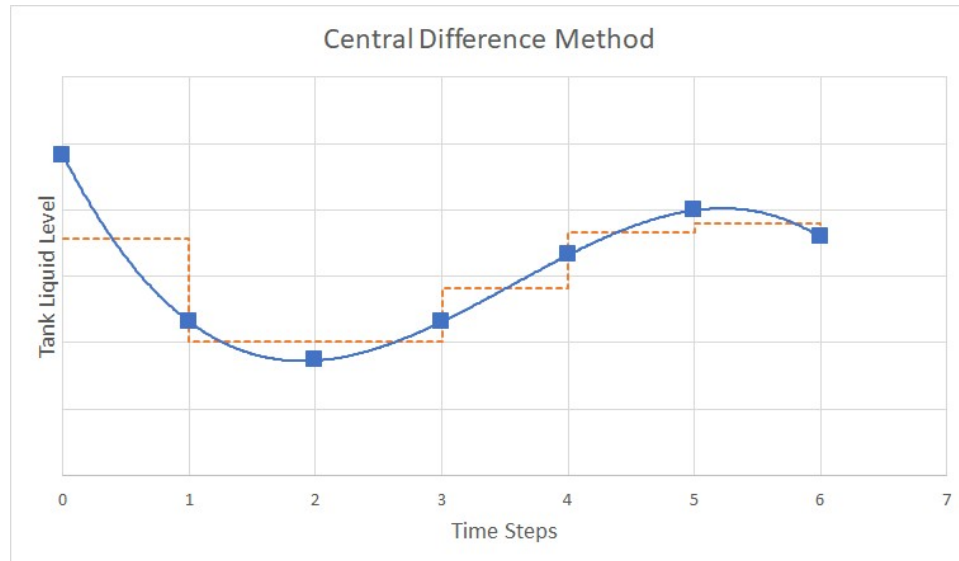
Figure 2: Central difference method for time calculation uses average liquid level during time step for calculation, requiring iteration for each time

There are several iteration control parameters needed by the central difference method, and these are input in the Simulation Duration panel(Figure 3). The central difference method attempts to converge on the liquid height at the next time step for each finite tank.

Relative and Absolute Tolerance

The relative and absolute tolerance inputs influence when AFT Fathom concludes it has con-verged on the new liquid height. Both of these criteria are applied, and if either one is satisfied then convergence has occurred.
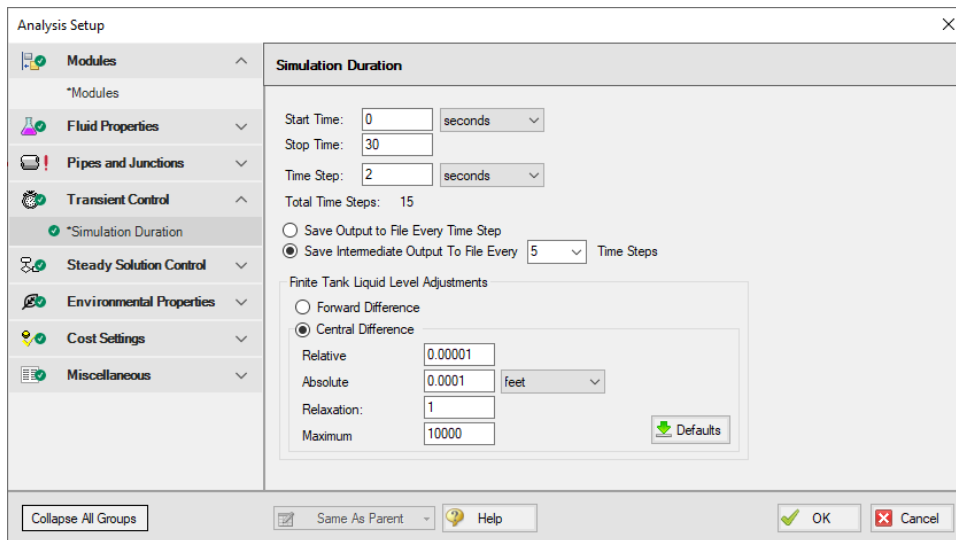
Figure 3: Iteration control parameters are required for the central difference method

The relative criteria is how much the liquid level calculation changes between iterations on a relative, or percentage basis. The absolute criteria relates how much the liquid level changes between iterations on an absolute basis of height. Hence the absolute tolerance has units of length associated with it.

The default values for relative tolerance is 0.00001, and the absolute tolerance is 0.0001 feet.

Relaxation

The relaxation applies to how fast AFT Fathom allows the liquid level to change during time steps. See the Iterations panel topic for a general discussion of relaxation.

Relaxation values must be greater than zero and less than or equal to 1. By default a value of 1 is used. This in fact means no relaxation.

Setting values closer to zero will improve convergence but slow the calculation. Values less than 0.01 should never be used unless directed by Applied Flow Technology.

Caution: The iteration control parameters for central difference time calculations should not be changed by the user unless they clearly understand their use or it has been recommended by Applied Flow Technology support. If changed indiscriminately, there is a danger of incorrect results being generated.

Maximum Iterations

This is the maximum number of attempts allowed for each time calculation iteration. If it reaches this value, the transient solution will stop due to failure to converge.

Changing Parameters During the Run

The Solution Progress window displays while the transient simulation is being performed. If a central difference method is used, because of its iterative nature the simulation can get stuck. If this happens then the simulation can be paused and Central Difference iteration parameters changed (such as reducing the relaxation parameter). With forward difference there are no parameters to change and hence this option is not available.

Output Values Displayed with Forward Difference

One advantage of the forward difference method is that it is easier to reconstruct the calculation for a given time step. For example, assume you want to run a separate steady-state AFT Fathom model to represent a particular time step calculation.

To do so, just look at the reservoir levels for that time step, enter these as the reservoir levels in the steady-state model, and all results should be the same as in the transient solution.

Output Values Displayed with Central Difference

When using the central difference method a somewhat strange result is that the reservoir liquid levels displayed in the output are never directly used to solve any time step (except for the initial "time zero" step). As discussed earlier, each time step is solved using an average liquid level. But this average liquid level is not displayed in the output. Instead, the liquid levels upon which the average was derived are displayed. This is depicted in Figure 3 where the liquid levels in the output do not correspond directly with the assumed liquid level for calculation. The squares represent the actual liquid level output values for central difference.

If for some reason you would like to construct a steady-state model to represent a time step from a transient simulation which used central difference, you must enter reservoir liquid levels which are the average of the desired time step and the previous time step.

Related Topics

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 Fathom 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

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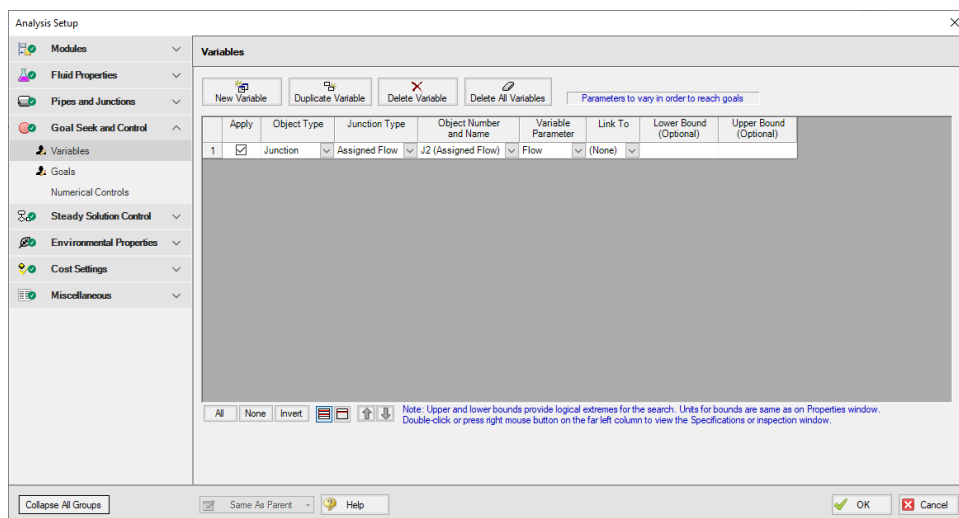
Figure 1: GSC Variables are input parameters to be automatically varied

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 Elevation Temperature	 Only with Heat Transfer
Assigned Pressure	Pressure Elevation Temperature	 Only with Heat Transfer
Branch	Imposed Flow	
Check Valve	Loss Value	
Control Valve	Control Setpoint	
General Component	Loss Factor	
Heat Exchanger	Loss Factor Heat Rate Discharge Temperature Temperature Rise Enthalpy Rise Heat Transfer Area Overall Heat Transfer Coefficient Secondary Flow Secondary Inlet Temperature	 Only with Heat Transfer Only with Heat Transfer Only with Heat Transfer Only with Heat Transfer Only with Heat Transfer Only with Heat Transfer Only with

		Heat Transfer Only with Heat Transfer
Orifice	Diameter/Area Loss Factor Elevation	
Pump	Speed Flow Head Rise Impeller Size Control Setpoint	
Relief Valve	Loss Value	
Reservoir	Liquid Level Surface Pressure Temperature	 Only with Heat Transfer
Screen	Open Area Loss Factor	
Spray Discharge	Spray Area K (Fire Sprinkler) Number Sprinklers Number Sparger Holes Elevation	
Three-Way Valve	Open Percentage	
Valve	Loss Value Elevation (Exit Valves)	 Only for Exit Valves

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 pump impeller trim as a variable for several pumps in parallel. It is often desirable to solve for a condition where the pump impeller trim is the same for all of the pumps. This can be accomplished by using variable linking. A pump impeller trim variable would be defined for each of the pumps. Then, one of the pumps would be selected as the basis pump, which just means no linking is specified for this pump. The other pumps would be linked to the basis pump by selecting the basis pump 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

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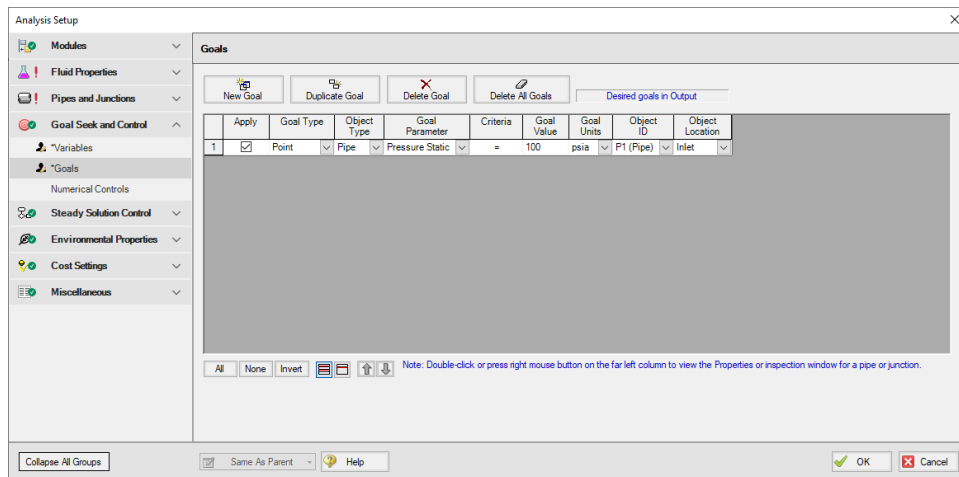
Figure 1: GSC Goals are output parameters to be achieved by changing the variables

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/Comments
Pipe	Energy Gradeline	
	Head Gradient	
	Head Loss	
	Heat Rate Into Pipe*	*Only With Heat Transfer
	Hydraulic Gradeline	
	Mass Flow Rate	
	Pressure Drop Friction	
	Pressure Gradient	
	Pressure Stagnation	
	Pressure Static	
	Pressure Static Maximum	
	Pressure Static Minimum	
	Temperature*	
	Temperature Insulation #1 Surface*	
	Temperature Insulation #2 Surface*	
	Temperature Insulation #3 Surface*	
	Temperature Loss*	
	Temperature Outer Surface*	
	Velocity	
	Volumetric Flow Rate	
	Wall Temperature*	

Area Change Bend Check Valve General Com- ponent Jet Pump Orifice Relief Valve Screen Valve Venturi Volume Balance	Energy Gradeline Hydraulic Gradeline Mass Flow Rate Pressure Stag- nation Pressure Static Temperature* Volumetric Flow Rate	*Only With Heat Transfer
Branch	Energy Gradeline Hydraulic Gradeline Mass Flow Rate* Pressure Stag- nation Pressure Static Temperature** Volumetric Flow Rate*	*Net Flow Rate **Only With Heat Transfer
Assigned Flow	Energy Gradeline Hydraulic Gradeline Pressure Stag- nation Pressure Static	
Assigned Pres- sure Reservoir	Mass Flow Rate Net into Jct* Vol. Flow Rate Net into Jct*	*Net Flow Rate
Control Valve	Cv* Energy Gradeline** Hydraulic	*If Open Percent Data Exists **Only for FCV ***Only for PRV, PSV

	Gradeline**	^Not PDCV
	Kv*	^^Only with Heat Transfer
	Mass Flow Rate***	
	Open Percentage*	
	Pressure Loss^	
	Pressure Stag-nation**	
	Pressure Static**	
	Temperature^^	
	Volumetric Flow Rate***	
Dead End	Pressure	
	Hydraulic Gradeline	
Heat Exchanger	Energy Gradeline	
	Heat Rate In*	*Only with Heat Transfer. For non-fixed heat rate exchanger models.
	Hydraulic Gradeline	
	Mass Flow Rate	
	Pressure Stag-nation	
	Pressure Static	
	Temperature**	**Only with Heat Transfer. For non-controlled tem-perature models.
	Volumetric Flow Rate	
Pump	% of BEP	
	Energy Gradeline	
	Head Rise*	*Only for non-fixed head rise pumps
	Hydraulic Gradeline	
	Mass Flow Rate**	
	NPSH Margin Abso-lute	**Only for non-fixed flow pumps
	NPSH Ratio	

	Power Overall	
	Pressure Rise*	
	Pressure Stagnation	
	Pressure Static	
	Temperature***	***Only with Heat Transfer
	Volumetric Flow Rate**	
Spray Discharge	Mass Flow Rate Out*	*Net Flow Rate
	Volumetric Flow Rate Out*	
Tee or Wye	Energy Gradeline	
Three Way Valve	Hydraulic Gradeline	
	Pressure Stagnation	
	Pressure Static	
	Temperature*	*Only With Heat Transfer

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 then 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

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.
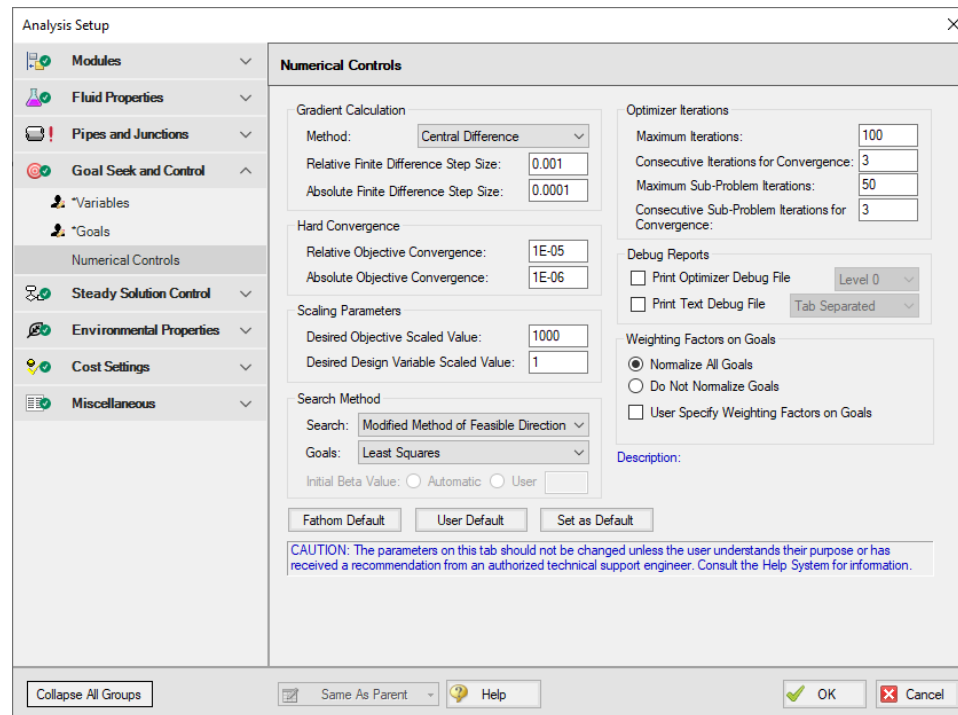
Figure 1: Default Numerical Controls panel setup in the Goal Seek and Controls Group

Related Topics

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 the AFT Support Team. **Poorly defined Solution Control settings can cause the model to converge on an incorrect solution without warning messages.**

- 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.

- [Temperature Limits panel](#) - Options to limit the temperature results in the Solver when [heat transfer is enabled](#).

Related Topics

Tolerance Panel

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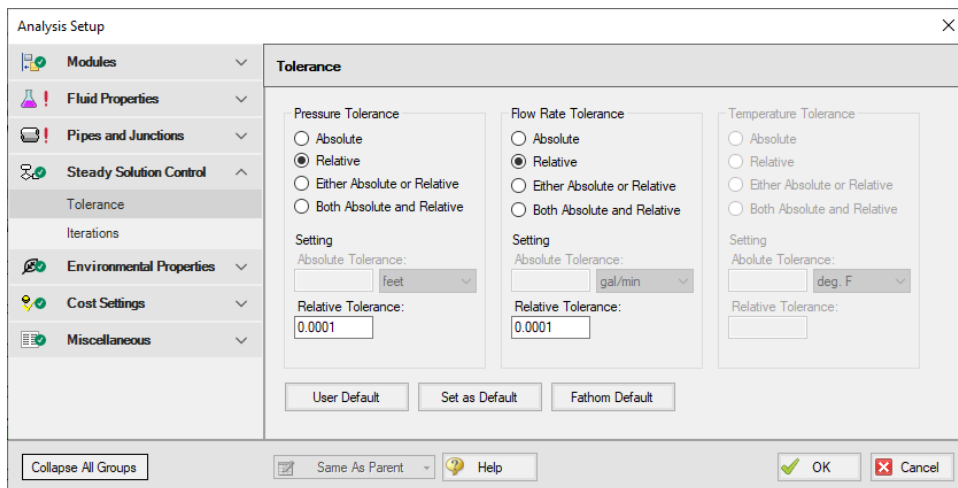
Figure 1: Tolerance panel in Analysis Setup

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 causes 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 tightened 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, on the other hand, 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

Iter-ation #	Mass Flow Rate (lbm/se-c)	Relative Change	Abso-lute Change (lbm/se-c)	Meets Abs. Tol-erance?	Meets Rel. Tol-erance?	Meets Abs. OR Rel. Tol-erance?	Meets Abs. AND Rel. Tol-erance?
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

Related Topics

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.
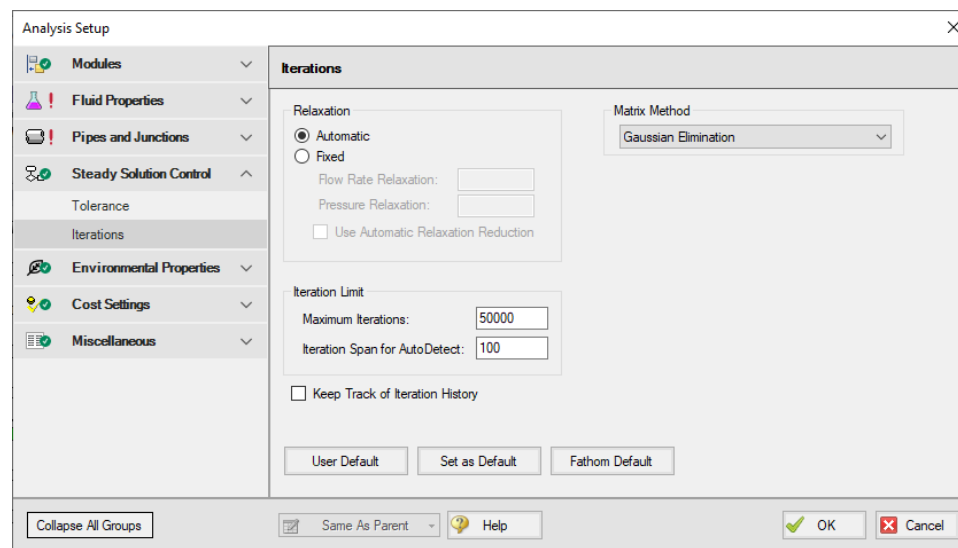
Figure 1: Iterations panel in Analysis Setup

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 <u>always</u> be set to 0.5 or 1 to maintain stability.

Iteration Limit

- **Maximum Iterations** - 50000 by default, sufficient for the vast majority of models. Most models converge in a relatively small number of iterations. This limit does not directly affect the solution.

- **Iteration Span for AutoDetect** - 100 by default, 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.

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 highly robust and usually provides the fastest convergence. The available Matrix Methods are:

- Gaussian Elimination
- Gaussian Elimination With Pivoting
- LU Decomposition

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.

Related Topics

Temperature Limits Panel

The Temperature Solver Limits panel allows users to set boundaries for temperature 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.

The Temperature Limits Panel is available if heat transfer is enabled.

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: Temperature Limits panel in the Analysis Setup window

Related Topics

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

Environmental Properties Panel

The Environmental Properties panel, shown in Figure 1, controls the definition of the atmospheric pressure, gravitational acceleration, and transition Reynolds number used to model the system.
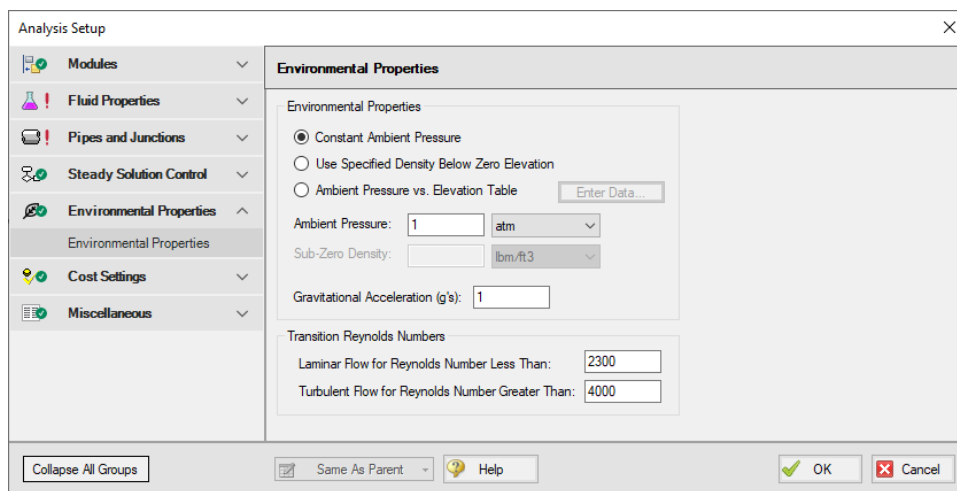
Figure 1: Default state of the Environmental Properties panel

Environmental Properties

Specify the ambient pressure and gravitational acceleration values for the solver to use in the calculations. By default, these values are 1 atm and 1 g, respectively.

Ambient Pressure

AFT Fathom 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. Fathom can automatically adjust atmospheric pressure using the Ambient Pressure vs. Elevation Table found in the Analysis Setup window. Here, users can input a pressure profile that will change the ambient pressure depending on a junction's elevation.

Gravitational Acceleration

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.

Transition Reynolds Number

AFT Fathom allows you to modify the transition Reynolds numbers. The default values assume that a Reynolds number of less than 2300 is laminar and a Reynolds number of greater than 4000 is turbulent. In the transition zone, a linear interpolation is assumed. This applies for friction factors and convection heat transfer.

You can modify these transition Reynolds numbers if you choose by manually typing in the values on the Environmental Properties panel in the Analysis Setup window.

Related Topics

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

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

The first step in any cost calculation is to enable it in the Cost Settings panel. The following options are available:

- **Do not calculate** - Selected by default.
- **Pump Energy Only** - Only calculate the costs associated with the power consumption of pumps. The energy cost and system life must be entered in Cost Settings.
- **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 costs calculated. This allows only analyzing the cost of certain parts of the model - useful if only one pump 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 below. 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

Energy cost data can be specified through an Energy Cost Library, or by directly entering an energy cost on the Cost Settings panel. 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

When Cost calculations are enabled the System Life is required. Interest Rate and inflation rate will default to zero, and may be changed by the user.

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

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

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 .
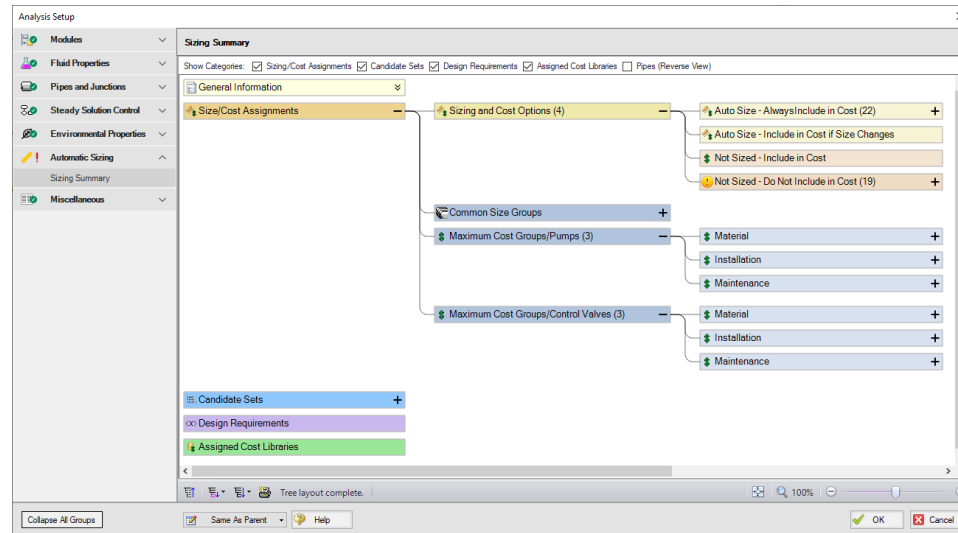
Figure 1: Sizing Summary panel with all tree nodes expanded

Related Topics

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 contains the following panels:

- NFPA panel - Allows NFPA to be activated to generate NFPA reports
- 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

NFPA Panel

The NFPA panel allows the user to enable NFPA settings, set the Hazen Williams factor, and define an ambient pressure for NFPA calculations.

NFPA stands for the National Fire Protection Association. This association develops codes and standards that are used by engineers designing safety related systems related to fire protection. AFT Fathom provides users the ability to specify remote paths and automatically calculate the hydraulics through that path as is recommended in NFPA 15 Section 8.5.3.1 (NFPA 15 (2012), pg 23). When coupled with the GSC module (Goal Seek and Control), AFT Fathom includes the ability to determine the hydraulically most remote path automatically.
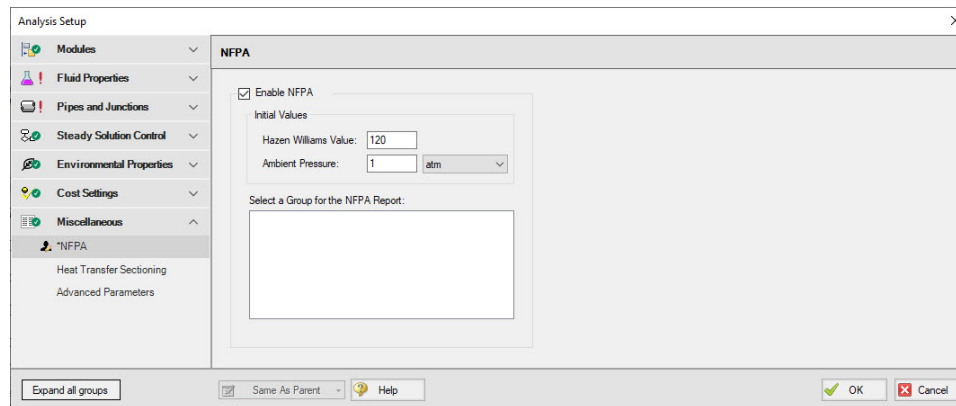
Figure 1: NFPA panel

NFPA Report

In order to generate the NFPA report, a group must be created using the Group Manger. This group must contain at least one spray discharge. It is best practice for this group to be a single continuous flow path connecting the most hydraulically remote spray discharge to the pressure source for the system. Once the group is created, a user will need to select the group on the NFPA panel above. Multiple groups can be created, but only one can be selected per run. To see how this is done, see the NFPA Firewater System Example. For more detail on NFPA reports consult the NFPA standard.

Terminology

The table below shows equivalent naming conventions between Fathom and the NFPA standard. Stagnation (total) pressure is equal to static (normal) pressure plus dynamic (velocity) pressure.

AFT Fathom	NFPA Standard
Stagnation Pressure (P_o)	Total Pressure (P_t)
Static Pressure (P)	Normal Pressure (P_n)
Dynamic Pressure	Velocity Pressure (P_v)

Differences Between Fathom and NFPA Standards

The NFPA standard was built as a method that engineers could use to solve complex fire water networks via hand calculations. To accomplish this the NFPA standard uses certain assumptions which differ from the computational methods and precise solution methodology that

Fathom uses in its Solver. Therefore, it is expected that Fathom and NFPA hand calculation results may differ. Some key differences are listed below:

- **Matrix Methods for Flow Splits** - Fathom uses a matrix of equations to calculate the exact flow splits, whereas the NFPA Standard uses conservative estimates for flow splits.
- **Velocity Changes at Flow Splits** - Fathom accounts for velocity changes at flow splits due to diameter changes and mass leaving the system at other nozzles.
- **Stagnation Pressure at Nozzle Exits** - Fathom uses stagnation pressure as the driving force for flow out of all nozzles (inline and end nozzles), whereas the NFPA standard uses static pressure for inline nozzles (total pressure minus velocity pressure). Therefore, Fathom does not adhere to NFPA 15 (2012) Section A.8.5.1.2 (pg 48).

Related Topics

Hazen-Williams NFPA

Some in industry use an alternate form of the Hazen-Williams formula which has this form (NFPA 13, 1999):

$$p = \frac{4.52 \, Q^{1.85}}{C^{1.85} \, d^{4.85}} \qquad (1)$$

where:

- p = Pressure drop gradient (psi per foot)
- C = friction loss coefficient
- Q = Volumetric flow rate (gpm)
- d = Internal diameter (inches)

This differs slightly from the Hazen Willams version used in AFT Fathom.

While both Equation 6 (and its variant Equation 7) in topic Pressure Drop in Pipes and Equation 1 (in this topic) are used heavily in industry, surprisingly they are not equivalent. The reason relates to Equation 1 being in terms of pressure drop but not having a fluid density in the equation. Equation 1 is intended for water, but it says that the pressure drop is independent of fluid density. This is contrary to Equation 2 and almost all other references.

The definition of pressure loss as it relates to head loss is

$$\Delta P = \rho \, g \, \Delta H$$

If this equation is used to equate Equations 1 and 2, the density must be 62.0933 lbm/ft3 to achieve equality. Hence any other density used in AFT Fathom will result in a pressure drop which does not equal to that of the Equation 1 NFPA formulation. Using the ASME Steam Tables, a density of 62.0933 lbm/ft3 at 1 standard atmosphere occurs at a temperature of 91.78 F.

Matching AFT Fathom Hazen Williams to NFPA Hazen-Willams

Users who want their results to match the NFPA version in Equation 1 can do so by one of the following two options:

- Use a density of 62.0933 lbm/feet3 in all calculations, or
- Use the desired density and then apply a design factor on each pipe to correct the pressure drop for density. The design factor should be equal to 62.0933 lbm/feet3 divided by the density on the Fluid panel in Analysis Setup. If variable properties are used, the design factor must be varied on a pipe-by-pipe basis.

Related Topics

Heat Transfer Sectioning Panel

If heat transfer is enabled, the number of thermal sections that are used to calculate heat transfer in Pipes and Heat Exchangers can be modified in the Heat Transfer Sectioning panel, shown in Figure 1.

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.
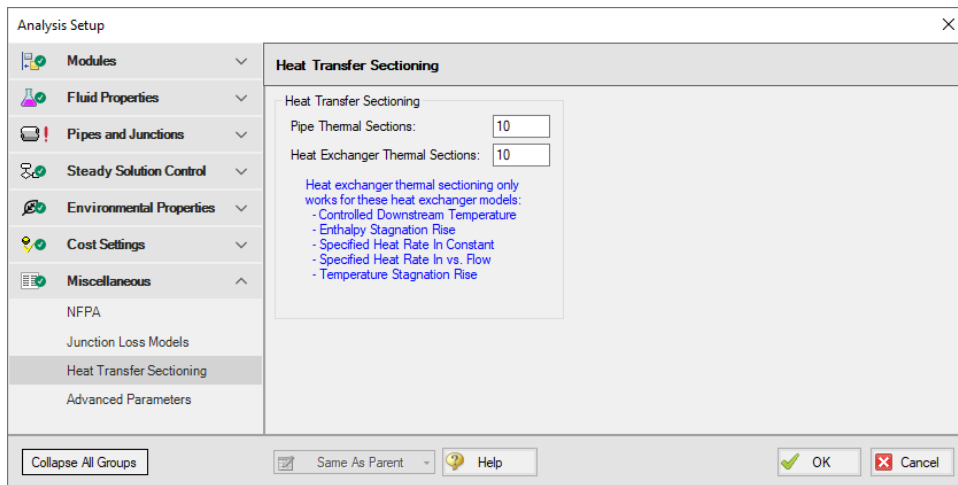
Figure 1: Default state of the Heat Transfer Sectioning panel

Related Topics

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.
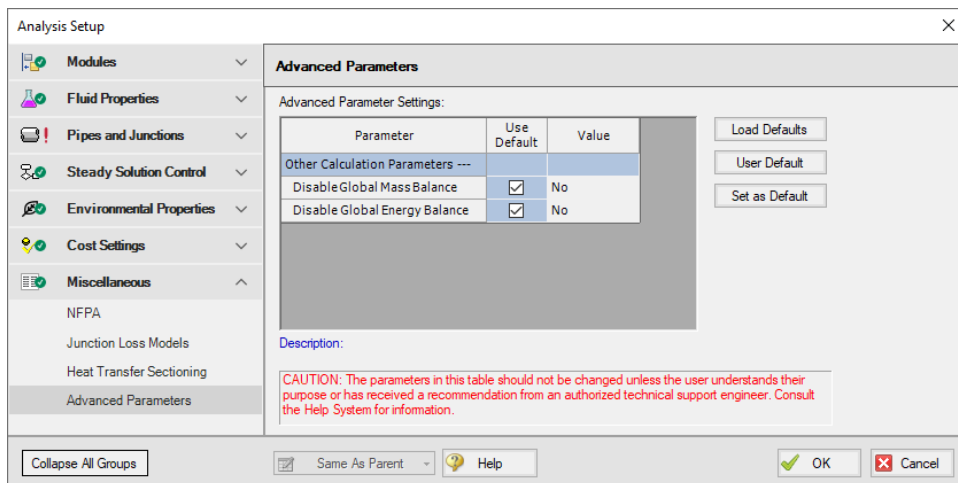
Figure 1: Default state of the Advanced Parameters panel

Related Topics

Working with Pipe and Junction Objects

This section focuses on interactions with the Pipe and Junction objects in the AFT Fathom workspace. Topics include how to create pipe 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

AFT Fathom 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

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 Properties
- Zero-Length Connectors
- Notes
- Undefined Objects
- User-Colored Pipes
- Transient Data Exists (XTS only)

- Groups
- Used as Goal or Variable
- Variable Fluid Properties
- 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

Finding 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 any 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

Groups

Groups of pipes and junctions can be named for later recall. This is done using the Group tools on the Edit menu. The easiest way to create a named Group is as follows:

1.   First select the pipes and junctions you want in the group.
2.   Choose Groups -> Create from the Edit menu or Toolbar.
3.   You will be prompted for a name for the group. Enter a meaningful name and click OK.
4.   The Group Manager then opens. The selection of pipes and junctions in the Group will match those currently selected on the Workspace. If you want to modify the selections further, 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

Pipe Drawing Modes

AFT Fathom 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.
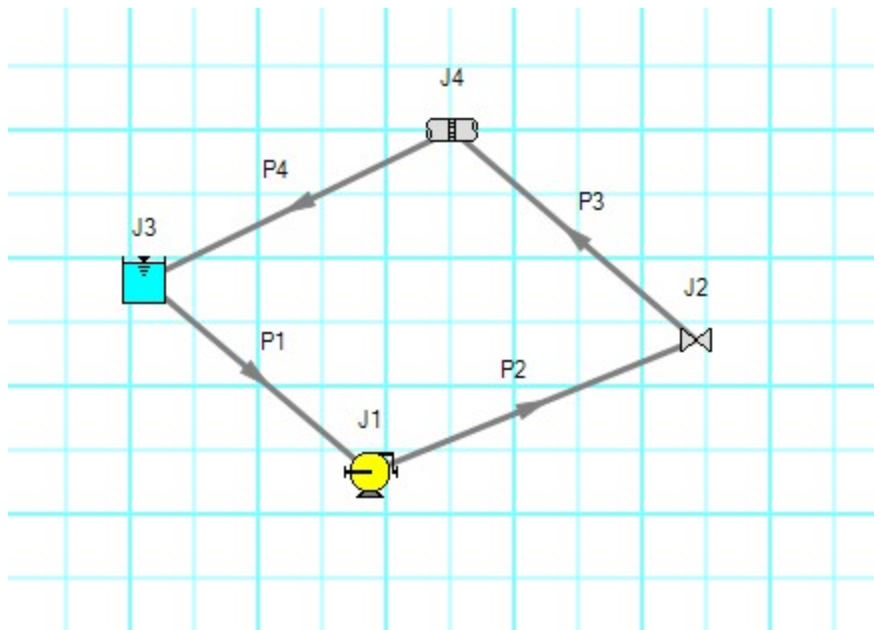
Figure 1: 2D Freeform with Snap to Grid off
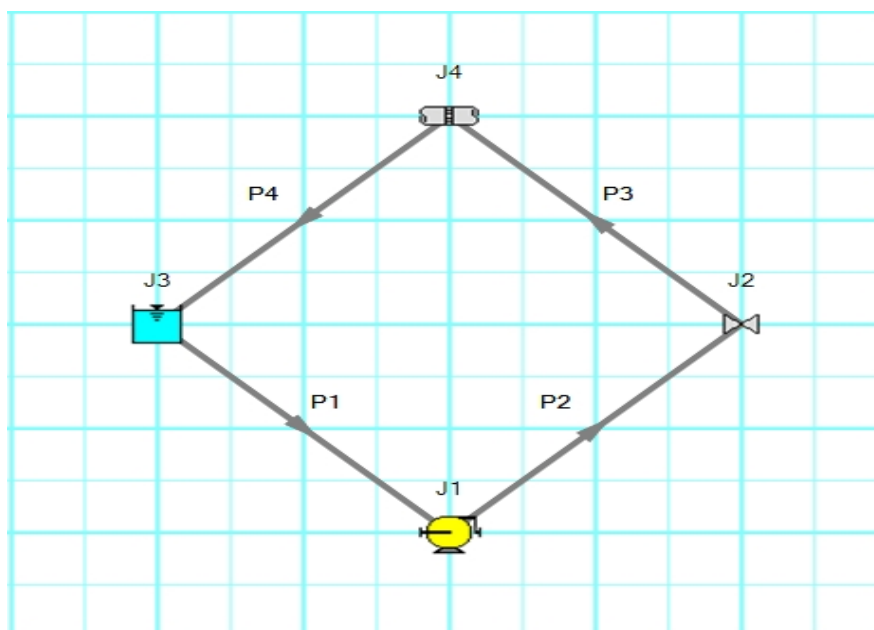
Figure 2: 2D Freeform with Snap to Grid on

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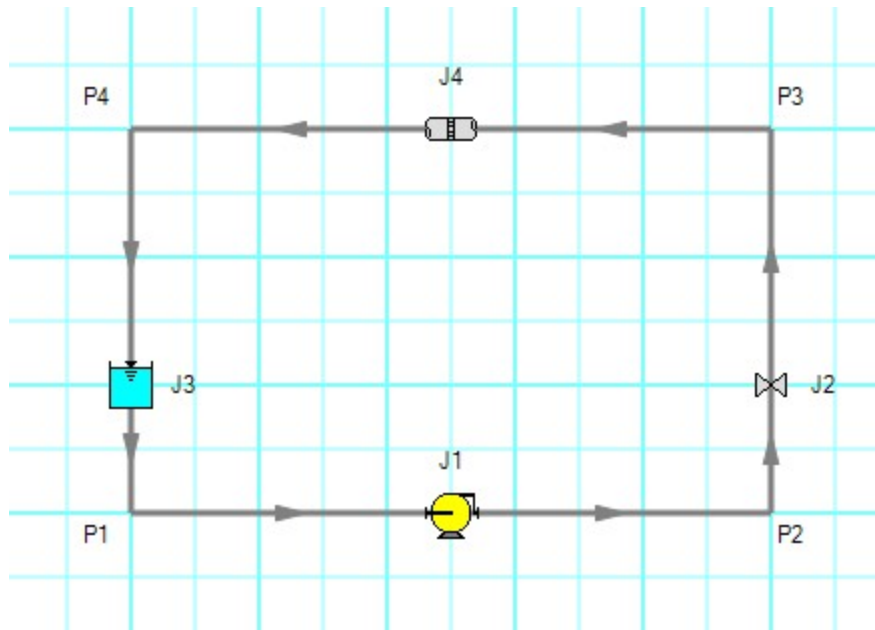
Figure 3: Model in 2D Orthogonal mode with Snap to Grid on

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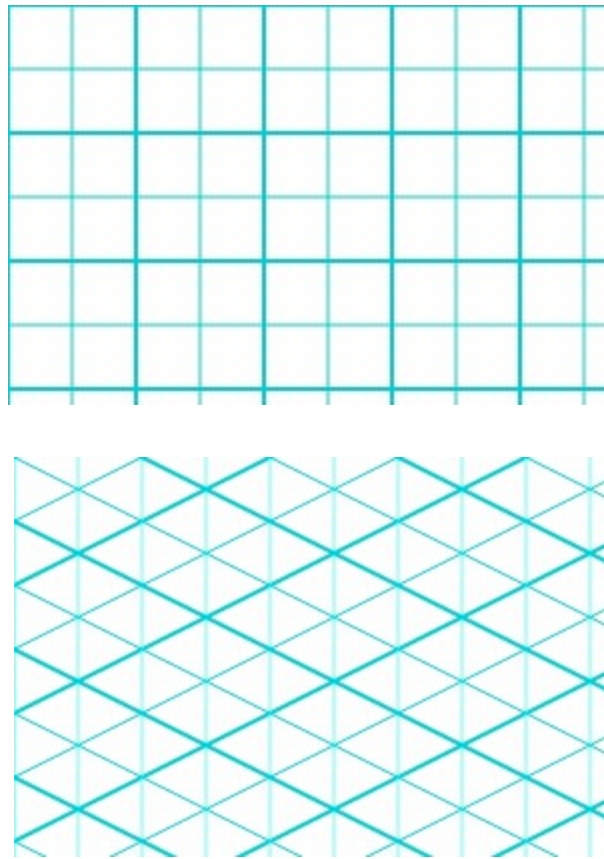
Figure 4: 2D (left) and isometric (right) grids

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.
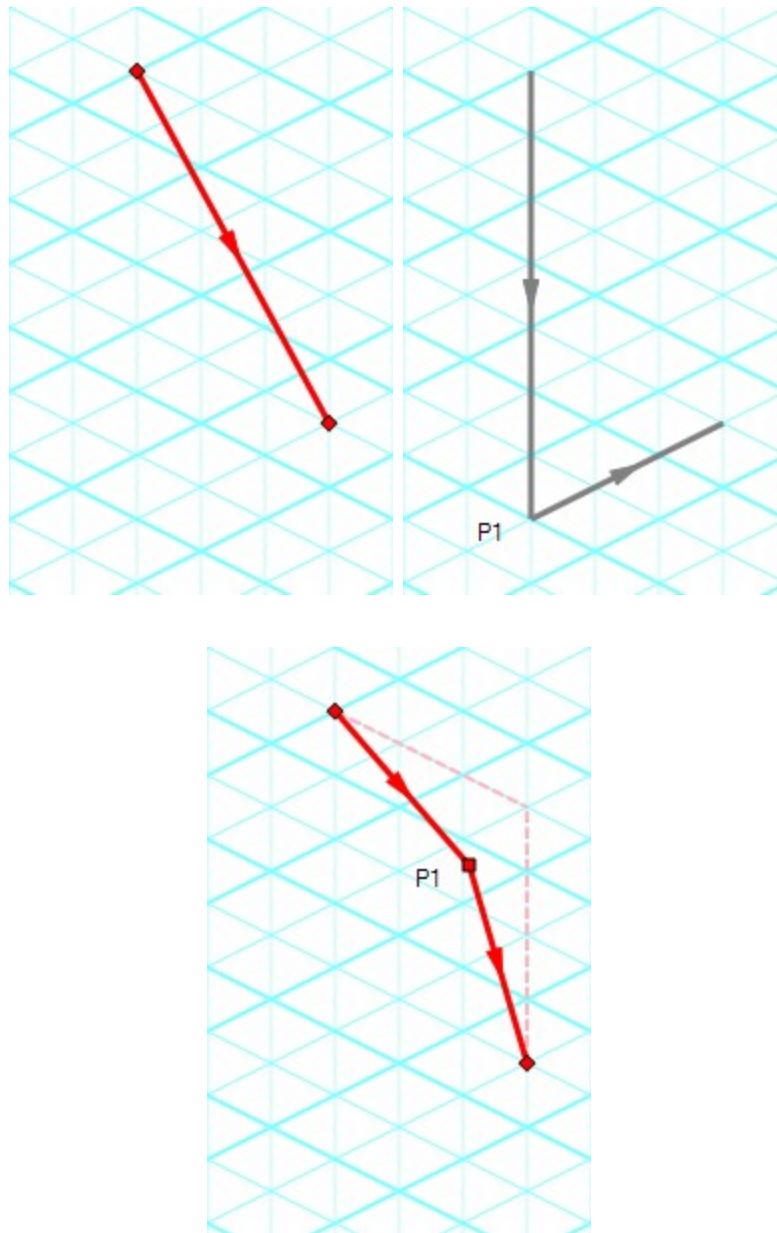
Figure 5: Automatic layout of isometric pipes

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 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 Fathom 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.
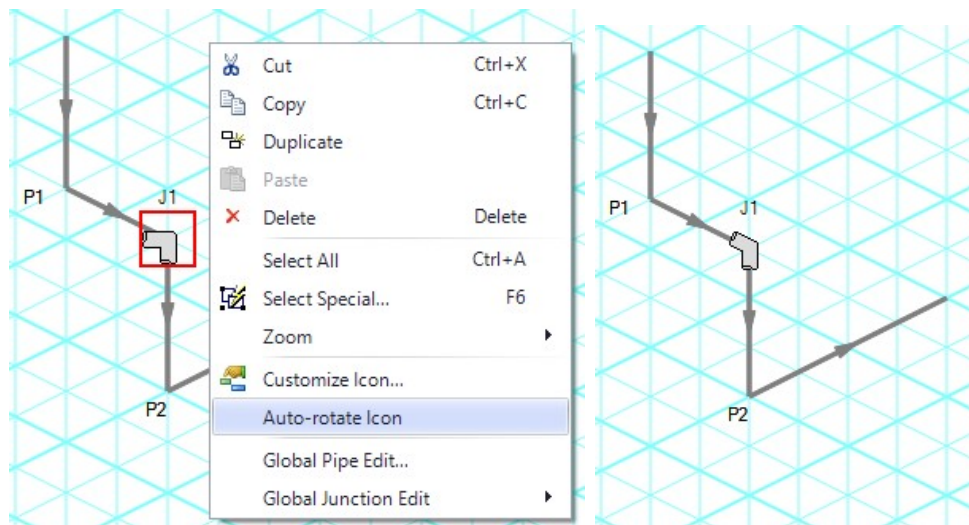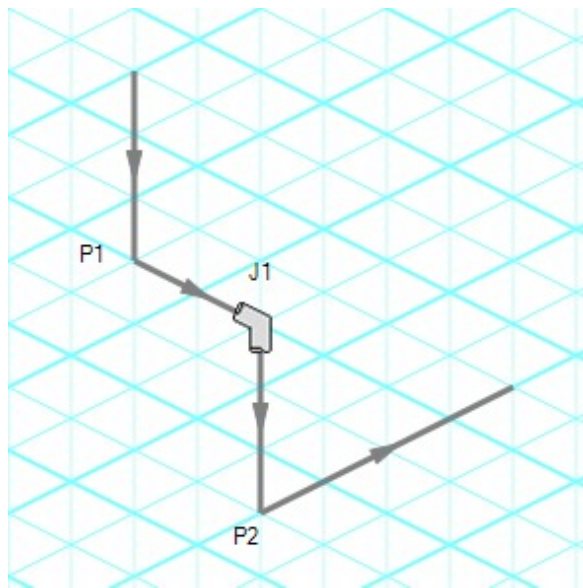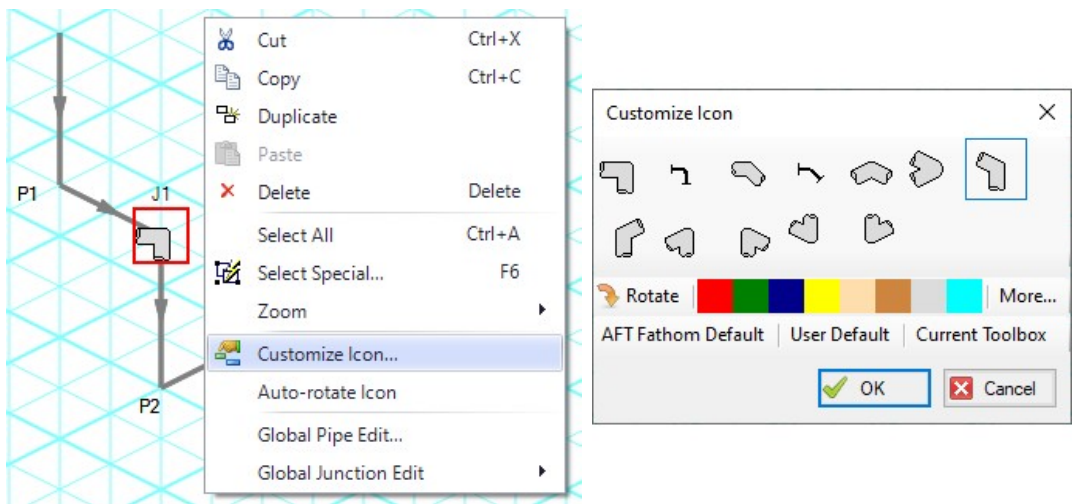
Figure 6: Icon rotations in isometric space

Related Topics

Global Edit

AFT Fathom 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 by using the Global Junction 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, from the Workspace toolbar, or from the right-click menu on the Workspace.
2. Select the pipes you want to edit from the Pipe list on the left (if pipes were selected in the Workspace, they will be automatically selected here).
3. Click the Select Pipe Data button. This will open a template Pipe Properties Window in which you can enter the data you want to change on each of the selected pipes. Alternatively, if you want to copy the data from an existing pipe, you can choose "Existing Pipe", then select the desired pipe number and skip to step 5.
4. Enter the pipe data you want to change and click the OK button in the Pipe Properties window.
5. The Global Pipe Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
6. Click the Apply Selections button.

Note: You cannot change some data, like the pipe ID number, using this global method.

Note: There is no undo for global pipe editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

Related Topics

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 or the right-click menu. From there, choose the category of junction you want to edit. The available categories include those junctions selected on the Workspace, any junction type used in the model, and common junction data. Global Junction Edit can also be launched by right-clicking on the Workspace.

Note: There is no undo for global junction editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

Editing Common Junction Data

You can open the Global Junction Edit window for common data editing in two ways. First, you can choose Common Data from the Global Junction Edit menu selection on the Edit menu, or you can choose one of the other specific junction types on the menu and then choose the Common option for the Junction Template in the upper left.

Here is how to globally edit junction common data:

1. Open the Global Junction Edit window.
2. Select the Common option for Junction Template in the upper left.
3. Select the junctions you want to edit from the Junction list on the left. Note that all junctions in the model are shown, no matter what the junction type.
4. Click the Select Common Junction Data button. This will open the Common Junction Data Edit window in which you can enter the data you want to change on each of the selected junctions.
5. Enter the junction data you want to change and click the OK button in the Junction Properties Window.
6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
7. Click the Apply Selections button.

Note: You cannot change some data, like the junction ID number, using this global method.

Editing Specific Junction Data

You can open the Global Junction Edit window for specific data editing in two ways. First, you can choose the junction type from the Global Junction Edit menu selection on the Edit menu, or you can choose Common Data on the menu and then choose the desired junction template, or Existing Junction data for the desired junction type.

Here is how to globally edit specific types of junctions:

1. Open the Global Junction Edit window.
2. Select the Junction Template option in the upper left. Alternatively, if you want to copy the data from an existing junction, you can choose "Existing Junction", then select the desired junction number and skip to step 6.
3. Select the type of junction you want to edit in the dropdown list in the upper left. For example, you could select the Spray Discharge junction.
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

Toolbox

The Toolbox is located on the left of the Workspace window by default, but can be undocked by either dragging or right-clicking the heading and selecting Float. The assortment of tools allows you to create pipes, junctions, and annotations.
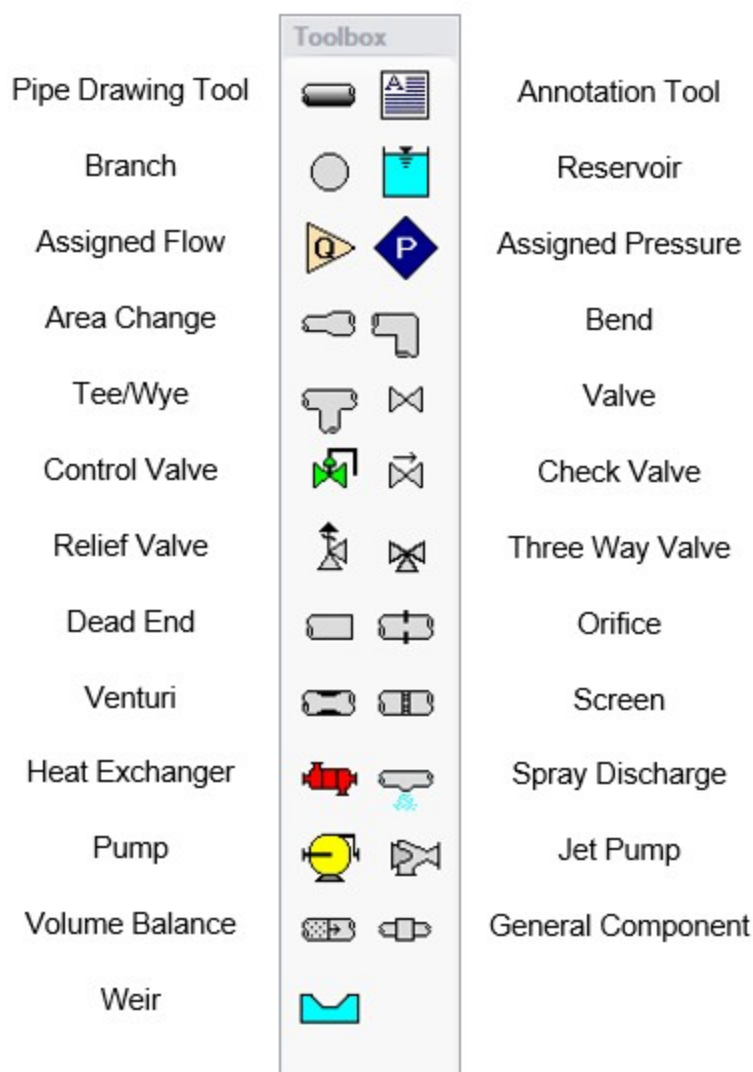
Figure 1: Toolbox items

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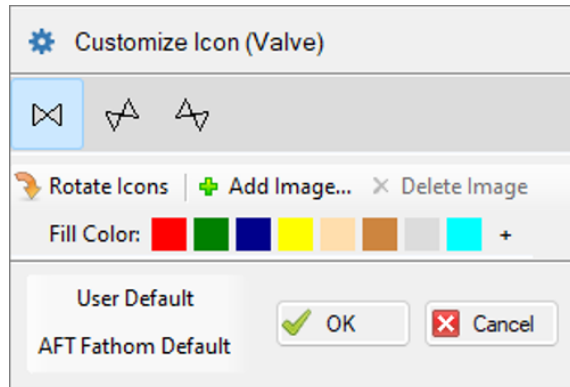
Figure 2: The Customize Icon window allows a custom image to be applied to any junction

Related Topics

Creating Pipes - the Pipe Drawing Tool

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 Fathom 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. These features can also be accomplished both from the right-click menu on a selected pipe as well as the Workspace toolbar. When a new pipe is derived from a previous pipe, the new pipe retains the input associated with the original pipe.

Each pipe must be connected to two junctions. There are no exceptions to this rule.

Pipe ID Numbers

When a new pipe is created, it is assigned a default ID number. When the pipe is displayed on the Workspace, the ID number is preceded by the letter "P" to indicate that it is for a pipe. From the Pipe Properties Window, you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the pipe and cannot be replicated in any other pipes on the Workspace. The pipe number remains with the pipe in descendant scenarios in the Scenario Manager. An arrow near the ID number indicates the reference flow direction for the pipe. The reference positive flow direction can be reversed by choosing Reverse Direction from the Arrange 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. This will open the Reset Label Locations window. From the Reset Label Locations window, 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 All Labels from the Reset Label Locations window.

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.

Related Topics

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. It will also have the elevation profile of the original pipes and junctions entered as Intermediate Pipe Elevations.

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 September 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 two different ways. First, holding the Shift key while placing a junction along the pipe 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.

Second, pipes with Intermediate Pipe Elevations can be split into separate pipes at any or all of those intermediate elevations. This can be done by selecting a pipe with Intermediate Pipe Elevations and clicking Edit -> Split Pipe at Intermediate Elevations. A window will open to select which intermediate elevations to make into new junctions.

The pipe can be split at all intermediate elevations, or at selected locations. This is specified by checking the box next to each elevation point the pipe should be split at. When the pipe is split, a branch will be created for each of the selected elevation points. If desired, a name can be entered for each new branch in the New Junction Name column of the table. Any points not selected will be retained in the intermediate elevation table of the appropriate pipe in the Workspace.

Note: When splitting a pipe at intermediate elevations, the pipe in the Workspace must be long enough to place the necessary number of branches in the pipe before the split can be completed. If insufficient space is available, the user will be required to exit the Split Pipe at Intermediate Elevations window and rearrange the model to create space before Fathom can perform the split action.

Figure 1: Split Pipe at Intermediate Elevations window

For pipes with a large number of intermediate elevations, it may be more convenient to define the split points in a spreadsheet. This can be done from the Edit Table button at the bottom of the window by selecting Export to File. This will export the table to a *.csv table which can be opened in Excel, as is shown in Figure 2.

The table can be updated by changing the "Split At" column value to Yes for each location that the pipe will be split at. Branch names can also be entered under "New Junction Name" column for the appropriate points. Note that it is important to not change any of the Length or Elevation columns in the *.csv file, as this will prevent the file from being imported. To import the file into Fathom, select "Import From File" from the Edit Table button, then browse to the updated *.csv file.

	A	B	C	D	E	F	G
1	Split At	Total Length (feet)	Segment Length (feet)	Elevation (feet)	New Junction Name		
2	No	100	100	-9			
3	No	200	100	-15			
4	No	300	100	-12			
5	No	400	100	-3			
6	No	500	100	0			
7	No	600	100	5			
8	No	700	100	12			
9	No	800	100	16			
10	No	900	100	20			
11	No	1000	100	15			
12	No	1100	100	12			
13	No	1200	100	5			
14							

Figure 2: Exported *.csv file for splitting a pipe at intermediate elevations

Related Topics

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

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 (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system), 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 Fathom will attempt to automatically connect pipes and junctions when they are moved. This eliminates unnecessary opening of the pipe and junction Properties windows. If Fathom 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 Fathom. It is most efficient to do this when you are entering the pipe's data in its Properties Window.

3. Open the Properties Window of the corresponding junction and accept the pipe connectivity determination. In most junctions, this pipe connectivity determination will be

located in the upper right hand corner of the Properties Window. In a few junctions, such as the Assigned Pressure Junction and Reservoir Junction, the pipe connectivity will be shown under the tab containing Loss Coefficients. You can do this as you enter the junction's data in its Properties Window.

If the model seems to not be connected properly, you can also use the Extended Model Check feature on the View menu.

When you accept AFT Fathom'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 Fathom disconnects the two objects. However, if you move a group of connected objects, their model connectivity is maintained.

Related Topics

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.

Keyboard Modifiers

If you hold down the CTRL key when dragging a junction, the endpoints of any connected pipes are not moved with the junction, thus breaking the graphical connection.

Related Topics

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. Note that not all Arrange actions can be undone.

The Copy Graphics feature on the Edit menu will copy the Workspace objects to the Windows clipboard. This image can then be pasted into other Windows applications.

Related Topics

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, select a junction from the Toolbox, and then 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

Defining Objects

To specify the required property data for an object, open its properties window. This can be done in six 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 valve 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 heat exchanger on the Workspace. A heat exchanger requires inlet and outlet piping. AFT Fathom reflects this by requiring each heat exchanger to be assigned an upstream pipe and a downstream pipe. Entering only the heat exchanger constants and setting an elevation is **necessary but not sufficient information to define the heat exchanger** according to AFT Fathom's requirements. Once you have entered the heat exchanger data and properly connected it, the heat exchanger 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 Workspace 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

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 above the tab interface 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 Fathom or when you are having difficulty obtaining a defined object status.

Related Topics

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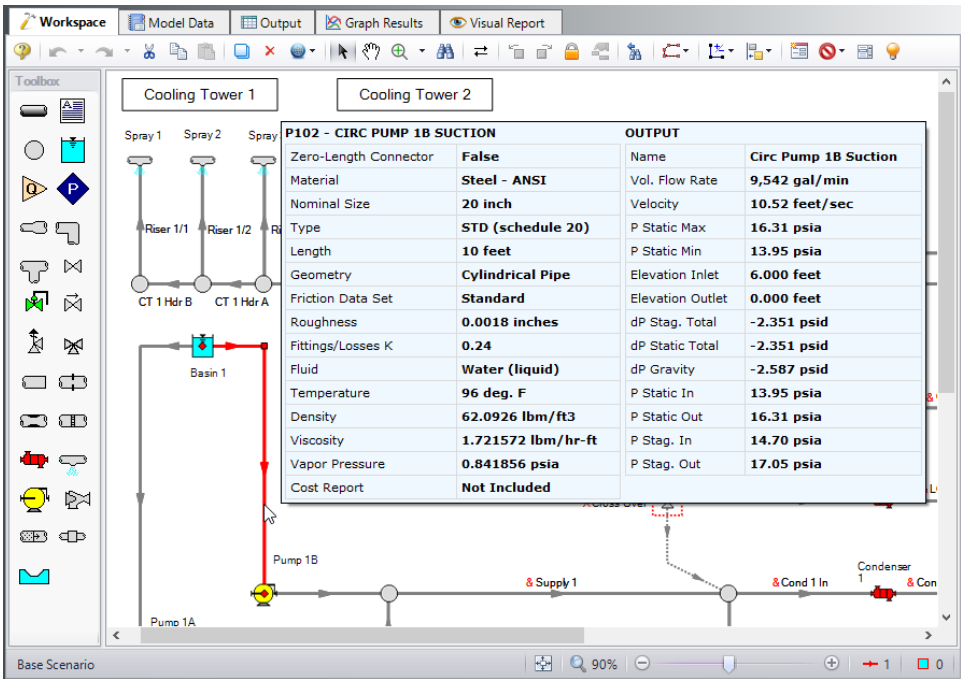
Figure 1: Holding the right mouse button on a Workspace object displays the associated data

Related Topics

Notes

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

Related Topics

Status

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

Related Topics

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

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

Special Conditions

Some junctions have Special Conditions which alter the normal state of the junction. For example, a valve can be closed, a pump 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 which have special conditions set are shown using a special symbol before the ID number (an "X" by default). This symbol can be customized in the User Options.

When the special condition results in a section of the model being closed or "turned off", the pipes which will have zero flow are display as dotted lines. This indicates visually that they are cut off from flow; the junctions are outlined with a dotted line. The default dotted line can be changed in the User Options.

The user can specify Special Conditions for Control Valves to be either closed or fully open. Similarly, Pumps can be turned off and either allow flow through or stop the flow.

Related Topics

Pipes

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

Contents

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

Related Topics

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 a child scenario, Copy Data From Pipe lists "Parent Pipe Data," allowing the user to copy all or some parameters from the parent pipe.

Connected Junctions

All pipe objects have a reference positive direction, as indicated on the Workspace by an arrow in the center of the pipe. The upstream and downstream junctions are identified with this dir-

ection. The connected junctions area displays these junctions. Every pipe must be connected to two junctions in order to be completely defined.

Related Topics

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:

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

    - **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 Pipe** - 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.

Theses models apply to Newtonian viscosity models. If a non-Newtonian model is selected in Analysis Setup, the below friction models may not be used.

- **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.

- **Hazen-Williams** - A method using an empirical factor to relate flow and pressure drop, common in water distribution and NFPA fire protection.

- **MIT Equation** - A model appropriate for crude oil.

- **Miller Turbulent** - A model appropriate for light hydrocarbons.

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

Related Topics

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 pump.

If you are using Equivalent Lengths, the table displayed will be a simpler form that allows the addition of specific bends and valves (the equivalent lengths displayed here are taken from NFPA13). You can also simply type in additional equivalent length.

The bottom of the table will show a Total Applied value which is a count of the number of losses that have been specified for the currently selected tab.
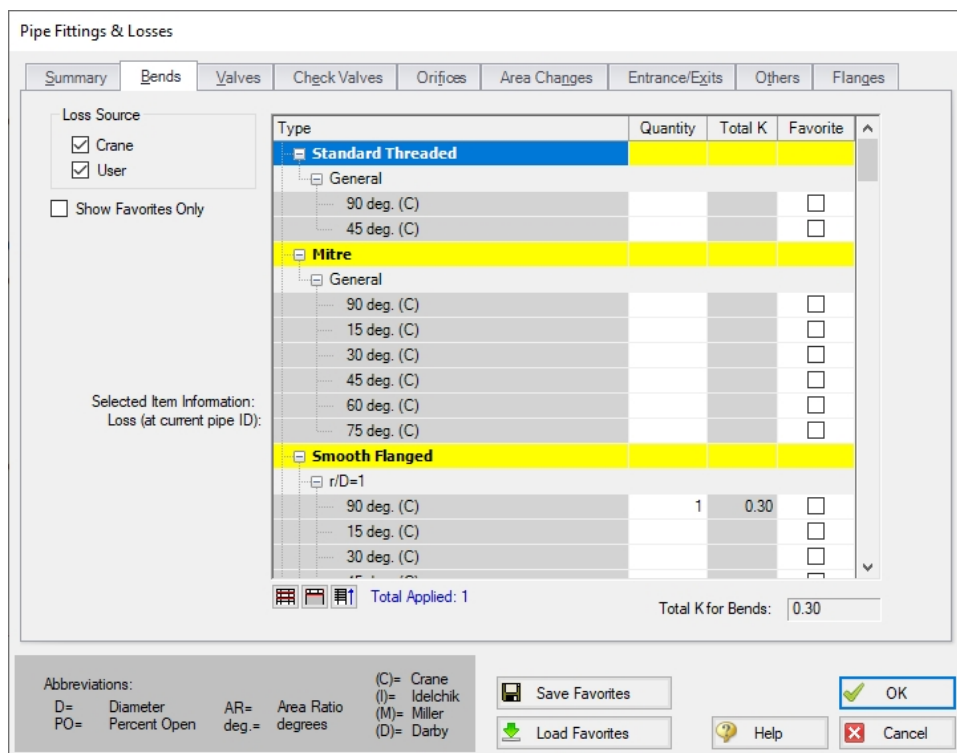
Figure 1: Pipe Fittings & Losses window

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 & Loses 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

Heat Transfer in Pipes

The Heat Transfer tab is only visible when heat transfer is being modeled. Heat transfer modeling is enabled in Analysis Setup on the Heat Transfer/Variable Fluids panel.

Each pipe responds thermally to the environment outside of the pipe. AFT Fathom 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 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	-50.00	-49.93	-49.46	-49.39	-49.16	-49.10	60.00
2	Show ...	Pipe	-49.93	-49.86	-49.39	-49.32	-49.10	-49.03	60.00
3	Show ...	Pipe	-49.86	-49.79	-48.81	-48.75	-48.52	-48.45	60.00
4	Show ...	Pipe	75.06	75.05	74.96	74.95	74.91	74.91	60.00
5	Show ...	Pipe	-49.86	-49.79	-48.81	-48.75	-48.52	-48.45	60.00
6	Show ...	Pipe	75.06	75.05	74.96	74.95	74.91	74.91	60.00
7	Show ...	Pipe	75.04	75.02	74.99	74.97	74.95	74.93	60.00

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:
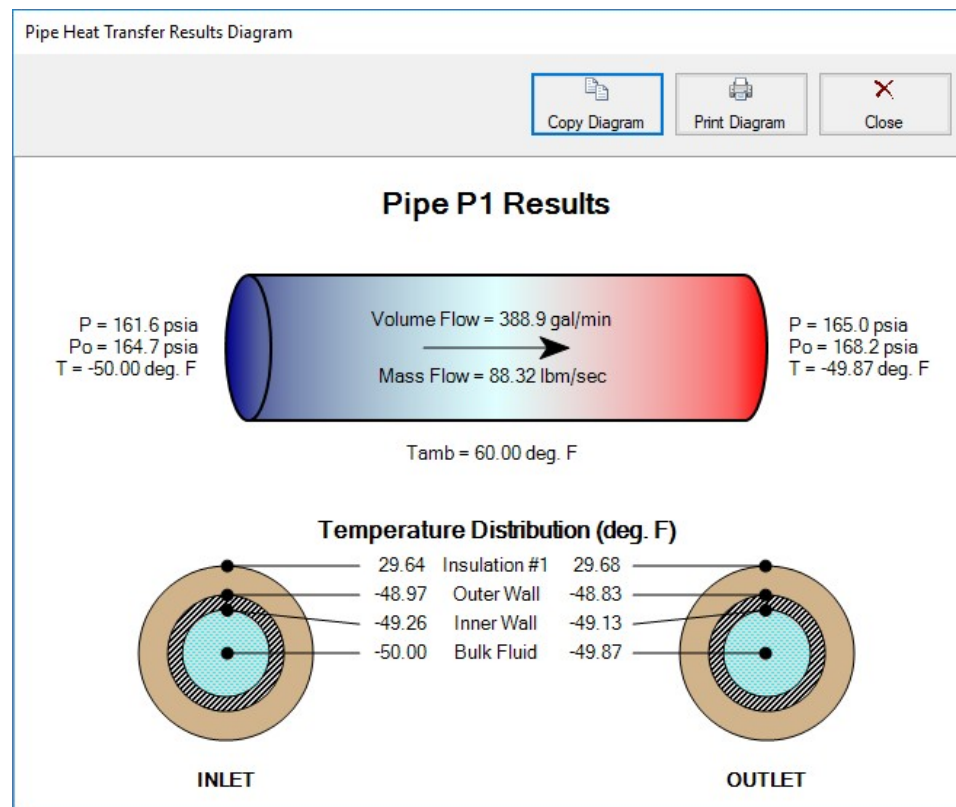
Figure 2: Pipe Heat Transfer Results Diagram

Related Topics

Constant Heat Transfer

There are several thermal models for pipes in AFT Fathom 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.

Note: The Heat Transfer tab will not be present unless heat transfer has been enabled from the Heat Transfer/Variable Fluids panel of the Fluid Properties group in the Analysis Setup window.

Figure 1: Constant Heat Flux

Related Topics

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 Fathom 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. If a Pipe Material is chosen on the Pipe Model tab, and that material has insulation data sets associated with it, an option is provided in the Conductivity/Convection Data Source column to select that insulation data set. The insulation data sets include the insulation thickness, and value is displayed in the thickness column.

  - **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

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 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.

Fathom 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 Fathom 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, Fathom models that heat flux based on the theory described below.
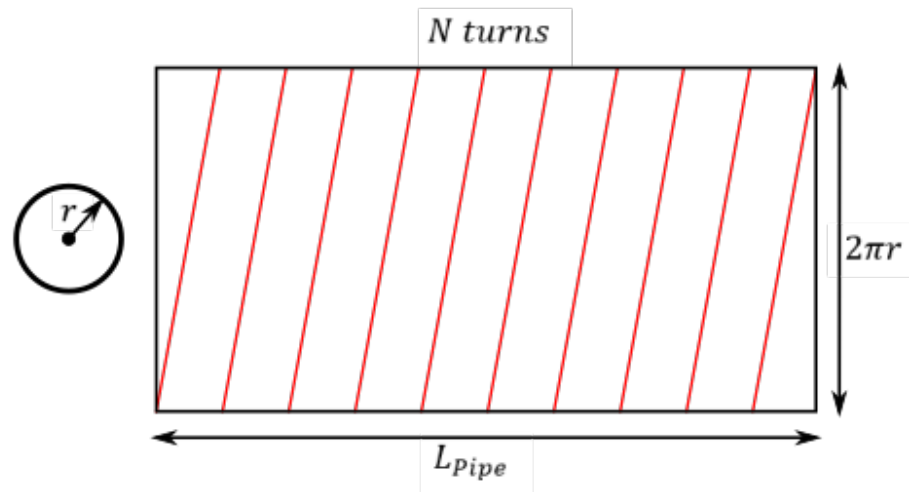
Figure 2: Wire tracing geometry

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:

$$N = nL_{Pipe}$$

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.

$$L_{Wire} = NL_{Turn}$$

Incorporating this relationship and rearranging:

$$L_{Wire}^2 = L_{Pipe}^2 + (2\pi rN)^2$$

Using the number of turns per unit length:

$$L_{Wire}^2 = L_{Pipe}^2 + (2\pi rnL_{Pipe})^2 = L_{Pipe}^2(1 + (2\pi rn)^2)$$

or:

$$L_{Wire} = L_{Pipe}\sqrt{1 + (2\pi rn)^2}$$

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:

$$Q'' = \frac{pL_{Wire}}{2\pi r L_{Pipe}}$$

Substituting in for L_{Wire}:

$$Q'' = \frac{pL_{Pipe}\sqrt{1 + (2\pi rn)^2}}{2\pi r L_{Pipe}}$$

Simplifying:

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

Related Topics

Heat Transfer in Pipes - Detailed Discussion
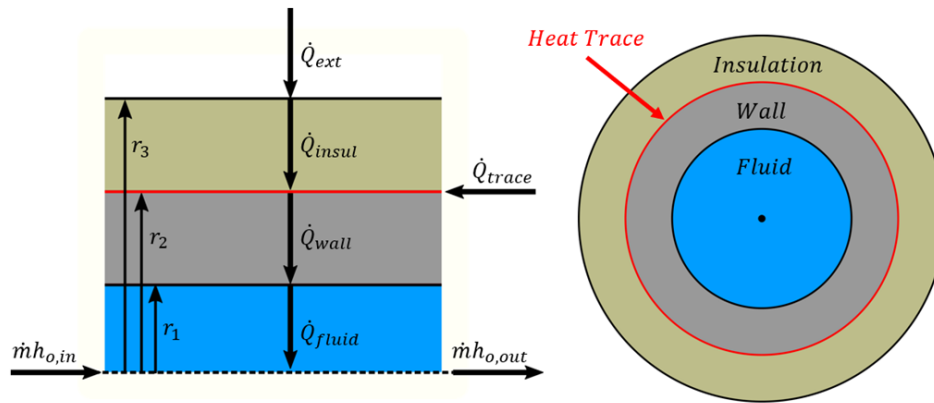
Figure 1: Diagram of a pipe with Heat Tracing and Insulation

There are many possible combinations of heat transfer that can be defined in AFT Fathom. 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 Fathom 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 Fathom 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

$$\Delta h = c_p \Delta T_{fluid}$$

To find that

$$q = c_p \Delta T_{fluid}$$

Rather than using specific quantities, it is useful to relate heat rate and mass flow. Multiply both sides by $\dot{m}$

$$\dot{Q}_{fluid} = \dot{m}c_p \Delta T_{fluid}$$

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:

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

We now have a system of equations that allow us to determine all the heat rates in the system given a mass flow (determined hydraulically), 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

$$R_{convection} = \frac{1}{hA}$$

$$R_{conduction} = \frac{\Delta r}{kA}$$

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

For conduction, the area and thus resistance changes with increasing *r*, so an integration must be done

$$\int_0^R dR_{conuction} = \int_{r_1}^{r_2} \frac{dr}{kA_{eff}} = \int_{r_1}^{r_2} \frac{dr}{kA_{ratio}2\pi rL}$$

$$R_{conduction} = \frac{\ln(r_2/r_1)}{kA_{ratio}2\pi L}$$

These resistances can all be calculated with user-defined data. AFT Fathom 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:

$$\boxed{e^{\frac{-UPL}{\dot{m}c_p}} = \frac{T_{external} - T_{out}}{T_{external} - T_{in}}}$$

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 found 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.

$$Nu = \frac{Convection}{Conduction} = \frac{h}{k/L}$$

These correlations also make use of the Prandtl number, a ratio of viscous diffusion to thermal diffusion.

$$Pr = \frac{Viscous\ diffusion}{Thermal\ diffusion} = \frac{\mu/\rho}{k/(c_p\rho)} = \frac{c_p\mu}{k}$$

Gnielinski

$$Nu = \frac{(f/8)(Re - 1000)Pr}{1 + 12.7(f/8)^{1/2}(Pr^{2/3} - 1)} \qquad valid\ for\ \begin{pmatrix} 0.5 \leq Pr \leq 2000 \\ 3000 \leq Re \leq 5 * 10^6 \end{pmatrix}$$

Dittus-Boelter

$$Nu = 0.023\ Re_d^{\ 4/5}\ Pr^n \quad where\ \begin{pmatrix} n = 0.4\ for\ heating \\ n = 0.3\ for\ cooling \end{pmatrix} valid\ for\ \begin{pmatrix} 0.6 \leq Pr \leq 160 \\ Re \geq 10^4 \end{pmatrix}$$

External convection behavior differs from the above. AFT Fathom 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.

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

Buried Pipe Heat Transfer

Buried Pipe heat transfer is modeled by solving directly for the thermal resistance of the soil layer. AFT Fathom 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 centerline, while r is the outside radius of the pipe.

Related Topics

Insulation for Pipes

In general, insulation data can be entered either on the Insulation tab or Heat Transfer tab. Only one of these tabs displays at a time. Insulation input depends on whether or not heat transfer is being modeled, and is an optional input. Insulation data is from the Nayyar Piping Handbook 2000 pages B.325-B.344.

If Heat Transfer Is Not Modeled

If heat transfer is not modeled, then a separate Insulation tab appears on the Pipe Properties Window. The insulation will have no thermal effect.

- **Insulation on Outside of Pipe or Duct** - If the insulation is on the outside of the pipe or duct, it will have no hydraulic effect, and will only affect cost calculations if they are enabled.

- **Insulation on Inside of Pipe or Duct** - If the insulation is on the inside of the pipe or duct, it will reduce the flow area and change the pressure drop calculations. AFT Fathom accounts for this diameter reduction in such cases. The user must enter the insulation friction characteristics.

If Heat Transfer Is Modeled

When heat transfer is modeled, insulation input occurs on the Heat Transfer tab. There is no Insulation tab in this case.

- **Convective Heat Transfer Not Used** - If a heat transfer model other than convective heat transfer is selected, the effect of insulation is the same as discussed above for the non-heat transfer case.

- **Convective Heat Transfer Is Used** - If convective heat transfer is selected, the effect of insulation is the same as discussed above for the non-heat transfer case except now the thermal effect of the insulation is accounted for.

Related Topics

Fluid Properties for Pipes

This folder tab is only visible when variable properties are being modeled. Variable property modeling is enabled in Analysis Setup on the Heat Transfer/Variable Fluids panel.

For models with variable fluid properties, you can assign different fluid properties to each pipe in the model. This is convenient for modeling systems that are subject to temperature variations and systems that have multiple fluids. The pipe fluid properties are displayed in the Pipe Properties Window, and can be modified by selecting the Fluid Properties button. Besides varying the fluid properties, the viscosity model can be varied from pipe to pipe. Thus some pipes in the model can be modeled as Newtonian, while others as non-Newtonian.

Related Topics

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 get the Solver going in the right direction so a converged solution results. If this is not the case, user defined guesses can help obtain a solution.

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.

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.

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 or Equivalent Lengths) 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.

Intermediate Pipe Elevations

The inlet and exit elevations of a pipe are equal to the elevations of the upstream and downstream junctions. The pipe elevation is assumed to vary linearly between the junctions. In other words, the pipe is assumed to be perfectly straight.

Modeling a large number of elevation changes with straight pipes would require a large number of junctions, as shown below:
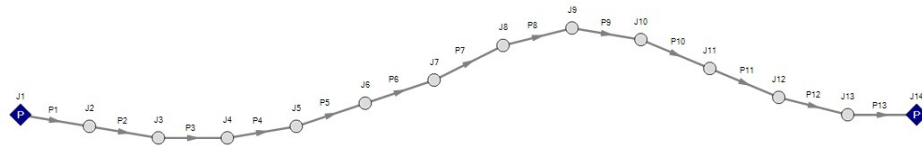
Figure 1: Accounting for elevation changes with many pipes and junctions

This is tedious and not required for overall hydraulic calculations - only the elevation change across a (fluid full) pipe is needed. However, intermediate pressure results along the pipe will be affected.

For example, Figure 1 shows elevation extremes both lower and higher than the elevations at the ends of the pipeline. Low points may exceed maximum allowable pressures, and high points could drop below vapor pressure. If this is modeled as a single straight pipe these issues would not be easily identified. In fact, if these pipes were replaced with a single pipe it would show no elevation change at all!

Instead of using a large number of pipes, these effects can be accounted for in a single pipe using intermediate pipe elevation feature.

With length defined, check Use Intermediate Pipe Elevations, choose the elevation units, the number of intermediate elevations (a 2 segment pipe has 1 intermediate elevation), and whether the entered values are Point to Point or Length Along Pipe.

Figure 2: Intermediate Elevations - Identical Profiles defined with both methods

AFT Fathom will ensure the total length matches the length on the Pipe Model tab. It does so by fixing the length of the final section to the difference between the sum of the lengths in the table and the actual length entered by the user. The end elevation is also fixed by the downstream junction. If the user later modifies the downstream junction elevation, the pipe's final elevation section will be automatically updated.

After the intermediate elevations have been entered, a graph showing the data can be displayed. Below is the profile graph of the pipe described above.

Note that a pipe with intermediate elevations entered can be split into multiple pipes connected with branches at the intermediate elevation points. See Joining and Splitting Pipes for more information.
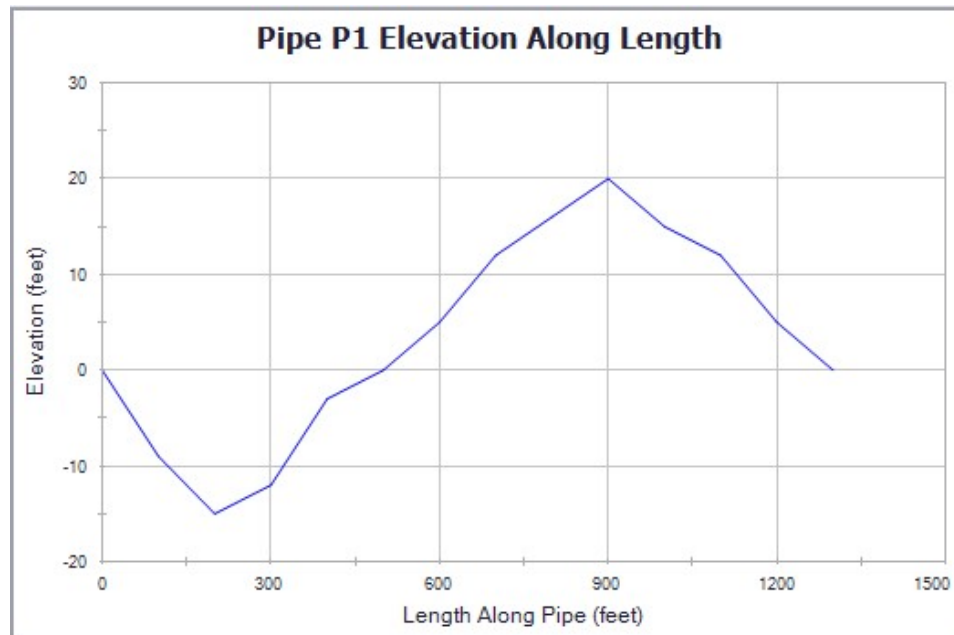
Figure 3: Intermediate Elevation Graph

Related Topics

Add/Remove Pipe Segment

Visual segments can be added to pipes.

To add or remove segments, use the Arrange Menu, Add/Remove 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.

Related Topics

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 Fathom 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 pumps 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

Pressure Drop in Pipes - Detailed Discussion

Depending on the viscosity model and pipe friction model selected, the calculation of pressure drop in a pipe differs.

Darcy-Weisbach Loss Model

AFT Fathom 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.

$$\Delta P = f \frac{L}{D} \frac{1}{2} \rho V^2 \tag{1}$$

This model requires the calculation of a friction factor, as described below.

- **Roughness-Based Methods** - This method calculates the friction factor based on the roughness of the pipe wall. Different equations are used based on flow regime - laminar flow uses the standard laminar relationship, while turbulent flow uses the implicit Colebrook-White equation. In the transition range between laminar and turbulent flow, a linearly interpolated value is used. The default transition Reynold's Numbers can be modified in the Environmental Properties panel.

$$f = \frac{64}{Re} \qquad\qquad Laminar,\ Re < 2300\ (default)$$
$$f = \left[1.14 - 2\log\left(\frac{\epsilon}{D} + \frac{9.35}{Re\sqrt{f}}\right)\right]^{-2} \qquad Turbulent, Re > 4000\ (default) \tag{2}$$

- **Absolute Roughness (default)** - The absolute average roughness height ε is specified directly.
- **Relative Roughness** - The roughness is specified as a ratio ε/D.
- **Hydraulically Smooth** - The ratio ε/D is set equal to zero.

- **Explicit Friction Factor** - The friction factor to be used in the Darcy-Weisbach equation is specified directly.

Frictionless

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

Resistance

The pipe resistance relates head loss to volumetric flow rate. In equation form, the head loss is:

$$\Delta H = RQ^2 \tag{3}$$

Using standard relationships, the friction pressure drop is related to volumetric and mass flow rate as follows:

$$\Delta P = \rho \, g \, R \left(\frac{\dot{m}}{\rho} \right)^2 = \frac{R \, g \, \dot{m}^2}{\rho} \tag{4}$$

Non-Newtonian Pressure Loss

The pressure loss behavior for Non-Newtonian fluids depends on the specific viscosity model selected in Analysis Setup. See Non-Newtonian fluids for more information.

Helical Tubes

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

$$\Delta P_{helical} = \Delta P_{straight} \left[Re \left(\frac{D_{pipe}}{2 \, r_{coil}} \right)^2 \right]^{0.05} \tag{5}$$

Hazen-Williams Method

AFT Fathom also offers the Hazen-Williams method of specifying irrecoverable loss information. The traditional Hazen-Williams formula (Zipparro and Hasen editors, 1993, page 2.7)

$$V = 1.318 \, C_{HW} \, R_h^{0.63} \, S^{0.54} \tag{6}$$

where

- V = Velocity (ft/s)
- C_{HW} = Hazen-Williams factor
- R_h = Hydraulic Radius (cross-sectional area divided by wetted perimeter)
- S = Hydraulic Gradient ($\Delta H/L$)

can be modified to a more convenient head loss form for AFT Fathom:

$$\Delta H = L \left(\frac{V}{0.55 \, C_{HW}} \right)^{1.85} D^{-1.17} \tag{7}$$

where

- V = Velocity (ft/s)
- C_{HW} = Hazen-Williams factor
- D = Inner Diameter in feet (four times R_h for circular pipe)
- L = Length of Pipe (feet)

Also see Hazen Williams NFPA

MIT Equation for Crude Oil

The MIT Equation is appropriate for crude oil and is given by the following equation (Pipe Line Rules of Thumb Handbook):

$$\frac{dP}{L} = \frac{\left(0.241 f s Q^2\right)}{d^5} \tag{8}$$

$$r = \frac{0.0119 Q}{dv} \qquad MIT \; Reynold's \; Number \tag{9}$$

$$f = \begin{cases} 0.00207 \left(\frac{1}{r}\right) & laminar \quad 0.1 \leq r \leq 0.135 \\[2mm] 0.0018 + 0.00662 \left(\frac{1}{r}\right)^{0.355} & turbulent \quad r > 0.4 \end{cases} \tag{10}$$

where:

- dP = pressure drop (psi)
- L = length (miles)
- f = friction factor, MIT
- Q = volumetric flow rate (barrels/day)
- s = specific gravity
- d = inside diameter (inches)
- v = kinematic viscosity (centistokes)

The friction factor calculated by the MIT equations is not the same as the Darcy friction factor. Once the pressure drop (dP) in the pipe is determined, AFT Fathom uses the Darcy equation to back calculate the equivalent Darcy friction factor.

The MIT equation defines the lower bound for the laminar equation as $r=0.1$, but offers no explanation as to the validity of calculations below this point. AFT Fathom continues to use the laminar flow equation for Reynolds numbers below 0.1. The older version of the MIT equations did not have this lower limit, and the results from AFT Fathom 11 agree closely with values from the older equations at these lower Reynolds number values.

For Reynolds numbers in the Indeterminate flow regime, the value for the MIT friction factor is determined by linearly interpolating between the friction factors for the highest laminar Reynolds number and the lowest turbulent Reynolds number.

Miller Turbulent Method

The Miller Turbulent method is appropriate for light hydrocarbons and is given by the following equation (Pipe Line Rules of Thumb Handbook):

$$Q = \frac{d^{2.5}\left(\frac{dP}{L}\right)}{5.9113\left(\frac{\rho}{\rho_{water}}\right)^{0.5}} \left[\log\left(\frac{d^3\left(\frac{\rho}{\rho_{water}}\right)\left(\frac{dP}{L}\right)}{\mu^2}\right) + 4.35\right] \tag{11}$$

where:

- dP = pressure drop (psi)
- L = length (miles)
- Q = volumetric flow rate (barrels/hour)
- ρ = density (lbm/ft3)
- ρ_{water} = density of water (62.3 lb$_m$/ft3)
- d = diameter (inches)
- μ = dynamic viscosity (centipoise)

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

Junctions

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

Contents

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

Table 1: Comparison of junction capabilities

Junction Type	Connecting Pipes	Special Conditions	Restricted Area	Custom Library	Window Format
Area Change	2			Yes	1
Assigned Flow	1	Yes		Yes	1
Assigned Pressure*	1/1-25			Yes	1
Bend/Elbow	2		Yes	Yes	1
Branch	2-25	Yes		Yes	2
Check Valve	2	Yes	Yes	Yes	1
Control Valve	2	Yes		Yes	1
Dead End	1				1
General Component	2	Yes		Yes	1
Heat Exchanger	2	Yes		Yes	1
Jet Pump	2	Yes		Yes	1
Orifice	2		Yes	Yes	1
Pump	2	Yes		Yes	1
Relief Valve	1-2	Yes	Yes	Yes	1
Reservoir	1-25			Yes	2
Screen	2		Yes	Yes	1
Spray Discharge	1-4	Yes	Yes	Yes	2
Tee/Wye**	3				2
Three-Way valve**	3	Yes		Yes	2
Valve	1-2	Yes	Yes	Yes	1
Venturi	2		Yes	Yes	1
Volume Balance	2				1

Weir	2	Yes	Yes	Yes	1

*Assigned Pressure junction connects to only one pipe for static pressure, and up to 25 for stagnation

**Tee/Wye and Three-way valve junctions have a unique format closer to Format #2 than Format #1

Related Topics

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 Fathom 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 pipe's reference positive flow directions specified in the physically correct directions. A good example of this is a pump junction. A pump will add pressure to the system, but it does so only in the direction of positive flow through the connecting pipes. The Pump junction interprets where to add the pressure based on the directions of the connecting pipes.
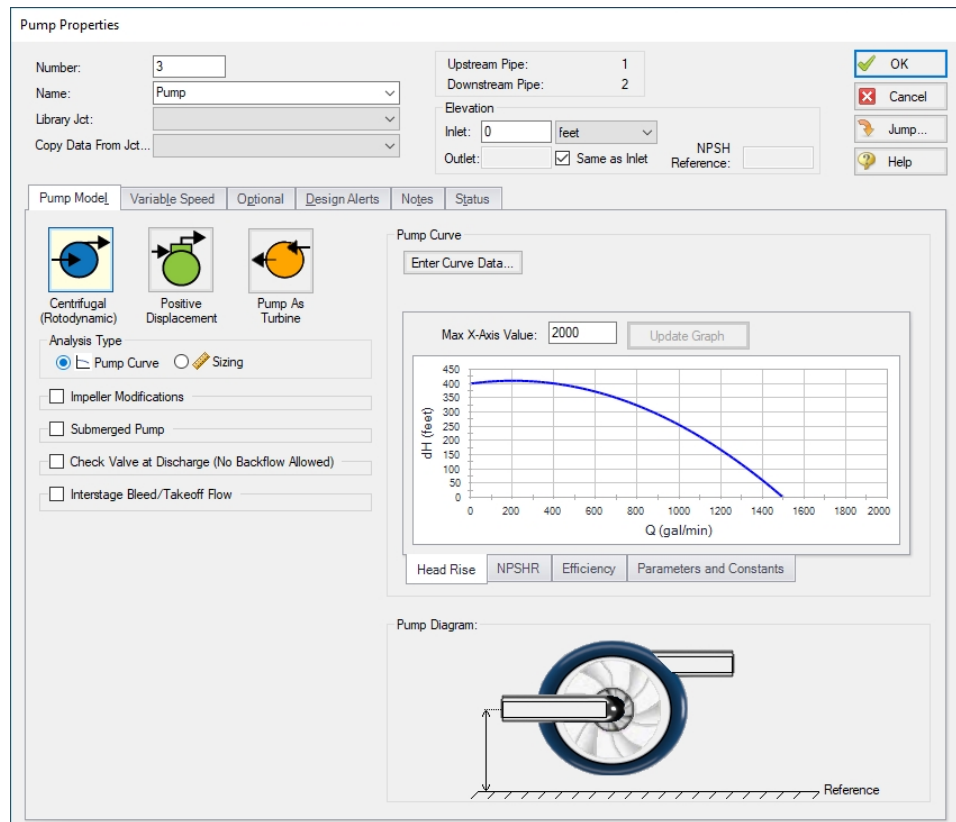
Figure 1: Properties window format #1 (fixed connecting pipe display)

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; the exceptions are the Three-Way Valve and Tee/Wye junctions, which allow 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 Fathom'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

Common Junction Input Parameters

Number

Every junction must have a unique ID number. When new junctions are created, a junction number is automatically allocated.

The junction number can be changed by the user to any unique integer value greater than zero and less than 100,000. Junction numbers are only identifiers and have no effect on the model behavior.

Name

A name can be assigned to each junction 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 junctions' properties to the current junction. Selecting an existing junction will prompt the user on what parameters should be copied over.

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).

If multiple pipes are connected to a single junction, there are multiple elevations that must be defined. By default, all connected pipe centerlines are assumed to be at the same elevation. A different inlet/outlet elevation can be specified by unchecking the "Same as Inlet" box for junctions with only two connecting pipes.

For junctions with more than two pipe connections, these elevations can be specified in the Pipe Depth or Loss Coefficients tab, depending on junction.

Elevations are used to account for pressure changes due to gravity. There can be a pressure changes due to elevation change in a pipe or a junction. Junction elevation changes will have no effect on the flow solution, but will offset the local static pressure by the hydrostatic pressure difference.

A default elevation can be specified in Parameter Options, automatically setting new junctions to the specified elevation.

Jump Feature

Quickly move to the next junction. See Jump Feature.

Related Topics

Optional Input for Junctions

Initial Guesses

A user defined initial guess can be entered for pressure (or head) and temperature (if heat transfer is modeled) 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 (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) 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 hydraulic analysis. For junctions, three types of design factors exist:

- **Pump Junctions** - head rise at the pump is multiplied by the design factor assigned to the pump.
- **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 pumps 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 pump junction will give an extra 10% pressure rise, effectively modeling the pump as having extra capacity than expected. To add a 10% safety margin to the pump 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.

Restricted Area

This is an optional parameter in several junction Properties windows which describes the effective flow area through the junction. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

Specifying Losses

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

Related Topics

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 Fathom 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, Reservoir, Assigned Pressure, and Assigned Flow. Other exceptions are Valves and Relief Valves specified as exit valves, and Orifices specified as an exit.

Related Topics

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 pump). 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 centifugal 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.

Pump curves are generally 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 at a fixed speed, for example, will be the same for water and liquid mercury.

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 by the component manufacturer. By default, AFT software will use the density of the working fluid in the model defined on the Fluid panel, but this can be changed using the Reference Density setting on certain junctions.
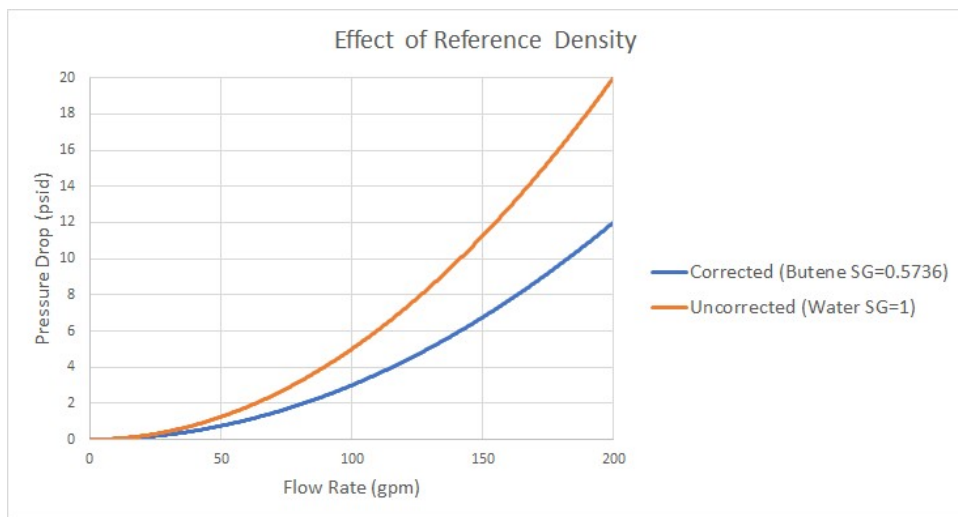
Figure 1: Visual comparison of Pressure Loss Curves for differing densities (head curves would be identical for this component)

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 pump 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

Pumps and Resistance Curves 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

Reference Density - density of the fluid that the manufacturer used during testing to generate the pressure loss vs flow data for the component.

System Fluid Density - density of the fluid defined within the AFT model on the Fluid panel.

If the System Fluid Density significantly differs from the Reference Density, then a correction should be made using the Reference Density option.

- **No Correction (default)** - This selection assumes that the component's pressure loss data was developed by the manufacturer using the same fluid that is defined in the model on the Fluid panel.
- **Water @STP** - The reference density is that of water at standard temperature and pressure. If the Analysis Setup Fluid differs, then the entered curve will be corrected.
- **User Specified** - This directly specifies the reference density. The user should enter the density of the fluid that the manufacturer used during testing to generate the pressure loss vs flow data. If the Analysis Setup Fluid differs, then the entered curve will be corrected.

Related Topics

Equal Percentage Valves

Equal Percentage valves have a valve plug shaped so that each increment in valve percent open (lift) increases the flowrate by a certain percentage of the previous flow.

The relationship between valve lift and orifice size is not linear but logarithmic.

It is expressed mathematically as follows:

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

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

where:

- τ = valve rangeability (ratio of the maximum to minimum controllable flowrate)
- H = valve percent open or lift (0 = closed; 1= fully open)
- $C_{v,max}$ = full open valve loss value

Related Topics

Junction Loss Models

There are a number of loss models available in AFT Fathom to account for frictional losses at the junctions in the system. See the below topics for more information on how losses are calculated for each of the available models.

K Factor Loss Model

AFT Fathom models K-factor 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.

Table 1 lists the sources for the loss models used in AFT Fathom. 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 (see the Resistance Curve topic for flow dependent loss applications).

Table 1: Loss model references

Junction Type	References
Bend	Crane 1988, A-29 Miller 1990, 207-209
Area Change	Crane 1988, A-26 Idelchik 2007, 246 & 254
Orifice	Idelchik 2007, 258-259 & 262
Screen	Idelchik 2007, 598
Tee/Wye	Idelchik 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.

Another lesser known source of loss factor information is Idelchik's Handbook of Hydraulic Resistance (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 role. The reference is voluminous in scope and full of tables, charts and equations for calculating loss factors for almost any pipe arrangement.

Miller's Internal Flow Systems (1990) is another reference offering good general purpose hydraulic data.

Related Topics

Kv and Cv Loss Model

AFT Fathom 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.

Manufacturer Testing in Standard Units

C_v (Coefficient of flow for Valves) is a single parameter which relates fluid density (given in units of specific gravity), pressure drop (in psi), and flow rate (in gallons per minute).

$$C_v = Q\sqrt{\frac{SG}{\Delta P}}$$

One can get a better understanding of the role of C_v by setting the pressure drop (dP) and SG to 1, in which case C_v is equal to flow in gallons per minute. C_v could be considered the flow capacity (in gpm) of a given fluid (i.e. a given SG) for a given pressure drop. Or, one may wish to calculate the pressure drop through a valve, with a given C_v at a given flow rate.

$$\Delta P = SG\left(\frac{Q}{C_v}\right)^2$$

Manufacturer Testing in Metric Units

If the engineer is provided with valve data in metric units, they are also likely to be provided a value for K_v and not C_v.

$$K_v = Q\sqrt{\frac{SG}{\Delta P}}$$

$$K_v = 0.8646\ C_v$$

$$C_v = 1.157\ K_v$$

The only difference is the units used for Q and dP. K_v uses flow units of m^3/hr and pressure drop units of bar.

If one wishes to calculate pressure drop through a valve, with a given K_v, at a given flow rate, the same relationship applies.

$$\Delta P = SG \left(\frac{Q}{K_v} \right)^2$$

Related Topics

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. Pressure loss can be alternatively entered as a head value.

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 man-ufacturer
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

3-K (Darby) Method

The 3-K method described in Darby, 2001, pp. 209-211, is used for high accuracy of predicted loss factors at low Reynolds numbers. The 3-K loss model is available in the valve and bend junctions where there is a dropdown menu to select from standard types of elbows and valves. After selecting an option in this menu, the 3 K's will be displayed at the bottom of the properties window. These values come from the chart in Darby on pages 210-211. The user can also select User Specified in the 3-K (Darby) dropdown menu to input their own values for K_1, K_i, and K_d.

$$K_f = \frac{K_1}{Re} + K_i \left(1 + \frac{K_d}{D_{n,in}^{0.3}} \right)$$

Where:

- K_f = Resistance coefficient
- K_1 = 3-K method constant
- K_i = 3-K method constant
- K_d = 3-K method constant
- Re = Reynolds Number
- $D_{n,in}$ = Inner diameter

Figure 1: Bend Properties window when the 3-K (Darby) method is turned on

Related Topics

Equivalent Length

The use of equivalent lengths is supported by AFT Fathom in the Valve junction, Bend junction, and the pipe Fittings & Losses. The options for bends and valves come from table 23.4.3.1.1 in NFPA 13. If you are not using the NFPA 13 standard then using a user-specified equivalent length is recommended. To enable NFPA reporting for use with equivalent length losses see the NFPA panel topic.

The loss factors in table 23.4.3.1.1 assume that a Hazen-Williams C-Factor of 120 and schedule 40 steel piping are being used. If your reference pipe is defined using a different Hazen-Williams factor or different pipe schedule, then the equivalent length will be adjusted by equation 6.2 (Equation 1 below) on page 103 of Brock, Pat D. 2012.

$$L_{eq} = L_1 \left(\frac{D_{eq}}{D_1} \right)^{4.87} \left(\frac{C_{eq}}{C_1} \right)^{1.85}$$

(1)

Where:

- L_{eq} = new equivalent length
- L_1 = original length
- D_{eq} = diameter of new equivalent length
- D_1 = original diameter
- C_{eq} = C-Factor for new equivalent length
- C_1 = original C-Factor

This equation is intended for taking an equivalent length and adjusting for a new equivalent length based off changes to diameter and C-Factor. This is a general equation, but NFPA table data from table 23.4.3.1.1 is only for schedule 40 steel and a C-Factor of 120. Therefore, the adjusted equivalent length can be written as shown in Equation 2.

$$L_{eq} = L_1 \left(\frac{D_{eq}}{D_{40}} \right)^{4.87} \left(\frac{C_{eq}}{120} \right)^{1.85}$$

(2)

Equation 2 (adjusted from Brock et. al. Equation 6.2) will predict an adjusted equivalent length that coincides with the NFPA 13 equivalent length modifier 23.4.3.1.3.1 and the C-Value Multiplier from table 23.4.3.2.1. Note that if the reference piping is using a friction model other than Hazen-Williams, C_{eq} is assumed to be 120 and the equivalent length will not be adjusted due to friction.

Related Topics

Area Change

The Area Change junction type must always have two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs at the area transition between two pipes, whether expansion or contraction.

The Area Change Properties Window follows the first of the two basic Properties Window formats, displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the two connecting pipes.

The Area Change Properties Window offers two standard types of area change losses: the Conical Transition and the Abrupt Transition.

The critical parameter that influences the magnitude of the area change loss is the area ratio between upstream and downstream pipes. Until you have input the information for the upstream and downstream pipes that allow for flow area determination, the standard Area Change junction loss factor is not calculated. For conical transition geometries, you also need to specify the conical angle. This angle has a range of 0-180 degrees, with a 0 degree angle representing a transition taking place over an infinitely long length, and a 180 degree angle representing an abrupt transition. A depiction of the angle is shown in Figure 1.

If the standard loss factors provided are inadequate for your application, you may specify your own custom loss factor referenced to the basis area you specify.
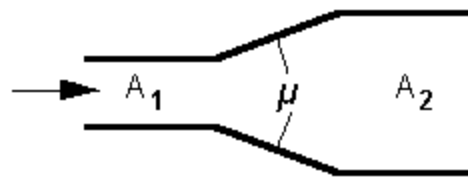
Figure 1: Angle for an area change

Related Topics

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

$$K_{up} = 2.6 \left(\sin \frac{\mu}{2} \right) \left(1 - \frac{A_{up}}{A_{down}} \right)^2 \qquad \mu < 45°$$

$$K_{up} = \left(1 - \frac{A_{up}}{A_{down}} \right)^2 \qquad \mu > 45°$$

The conical contraction correlation (Crane 1988, A-26) is

$$K_{up} = \frac{0.8(\sin \frac{\mu}{2})(1 - \frac{A_{down}}{A_{up}})}{\left(\frac{A_{down}}{A_{up}} \right)^2} \qquad \mu < 45°$$

$$K_{up} = \frac{0.5 \left(1 - \frac{A_{down}}{A_{up}} \right) \sqrt{\sin \frac{\mu}{2}}}{\left(\frac{A_{down}}{A_{up}} \right)^2} \qquad \mu > 45°$$

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

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. 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 volumetric or 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.

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.

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.

Related Topics

Assigned Pressure

The Assigned Pressure junction allows for the specification of a fixed boundary pressure and temperature (if heat transfer is enabled).

This junction is very similar to the Reservoir 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 Reservoirs 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 Reservoir 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.

When heat transfer is modeled, a temperature must also be specified. 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.

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

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.

Standard Loss Types

The Bend Properties window offers several standard types of bend losses: the Smooth Bend, the Standard Elbow (the standard short radius threaded elbow), the Mitre Bend, and the User Specified K-factor. The window shows a schematic of the selected geometry for these three standard loss types. 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 divided by the pipe diameter. At *r/D* values 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 select the radio button for Non-90 Degrees, enter the angle (0-180 deg), and the standard loss factor will be calculated.

Special Loss Types

The Bend also offers specialized loss types: the 3-K (Darby) method and Equivalent Length.

Additional Options

Basis Area for Loss Model - By default, the loss will be referenced to the upstream pipe area. This can be changed using the Basis Area for Loss Model dropdown.

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.

Restricted Flow Area - Optional input that is used only for checking if cavitation exists in the Bend junction.

Related Topics

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 values for 90 degree smooth flanged bends with r/D > 1

r/d	K
1	$20\,f_T$
1.5	$14\,f_T$
2	$12\,f_T$
3	$12\,f_T$
4	$14\,f_T$
6	$17\,f_T$
8	$24\,f_T$
10	$30\,f_T$
12	$34\,f_T$
14	$38\,f_T$
16	$42\,f_T$
20	$50\,f_T$
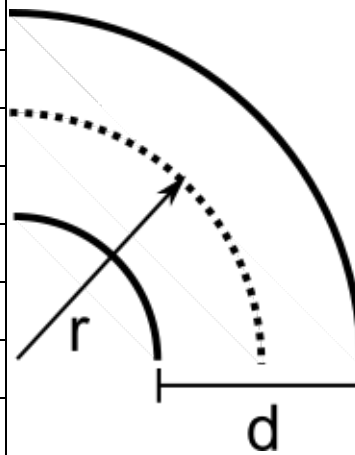
The **standard threaded elbow** is given by

$$K = 30\ f_T$$

where f_T is the turbulent friction factor given in Table 2.

Table 2: Pipe friction factors used for Crane formulas

Nominal Size	Friction Factor f_T
1/2"	0.027
3/4"	0.025
1"	0.023
1 1/4"	0.022
1 1/2"	0.021
2"	0.019
2 1/2",3"	0.018
4"	0.017
5"	0.016
6"	0.015
8"-10"	0.014
12"-16"	0.013
18"-24"	0.012

The **Mitre bend** is given by

$$K = 60\, f_T$$

Non-90 Degree Bends

The two non-90 degree bend loss correlations are for turbulent Reynolds numbers (Crane 1988, A-29).

A **smooth flanged bend** with $r/D \geq 1$ is given by
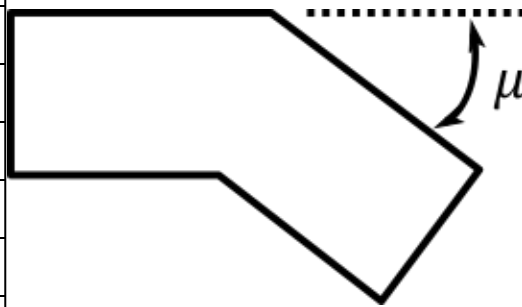
$$K_B = (n-1)\left(0.25\,\pi\, f_T\, \frac{r}{D} + 0.5K\right) + K$$

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.

$$C_{Re} = \frac{K_b^*}{K_b^* - 0.2 \left(C_{Re} \; for \; r/D = 1 \right) + 0.2}$$

where the C_{Re} for *r/D* = 1 is also found from Miller Figure 9.3.

Related Topics

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. The pipe 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 distance of each connecting pipe (if any difference exists) from the Branch elevation can be entered in the pipe table.

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 Fathom's convention, a source of flow into the system is defined as positive, while a flow out of the system (a sink) is defined as negative. Along with the flow rate, you must specify the temperature when heat transfer is modeled.

Transient

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

Related Topics

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 Fathom assumes that the check valve is initially open. If the flow solution indicates that the specified Forward Velocity to Close is not met, AFT Fathom closes the valve and solves the flow distribution again. By default, the Check Valve will close at a velocity of 0. If a negative velocity value is specified, some amount of reverse flow will be allowed to occur before the valve closes.

A positive velocity value means that the check valve will close while there is still forward flow through the valve.

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. When you select Resistance Curve, the Check Valve Properties window makes additional features available. Using these features, you can input Check Valve 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. When a variable loss is specified, AFT Fathom dynamically modifies the loss factor in the Solver to agree with your input. You can choose any of the optional flow and pressure parameters provided; you can also specify the most convenient units.

An optional input in the Check Valve Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the valve. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from the manufacturer or from test data.

Related Topics

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), Pressure Sustaining Valves (PSV), Flow Control Valves (FCV), and Pressure Drop Control Valves (PDCV). 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.

- A **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 and Flow Area can be specified vs. the Open Percentage of the valve. This data does not affect AFT Fathom'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. Tabulated data can be pasted or imported to the table by using the Edit Table menu to the right of the table. A graph of the open percentage, C_v, and flow area data can be displayed by selecting Show Graph to the right of the Open Percentage Data table.

A C_v vs Open Percentage table can be generated quickly for common C_v vs Open Percent profile types by choosing the Create C_v/Open %... button next to the table. This then opens a window in which a curve can be defined using one of the following options:

- **Transferred** - If a table has already been defined on the Optional tab of the Properties window, it can be copied over
- **User Defined** - Data can be entered directly into the table, or imported from existing data
- **Linear** - A linear curve is generated based on the user defined full open C_v and percent increment
- **Equal Percentage** - Generated as an equal percentage curve
- **Pre-Defined** - Pre-defined curves for several common valve C_v vs Open Percent profiles can be chosen and adjusted for a defined full open C_v value based on data from Val-Matic 2009.

Related Topics

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 pump. 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. The pressure added to control can be seen in the Warnings section, or in the Valve Summary table on the Output window.

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 Fathom 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 Fathom 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

because analysis is transient...

Dead End

The Dead End junction must always have only one connecting pipe. This junction type is one way to define flow in a pipe as truly zero. Use of this junction allows AFT Fathom to skip 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

General Component

The General Component junction type is internal to the piping system and requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through equipment that may not be explicitly represented on the Toolbox and to specify loss factors as a function of a flow parameter.

The General Component Properties window follows the first of the two basic Properties window formats, displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

When you select Resistance Curve, the General Component Properties Window makes additional features available. You can specify polynomial constants, fit a curve to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom 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.

Related Topics

Heat Exchanger

The Heat Exchanger junction type is internal to the piping 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.

When you select Resistance Curve, the Heat Exchanger Properties Window makes additional features available. Using these new features you can input heat exchanger 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.

When a variable loss is specified, AFT Fathom 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.

Tube Configuration Model

The Heat Exchanger junction offers a special model for tube side pressure drop. The tube configuration model allows the user to specify tube properties and quantities, scaling and frictional characteristics, multiple passes, blocked tubes and K factors for turnarounds. The pressure drop is then calculated based on this data.

Heat Transfer

When heat transfer is modeled, the Thermal Data tab becomes active. See Heat Exchanger Heat Transfer to read about how this is modeled, and how to specify various configurations.

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

Heat Exchanger Heat Transfer

There are eleven different heat exchanger models available in AFT Fathom.

- 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
    - Enthalpy Stagnation Rise
    - Specified Heat Rate In Constant
    - Specified Heat Rate In vs Flow
    - Temperature Stagnation Rise

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.

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 specified in a heat exchanger properties window, the user must specify the heat transfer area and overall convection coefficient. AFT Fathom will force the two heat exchanger junctions to use the same value.

This capability can also be used to model heat transfer between loops with different fluids (see Fluid Groups).

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:

$$C_h = (\dot{m}C_p)_h$$
$$C_c = (\dot{m}C_p)_c$$
$$C_{min} = smaller \ of \ C_h, C_c$$

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:

$$q_{max} = C_{min} \left(\Delta T_{max} \right)$$

The effectiveness of a heat exchanger is the ratio between the actual heat transfer rate and the maximum possible heat transfer rate:

$$\epsilon = \frac{q}{q_{max}} \quad where \quad q = C_h \left(T_{h1} - T_{h2} \right) = C_c \left(T_{c2} - T_{c1} \right)$$

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:

$$q = \epsilon \, C_{min} \left(T_{h1} - T_{c1} \right)$$

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:

$$C = \frac{C_{min}}{C_{max}}$$

$$NTU = N = \frac{UA}{C_{min}}$$

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:

$$Parallel\ Flow \quad \epsilon = \frac{1 - e^{-N(1+C)}}{1 + C}$$

$$Counterflow \quad \epsilon = \frac{1 - e^{-N(1-C)}}{1 - Ce^{-N(1-C)}}$$

$$Crossflow,\ Both\ Fluids\ Unmixed \quad \epsilon = 1 - e^{\left(\frac{N^{-0.22}}{C} \right) \left(e^{-CN^{0.78}} - 1 \right)}$$

$$Crossflow,\ Fluid\ with\ C_{max}\ mixed \quad \epsilon = \frac{1}{C} \left(1 - e^{-C(1-e^{-N})} \right)$$

$$Crossflow,\ Fluid\ with\ C_{min}\ mixed \quad \epsilon = 1 - e^{\frac{1}{C}(1-e^{-CN})}$$

$$Shell\ \&\ Tube \qquad \epsilon = 2\left\{1 + C + \sqrt{1+C^2}\left(\frac{1+e^{-N\sqrt{1+C^2}}}{1-e^{-N\sqrt{1+C^2}}}\right)\right\}^{-1}$$

Related Topics

Jet Pump

The Jet Pump junction is internal to the system and requires two connecting pipes, representing the suction and the discharge. The supply flow is specified explicitly and modeled internally in the junction type.

The Jet Pump 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.

The Jet Pump junction is useful for modeling jet pumps, also known as eductors. Jet pumps are devices which use a high pressure injected fluid to entrain a lower pressure fluid and induce motion, thus pumping the lower pressure fluid. The high pressure fluid is referred to as the supply fluid, the low pressure fluid is the suction fluid, and the combined mixture is the discharge fluid. If the supply and suction fluid are different, the variable physical properties model should be used (selected in Analysis Setup on the Heat Transfer/Variable Fluids panel). In such a case, the data for the jet pump will be best entered on a mass flow rate basis, which includes the effect due to density.

To describe the performance characteristics, data must be entered for the pressure ratios at maximum and minimum flow, as well as the flow ratio at maximum flow. Additionally, the supply fluid pressure and flow rates are required.

If the pipe flow model includes heat transfer, a supply temperature is required.

More information on Jet Pump calculations can be found in Karassik, et. al, 1986 on pages 4.2-4.9.

Related Topics

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 irre-

coverable 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 an exit 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, then specify the exit pressure and, if a heat transfer fluid model is used, the exit temperature.

Orifice Types

AFT Fathom 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 Fathom provides, you may 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 specify polynomial constants, fit a curve to available data, or use interpolated x-y data. When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) to agree with the solution. Here you can specify any of the available optional flow and pressure parameters using your preferred units.

An optional input in the Orifice Properties window is the Restricted Flow Area. This parameter describes the effective area restriction in the orifice. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

Orifice Loss Model

Sharp-Edged Orifice with Identical 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):

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}}\right)^2 \left[0.707\left(1 - \frac{A_{orifice}}{A_{up}}\right)^{0.375} + \left(1 - \frac{A_{orifice}}{A_{down}}\right)\right]^2$$
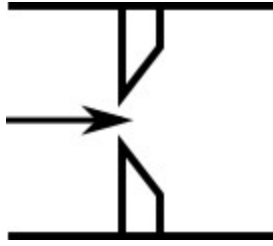
Figure 1: Sharp-edged orifice (in this case $A_{up} = A_{down}$)

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

$$K' = 0.03 + 0.47 * 10^{-7.7\, r_{edge}/D_{orifice}}$$
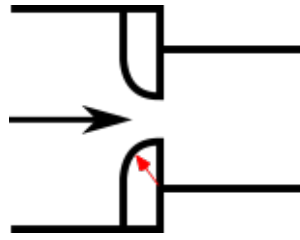
Figure 2: Round-edged orifice with different upstream and downstream pipe areas

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

$$K' = 0.03 + 0.47 * 10^{-7.7\, r_{edge}/D_{orifice}}$$
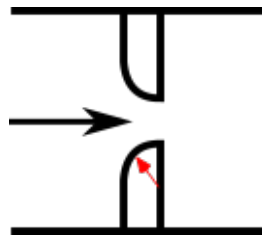
Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

For other orifice configurations, see Chapter 4 of Idelchik's Handbook of Hydraulic Resistance (2007).

Orifice Discharge Coefficient

The relationship between orifice C_D and K is given by Crane A-20:

$$K = \frac{(1 - \beta^2)(1 - \beta^4)}{C_D^2 \beta^4}$$

where β is the diameter ratio of the orifice diameter to the upstream pipe diameter, or

$$\beta = \frac{d_{orifice}}{d_{up}}$$

Related Topics

Pump

The Pump junction allows you to model the pressure added to a system by a pump, fan, or blowers (ignoring compression effects). Because all pumps are different, AFT Fathom does not provide any standard pump types. Pressure and flow data that describes the pump must be obtained from the manufacturer or from test data. To reduce repeated data entry, frequently-used pumps can be added to the Component library.

Common Input Information

Like all junctions, pump junctions require a junction number, junction name, and inlet and outlet elevations. Similar to other junctions, pumps can have Design Alerts and Notes applied to them. They require two connecting pipes, unless the Submerged Pump option is enabled. Note that the direction of the connected pipes define what side of the pump is suction and which is discharge.

NPSH Reference Elevation

The NPSH Reference Elevation is the location where the pump manufacturer measured the NPSHR/A data, referenced to the same common elevation datum used by the rest of the model elevations. For most pumps this represents the centerline of the inlet suction flange. This input is required if the junction has different inlet and outlet elevations.

Pump Model

This tab requires the selection and definition of what type of pump the junction represents. There are three models available:

- **Centrifugal (Rotodynamic)** - Represents a typical centrifugal pump where the head added to the fluid is related to the flow through the pump via a pump curve.
    - **Interstage Bleed/Takeoff Flow** - Represents a multiple stage pump that has some amount of flow removed between the upper and lower stages. Centrifugal (Rotodynamic) must be selected to enable this model.
- **Positive Displacement** - Represents a steady state approximation of a positive displacement pump. The flow is fixed to a constant value, and whatever head is necessary to obtain that flow will be supplied.
- **Pump As Turbine** - Represents a pump as a turbine which allows the pump to produce electrical power when the pump sees reverse flow.

Variable Speed

This tab allows the speed of the pump to vary either by a fixed amount, or dynamically. These options are only available for a centrifugal pump. A minimum speed can be entered to limit how much the speed is varied.

- **Fixed Speed (%)** - Directly modify the pump curve according to the Affinity Laws. The Pump Curve entered in Pump Configuration is always assumed to be the 100% speed curve.
- **Controlled Pump (Variable Speed)** - Dynamically change the speed of the pump to meet a specified control setpoint. Instead of allowing the flow or head to vary with a fixed speed, the speed of the pump (and thus Pump Curve, according to the Affinity Laws) varies to meet a fixed flow or head. If modeling a suction or discharge pressure, control can optionally be enforced only if the value is above/below the setpoint.

Thermal

If heat transfer modeling is enabled, the Thermal tab will appear and heat added to the fluid by the pump due to inefficiency can be modeled. Three options are available:

- **Percent of Ideal Efficiency** - Enter the percentage of ideal energy required by the pump that goes into heat addition. Typically, this would equal the actual efficiency subtracted from 100%. For example, if the pump was 70% efficient, the entry here would be 30%.
- **Percent of Inefficiency** - Enter the percentage of the lost energy (i.e., the inefficiency) that goes into fluid heating. Typically, this would be 100%, which says that all inefficiency goes into heat.
- **User Specified Heat Rate** - Directly enter a heat load which goes into the fluid.

Slurry De-Rating

([SSL Module](#) Only)

Pump head degradation occurs when pumping solids. This is known as de-rating. See the [Slurry Pump De-rating](#) topic for more information.

Transient

([XTS Module](#) Only)

The pump junction allows modeling of transients to occur in the following pump parameters:

- Speed (if using a pump curve) for both Pumps and [Pump as Turbine](#)
- Flow rate (if modeling a fixed flow rate)
- Pressure/head (if modeling a fixed pressure or head rise)
- Control setpoint
- [Load Rejection](#) for Pump as Turbine

For more information on transient data, including event transients, see [Junction Transient Data](#).

Optional

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

Also present are several pump junction specific options:

- **Special Condition**
    - **None** - The pump operates normally.
    - **Pump Off No Flow** - The pump is shut off and blocked. This is identical to a closed valve in Fathom.
    - **Pump Off With Flow Through** - The pump is shut off but allows flow to continue through. This is identical to a lossless connection in Fathom.
- **Number of Pumps at This Location** - Several pumps can be represented by one pump junction. This simplifies the model by reducing the number of pipes and junctions present, as well as automatically calculating the resulting composite pump curve based on the user-defined curve. When this option is selected the pump summary will show the total flow/head from the combined pumps.
- **Use [Viscosity Correction](#)** - Pump performance changes if the system fluid viscosity is different than the pump test fluid viscosity. Viscosity Corrections allow the user to account for these changes.

- **Additional Efficiency Data** - Power and efficiency information for each stage in a VFD-Motor-Pump assembly.

Related Topics

Centrifugal (Rotodynamic) Pump

Analysis Type

This topic pertains to the Pump Curve Analysis Option which represents the real behavior of a centrifugal pump. For Sizing Analyses, see Pump Sizing.

Pump Curve

A Pump Curve is required to model a centrifugal pump. This is the typical Head or Pressure vs. Flow information that will be available from any pump manufacturer. Certain features also require the definition of an NPSHR curve, and Power or Efficiency curves.

To enter curves, click Enter Curve Data. Once this data has been defined, previews of the applied curves can be seen here.

Trimmed Impeller

These values allow representation of different impeller trims without the need to input new curves.

- **Ratio as Percent** - This option assumes that the pump curve represents 100% trim. Entering different values adjusts the pump curve with the Affinity Laws.
- **Actual Impeller** - Only available if Multiple Configurations are defined. In this case, a dimensional value for impeller size can be entered here.

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See Submerged Pump.

Check Valve at Discharge (No Backflow Allowed)

Model a check valve that can be considered part of the pump itself. If using the XTS Module, a check valve junction will not close until the time step after the reverse flow occurs. However, when used as part of the pump junction it will never allow backflow at any time step.

- **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 reverse flow through the pump.
- **Delta Head to Re-Open** - The differential head or pressure (upstream minus downstream) required to re-open the check valve.
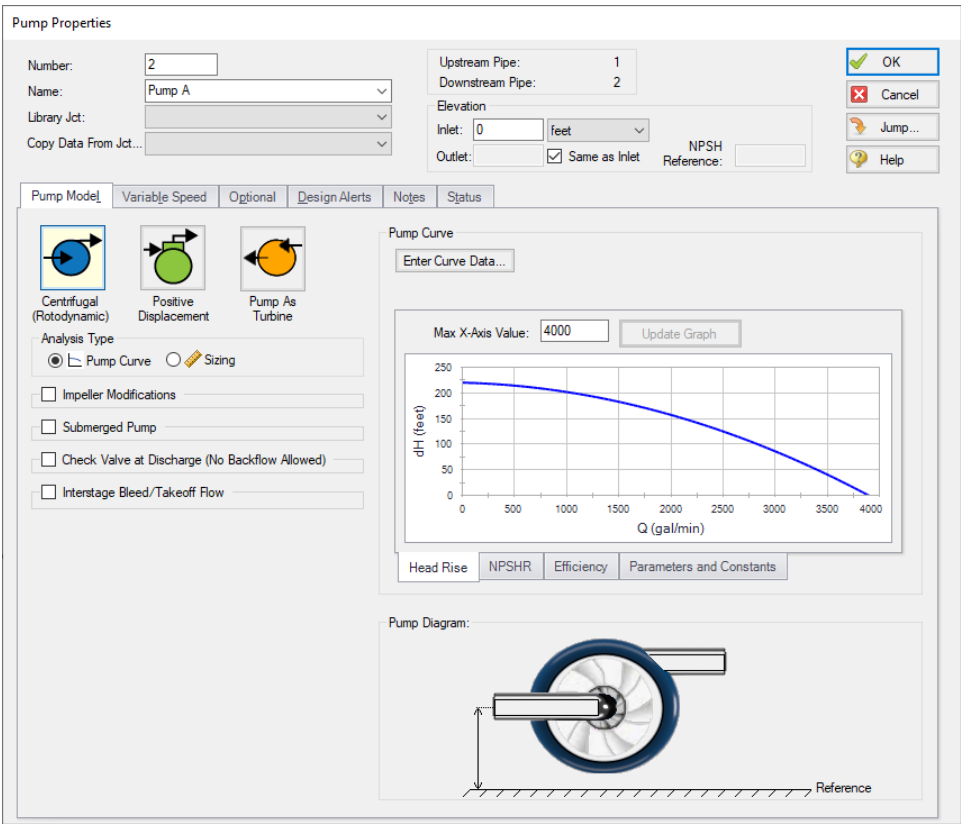
Figure 1: Centrifugal model with a pump curve defined

Interstage Bleed/Takeoff Flow

Represents a multiple stage pump that has some amount of flow removed between the upper and lower stages. See Interstage Bleed/Takeoff Flow.

Related Topics

Pump Configuration

The pump configuration window is used to define the specific operation of a centrifugal pump. It is opened with the Enter Curve Data button from the Pump Properties Pump Model tab when Centrifugal is selected.

Figure 1: Enter Curve Data

Configuration Method

By default a pump is defined as a Simple configuration. This indicates that the pump junction can represent only one pump - defined by the curves in pump configuration.

If the Configuration Method is changed to Multiple, additional input becomes available on the right of the configuration window. This allows defining different operating speeds or impeller diameters for this pump, along with those configuration's specific curves. See a detailed description in Multiple Configurations.

Pump Data

The first tab of the pump configuration controls the pump's most basic behavior - the pump curves.

Figure 2: Pump Data

A Pump can have three curves defined:

- **Flow vs. Head/Pressure** - Required. This is what is commonly referred to as the "pump curve" and defines how much head a pump can add to a fluid at any given flow rate.

- **Flow vs. NPSHR** - Optional. The NPSHR varies with flow and can be specified for a pump. Without specifying this curve, it is impossible to determine if cavitation may exist within the pump.

Note: NPSHR calculations require a reference elevation, which is defined in the Pump Properties window. For most pumps, the NPSH Reference elevation is the centerline of the suction of the pump. However, this should be checked with the pump manufacturer to determine where the NPSH curve is referenced.

- **Flow vs. Efficiency/Power** - Optional. The efficiency or power curve can be used to calculate the Best Efficiency Point (BEP), which is used for a variety of purposes. The definition of this curve also allows for the calculation of the actual power used or efficiency of the pump at its operating point. Without this curve, pump efficiency is assumed to be 100%.

After entering the raw values into the table, the data needs to be curve fit.

If Mass Flow, Pressure, or Power are used in the curve definitions, Reference Density becomes an additional required input.

Configuration Data

This tab controls some additional behavior or input for a pump junction. Entering or changing values on this tab is optional.

Figure 3: Configuration Data

- **Best Efficiency Point (BEP)** - The Best Efficiency Point (BEP) by default is calculated from the entered efficiency or power curve. This is generally the preferred option as it best relates to the true meaning of BEP. A User Specified value can be entered if no efficiency or power curve is available, or if it is desired to force the BEP to a certain flow.

- **End Of Curve Flow Rate (optional)** - This value is used to create a warning in the output if it is exceeded. It does not affect the solution - the model will behave identically however this value is defined.

- **NPSHR Affinity Exponent At Operating Speed** - Pump curves - flow, head, and power - generally follow the affinity laws. However, NPSHR does not always follow a standard relationship. The default is to assume the NPSHR varies with the square of the operating speed. If this is not the case, the value can be adjusted here.

$$NPSHR_{adjusted} = NPSHR_{original} * (Speed\ Ratio)^{Affinity\ Exponent}$$

$$Speed\ Ratio = \frac{Speed\ adjusted}{Speed\ original}$$

- **Preferred Operating Region (POR)** - The region where the pump operates most effectively. Percent of BEP values falling outside of the defined range will generate a warning in the model output. This range can be taken from HI 9.6.3-2017 based on the specific speed of the pump, from ANSI/API 610 (2010), or can be defined by the user.
- **Allowable Operating Region (AOR)** - When an AOR is defined a warning will be created in the output when the user specified flow range is not adhered to. This is usually provided by the manufacturer.

Graphs

The following tabs display the curves that have been fit to the given data. If multiple curves are defined, the Composite Graph tab becomes available. The other graph tabs show the individual curves, along with points representing the raw data. It is a good idea to check that these graphs to confirm the curve fit is reasonable for any potential operating region.
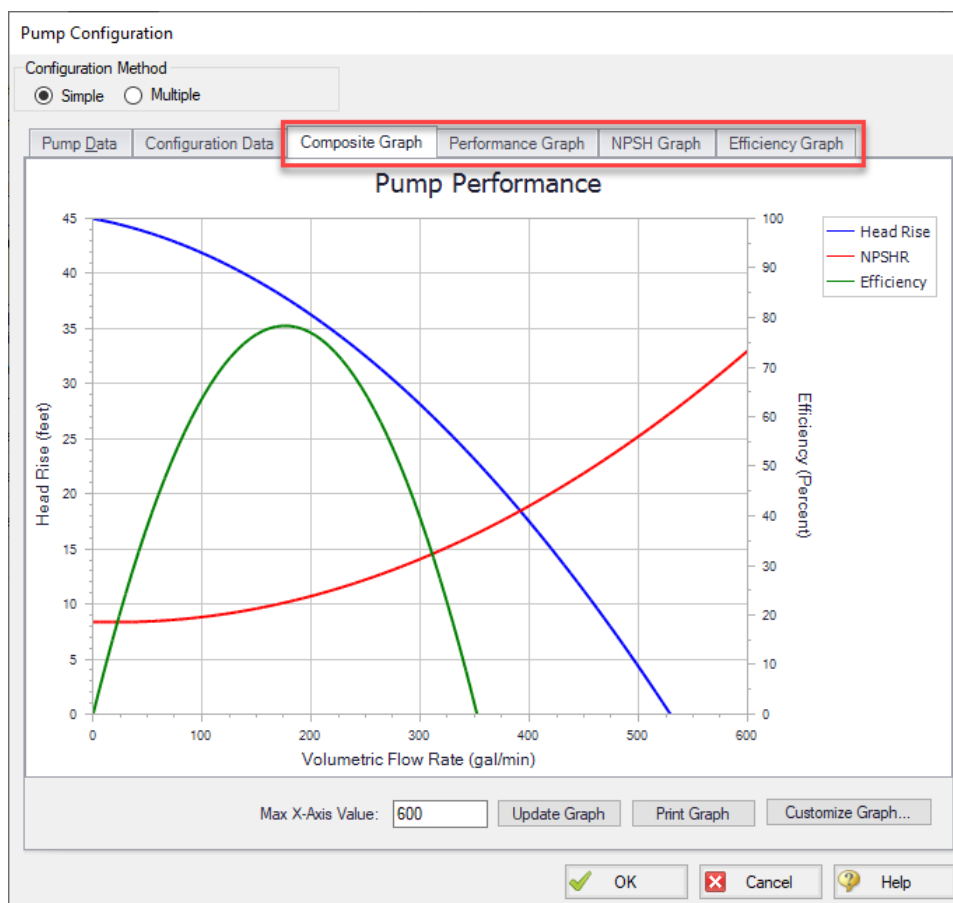
Figure 4: Composite Graph

Related Topics

Multiple Configurations

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

Multiple configurations also allow for an analysis of [pump reliability factors](#).

Defining Multiple Configurations

When opening the [pump configuration](#), change the Configuration Method from Simple to Multiple. When doing so, an additional user input panel appears on the right hand side of the window.

Figure 1: Multiple Configuration Method Selected

In order to make use of the multiple configuration feature, an RPM and Impeller Diameter under Configuration Information should be filled in. A Description can optionally be added. After modifying any information that affects the definition of this configuration the Update Configuration box turns yellow:

Figure 2: Update Configuration

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

Figure 3: Current Pump Configurations

To create an additional configuration, click Create.

Figure 4: Creating a new Configuration

The pump curves must be created and curve fit, as with a single configuration pump. After completing this and clicking Update Configuration Now, the new configuration will be created and shown in the Pump 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 Pump Model tab:

Figure 6: Selection of a Configuration

Related Topics

Pump Reliability Factors

Reliability factors from Bloch & Geitner (1994) can be used to help assess the long-term reliability of pumps.

Setting up a Pump to Calculate Reliability

In order to calculate overall reliability, three pieces of pump data are needed:

- Efficiency vs. Flow
- Speed in RPM
- Actual, maximum, and minimum impeller diameters

Figure 1 shows the Pump Configuration window which is opened from the Pump Specifications window. In the table at the left of Figure 1, you can see where efficiency data is entered. The best efficiency point (BEP) is computed based on this data. If you would rather enter the BEP directly you can do so on the second tab Configuration Data. The BEP data is used to compute the reliability factor based on BEP. AFT Fathom calls it Reliability Factor BEP or **Reliability BEP**.

Figure 1: Pump Configuration window opened from Pump Specifications window

At the top left of Figure 1 you can see options for Configuration Method. To calculate the reliability factors for speed and impeller you need to use the Multiple Configurations option as shown in Figure 1. When you choose this, the window expands to the right and you can see the areas at the right of Figure 1 where RPM and impeller size are specified. Each RPM and impeller size combination is referred to as a pump configuration. To make the reliability factor calculations work correctly here is the data that must be entered for the pump:

- One configuration for the *maximum* impeller diameter must be entered
- One configuration for the *minimum* impeller diameter must be entered
- One configuration for the *maximum* speed in RPM must be entered

Using these three configurations, AFT Fathom searches all configurations for the pump and finds the maximum and minimum diameters to use for the impeller diameter based reliability factor which we call **Reliability Factor Diameter**. AFT Fathom also searches for the highest RPM value and assumes that as the design value. This value is then used for the **Reliability Factor Speed** calculation.

The **Reliability Factor Total** is simply the product of the BEP, Speed, and Diameter reliability factors.

Setting Pump Reliability in Output

The reliability factors can be selected for viewing using the Output Control window. Choose Output Control from the Tools menu or click the Output Control button on the toolbar.

Choose the Pump button at the top of the Display Parameters tab, and from the list on the left select the reliability parameters you want to see in your output. Add them to the list on the right. When you run your model you will see the reliability factor output as in Figure 3.
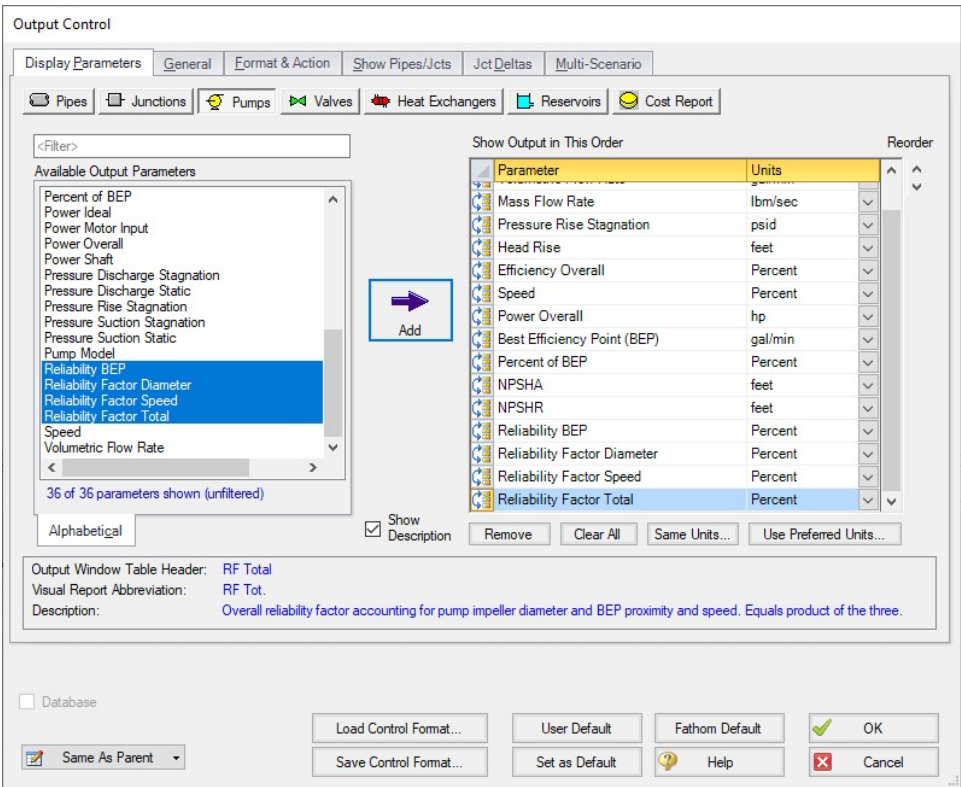
Figure 2: Output Control Window with Reliability Factors added

Figure 3: Reliability Factors in the Pump Summary

Related Topics

Pump Sizing

The Centrifugal (Rotodynamic) Pump Model offers the option of Sizing a pump. By entering a Pump Sizing Parameter, the corresponding operating conditions are calculated and displayed in the Pump Summary section of the Output window . By inputting a flow at a pump, determining the pump size requirements is straightforward. The Walk Through Examples have an example of using AFT Fathom to size a pump.

Pump Sizing with a Flow Control Valve

Using the sizing feature on a pump (fixing the flow rate) with a Flow Control Valve (which also fixes flow rate) downstream results in a Reference Pressure error - the system no longer has a unique solution. While the pump is being sized, the control valve must be temporarily defined as a Constant Pressure Drop Control Valve to avoid these errors. Read more about this in Role of Pressure Junctions - Detailed Discussion.

Suggest Pump & Intelliquip™

Once you have sized a pump, you can use the Suggest Pump feature which is opened from the Library menu > Suggest Pump or the Output Toolbar to find an actual pump that meets the requirements. The Suggested Pumps window allows you to specify search criteria and to select the libraries you want to search. Search criteria includes pump head, NPSH, and BEP proximity.

If you or a colleague has previously saved various manufacturer pump models and performance curves to a junction library that you have connected in the Library Manager, you can search this library to see if any saved pump curves meet the requirements. This is useful if an Enterprise-Wide Network Library System is set up for your company.

Click Search once the criteria and Source Library are defined (see below for instructions on using the Internet Library Source). Pumps that meet the criteria will be shown in the table below. The table can be sorted by proximity to operating head, BEP, or alphabetically. The Show Pump Curve button allows for quick viewing of the composite curve for a selected pump. The operating point is overlaid on the graph such that you can determine if the pump is a good fit. If the performance curve is determined to be appropriate for the model, the Apply Pump Data button can be used to open the Apply Pump Data window where one or more pump junctions can be updated to use the performance curve.

Intelliquip™ - In addition, the Internet Library Source allows you to pass operating point data to Internet hosted pump supplier websites which use Intelliquip™.

1. Select the Internet radio button under Source Library, and select Intelliquip from the drop-down menu.
2. Click Search to open the Intelliquip Search window.
3. Select Display Units and the Frequency of the electrical grid that the pump will operate on (60 Hz or 50 Hz).

4. Click Open Web Browser to be taken to the AFT Intelliquip landing page.

5. Select a pump supplier website.

6. Browse the catalog and adjust the search filters. Each supplier's website may operate differently and might require different data inputs to be adjusted.

7. Once a pump model is found to be appropriate, look for a download button and download the *.iqx file.

8. Return to the Intelliquip Search window in Fathom, click the radio button for Step 2, and click Open File(s).

9. Select the *.iqx file for the pump. The pump will now appear in the table.

10. Select the pump in the table, click Apply Pump Data, and update the desired junctions in the model. The Add to Library option allows you to save the pump curve to a connected library for future use.

For an example of pump sizing in Fathom, see the Sizing a Pump example.

Related Topics

Interstage Bleed/Takeoff Flow

Some multiple stage pumps can have flow removed at some intermediate point. For such pumps, the manufacturer provides two pump curves:

- **Overall Pump Curve** - The total dynamic head from the suction of the first stage to the discharge of the last stage *when the takeoff flow is zero*.

- **Takeoff Pump Curve** - The total dynamic head of the stages *before* the takeoff point.

Note: The **Overall Pump Curve** is an *effective* curve that describes the operation of all pump stages together. This curve cannot be directly used to determine the overall head when the takeoff flow is anything other than zero.

Defining the Model

First, place two pump junctions onto the Workspace, representing the upstream and down-stream stages of the multistage pump.
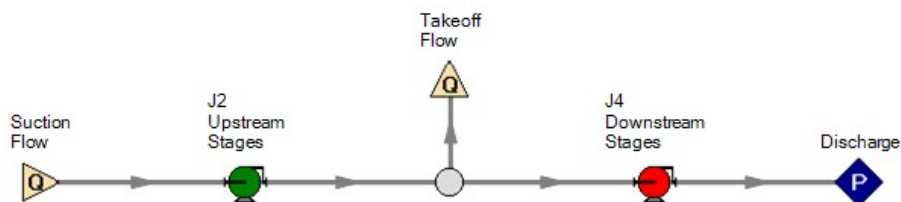
Figure 1: Example Workspace layout - color for emphasis only

Both pumps need to be defined as "Interstage Bleed/Takeoff Flow" as shown below. The "Upstream Stages" pump should have the Takeoff Curve entered. The "Downstream Stages" pump should have the Overall Pump Curve entered.

Keep in mind that the pipes connecting the two pumps **do not exist in reality**. For this reason, it is reasonable to make these pipes have as little impact as possible. This can be done by making them short and/or frictionless. Otherwise, there will be some nominal frictional head loss in the section where effectively none would exist in the real system.

Additionally, the two pumps need to be "connected" by setting the junction of the corresponding pump junction in this area.
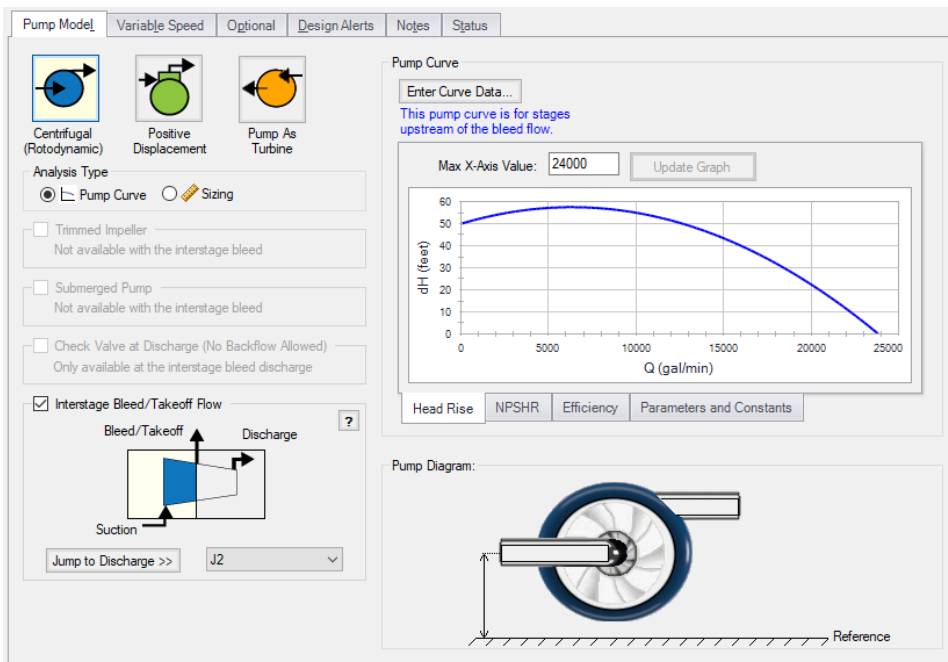
Figure 2: Interstage Configuration

Methodology

As can be seen in Figure 3, the Suction Flow is always equal to the sum of the Takeoff and Discharge Flows. Also important to note is that the Suction Flow is always equal to the discharge flow of the upstream stages - that is, adjusting the takeoff flow does not directly affect the behavior of the upstream stages.

From this, we can determine the head that is added by the upstream stages by examining the Takeoff Pump Curve at the Suction Flow. This is shown in Figure 4 as Upstream Stages Head.

Next, we need to determine the head contributed by the Downstream Stages. However, we do not know the pump curve for these stages! By subtracting the Takeoff Pump Curve from the Overall Pump Curve, we can determine the Downstream Stages Pump Curve. Knowing the Discharge Flow (Suction Flow - Takeoff Flow), we can now look up the Downstream Stages Head, as shown in Figure 4.

Adding these two values brings us to the Overall Effective head across the entire pump at the given Discharge Flow. This value will be smaller than the Overall Pump Curve value due to the head lost to the Takeoff Flow.

Similarly, adding the overall power calculated at each of the pump junctions will give the ideal Overall Effective power for the entire pump.
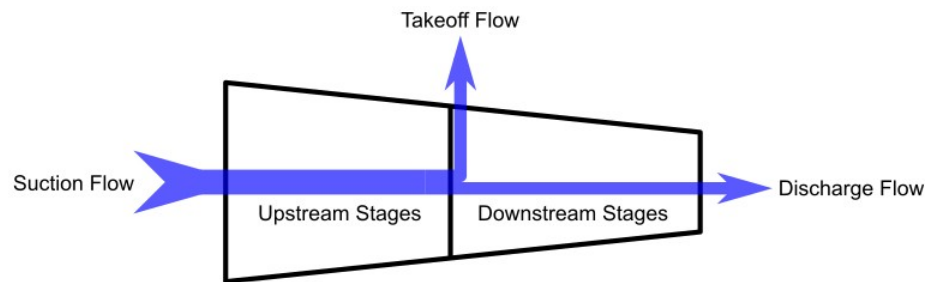
Figure 3: Interstage Diagram
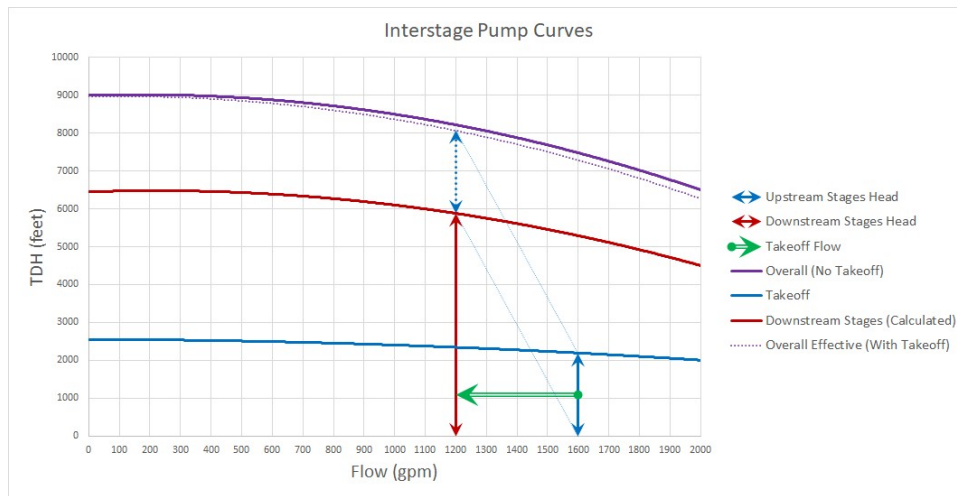
Figure 4: Interstage Curves

Related Topics

Viscosity Correction

Most pump curves are developed with pure water; however, pumping a fluid more viscous than water will result in a reduction of pump performance - even if the fluid reference density is the same as water. The Viscosity Correction feature employs the methods described in ANSI/HI 9.6.7-2021 Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance to correct for losses in performance due to viscosity.

Viscosity Corrections should only be used if the pump curve entered into the Pump Properties window is a water curve, since this is the only situation covered by ANSI/HI 9.6.7-2021. Viscosity Corrections will effectively transform a water pump curve into a viscous pump curve. If the manufacturer provides a pump curve for your working fluid at your desired viscosity, then it should be entered directly and the Viscosity Correction feature should remain disabled, otherwise the effects would be double counted. If the manufacturer provides a pump curve based on your same working fluid, but at a different viscosity, then the Viscosity Correction

feature should still remain disabled because unfortunately that case falls outside the scope of the ANSI/HI 9.6.7-2021 standard.

Correction factors referred to as C_H (for head rise), C_Q (for volumetric flow), and C_E (for efficiency) are defined and can be displayed in the Pump Summary section of the Output:

$$C_H = \frac{H_{viscous}}{H_{water}} \qquad C_Q = \frac{Q_{viscous}}{Q_{water}} \qquad C_E = \frac{\eta_{viscous}}{\eta_{water}}$$

To calculate the correction factors, the water BEP (head and flow at BEP) and rated pump speed are required.

- **Assume Pump is Operating at BEP** - This option is useful when the pump is in Sizing mode, and will assume that the head rise and flowrate calculated by the model are at BEP. This option is also useful when a water pump curve is entered but no efficiency data is present.
- **Use Water BEP**
    - **From Pump Configuration** - The water BEP flowrate can be determined from the pump curve data entered on the Pump Configuration window. This option will be disabled if efficiency data is not present.
    - **User Specified** - Alternatively, the user can directly specify the water BEP flowrate.

Scope of Correction

It is important to keep in mind that, like all mathematical models, this standard is not universally applicable. Below are the primary requirements:

- Single or multistage centrifugal pump
- Essentially radial discharge with specific speed under 3000 (US units)
- Newtonian Fluid
- Kinematic viscosity between 1 and 4000 cSt

For detailed information regarding these limitations and the application of the standard, please consult ANSI/HI 9.6.7-2021.

Related Topics

Positive Displacement Pump

Positive Displacement pumps can be used to provide a fixed flow for steady state analysis.

Parameter

- **Volumetric Flow Rate** - Fix the volumetric flow rate through the pump.
- **Mass Flow Rate** - Fix the mass flow rate through the pump.

Fixed Condition

- **Fixed Flow Rate** - The value and unit of the fixed flow amount.
- **Nominal Efficiency** - A fixed efficiency, useful to calculate power requirements. This is an optional input.
- **Nominal NPSHR** - Net Positive Suction Head Required for the pump at the specified flow. This is an optional input.

Acceleration Head/Pressure Parameters

Acceleration head is the energy required to change the velocity of the fluid in the system from zero to some non-zero velocity. This requires additional energy beyond the amount required simply to overcome the frictional losses in a system. This is reflected when starting a centrifugal pump - the peak power requirement is higher than during steady operation. Similarly, positive displacement pumps are constantly accelerating and decelerating the fluid in the suction line. The value for acceleration head can be added to the Pump Summary via Output Control, and is visible in the Applied Standards tab.

- **Type of Pump (C)** - The type of Positive Displacement Pump, for the purposes of calculated the acceleration head pump type coefficient, as described below. If no predefined types are applicable, a user specified value must be entered.
- **K** - The compressibility factor of the fluid, as described below.
- **Pump Speed** - The nominal RPM of the Positive Displacement Pump.

This suction line acceleration head can by calculated with the method defined in ANSI/HI 6.1-6.5-2015. This method requires the user definition of pump speed, pump type coefficient, and a fluid compressibility factor. Suction line length and fluid velocity are also required and are calculated by AFT Fathom.

$$h_{acc} = \frac{lvnC}{Kg}$$

- h_{acc} - Acceleration head (ft)
- l - Length of suction line (ft)
- v - Fluid velocity in suction line (ft/s)
- n - Pump speed (rpm)
- C - Coefficient determined by type of reciprocal pump:
  - Simplex (Single-acting) = 0.628
  - Simplex (Double-acting) = 0.200
  - Duplex (Single-acting) = 0.200
  - Duplex (Double-acting) = 0.115
  - Triplex = 0.066
  - Quintuplex = 0.040
  - Septuplex = 0.028
  - Nonuplex = 0.022
- K - Factor representing relative compressibility of fluid. Some common compressibility factors are provided.
  - Hot Water = 1.4
  - Hot Oil = 2.5
- g - Gravitational constant (ft/sec^2)

AFT Fathom calculates the total acceleration head by summing the calculated acceleration head for each individual pipe. The solver will attempt to automatically determine the pipes in the suction line by tracing the flow upstream of the pump back to an Assigned Pressure or Reservoir junction. If at any point upstream of the pump there is more than one potential path (for example, two inflows merging together), this calculation will not proceed because the suction line is indeterminant. However, the flow can split into multiple outflows. As long as there is one continuous and unique inflow path to the pump, acceleration head can by calculated.

Note: Though the ANSI/HI 6.1-6.5-2015 standard states that the NPSHA should be corrected to account for acceleration head, Fathom will not adjust the NPSHA shown in the output. The user should subtract the acceleration head from the displayed NPSHA to comply with the standard.

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See Submerged Pump.
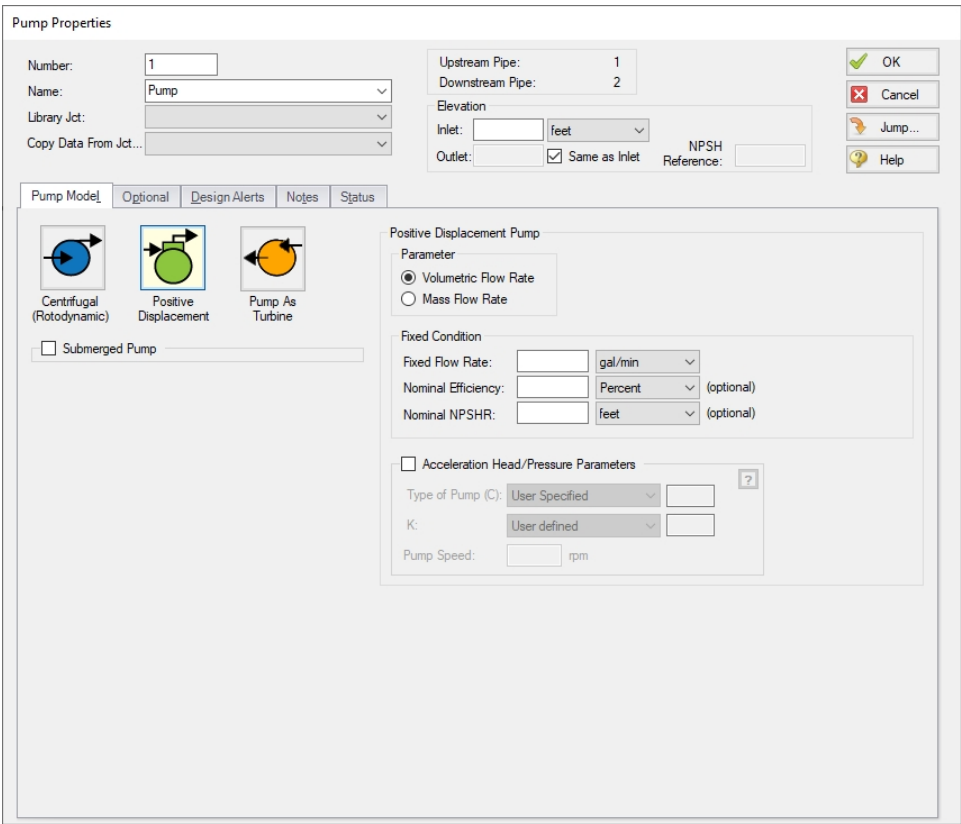
Figure 1: Positive Displacement Pump Model

Related Topics

Pump as Turbine Overview

Running a pump as a turbine allows the pump to produce electrical power when the pump sees reverse flow. During reverse flow, the motor coupled to the Pump as Turbine (PAT) spins in

reverse, causing the magnetic rotor to induce a current in the motor coils, turning the motor into a generator.

Note: The positive flow direction for the pipes connected to the PAT should be the direction in which flow will travel when the pump is operating as a turbine, rather than the direction in which flow will travel when the pump is operating as a pump. Thus, while the physical pump equipment will see reverse flow while operating as a turbine, the pump as a PAT will see positive flow.

Standard Turbine Curve

A Standard Turbine Curve is required to model a PAT and must include both head and power data. The head loss vs flow curve defines the head loss seen at the PAT while coupled to the motor and generating power. The power versus flow curve defines the power generated by the PAT while coupled to the motor and generating power.

To enter these curves, click Enter Curve Data. Previews of the applied curves can be viewed in the Standard Turbine Frame once the data has been defined.
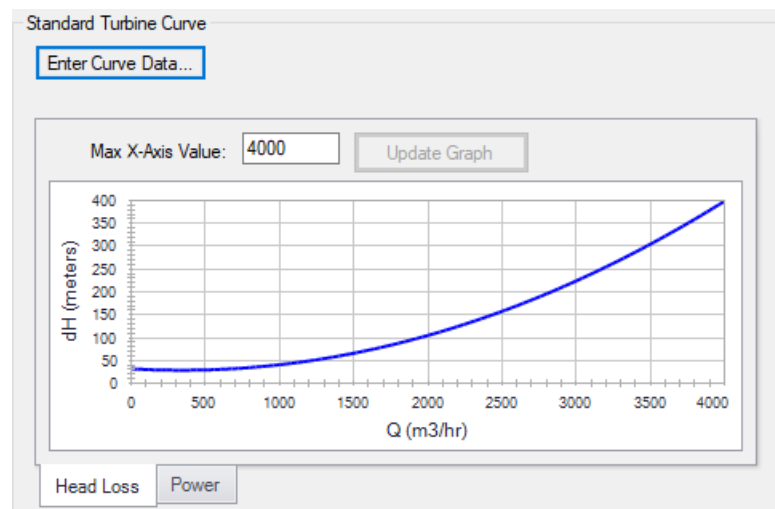
Figure 1: Standard turbine curve

Flow Directional Behavior

A visual representation of the modeled PAT is presented in the lower right of the properties window to provide immediate feedback on what the various options represent. In the PAT depicted in Figure 2, head loss and power information have been defined, and the PAT is in normal operation, with reverse flow seen at the pump.
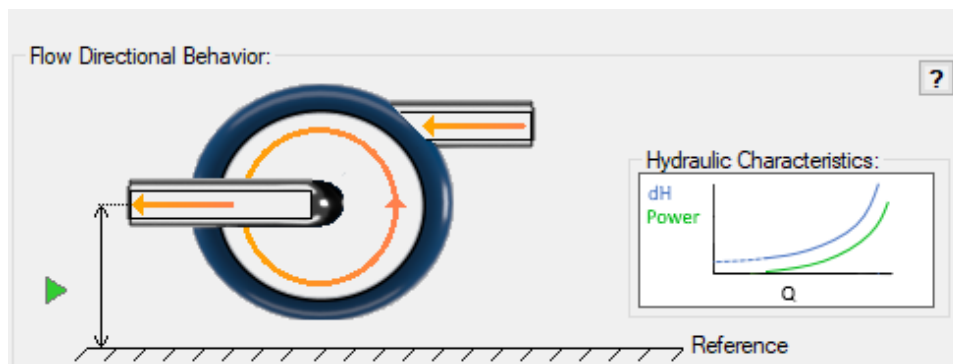
Figure 2: Flow Directional Behavior image for an example PAT

PAT Configuration

The Pump as Turbine Configuration window is used to define the specific operation of a Pump as Turbine. It is opened by clicking Enter Curve Data from the Pump Properties window on the Pump Model tab when Pump as Turbine is selected.
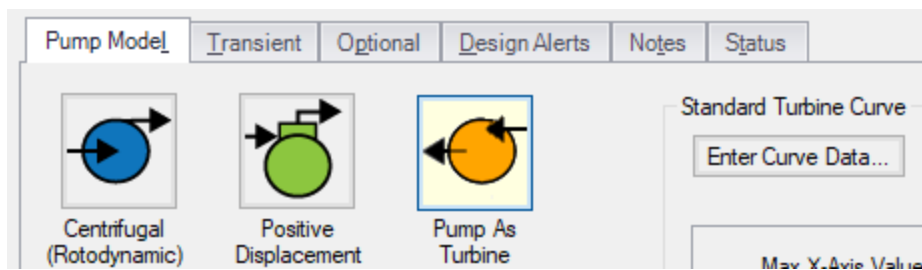
Figure 1: Enter Curve Data on the Pump Model tab

Turbine Data

The first tab of the PAT configuration defines the PAT's most fundamental behavior - the turbine curves.
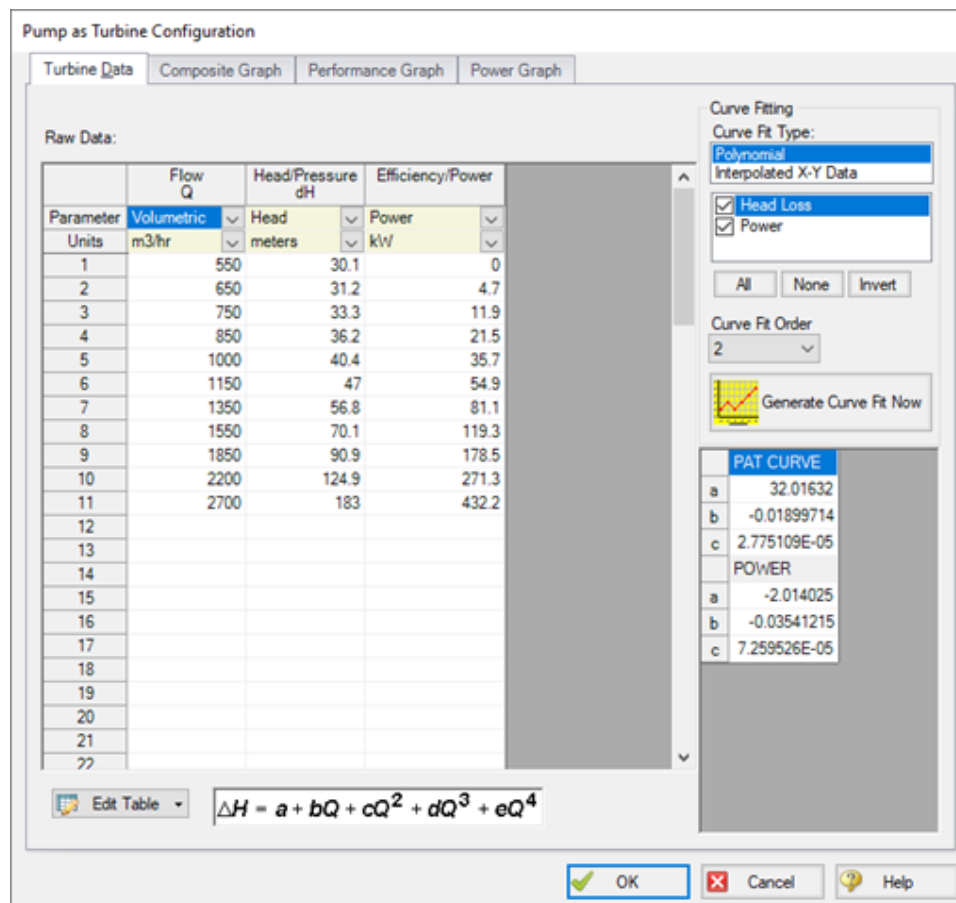
Figure 2: Pump as Turbine Configuration window

A PAT can have two curves defined:

- **Head or Pressure vs. Flow:** Required. This curve is commonly referred to as the "turbine curve" and defines the head loss across the PAT at any given flowrate.
- **Power vs. Flow:** Required. The power curve is used for two purposes for the PAT. First, it is used to determine the power generated while the pump is operating as a turbine. Second, it is used to determine if or when the pump and motor decouple.

Graphs

The following tabs display the curves that have been fit to the given data.

- **Composite Graph:** Shows the curve-fit PAT Performance and Power curves.
- **Performance Graph:** Shows the curve-fit PAT Performance curve overlaid on the raw data points. The solver will ignore the raw data points and operate on the curve fit.

- **Power Graph:** Shows the curve-fit Power curve overlaid on the raw data points. The solver will ignore the raw data points and operate on the curve fit.

PAT Operation as a Turbine

The main function of the PAT model is to allow a pump to operate as a turbine. In this mode of operation, the PAT generates power by removing energy in the form of a head loss across the pump, compared to normal centrifugal pump operation where the pump adds energy in the form of a head rise across the pump.

The turbine operating point is determined using the Turbine Performance Curve defined by the user. This curve relates head loss to flow rate similar to how the Pump Curve relates head rise to flow rate. Note that the PAT uses the turbine's frame of reference for this data rather than the pump's. Head loss and flow rate should be entered as positive values for the PAT. The Turbine Performance Curve should have a head loss greater than zero at zero flow and should see increasing head loss with increasing flow. An example Turbine Performance Curve is shown in Figure 1 below.
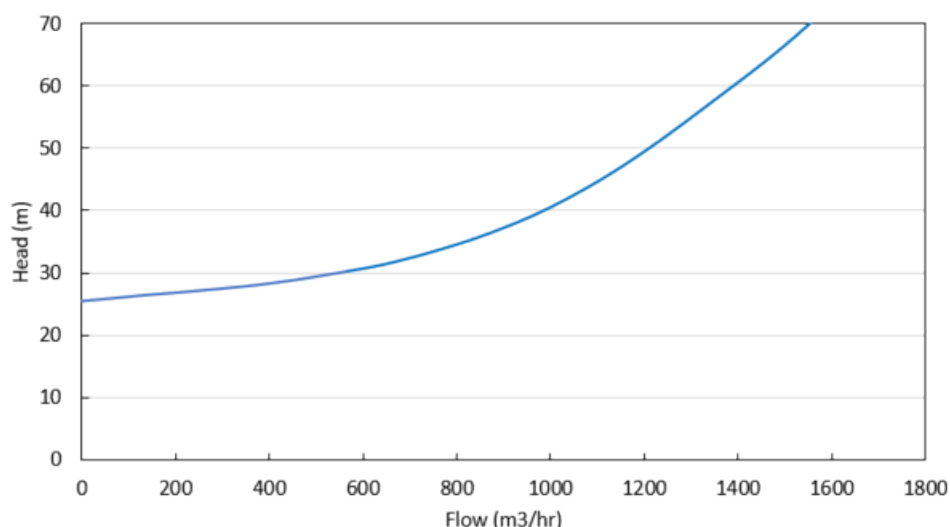
Figure 1: Example Turbine Performance Curve

The power generated by the PAT is determined by the user-defined Turbine Power Curve. This curve relates power generated to flow rate, similar to how the Pump Curve relates pump power to flow rate. Note the PAT uses the turbine's frame of reference for this data rather than the pump's. The Turbine Power curve should have zero power generated at some non-zero flow rate and should see increasing power output with increasing flow. An example Turbine Power Curve is shown in Figure 2 below.
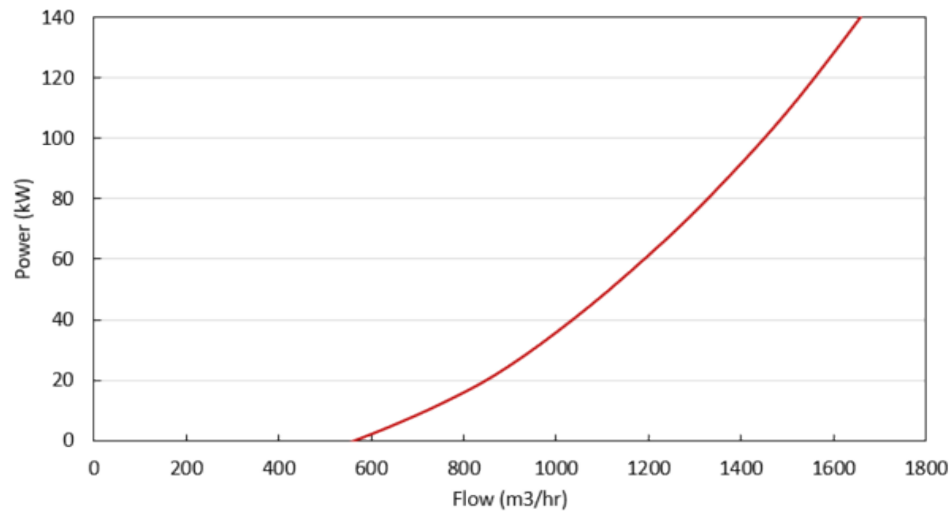
Figure 2: Example Turbine Power Curve

The flow rate at which the PAT generates zero power is called the 'no load' point. At this flow rate, the pump impeller is spinning with the fluid, neither adding nor removing energy from the fluid. At higher flow rates, the impeller's speed is resisted by the motor, removing energy from the fluid and generating electricity. At lower flow rates, the impeller's speed is resisted by the fluid, requiring energy to be added via the motor. These operating conditions are shown in Figure 3 below.
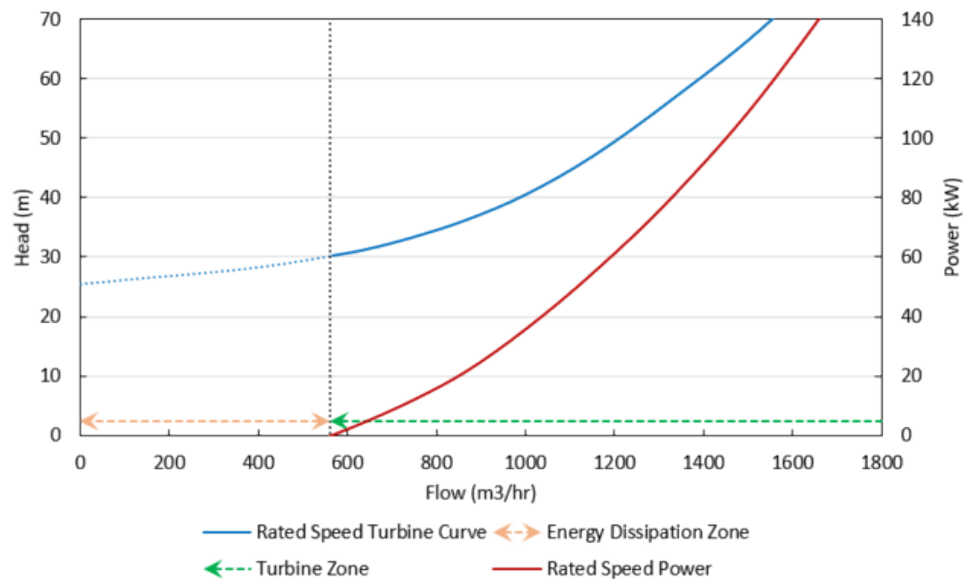
Figure 3: General turbine operating conditions

PAT Operation While Decoupled

Operation as a Turbine means the PAT is coupled to the motor, but the PAT can also operate while decoupled from the motor, meaning the pump's impeller is spinning freely in the fluid stream. The PAT will be decoupled from the motor when it is operating below the No Load point (the flow rate with zero power generation) on the Turbine Curve, since below that flow rate the motor would be required to input power to the system.

Figure 1 below shows these operating conditions. Since the PAT is decoupled from the motor, the generator does not impart a torque on the pump impeller to counteract the torque applied by the fluid, and the pump impeller becomes free-spinning.
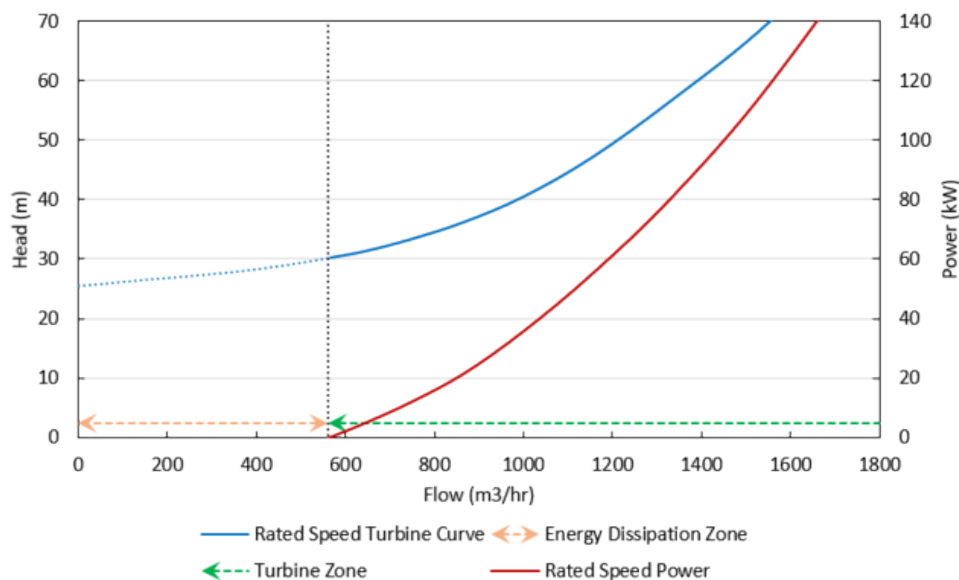
Figure 3: General turbine operating conditions

While the PAT is free-spinning, the impeller will spin at a speed dictated by the fluid such that no torque is applied on the impeller by the fluid. This speed may change based on other transients in the system and is called the runaway speed.

The pressure loss of a free-spinning PAT is determined by the Turbine Performance curve, adjusted by the Affinity Laws. The user-defined Rated Turbine Speed is considered 100% speed, and the Affinity Laws are used to adjust the curve based on speed. The Affinity Laws adjust the Turbine Performance curve in the same direction they adjust a Pump Performance curve (i.e., head loss and power increase with impeller speed).

The speed of a free-spinning impeller is determined by the system flow rate and the No Load Curve.

No Load Curve

Each runaway speed of the PAT has a different No Load point, meaning a No Load Curve can be created to describe how head loss is related to flow across the range of potential runaway

speeds. Each point on the curve represents a different runaway speed and different steady state. Figure 2 below shows an example No Load curve overlaid with a Turbine Performance Curve and a Turbine Power Curve. Notice how the No Load curve intersects the Turbine Performance Curve at the point where the Turbine Power Curve has zero power.

This No Load curve can be created by plotting the head loss for multiple Turbine Performance Curves (adjusted for speed) at the flow where their corresponding Turbine Power Curves cross the zero power line. Note that AFT Fathom does not calculate, use, or require the No Load curve for its calculations. Rather, the No Load curve is merely an observable consequence of the physics the PAT follows.
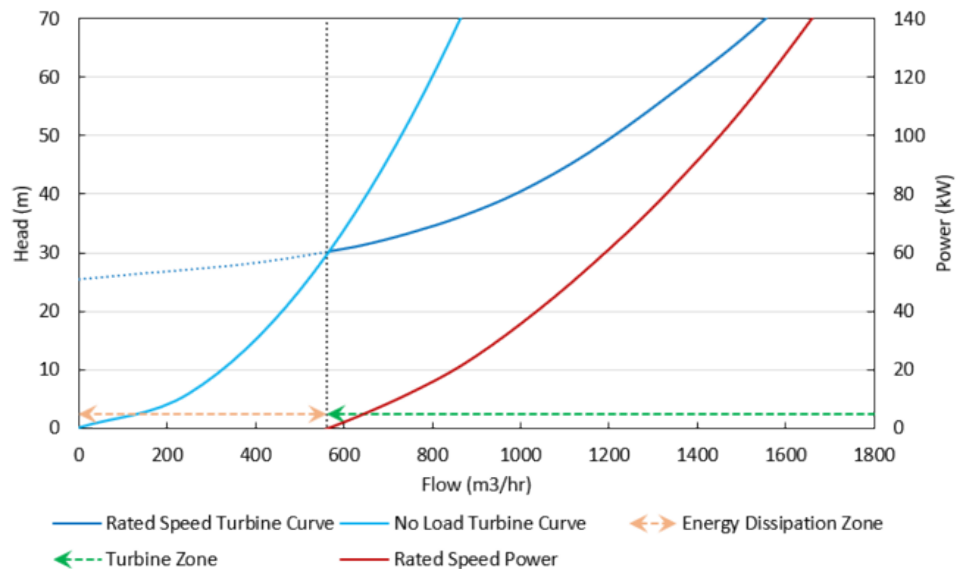
Figure 2: Example No Load Curve showing general turbine operating conditions

PAT Operation During Decoupling

The PAT transitions between the turbine and decoupled operating conditions in two situations: when manually decoupled or whenever the power generated is equal to zero. The PAT cannot recouple after becoming decoupled during a simulation.

The **first** transition from coupled to decoupled is referred to as Load Rejection and can be set as a transient event in the Transient tab of the Pump Properties window.

When this transient event is initiated, the PAT is immediately uncoupled from its motor at the onset of the transient simulation, the speed is allowed to change as dictated by the fluid, and the PAT speed is determined for each time based on the No Load Curve. The uncoupling event occurs instantaneously and the PAT will immediately begin operating on the No Load Curve at steady-state.

The **second** transition from coupled to decoupled occurs when the PAT reaches the No Load point due to other transient events in the system. This transient event is always ready to occur regardless of the transients defined for the PAT. When the flow rate through the PAT becomes low enough such that zero power is generated by the PAT, the PAT will immediately become

uncoupled from the motor and become free-spinning with its speed determined by the system flow rate and the No Load Curve.

Affinity Laws

Pumps can operate at variable speeds (adjustable in the Variable Speed tab in the Pump Properties window). Affinity laws (also called homologous laws) allow an estimation of the pump performance at speeds other than the design speed.

To a somewhat lesser degree, impeller diameter changes (Impeller Modifications) follow the same laws as speed changes. Substituting impeller changes for speed changes below can be used to see the effect of an impeller change or for a small impeller trim (larger trims deviate significantly from this estimation).

Before discussing the affinity laws, it should be noted that the affinity laws are an approximation. In many cases, pump manufacturers will have performance data for a number of speeds. Manufacturer-provided data should be used if available. Multiple Configurations can be entered to save pump curves at different speeds. When data is not available for varying speeds/impeller trims, the affinity laws can be used with an understanding that an approximation is being made.

Pump curves can generally be curve fit to a polynomial:

$$\Delta H = a + b\,Q + c\,Q^2 + d\,Q^3 + e\,Q^4$$

The speed ratio is defined in terms of the design impeller speed, N_1 and the desired operating speed N_2 as:

$$s = \frac{N_1}{N_2}$$

Affinity Laws

For the following ratios, subscript 1 represents the parameter from the defined curve (assumed to be at 100% speed). Subscript 2 represents the given parameter at the desired operating speed N_2.

- **Flow rate** -

$$\frac{Q_1}{Q_2} = s$$

- **Pump head** -

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

Note: By default NPSHr will follow the same relationship as pump head rise. The speed ratio exponent used for NPSHr can be changed on the Configuration Data tab in the Pump Configuration window.

- **Power consumption** -

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

- **Efficiency** -

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

Related Topics

Submerged Pump

A pump junction can be defined as submerged by clicking the Submerged Pump check box in the Pump Properties window on the Pump Model tab. This is convenient to avoid the need of modeling suction piping when the suction head or pressure is a known and fixed value.

There are three options to choose from to define a submerged pump: Head (HGL), Pressure, and Surface Conditions.

Head (HGL)

With the Head (HGL) option selected, the only required value is Liquid Surface HGL. This value refers to the *liquid* surface elevation **relative to the reference elevation**, as can be seen in Figure 1. This is important when the elevation of the pump is a non-zero value. The specified elevation for the pump is then used to determine the depth of the submerged pump.
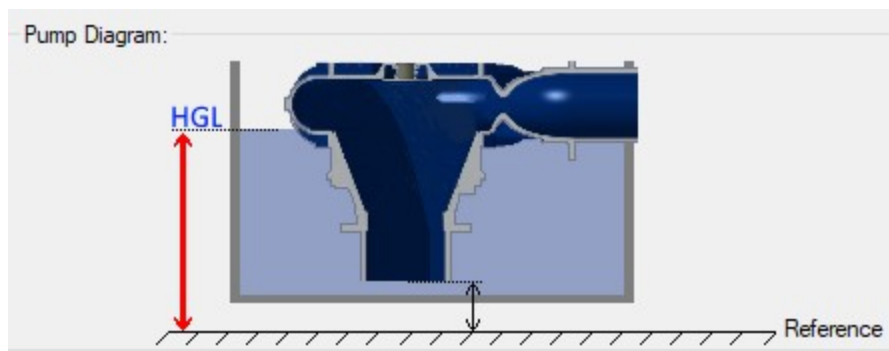
Figure 1: Diagram of Submerged Pump, Liquid Surface HGL input

Pressure

With the Pressure option selected, the only required value is Suction Pressure. This value refers to the fluid pressure at the inlet of the submerged pump.
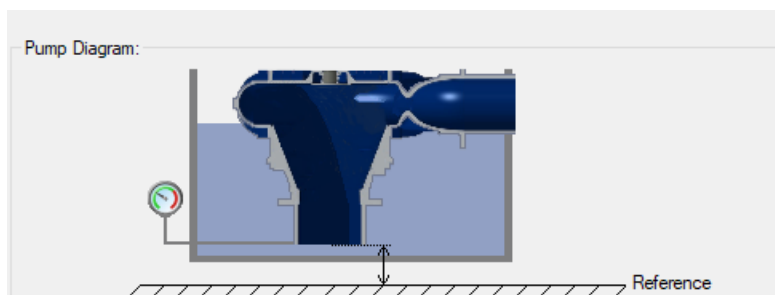
Figure 2: Diagram of Submerged Pump, Pressure input

Surface Conditions

With the Surface Conditions option selected, there are two required values: Surface Pressure and Liquid Depth. Similar to how a reservoir junction is defined, the Surface Pressure is the pressure of the fluid at the surface, which is helpful when the liquid surface is not at atmospheric pressure. The Liquid Depth is the distance between the liquid surface and the inlet of the submerged pump.
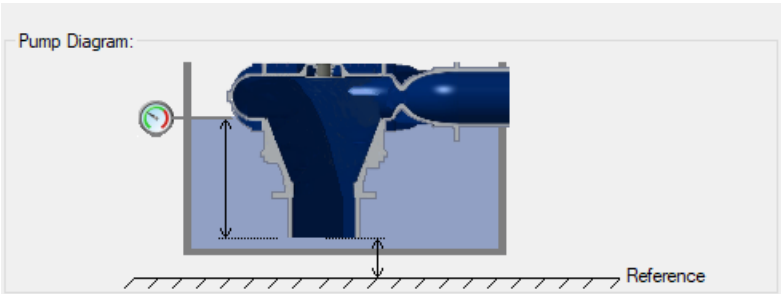
Figure 3: Diagram of a Submerged Pump, Surface Conditions input

Related Topics

Pump Efficiency Data

Best Efficiency Point (BEP)

By defining an efficiency or power curve, the Best Efficiency Point of the pump can be found. Optionally, the calculated value can be overridden with a user-defined value in the Pump Configuration window. It is generally desirable to operate pumps as close to the BEP as possible. The Pump Summary tab in the Output can display the BEP, and the % of BEP which indicates how close the operating point is to the BEP.

Power and Efficiency Calculations

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

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

The Solver will determine the total dynamic head (ΔH) and flow (Q) through the pump via iteration for the unique system. Then, the **Ideal Power** (also called hydrodynamic power) can be directly calculated using the equation below:

$$P_{Ideal} = Q\rho g \Delta H$$

where ρ is the fluid density at the pump and g is the gravitational acceleration. When a pump efficiency or power curve is entered on the Pump Configuration window, the power supplied to the pump, or **Shaft Power**, can be calculated:

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

where the pump efficiency comes from the pump curve at the given flow. On a pump's optional tab, further efficiency information can be defined:

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 and impeller. 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 calculated 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 Pump Summary along with the other above values. Whenever an efficiency value is left undefined, it is assumed to be 1 (perfectly efficient).

$$\eta_{overall} = \eta_{VFD}\eta_{Motor}\eta_{Pump} \qquad P_{Overall} = \frac{P_{Ideal}}{\eta_{overall}}$$

Related Topics

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 **(Constant):** 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 determines 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.

Restricted Flow Area

An optional input in the Relief Valve Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the valve. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from the manufacturer or from test data.

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

Reservoir

The Reservoir junction type allows you to connect up to twenty-five pipes. At least one connecting pipe is required. The Reservoir junction is convenient for specifying a fixed pressure in your system.

A Reservoir junction applies a defined pressure and temperature (if using heat transfer) in the model at the junction location. When solving a pipe flow system, a Reservoir causes the rest of the system to distribute the flow in a manner consistent with the defined pressure.

The Reservoir Properties Window follows the second of the two basic window formats for junctions.

The Liquid Surface Elevation of the Reservoir must be defined - this value will be used along with the Pipe depth to determine the hydrostatic pressure contribution at the pipe inlet.

Surface Pressure is required, and fluid temperature is required when heat transfer is modeled.

Specifying Pipe Connectivity

If there is a single pipe connected to the junction, its elevation can be directly specified on the Reservoir Model tab. The elevation can be specified either as an absolute Elevation, or as a depth below the liquid surface. This means there are two ways to define the same pipe elevation.

If multiple pipes are connected, they can each have a unique elevation. This is displayed on the Pipe Depth and Loss Coefficients tab. In this table, all elevations must be specified as either depth or elevation, as is specified by the radio button selection below the table. Additionally, losses such as re-entrant losses can be added to the pipe connections in this table. Loss factors can optionally be specified for flow into and flow out of the pipe.

Pipes Connected Above Liquid Surface

A pipe elevation can be specified higher than that of the liquid surface. Pipes that empty into the Reservoir above the liquid surface are assumed to have liquid free fall to the liquid surface. AFT Fathom applies the proper boundary condition for above the liquid surface.

If the pipe is above the liquid surface, the only physically realistic condition is for the fluid to be flowing from the pipe into the reservoir. However, AFT Fathom will assume the fluid flowing into the pipe is the same as the reservoir fluid in order to solve the system. If this occurs a warning will be shown in the Output.

Infinite and Finite Reservoirs with the XTS Module

With the standard AFT Fathom, all reservoirs are treated as infinite. With the XTS Module, reservoirs can be infinite or finite.

- **Infinite reservoir**
    - If modeling an infinite reservoir, the user can vary the surface level or surface pressure with time. Note that the liquid level is not calculated in this case, but is input by the user.
- **Finite tanks**
    - Finite tank reservoirs can be open or closed. Finite reservoirs require specification of tank geometry. Typically, the liquid level and surface pressure are defined initially, and then calculated over time based on the tank geometry.
    - Open tanks do not vary the surface pressure unless the user does so through the Transient tab. Closed tanks vary the surface pressure in accordance with gas thermodynamic laws and a user specified polytropic constant.

- For closed tanks, users can enter maximum and minimum pressures. These represent pressure control systems on the tank such as relief valves. Liquid overflow lines can be modeled with normal pipes. In such cases use of valves with event transients can be used.
- **Known parameters initially**
    - In some cases, the initial liquid level or pressure may not be known, and it is desired to have this calculated based on a mass balance in the system. This can be selected using the check box options provided.

If an XTS model is run, and the pipe is no longer submerged with flow into the pipe, AFT Fathom will stop the flow in that pipe on subsequent time steps because there is no liquid to supply it.

A potential confusion exists when pipe connection data is entered as depth below the liquid surface, and then the liquid surface changes during a transient. This is not an issue in that all depth data is converted internally to absolute elevation at the beginning of the run so that liquid level changes will be handled properly.

> **Tip:** Since transient models allow the liquid level to change, and the user may desire to run different scenarios with different initial liquid levels, it is better to specify reservoir pipe connections based on elevation and not depth.

Energy Balance

The energy balance feature is displayed only when heat transfer is being modeled. 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. The temperature can be allowed to vary at a junction by checking the Balance Energy option. See Open vs. Closed Systems - Balancing Energy for detailed discussion.

Related Topics

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 Fathom provides four standard screen types for your use:

- Sharp-edged perforated plate
- Round-edged perforated plate
- Circular wire screen
- Silk thread screen

The loss factors for these screens are taken from handbook values. In addition to the standard screens that AFT Fathom provides, you may enter your own custom screen loss factor that is consistent with the application.

When you select Resistance Curve in the Loss Model section on the Loss Model tab, the Screen Properties window makes additional features available. Using these features, you can input pressure drops that vary with flow. To enter this information, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom 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.

An optional input in the Screen Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the screen. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

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):

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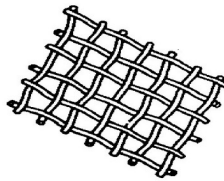
Figure 1: Circular metal wire screen

The **silk thread screen**, shown in Figure 2, is given by the following equation for screen area Reynolds numbers above 500 (Idelchik 2007, 577):

$$K_{up} = 2.1 \left(1 - \frac{A_{screen}}{A_{up}} \right) + \left(\frac{A_{up}}{A_{screen}} - 1 \right)^2$$

For other screen configurations, see Chapter 8 of Idelchik's Handbook of Hydraulic Resistance (2007).
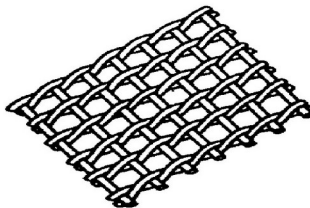
Figure 2: Silk thread screen

Related Topics

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. If a heat transfer fluid model is used, then exit temperature must also be entered.

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 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.

The distance of each connecting pipe (if any difference exists) from the specified junction elevation can be entered in the pipe table.

Loss Model

The discharge coefficient for the junction is entered under the Loss Model tab in the box near the bottom of the window. The physical area of the exit flow can also be entered under the discharge coefficient input box. 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.

The K Fire Sprinkler model uses the *K* value as defined by the fire sprinkler industry. Whereas the standard *K* loss value is dimensionless, the *K* Fire Sprinkler has units associated with it. You can adjust these units according to your data.

If you select to model the spray discharge pressure losses with a Resistance Curve, the Spray Discharge Properties window makes additional features available. Using these 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.

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 CdA or *K* 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

Spray Discharge Fire Sprinklers

The definition of *K* value for fire sprinklers is:

$$K_{sprinkler} = \frac{Q}{\sqrt{\Delta P}}$$

where Q and ΔP can be in the units of your choice. The preference in the USA is gal/min for flowrate and psid for pressure. In Europe it is frequently liter/min for flow rate and bars for pressure. Note that $\textbf{\textit{K}}_{\textbf{\textit{sprinkler}}}$ **is not dimensionless**, but has units associated with it. Also, note that sprinkler vendor values for $K_{sprinkler}$ implicitly assume that water is the fluid, so there is a built-in density of 62.3 lb_m/ft^3 or 1,000 kg/m^3. If this sprinkler is used for something other than water, the $K_{sprinkler}$ value will need to be adjusted.

Also note that if the spray is into the atmosphere, as is typical, the ΔP is the same thing as the gage pressure. That is why the previous equation is stated by some vendors as just P in the denominator, rather than ΔP. When stated as just P, the pressure is gage.

Finally, note that the Exit Pressure, as defined by AFT Fathom, is different than the concept of Discharge Pressure, an often used parameter in the Fire Protection industry. AFT software defines exit pressure as the pressure of the ambient surroundings, which in an unpressurised location at sea-level, is 1 atm.

Note: The fire sprinkler K is not the same as the dimensionless K value as given by Equation 1 in the Loss Model Summary topic and used extensively throughout AFT Fathom.

The $K_{sprinkler}$ is related to the discharge coefficient. It can be shown that the relationship is given by

$$C_D\, A = \sqrt{\frac{\rho_{water}}{2}}\, K_{sprinkler}$$

Related Topics

Spray Discharge CD (Discharge Coefficient)

The pressure drop across a spray discharge which uses a discharge coefficient is discussed here. The mass flow rate through a restriction is given as follows:

$$\dot{m} = \rho\, C_D\, A\, V$$

The pressure drop across a discharging restriction is given as:

$$\Delta P = \frac{1}{2}\rho\, V^2$$

Combining the two yields:

$$\Delta P = \frac{1}{2}\rho\left(\frac{\dot{m}}{C_D\,A\,\rho}\right)^2$$

or

$$\Delta P = \frac{1}{2\,\rho(C_D\,A)^2}\dot{m}^2$$

This equation is used for spray discharge junctions in AFT Fathom, with the *ΔP* being the difference between stagnation pressure at the spray discharge inlet and the exit pressure specified for the junction.

Related Topics

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, Fathom will automatically define the straight and branch run pipes based off of the arrangement in the Workspace. If Fathom 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 Fathom 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 Tee Losses.

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

Tee/Wye Loss Model

The Simple loss model assumes there are no losses across the tee/wye. The Detailed loss model uses various correlations and data to calculate the loss coefficients across the different flow paths of the tee/wye. Generally, the correlations depend on the flow split, ratio of pipe cross-sectional areas, and angle of the connecting pipes.

- **Sharp Tee** - This option represents tees with sharp corners (r/d=0) that have equal areas in the header and an equal or smaller area in the branch ($A_c =< A_a = A_b$).

- **Rounded Tee** - This option represents tees with rounded or smooth corners (r/d=0.1) that have equal areas in the header and an equal or smaller area in the branch ($A_c =< A_a = A_b$).

- **Area Change Tee** - This option represents tees with sharp corners (r/d=0) which have an area sum ($A_a + A_c = A_b$).

- **Symmetrical Wye** - This option represents symmetrical wyes with sharp corners (r/d=0) which have an area sum ($A_a + A_b = A_c$).

If the cross-sectional area requirements are violated, then the software will still provide results because it accounts for different basis areas and effectively "corrects" the loss coefficient by using velocity ratios. However, it is important to note that in these cases the correlations are still being used outside of their original ranges, so there is inherently some additional uncertainty.

Nomenclature

The correlations and data used by the Detailed loss model come from two different sources: Idelchik's Handbook of Hydraulic Resistance and Miller's Internal Flow Systems (see References below). Each source has their own naming conventions and nomenclature. Furthermore, both nomenclatures assume that the user knows the flow directions ahead of time. For these reasons, AFT has selected a more general nomenclature to describe the names of each pipe in the tee/wye. AFT's nomenclature also accounts for the fact that the user may not know the flow directions ahead of time, and might not be able to properly specify the tee using Idelchik's or Miller's conventions. Since most of the correlations come from Idelchik, and many of our users are familiar with the Idelchik text, we will first describe some of the important nomenclature here. Then, the AFT nomenclature will be depicted below. For reference, Idelchik uses several unusual parameter names; the greek letter Zeta represents the loss coefficient (K factor), "F" represents the cross-sectional area of the pipe, and "w" represents the velocity.

Figure 1 depicts the Idelchik nomenclature, but using "A" for area instead of "F". The pipes are labeled as combined flow (c), straight-through flow (st), and side branch (s). The combined flow

pipe always carries the total flow, and depends on if the tee has a diverging or converging flow pattern (flow patterns discussed further in the next section). Loss coefficient subscripts refer to a basis area 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 and velocity. An issue with applying the Idelchik convention to a general software is that sometimes the flow directions are not known ahead of time, or may change based on multiple operational configurations, transient events, etc.
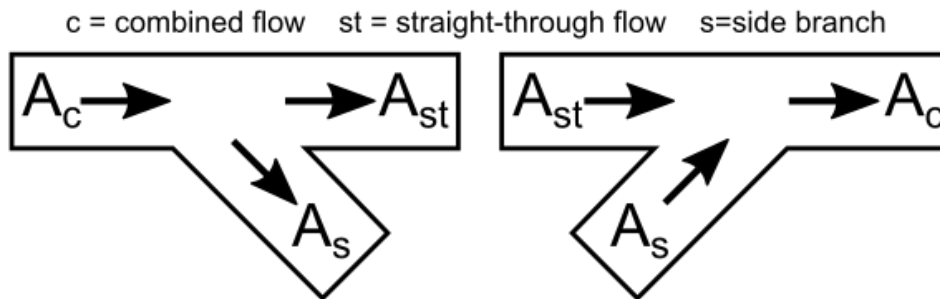
Figure 1: Idelchik's nomenclature for diverging (left) and converging (right) flow through a Tee/Wye junction

Figure 2 depicts the AFT nomenclature. The pipes are labeled as *a*, *b*, and *c*. The *c* pipe is always the branching pipe (equivalent to the "s" pipe in Idelchik). This convention allows AFT to cover all flow pattern possibilities discussed below and listed in Table 1 without specifying exactly which pipe is the combined flow pipe.

Figure 2: AFT's nomenclature in the Tee/Wye Properties window where the c pipe is always the branch pipe

Flow Patterns & Correlation Selection

There are four general types of flow patterns in tees which depend on the directions of the flow streams:

1. **Diverging From Header** - flow comes from a single upstream pipe in the header, and a portion of flow diverges into the branch pipe while the remaining portion continues in the straight run.
2. **Converging Into Header** - flow comes from two upstream pipes, one of which is the header and other is the branch pipe, and the two streams converge into the downstream portion of the header.
3. **Dividing From Branch** - flow from the branch pipe divides in opposite directions in the header.
4. **Combining Into Branch** - flow comes from opposite directions in the header and combines into the branch pipe.

The Solver will automatically determine which flow pattern is applicable based on the actual flow directions in the model. The Diverging and Converging patterns use equations or tabulated data

with interpolation that come from Idelchik. The Dividing and Combining patterns use tabulated data with interpolation that comes from Miller.

Angle

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. AFT will always define the angle from the *a* pipe to the *c* pipe. The user should define the angle according to this definition and be consistent with their physical layout of the pipes. If the actual flow directions in the final solution are different, then the Solver will automatically handle this in the backend.

Some Idelchik correlations are for specific, discreet angles. If the user-entered angle is between these values, then the software will interpolate. Other Idelchik correlations provide equations with the angle as an input, and are valid for a specified angle range. If the angle falls outside of the range, then it will be treated as either the maximum or minimum angle.

The data referenced from Miller was only for 90 degree tees, and therefore, the angle has no effect and is always assumed to be equal to 90 degrees for the Dividing From Branch and Combining Into Branch flow patterns.

Convergence Issues

Sometimes intermediate iterations of the Solver cause a different flow pattern to be selected, and sometimes the system may experience instability as the Solver gets stuck switching back and forth between two different flow patterns. Other times, having many Detailed tees in a pipe network can cause numerical complexity due to the effects of the tees coupling together in such a way that the Solver cannot converge on all of the flow splits. The tee loss coefficients are dependent on the flow splits, but the flow splits are dependent on the other losses in the system. Therefore, if a pipe network contains many tees using the Detailed loss model, when one tee's loss coefficient is updated from an intermediate Solver iteration, the entire network needs to be re-calculated to determine how the flow splits changed. Sometimes this process is not able to terminate within the Tolerances specified by the model.

See the [Tee/Wye Junction Complexity](#) help topic for more information and suggestions. In general, if the losses from the tees are small in comparison to the total system losses, or other components that contribute significant pressure loss, then it may be acceptable to assume the tee is lossless and use the Simple loss model.

Negative Loss Coefficients

Under the right conditions it is possible to see a negative loss coefficient for one of the flow paths in the tee. This is physically realistic and is supported in literature by both Idelchik and Miller. In summary, a negative loss coefficient represents an energy/pressure gain for said flow stream of the tee and is achieved by having a portion of kinetic energy transferred to it from the other flow stream. The other flow stream sees an increased loss coefficient (higher than it would have been

otherwise), and the net result is still an irrecoverable energy loss across the entire tee control volume. Some specific details are discussed below.

- **Converging Tee Side Branch:** Turbulent mixing and momentum exchange takes place between the particles which favors equalization of the velocity field within the pipe which carries the merged flow. In other words, there is a stream with higher energy (velocity/kinetic energy), and a stream with lower energy, and when they merge together they level out to some equilibrium that is somewhere between the high and low energy in the combined flow pipe. The stream which had the higher velocity effectively transfers part of its kinetic energy to the stream with the lower velocity.
  - Evidence from Idelchik 3rd Ed 1994: Diagrams 7-1 to 7-9, the tables and graphs show negative $K_{c,s}$ values.
  - Evidence from Miller 2nd Ed 1990: Figure 13.10 pg 311, the chart plots lines of constant K factor, some of which have negative values.
- **Diverging Tee Straight Run:** At certain flow rate ratios (Qs/Qc), a portion of the slowly moving boundary layer adjacent to the wall passes into the side branch while transferring part of the momentum into the straight passage. The increase in energy in the straight passage is associated with a negative K_{st}, while also accompanied by an increased positive value of K_s such that the whole process is a net irrecoverable pressure loss.
  - Evidence from Idelchik 3rd Ed 1994: Diagrams 7-1 to 7-9, the tables and graphs show negative $K_{c,st}$ values.
  - Evidence from Miller 2nd Ed 1990: Figure 13.23 pg 319, the curve dips below zero for a range of flow ratios between 0.1 and 0.4.

Larger Branching Flows Down a Header

There is an unintuitive, but realistic, situation that can occur due to modeling tees with the Detailed loss model. In some cases, as flow goes down a header pipe past a series of tees, there may be larger flow rates out of the final branching pipes compared to the initial branching pipes (see Figure 3). This is a realistic phenomenon that is caused by the "turning resistance" being larger than the "straight-through resistance"; or in other words, the loss factor for the flow to turn and go out of the branching pipe is larger than the loss factor for the flow to continue straight through the tee. Depending on the boundary conditions and losses in the remaining downstream parts of the network, this may influence the flow to take the path of least resistance and go straight through the tee as opposed to turning and flowing out of the branching pipe.
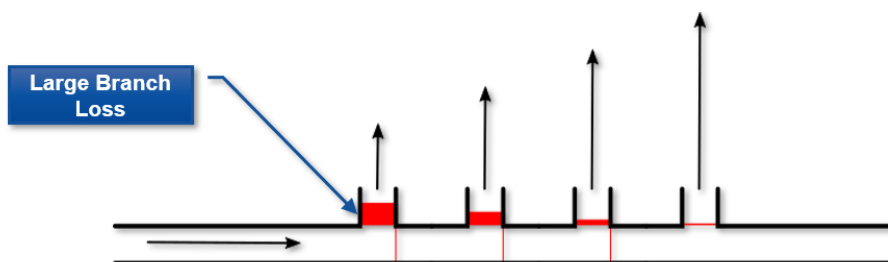
Figure 3: Illustration of branching flow increasing down the length of a header with a series of Detailed tees

By comparison, a tee modeled as lossless (Simple loss model) has no turning resistance or straight-through resistance. Stagnation pressure decreases continuously down the length of the header due to friction. As a result, the branching flow rates will decrease correspondingly (see Figure 4). The results in Figure 4 with decreasing flow down the header are also possible with the Detailed loss model, and whether you see results more similar to Figure 3 or Figure 4 depend on the pipe network as a whole, including the other losses in the system and the boundary conditions.
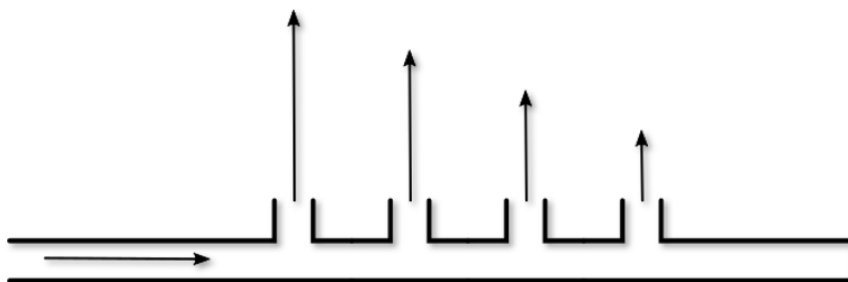
Figure 4: Illustration of branching flow decreasing down the length of a header (possible with Simple or Detailed loss models)

References

The correlations used by the Detailed loss model are taken mostly from Chapter 7 of Idelchik's Handbook of Hydraulic Resistance (Idelchik 2007), with some additional data taken from Miller's Internal Flow Systems (Miller 1990). Table 1 lists which references are used based on the user's selection in the Tee/Wye Properties window and which flow pattern is applicable as determined by the Solver.

Table 1: References for Tee/Wye junction correlations

Geometry	Type	Direction	Reference	Page	Diagram / Figure
Constant Area, Sharp (r/D = 0)	Diverging	Side Branch	Idelchik 2007	523	7-18
	Diverging	Straight Run	Idelchik 2007	525	7-20 No. 1
	Converging	Side Branch	Idelchik 2007	501-504	7-1 to 7-4
	Converging	Straight Run	Idelchik 2007	501-504	7-1 to 7-4
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d=0)
	Combining	Into Branch	Miller 1990	315*	13.16 (r/d=0)
Constant Area, Rounded (r/D = 0.1)	Diverging	Side Branch	Idelchik 2007	531-533	7-23 No. 1
	Diverging	Straight Run	Idelchik 2007	533 -> 526-527	7-23 -> 7-20 No. 1
	Converging	Side Branch	Idelchik 2007	511-515	7-11 to 7-13
	Converging	Straight Run	Idelchik 2007	511-515	7-11 to 7-13
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d-d=0.1)
	Combining	Into Branch	Miller 1990	315*	13.27 (r/d-d=0.1)
Area Sum, Sharp (r/D = 0)	Diverging	Side Branch	Idelchik 2007	524	7-19
	Diverging	Straight Run	Idelchik 2007	526-527	7-20 No. 2
	Converging	Side Branch	Idelchik 2007	505-509	7-5 to 7-9
	Converging	Straight Run	Idelchik 2007	505-509	7-5 to 7-9
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d=0)
	Combining	Into Branch	Miller 1990	315*	13.16 (r/d=0)
Wye	Diverging	-	Idelchik 2007	545 -> 524	7-30 -> 7-19

Geometry	Type	Direction	Reference	Page	Diagram / Figure
	Converging	-	Idelchik 2007	545	7-30
* Estimates have been made on available data.					
-> Means the first page number points the reader to a different Diagram.					

Related Topics

Three Way Valve

The Three Way Valve junction type always connects to three pipes. This junction type allows you to model the irrecoverable losses that occur through a valve with multiple inlets or outlets.

The Three Way Valve Properties Window does not follow either of the two basic Properties Window formats, but instead shows the connecting pipes in drop-down lists. This allows specification of which pipe is the combined flow pipe and which two are the split flow pipes.

All losses in a Three Way Valve are based on C_v (or K_v). The C_v data is entered for various percent open positions, and the user ultimately specifies the actual percent open to be used in the model. The resulting C_v values for the two flow directions are calculated and displayed. A graph of the open percentage, C_v, and flow area data can be displayed by clicking the Show Graph... button located below the Open Percentage Data table.

Special Conditions

When the specified percent open value input into the Actual Percent Open box corresponds to a C_v of one or both flow directions to be zero, a Special Condition is applied to the corresponding flow direction(s), causing the valve to close in any direction in which the C_v is zero.

Transient

(XTS Module Only) On the Transient tab, a transient open percent can be entered to simulate the position changes of the valve over time. For more information on transient data, including event transients, see Junction Transient Data.

Related Topics

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
- *K* Factor
- Resistance Curve
- Equivalent Length
- 3-K (Darby)

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. If a heat transfer fluid model is used, the Exit Temperature must also be specified to define the thermal boundary condition.

Restricted Area & Optional Losses

The Restricted Area is optional input available for the User Specified and Handbook loss sources which describes the effective area restriction in the valve for the purpose of assessing cavitation. By default, Fathom will check for cavitation in pipes since an area is specified as part of the required pipe properties. Cavitation in a junction can only be checked if a Restricted Area is provided. In most cases, information on the size of the restricted area must be obtained from the manufacturer or test data.

> **Note:** Fathom does not model cavitation but can provide a warning if the static pressure is below the vapor pressure. If a User Defined fluid is used, the optional input parameter for Vapor Pressure must be defined in the Fluid Properties. Fathom only models single phase flow, and assumes all pipes are fluid-full as stated in the Engineering Assumptions.

The Basis Area is available for the K Factor, Resistance Curve, Equivalent Length, 3-K (Darby), and Reduced Port loss models and describes the area from which loss factors are referenced from.

Open Percentage Data

Optional data may be entered for special valve characteristics. Specifically, the Open Percentage vs C_v 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 or K Factor 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

Valve Modeling Closures with K Factors

A closed valve would have an infinite K-factor. Since infinity is not a possible input, AFT Fathom uses the convention of K = -1 (negative one) to represent a closed valve.

This input is only valid when defining a valve transient while using the Fathom XTS Module. To represent a valve that is closed in steady state, use the valve Special Conditions in the Optional tab.

Related Topics

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 Fathom 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 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 Fathom 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.

Related Topics

Volume Balance

Note: The Volume Balance junction must be used with great caution. It is only intended to model specific situations and when used incorrectly will give incorrect results. When modeling mixing fluids with varying fluid properties, the Output may show a Caution message about the density change. This does not necessarily mean that a Volume Balance should be used at the mentioned location. Refer to Figure 1 below for an example of when the Volume Balance junction can be used appropriately.

The Volume Balance junction type is internal to the piping system and requires two connecting pipes.

The Volume Balance 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.

The Volume Balance junction is useful for modeling pipelines with a moving interface between fluids of different density. Such a scenario is, in reality, a transient application, because the mass in the system is not steady with time. Fathom is designed to balance mass at all branches, except for the Volume Balance junction. For the Volume Balance, the volume is balanced between the upstream and downstream pipes. This is consistent with the fact that an equal velocity exists at the interface, resulting in a volume balance at the interface.

Volume Balance junctions should be used in conjunction with the Variable Fluids feature, without heat transfer, but with variable density (see Heat Transfer/Variable Fluids panel in Analysis Setup).

The usage of a Volume Balance junction is illustrated with the Slurries With Variable Fluid Properties - SSL example (English, Metric). Figure 1 below provides a summary of the four snapshots in time that occur in the Slurries with Variable Fluid Properties example. In reality, injecting slurry into a pipeline where there are two separate fluids with an interface between is a transient situation.
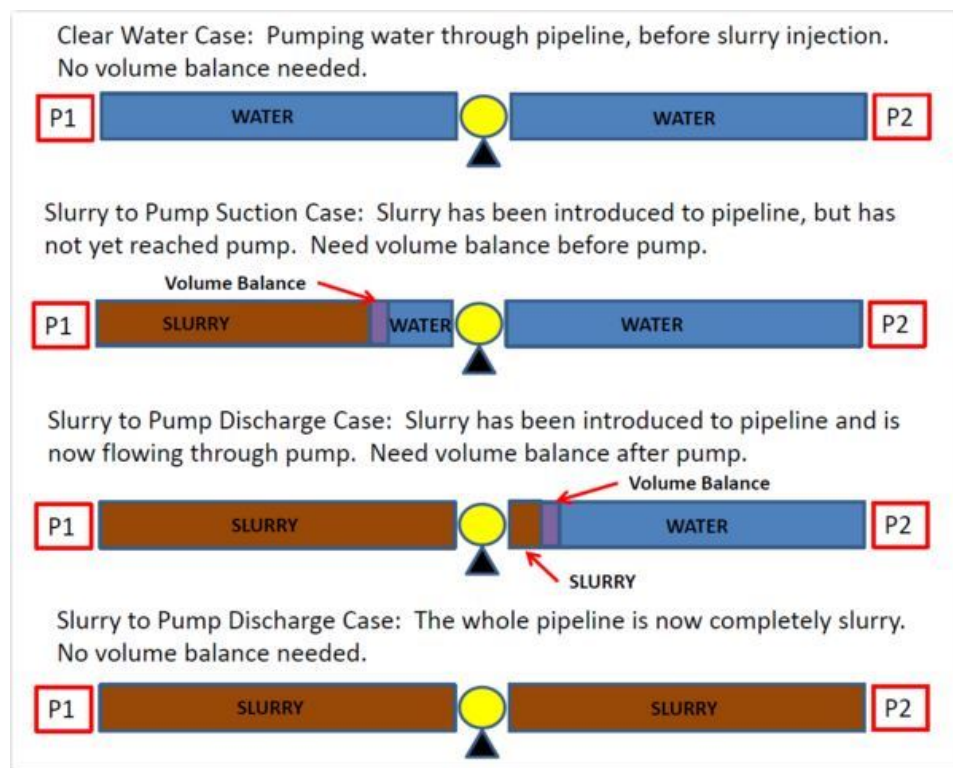
Figure 1: Four snapshots in time and when to use the volume balance junction is shown for slurries and variable fluid properties systems

Related Topics

Weir

> **Note:** The Weir junction must be used with great caution. A weir inherently is for open-chan-
> nel flow. However, Fathom always assumes flow is in a pipe, and that the pipe is liquid
> full. The loss models for the Weir junction will give a good estimate of losses associated
> with a weir in a system, but does not truly model open-channel flow.

The Weir junction type requires one or two connecting pipes. The Weir Properties window fol-
lows 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.

When using the Weir junction, keep in mind that AFT Fathom only models fluid flow in liquid-full
pipes. Use of the Weir junction will not allow AFT Fathom to model open-channel flow.

Weir Types

There are three weir configurations available. The following input data is required for each.

- **Rectangular Sharp Crested** -
    - Surface Pressure
    - Notch Elevation (Z_n)
    - Notch Width (L)
    - Channel Width (B)
    - Number of contractions
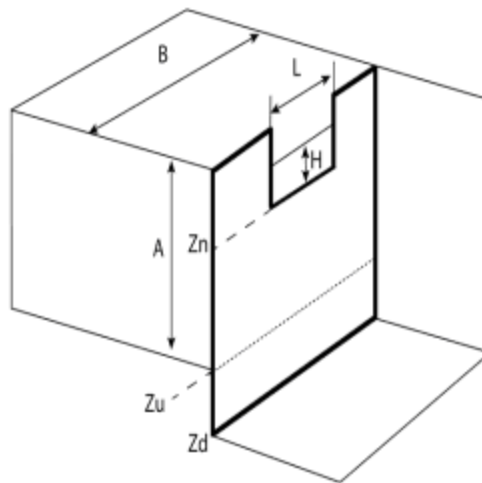
Figure 1: Rectangular Sharp Crested Weir Model

- **V-Notch Sharp Crested** -
    - Surface Pressure
    - Notch Elevation (Z_n)
    - Notch Angle (θ)
    - Channel Width (B)
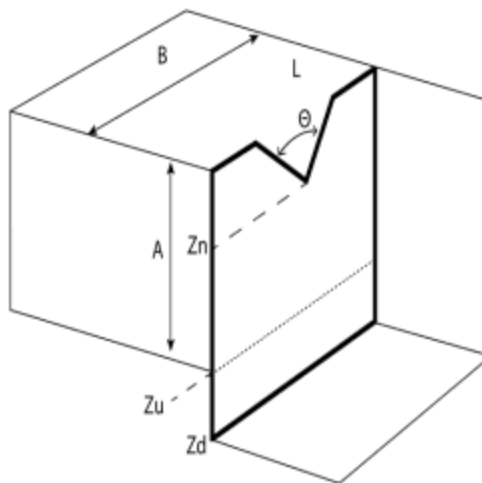
Figure 2: V-Notch Sharp Crested Weir Model

- **Cipolletti, where the vertical angle is a 1:4 ratio** -
    - Surface Pressure
    - Notch Elevation (Z_n)

- Notch Width (*L*)
- Channel Width (*B*)
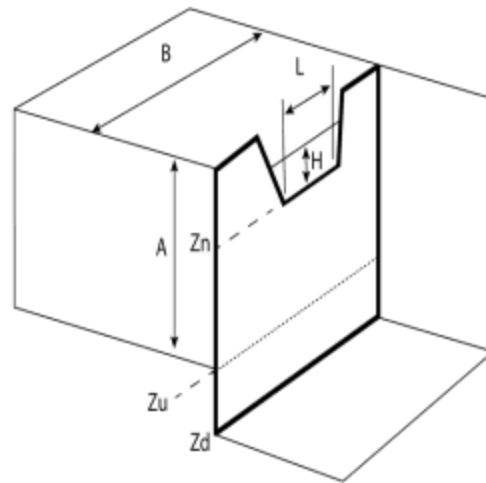
Figure 3: Cipolletti (1:4 ratio) Weir Model

Notch elevation (Z_n)

The absolute elevation of the bottom of the notch using the same reference frame as the pipe elevation.

Elevation of the Upstream and Downstream Areas (Z_u and Z_d)

The absolute elevation of the bottom of the the upstream (Z_u) tank area and downstream (Z_d) tank area using the same reference frame as the pipe elevation.

Number of Contractions

For a Rectangular Sharp Crested weir the notch area can be contracted from the sides of the weir. This can be from one side (1 contraction) or both (2 contractions). If the weir notch extends the full length of the channel then there are no contractions.

Upstream and Downstream Area Geometry

If the Cross-Sectional Area is specified as Constant, the weir area is assumed to be a vertical tank with constant cross-sectional area. If the cross-sectional area changes with height, this can be entered by selecting the variable option and entering the cross-sectional variation in the table.

The Tank Cross-Sectional Area is used for the purpose of determining the liquid volume. It will determine how high the liquid will rise or drop, since a given volume of liquid will flow into the weir during a time step.

Related Topics

Interface and User Options

This section focuses on the how the items within the various menus and tabs that comprise AFT Fathom can be used to customize the user's experience within the software. Topics discussed include the options present in each of the AFT Fathom 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

Many of the actions required to set up an AFT Fathom model can be completed from the various menus located at the top of the AFT Fathom 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](#)
- [Edit menu](#)

- [View menu](#)
- [Analysis menu](#)
- [Tools menu](#)
- [Library menu](#)
- [Arrange menu](#)
- [Window menu](#)
- [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 a pre-existing model can be selected for opening.
- **Save** - Saves input model using current model file name.
- **Save As** - Opens Save As common dialog so you can specify a file name to save model.
- **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.
- **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.
- **Import Graph Data** - (Graph Results Only) - Allows for data as .txt and .dat to be imported for creating graphs.
- **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](#).
- **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.
    - Exporting the Model Data as a Preformatted Report will generate an *.inp file, which is not to be confused with [EPANET files](#) that have the same file extension. These files can be useful for performing text comparison to find differences which is similar to the [Scenario Comparison Tool](#).
- **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.
- **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 Fathom 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 Fathom.

Related Topics

Edit Menu

- **Undo** - (Workspace Only) - Undoes the previous workspace manipulation action. Fathom will keep track of a historical list of changes, and users can revert the workspace back to recent states.

- **Redo** - (Workspace Only) - Reverts the workspace following Undo actions. Fathom will keep track of undo actions, and users can redo any recent 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.
- **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 Fathom. To enable this option, two junctions must be selected in the Workspace which Fathom 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.
- **Split Pipe at Intermediate Elevations** - (Workspace Only) - A pipe with intermediate elevations can be split into separate pipes at the intermediate elevation points. 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 and/or unctions with data from those objects in the current scenario's

parent.

- **Clear Initial Guesses** - Clears all initial guesses in pipes and junctions.

Related Topics

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 work-space.

- **Show Magnifier** - (Workspace Only) - Turns on the Magnifier tool in the Workspace.

- **Show Object Status** - (Workspace Only) - Toggles Show Object Status, which checks status of all pipes and junctions in model.

- **List Undefined Objects** - (Workspace Only) - Opens Undefined Objects window which displays undefined pipes and junctions and their properties.

- **Extended Model Check** - Performs and exhaustive check of model consistency and com-pleteness.

- **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

Analysis Menu

- **Analysis Setup** - Opens the Analysis Setup window.
- **Time Simulation (XTS Module Only)** - Allows selection of either steady-state run or transient run.
- **Goal Seek & Control (GSC Module Only)** - Allows selection of whether to use the Goal Seek & Control parameters in the current run.
- **Slurry (SSL Module Only)** - Allows selection of whether to use normal fluid calculation or settling slurry calculation in the current run.
- **NFPA Configuration** - NFPA must first be enabled in the NFPA panel. Its use can then be toggled on and off here.
- **Automated Sizing (ANS Module Only)** - Allows selection of whether to use Network Sizing calculations in the current run.
- **Fluid Properties...** - Opens the Fluid panel in Analysis Setup.
- **Transient Control... (XTS Module Only)** - Opens the Simulation Duration panel in Analysis Setup.
- **Run Model...** - When the model is complete (as determined by the Analysis Setup window), the Run Model selection sends the model to the Solver for solution.

Related Topics

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 Fathom 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

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.

- **Edit Solids (SSL Module Only)** - Opens the Edit Solids window where you can add, delete, modify or review solids properties for slurry modeling.

- **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.

- **Suggest Pump** - Opens the Suggested Pumps window which allows the user to search libraries created in Fathom or the Intelliquip™ online database of pump manufacturers to find and select a pump (and the associated performance curve) based on the operating point. See the Pump Sizing topic for more information.

- **Library Manager** - Opens the Library Manager window.

Related Topics

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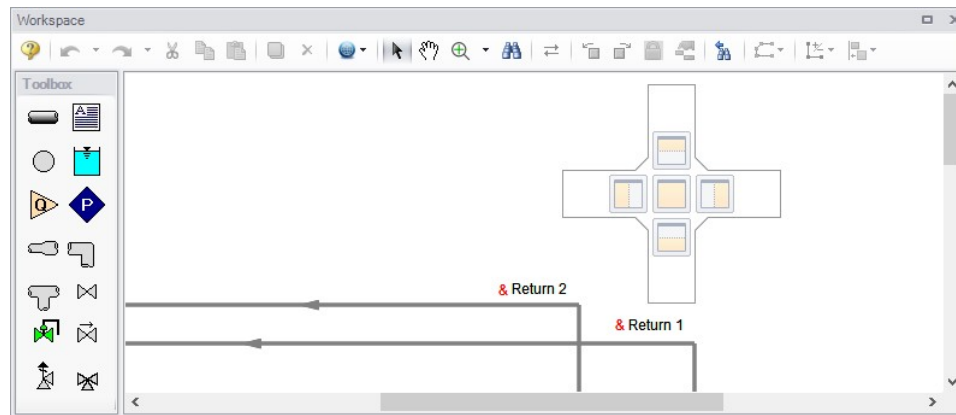
Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time

- **Sort Table By** - (Output Only) - Allows user to order tables in order of any variable and in ascending or descending order.

- **Rotate Icon Right or Left** - (Workspace Only) - Rotates icons for all selected junctions on the Workspace.

- **Auto-rotate Icon** - (Workspace Only) - Rotates the selected icon(s) to align with the connected pipes if possible.

- **Rotate Selection Right or Left** - (Workspace Only) - Rotates selected pipes and junctions as a group.

- **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.

- **Pipe Drawing Mode** - (Workspace Only) - Select the type of grid layout that the model will use.

    - **2D Freeform** - Junctions can be placed anywhere on the Workspace.

    - **2D Orthogonal** - Toggles orthogonal drawing capability on the Workspace. This toggle is a shortcut to enabling the feature in User Options window.

    - **Isometric** - Uses an isometric grid.

- **Snap to Grid** - (Workspace Only) - Available for 2D grids (on by default for isometric grid). Toggles Snap to Grid feature on the Workspace. When ON, all Workspace pipes, junctions and annotations snap to a grid specified in User Options window.

- **Show Grid** - (Workspace Only) - Toggles Show Grid feature on the Workspace. Shows or hides grid that is specified in User Options window.

- **Make Selected Pipes and Jcts Orthogonal** - (Workspace Only) - Makes all selected pipes and pipes connected to selected junctions on the Workspace orthogonal.

- **Flip Vertex** - (Workspace Only) - Available when orthogonal drawing is enabled. This will flip the vertex of all selected pipes on the Workspace.

- **Scale/Flip Workspace** - (Workspace Only) - Scales (shrinks or expands) or flips either selected or all pipes and junctions on the Workspace.

- **Set Pipe and Junction Locations to Parent** - (Workspace Only) - Sets locations of pipe and junction labels to the same position as the parent scenario.

- **Reset Label Locations** - (Workspace Only) - Sets locations of pipe and junction labels to their default locations.

- **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.

- **Add/Remove Segments** - (Workspace Only) - Allows user to Add Segment (visual bend in pipe), Add Segment at Right Angle, Add Multiple Segments, Remove Segment, or Remove all Segments.

- **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.

- **Reverse Direction** - (Workspace Only) - Reverses the reference positive flow direction of all selected pipes on the Workspace.

Related Topics

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

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 pdf in English or metric units
- **Quick Start Video -** Opens quick start video series on aft.com
- **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 in help system
- **Show Examples -** Opens help system with links to various example problems. Note that the Examples PDF must be downloaded from the PDF Downloads page on the online help site if the Use Offline Help option is active.
- **Video Tutorials -** Opens video tutorials on aft.com in English or metric units and English or Spanish language
- **Technical Seminars -** Opens information on training courses on aft.com.
- **Flow Expert Package -** Opens information on aft.com for individualized help from AFT engineers for training, modeling, consulting, and more
- **Online Learning Center -** Opens learning center on AFT website with links to resources such as blogs, video tutorials, webinars, etc.
- **AFT Home Page -** Opens home page on aft.com
- **AFT Blog -** Opens blog home page on aft.com
- **Send Email to Support -** Links to AFT support in your default email program
- **Latest Release -** Opens page on aft.com to download the newest AFT Fathom release
- **AFT Notifications -** Opens page on aft.com where you can sign up for email notifications for new software updates and releases
- **About AFT Fathom -** Opens window in AFT Fathom with information on current AFT Fathom version, license number, etc.

Related Topics

Toolbars

The Toolbars offer quick access to the features used most frequently in AFT Fathom. 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 in AFT Fathom. 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.
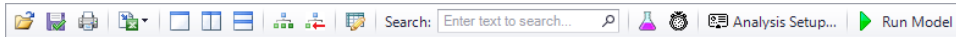
Figure 1: Common Toolbar

Figure 2: Workspace Toolbar

Figure 3: Model Data Toolbar

Figure 4: Output Toolbar

Figure 5: Graph Results Toolbar

Related Topics

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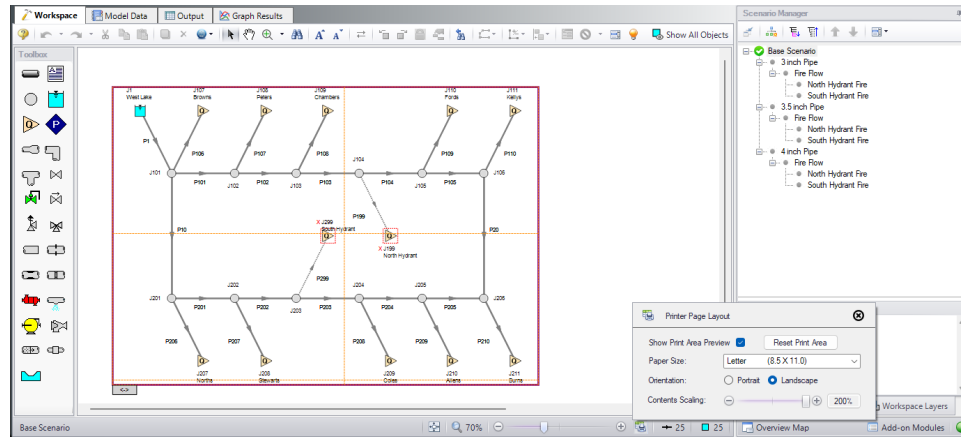
Figure 1: The Printer Page Layout options control how the model is split between pages on the Print Preview

- **Show Print Area Preview** - When checked, a purple box will appear on the Workspace which dictates what areas will be printed. Clicking on this purple box will cause sizing handles to appear which can be used to re-size the area. Additionally, the gray box in the lower-left of the purple print area can be used to drag and re-position the area relative to the objects on the Workspace. The purple print area is affected by the Snap to Grid option from the Arrange menu. Furthermore, dotted yellow lines may also be drawn within the purple print area depending on the other settings described below. The dotted yellow lines represent different pages that will be printed. 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 across the model, regardless of the size of the boxes, the model will be divided such that everything in a single yellow box will appear on its own page once the Print Preview is used.

- **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 Print Area Preview" 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 Print Area Preview" 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 dotted yellow boxes from the "Show Print Area Preview" are shrinking or growing; however, 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

Primary Window Overview

AFT Fathom 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 Fathom 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

Status Bar

AFT Fathom's Status Bar displays contextual information about the current scenario or operation.
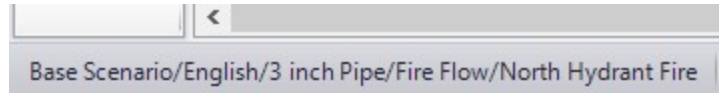
Figure 1: Current scenario displayed in the Status Bar

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.
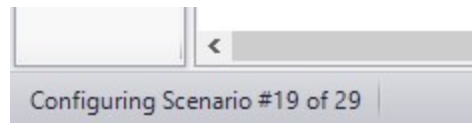
Figure 2: Status Bar displaying progress when opening a model

Related Topics

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
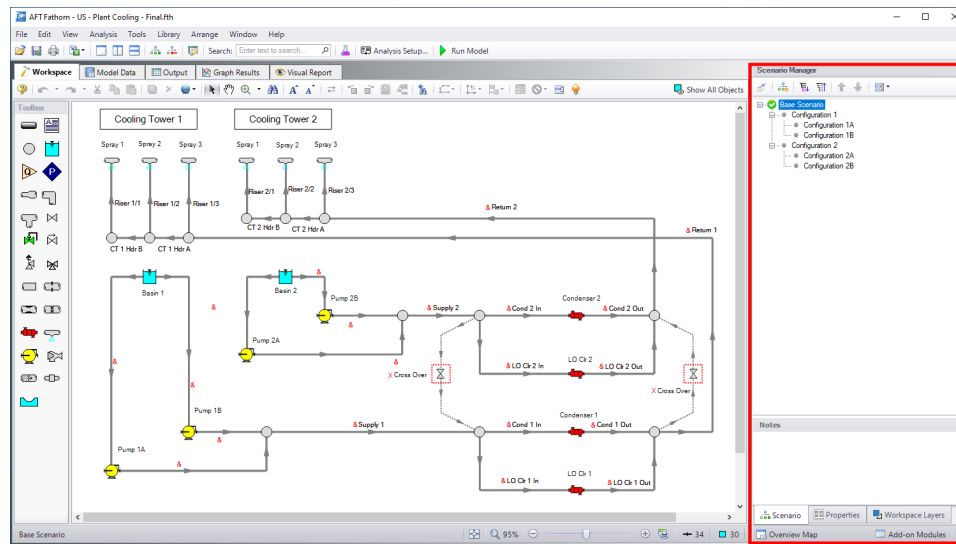
Figure 1: The Quick Access Panel

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.
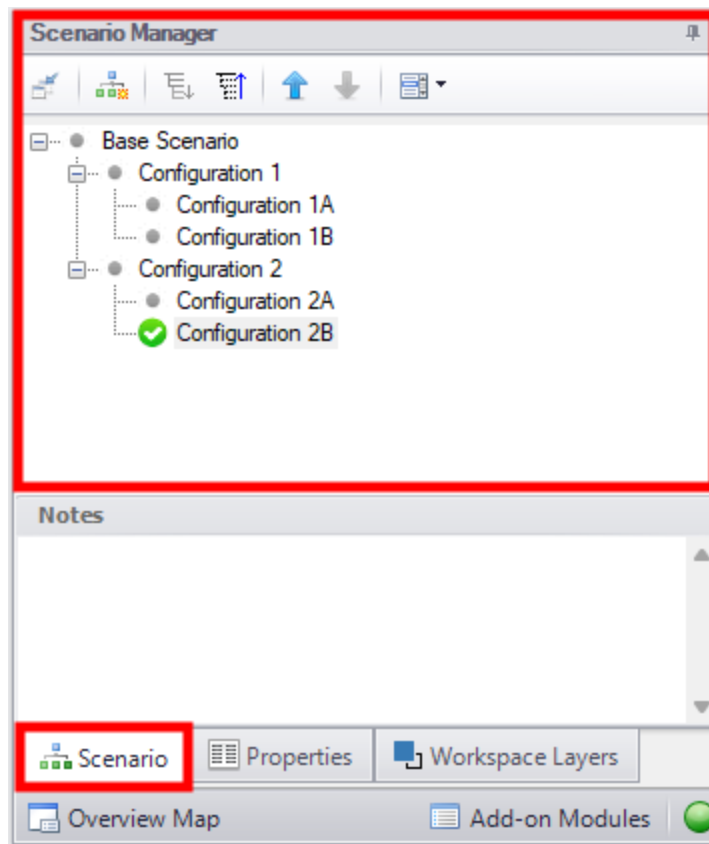
Figure 2: The Scenario Manager

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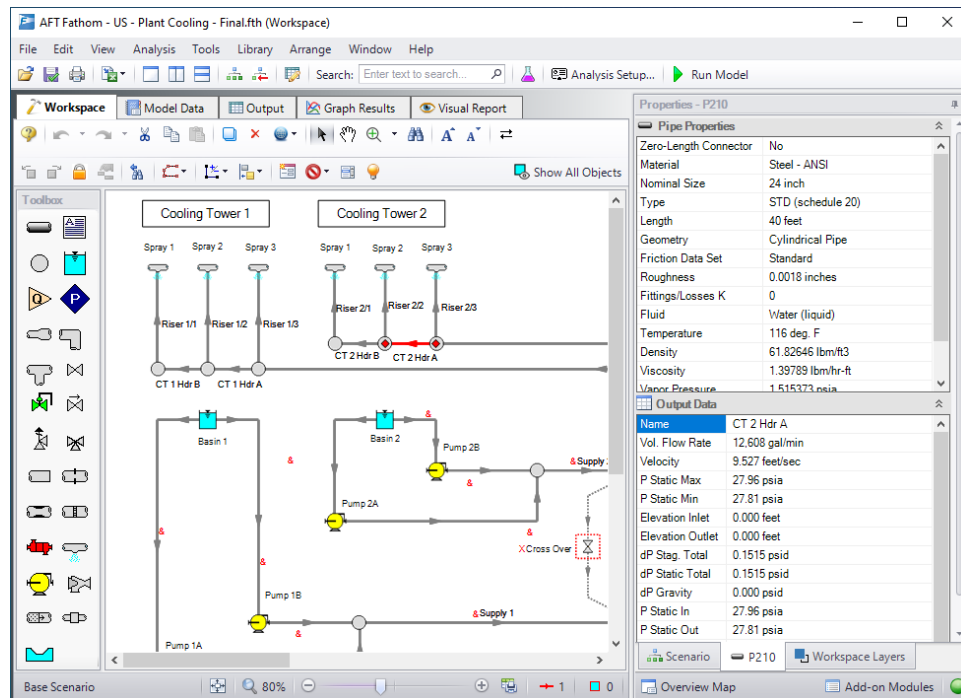
Figure 3: Viewing the Pipe/Junction Properties and Output Data from the Quick Access Panel

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.

Properties - Annotation 1

Enter text to search...   Defaults

Annotation (ID = 1)

Name	Annotation
Text	Group B
Text Alignment	TopLeft
Text Color	Black
Text Font	Microsoft Sans Serif, 18pt
Text Show	✓

Behavior

Anchor Object Tag	0
Anchor Type	Workspace
Workspace Allow Select	✓
Workspace Keep In Back	☐

Format

Border Color	Black
Border Style	Solid
Border Thickness	1
Shape	Rectangle 1:1
Shape Color	White
Shape Show	✓

Image

Image	
Image Keep Aspect Ratio	☐
Image Stretch	✓

Line

Arrow At End	✓
Arrow At Start	☐
Arrow Size	3
Arrow Style	Fill
Line Color	Black
Line Show	☐
Line Start Location	TopLeft
Line Style	Solid
Line Thickness	1

Scenario   Annotation 1   Workspace Layers

Figure 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.
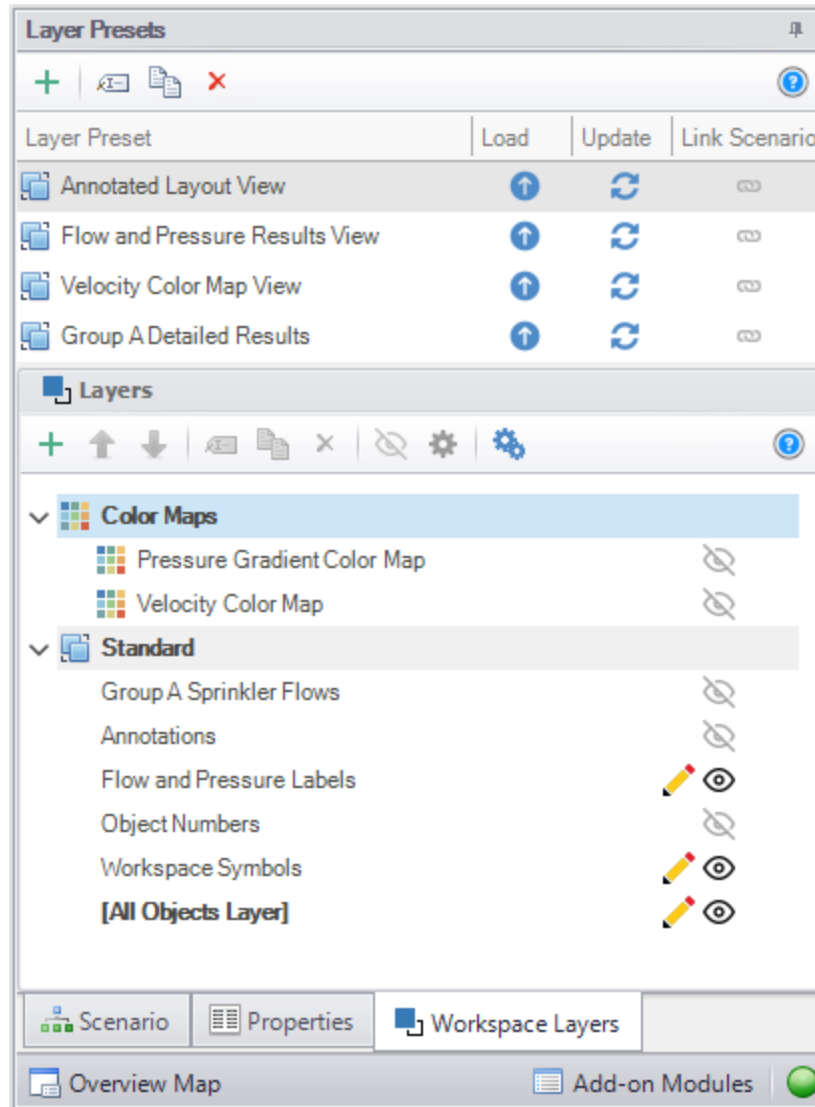
Figure 5: View of the Workspace Layers panel structure

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.
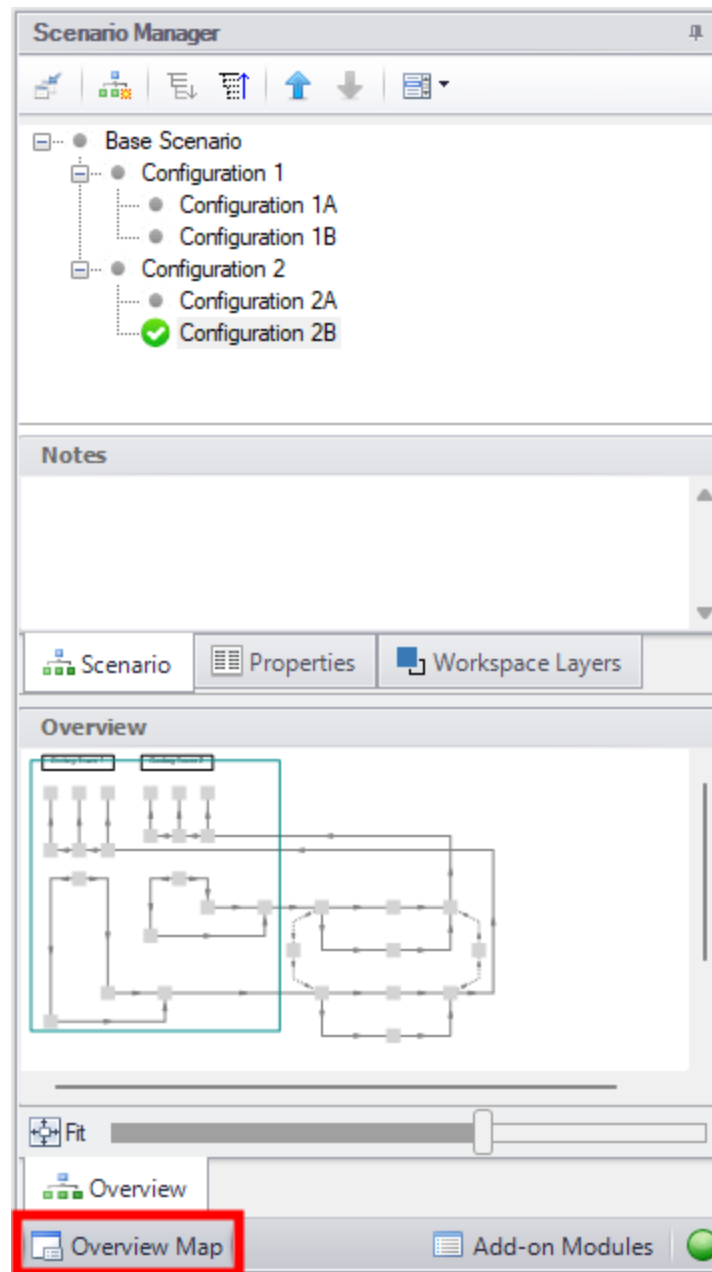
Figure 6: Workspace Overview Map

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.
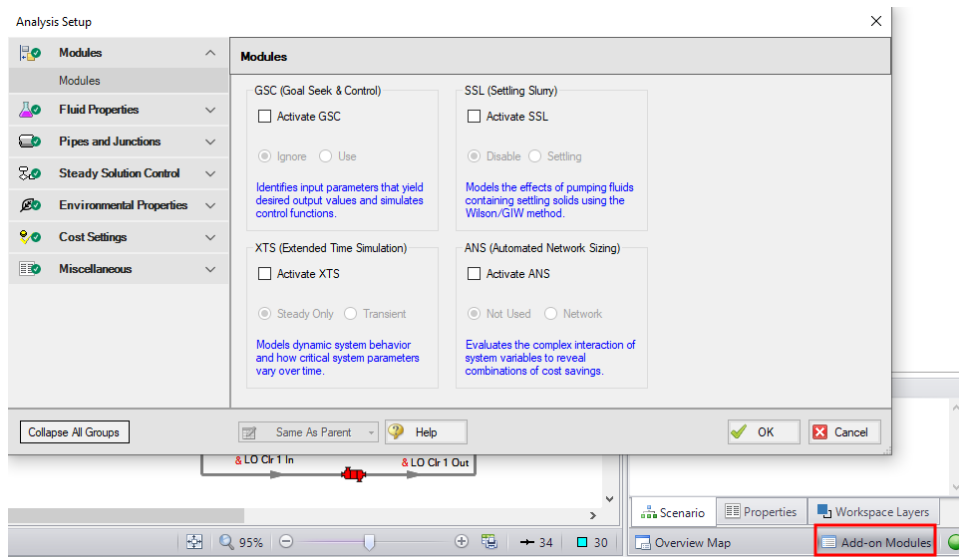
Figure 7: Activating the Modules from the Quick Access Panel

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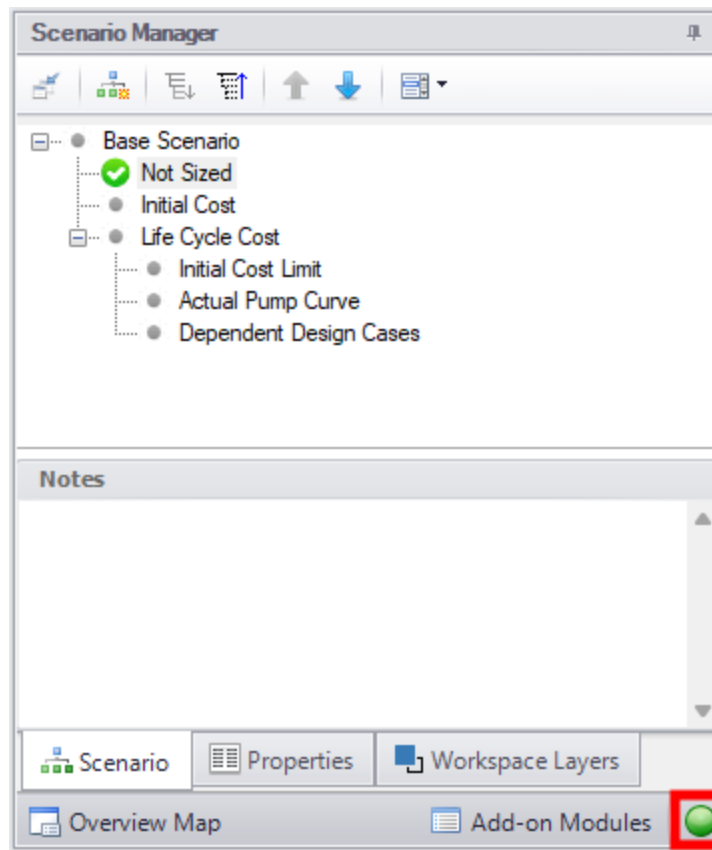
Figure 8: The Model Status Light is located at the bottom-right of the Quick Access Panel

Minimizing the Quick Access Panel

By default, AFT Fathom 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 Fathom 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.
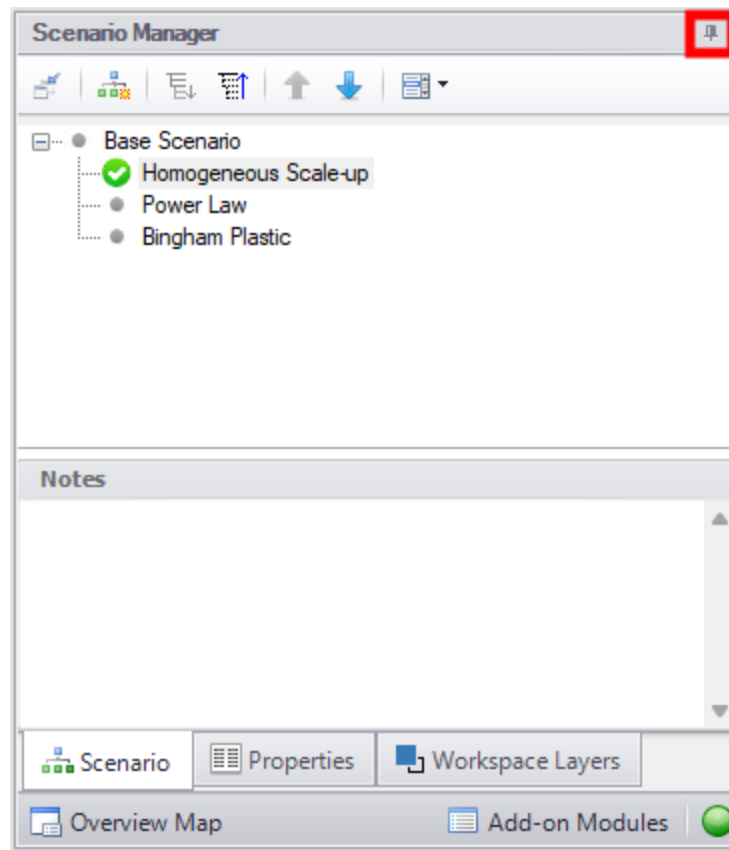
Figure 9: The Workspace with the Quick Access Panel expanded

Figure 10: The Workspace with the Quick Access Panel collapsed

Related Topics

Model Status Light

The light at the right of the Status Bar located at the bottom of the AFT Fathom window, which indicates whether or not the model can be sent to the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system).

Clicking on the Model Status Light will show the Analysis Setup window, which indicates incomplete inputs with a red exclamation mark: !

Related Topics

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

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).
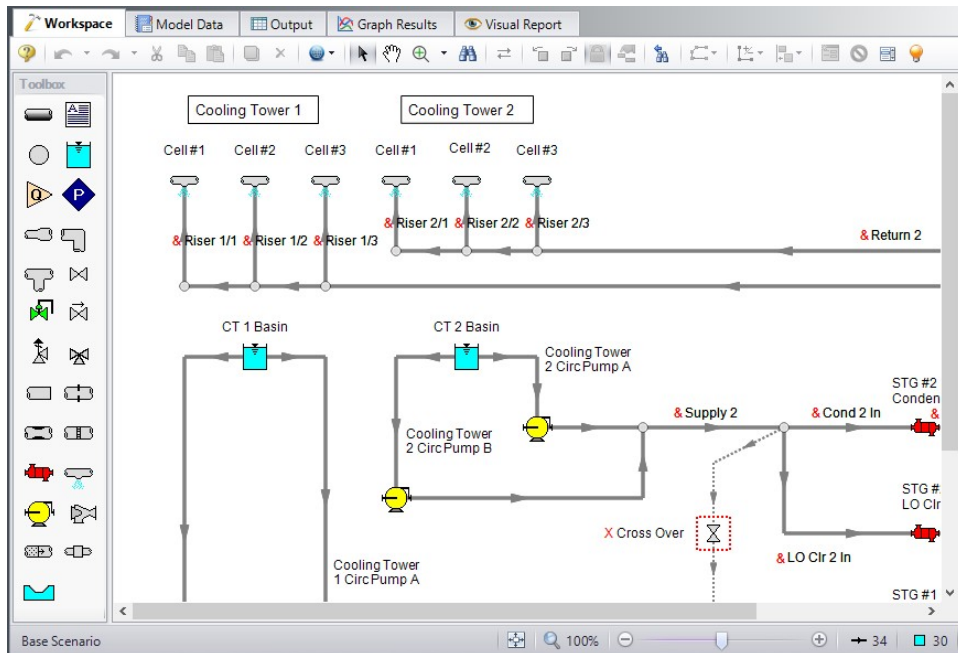
Figure 1: The Workspace window

Printing

The Workspace image can be:

- Copied to the clipboard (Note: cannot be done from print preview)
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The Print Preview window allows the above outputs to be applied to all or certain selected Workspace objects.

Related Topics

Workspace Layers

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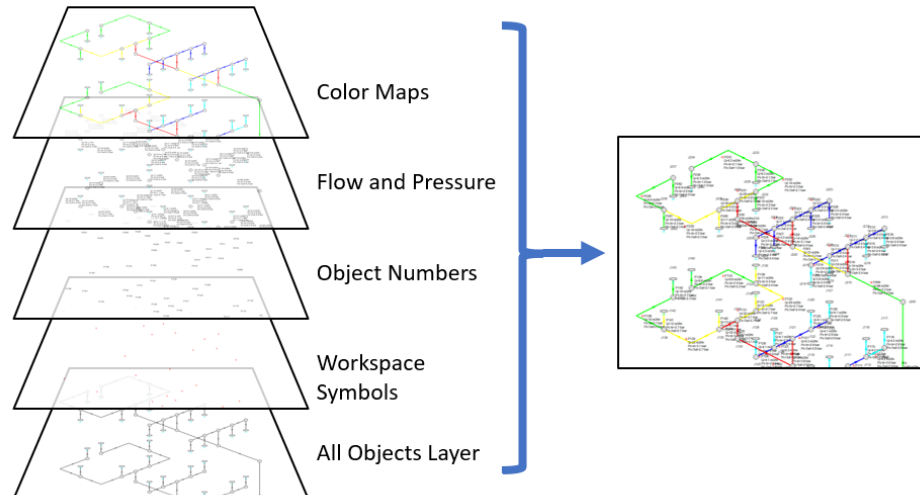
Figure 1: Information from all currently visible layers combines to form the view displayed on the Workspace

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.
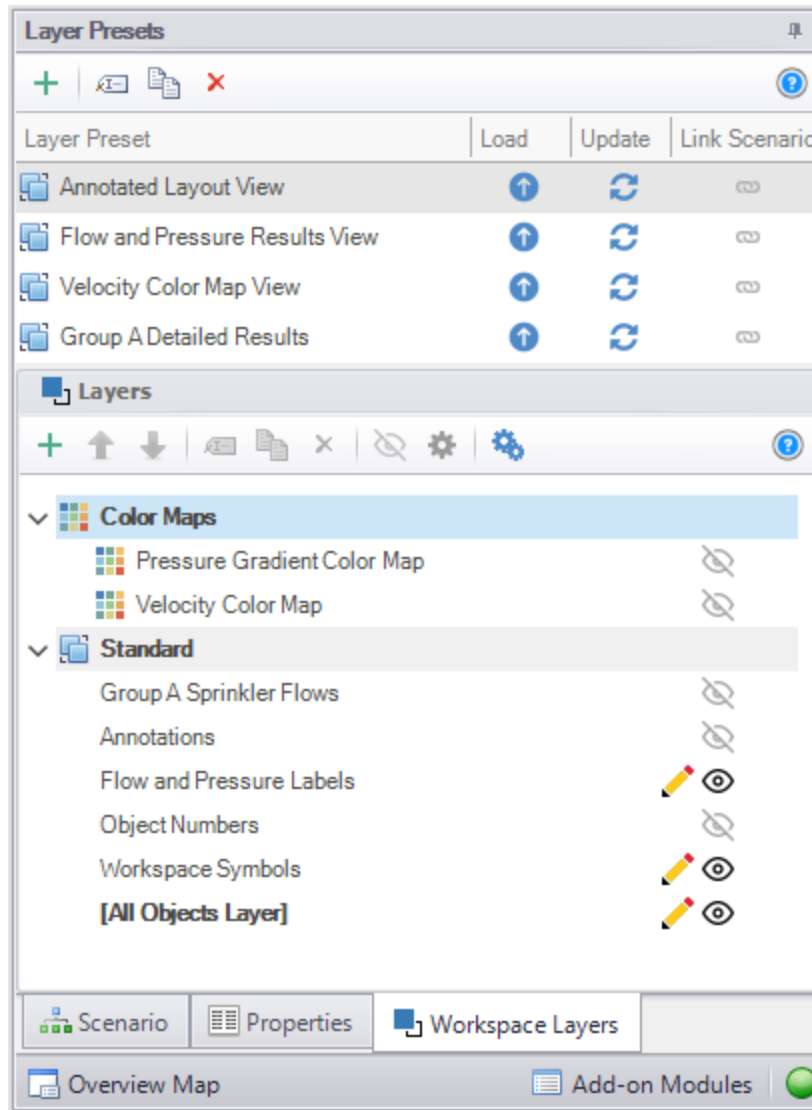
Figure 2: View of the Workspace Layers panel structure

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
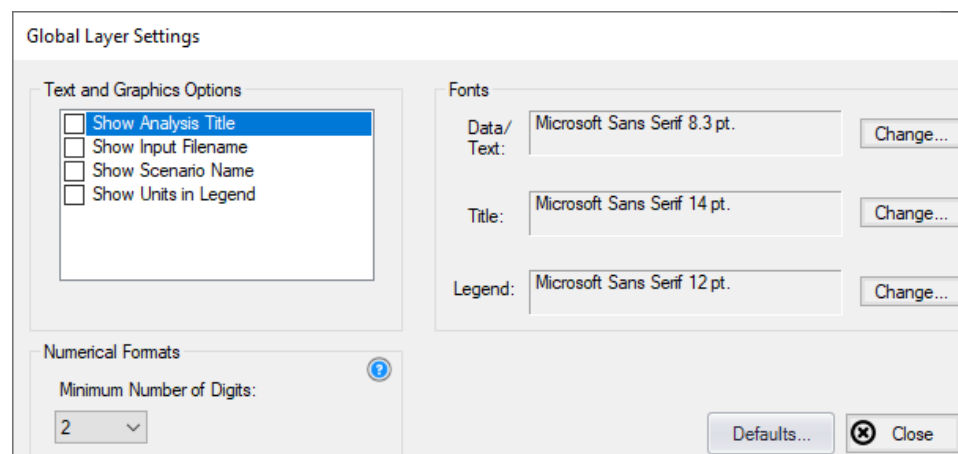
Figure 3: Global Layer Settings window

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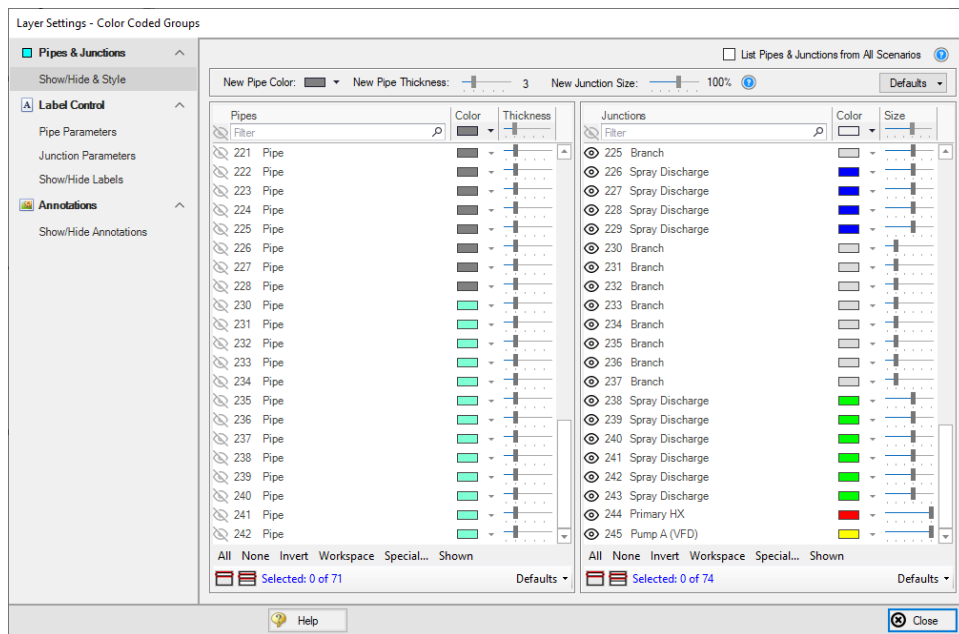
Figure 4: The Show/Hide & Style panel allows the appearance and visibility of pipes and junctions to be managed

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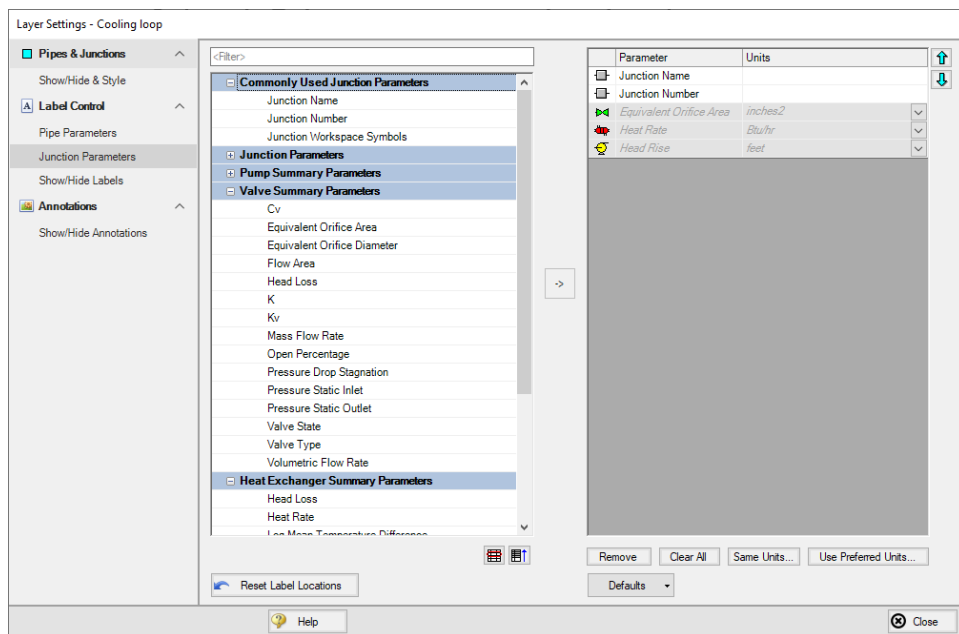
Figure 5: The Junction Parameters panel within the Label Control group of Layer Settings

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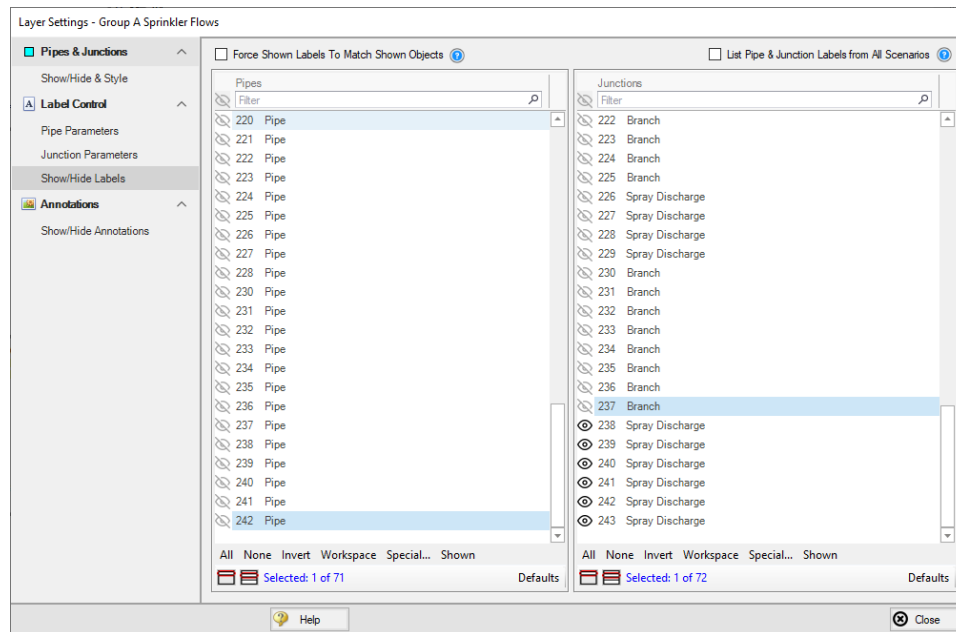
Figure 6: The Show/Hide Labels panel determines which objects will have visible labels for a given layer

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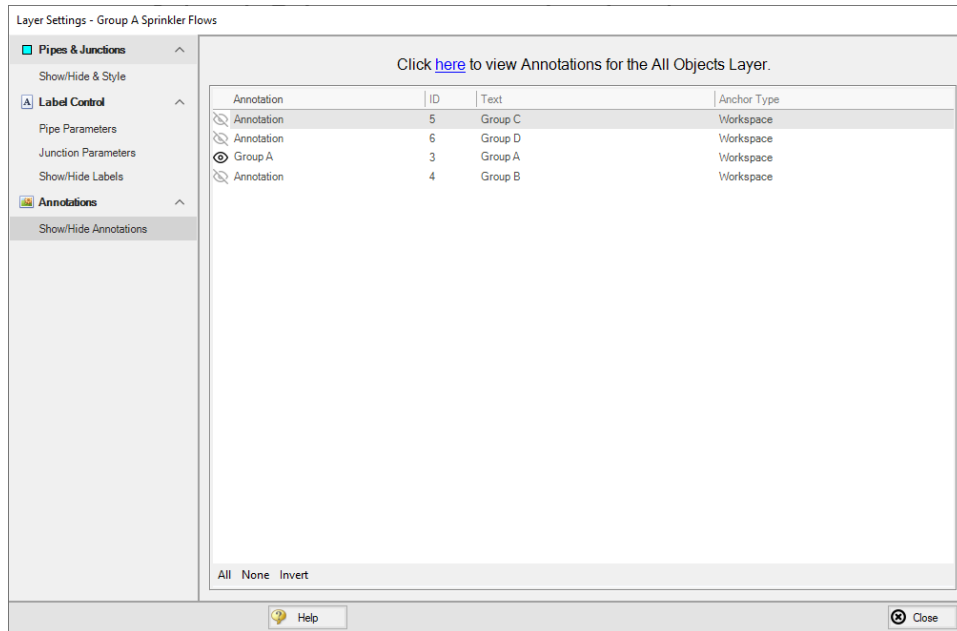
Figure 7: The Show/Hide Annotations panel shows all annotations that exist in the model file and allows their visibility to be controlled on a per-layer basis

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.
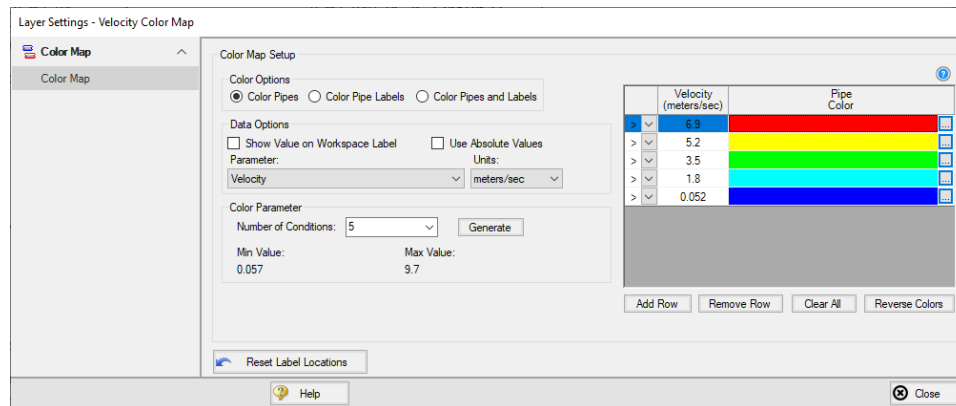
Figure 8: Layer Settings window for a Color Map

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. >, >=, =, <=, <) 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. Fathom 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.

Save/Load Layers to File

There are two buttons on the Layers Toolbar that can be used to save or load Layer Template Files which enable teams to create consistent layer formatting across different models.

Save Layers to File - Specific layers can be selected, including Color Maps/Animations, which will be exported to a Layer Template File (*.ltf). You can optionally choose for Global Layer Settings to be saved in the template file. The All Objects Layer cannot be selected or saved.

Load Layers from File - A Layer Template File (*.ltf) can be imported which will add layers on top of any existing layers in the model and overwrite the Global Layer Settings if they were selected to be saved in the template. If the template file contains parameters that are not available then they will be ignored and not loaded. It is also possible to load old Visual Report Control Files created from previous versions of the software. Model files themselves can also be selected as a supported file type which will copy all available layers from the model.

Workspace Layers Import Assistant

The Workspace Layers Import Assistant can be used when importing a model from Fathom 12 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

Annotations

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

New annotations are created using the Annotation Tool in the Toolbox. The Annotation Tool is located to the right of the Pipe Drawing Tool. With this tool, the user can draw a line or a shape on the Workspace. Shapes can display an image or text on the Workspace or can be used without text to display boundaries around various regions of the model.
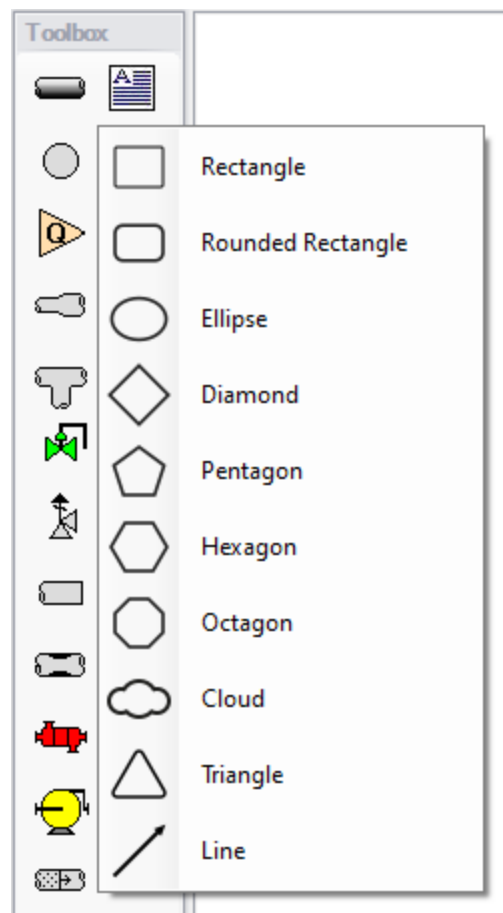
Figure 1: After clicking the Annotation Tool, select the shape for the new annotation

Properties

Properties - Annotation 1

Enter text to search...   Defaults

Annotation (ID = 1)

Name	Annotation
Text	Group B
Text Alignment	TopLeft
Text Color	Black
Text Font	Microsoft Sans Serif, 18pt
Text Show	✓

Behavior

Anchor Object Tag	0
Anchor Type	Workspace
Workspace Allow Select	✓
Workspace Keep In Back	☐

Format

Border Color	Black
Border Style	Solid
Border Thickness	1
Shape	Rectangle 1:1
Shape Color	White
Shape Show	✓

Image

Image	
Image Keep Aspect Ratio	☐
Image Stretch	✓

Line

Arrow At End	✓
Arrow At Start	☐
Arrow Size	3
Arrow Style	Fill
Line Color	Black
Line Show	☐
Line Start Location	TopLeft
Line Style	Solid
Line Thickness	1

Scenario   Annotation 1   Workspace Layers

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

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

- **Annotations** - The name of the annotation, its text content, text style, and text visibility can be edited within the Annotations section. The name of the annotation is not displayed on the Workspace but is shown within the Layers Settings where it can be used to communicate the purpose of an annotation.
- **Behavior** - The Behavior section allows the behavior of the annotation to be customized. Annotations can be attached to the Workspace, a pipe, or a junction. When attached to the Workspace, the annotation always remains in the same location until you move it. When attached to a pipe or junction, the annotation will move when the pipe or junction is moved.

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

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

Interaction with Scenarios and Layers

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

Related Topics

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.

Fathom 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

Merging Models

Two models can be merged together using the command on the File menu. Here's how:

- Open the first model
- Choose Merge on the File menu
- From the file list, select the model to be merged

If there are duplicate numbers, the objects in the second model will be changed. The second model will be selected (highlighted) so that you can move it into position (be careful not to click off the selected portions or the second model may be deselected).

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

Tip: To move a selected group of objects, use a junction icon, instead of a pipe, to drag the group. This will avoid missing the pipe and clicking on an empty area of the Workspace which will deselect your group.

Merging Models with Multiple Scenarios

If the model you are merging from has multiple scenarios, only the base scenario can be merged. If you want to merge one of the children of that model, do the following:

1. Open the model which you want to merge.
2. Open the Scenario Manager.
3. Select the child scenario you want to merge.
4. On the Other Actions button, choose the Save Scenario to File Without Children.
5. Enter a name and click OK. This process creates a new model with the desired scenario as the base scenario.
6. Open the original model and merge in the new model file created in Step 5.

Related Topics

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

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
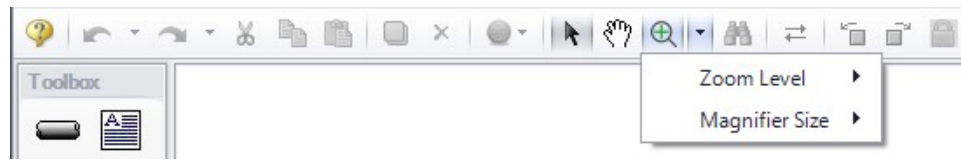
Figure 1: Magnifier tool on the Toolbar

Related Topics

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

Undefined Objects Window

This window is launched by clicking List Undefined Objects from the View menu.

This window displays all undefined pipes or junctions in the model. Clicking on the undefined object will display the undefined properties in the list below. All undefined data for all objects can be saved to the file or copied to the clipboard.

Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.

Related Topics

Keyboard Shortcuts for Workspace

Selection Box

- Select all objects completely inside box by dragging cursor from left to right
- Select all object completely or partially inside box by dragging cursor from right to left

Pipe Drawing Tool

- If you double-click the pipe drawing tool, it will allow you to draw multiple pipes at a time. Clicking Esc or clicking the pipe drawing tool again will deactivate the tool.
- When orthogonal drawing is off and you hold down the SHIFT key while drawing pipes on the Workspace, you can draw pipes that are perfectly horizontal or vertical. When the orthogonal drawing is on and you hold down the SHIFT key down while drawing new pipes on the Workspace the pipe vertex will flip. If you move an existing pipe and hold the shift key while selecting a handle at one of the pipe endpoints a new vertex will be inserted at that point endpoint.
- If you hold down the CTRL key when completing the pipe drawing (just before releasing the mouse button), the Pipe Drawing Tool remains active, and you can draw a series of pipes without returning to the Toolbox each time.
- To add pipe segments, use CTRL+SHIFT+Y
- To remove all pipe segments, use CTRL+SHIFT+N

Junctions

- If you double-click any junction in the Toolbox, you will be able to place multiple of that junction on the Workspace by clicking the Workspace where you would like to place each junction. Clicking Esc or clicking the junction icon in the Toolbox again will deactivate the tool.
- If you hold down CTRL key and select a junction from the Toolbox and drop it onto an existing Workspace junction, the old junction will "morph" into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.
- If you hold down the CTRL key then drag a junction on the Workspace which has pipes connected to it, the junction will move but the pipes will not move with it
- If you hold down the SHIFT key while selecting a junction from the Toolbox and then drop the junction onto an existing pipe, the pipe will split into two pipes. You will specify the length of the first pipe and the second pipe will be automatically set to the remaining length from the original pipe. Thus the sum of the two pipe lengths will equal the length of the original pipe. You can also choose which pipe contains any Fittings & Losses that were specified within the original pipe.
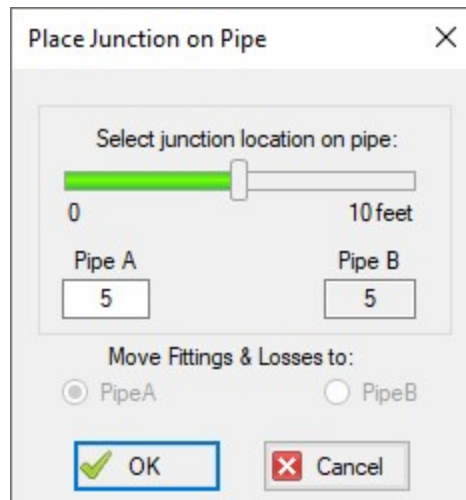
Figure 1: Split pipe keyboard shortcut

Pipe and Junction Properties Window

- When in a pipe or junction Properties Window you can toggle the highlight feature by pressing the F2 function key. You can also toggle this from the View menu.

- When in a Pipe Properties window, if you press the F5 function key, you will jump to the next highest numbered pipe. If you press the CTRL key while pressing the F5 function key, you will jump to the next lowest numbered pipe. The F5 function key also behaves this way for junction Properties Windows.

Pan and Zoom

- The Workspace position can be moved with CTRL + arrow keys.
- The Zoom level can be adjusted with CTRL + (+/-)
- The Zoom level can be reset to default with CTRL + 0
- Mouses with scroll wheels can be used to zoom by scrolling, or to pan by clicking and holding the scroll wheel

Switching Primary Windows

- Go to the Workspace window using CTRL + W
- Go to the Model Data window using CTRL + M
- Go to the Output window using CTRL + U
- Go to the Graph Results window using CTRL + G
- 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
- Redo the last action: CTRL + Y
- 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

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. Note that this clears Output for the current Scenario.

Related Topics

Raw Data Changes

When other objects in the model use the same base raw data as the current object and the data is changed, you can decide whether to use the new data on the other model objects.

This frequently happens when you have multiple junctions based on the same x-y data for pump curves or junctions resistance curves, and later change that x-y data. The question is whether you want to have the current junction you are working on updated, or all junctions that use that same base data.
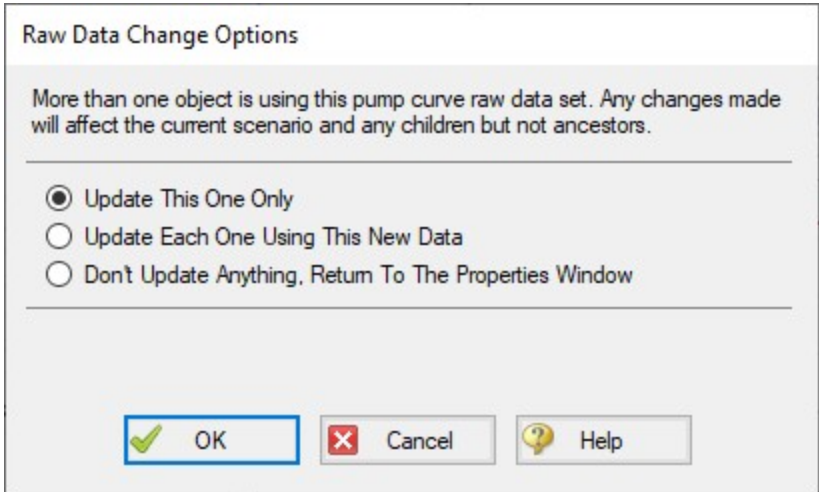
Figure 1: Raw Data Change Options

Related Topics

Model Data Window

The Model Data window offers a text-based perspective on all engineering information in the pipe flow model. This window is immensely useful for obtaining a global view of the model and rapidly checking the input.

The Model Data window does not offer any tools to build or add to a model; all model assembly must happen in the Workspace.

The Model Data window allows you to manipulate *existing* elements from the Workspace. Therefore, a complete model can be assembled in the Workspace window without ever opening the Model Data window. In fact, for small models this may be the preferred approach.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to 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.
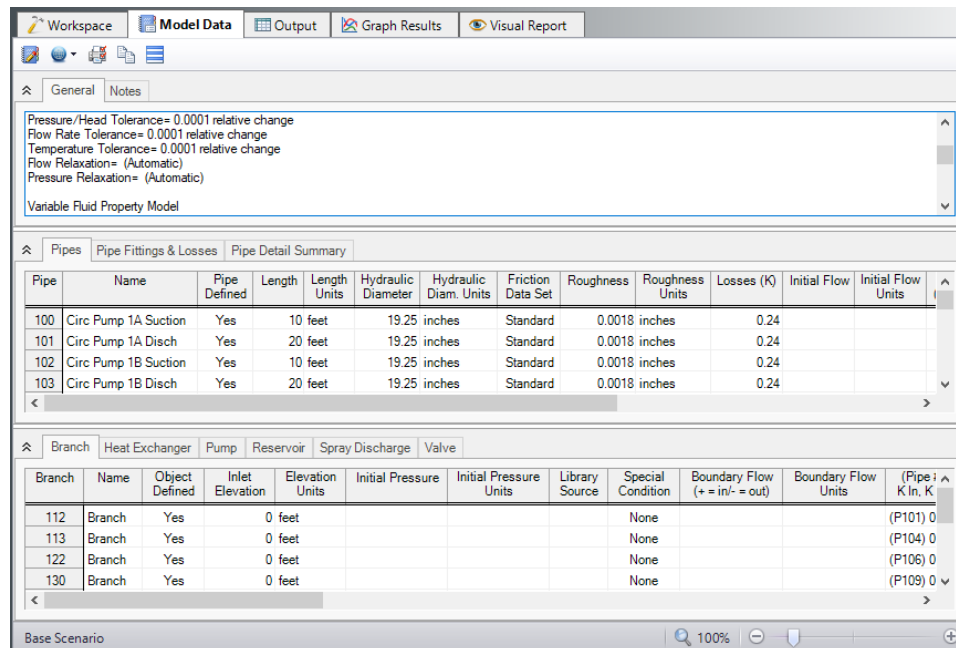
Figure 1: The Model Data window displays input in text form

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 anywhere in the row for the pipe or junction to be edited. Alternatively, you can use the Global Edit windows opened from the Edit menu.

- **XTS Module tables** - When using the XTS Module, a Transient Data table in the Junction area displays all transient data entered for junctions. If there is no transient data, the table will not be displayed.

- **GSC Module tables** - When using the GSC Module, if GSC data has been entered in the Goal Seek and Control Group in Analysis Setup, the variable and goal data will be displayed in the General section in a "Goal Seek & Control" table.

Display and Printing Features

Model Data Control lets you specify the combination of pipe, junction, and general information you want to see in the Model Data Window.

The Model Data Print Content window lets you specify the content to include in the printed Model Data report and select the font to be used for printing. This window is opened from the Tools menu (or Toolbar).

The Model Data window contents can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Saved directly to an Adobe PDF file
- Sent to the printer

Related Topics

Model Data Control

The Model Data Control allows you to customize what data to show for the pipes and for each junction type and also which pipes and junctions to show (see Figure 1). It can be opened from the Tools menu (or Toolbar).

Pipe Data Display

The data to be shown for the pipes is selected on the left side of the Model Data Control window. Only the items with check marks will be shown in the Model Data window. All pipes shown will report the selected items.

Junction Data Display

The data to be shown for the junctions is selected on the right side of the Model Data Control window. The data is grouped by junction type. However, some data is general and applied to each of the junction types. Select the junction type from the list and then select the data for that junction type. Each type also includes the general data which may be selected/deselected.

You can select/deselect the general data for all junction types simultaneously by clicking Set Common.
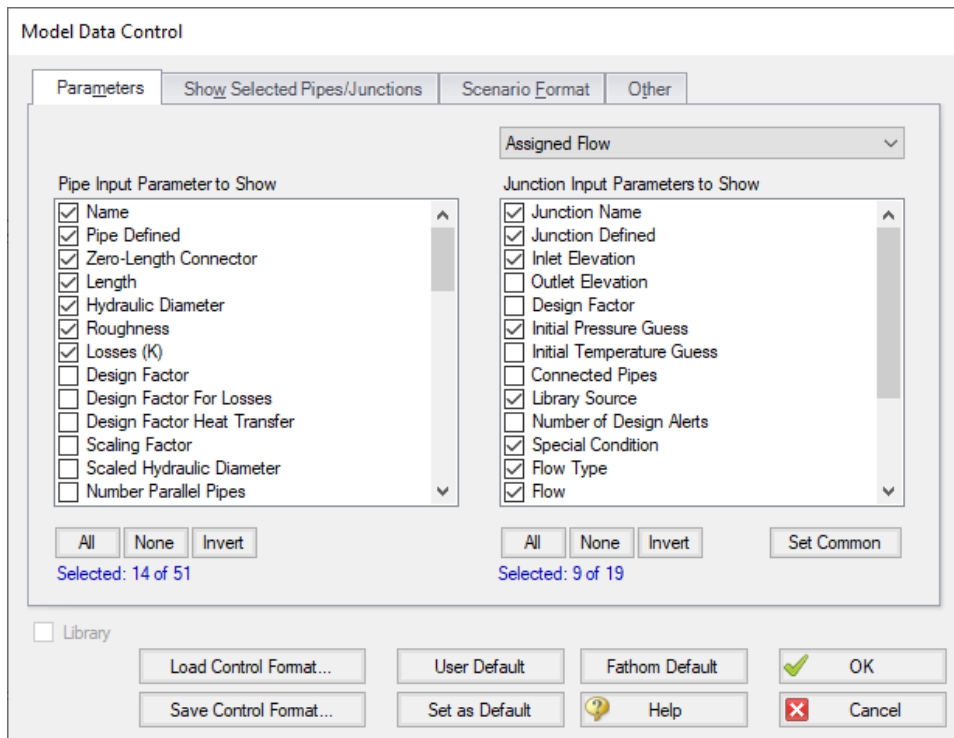
Figure 1: The Model Data Control window allows selection of the parameters to show in the Model Data window

Model Data Format

The Model Data Control format can be saved to a file so that the format can be quickly applied to other model files. There are multiple ways to do this. The Model Data Control format can be saved to the internal user library by clicking the Set As Default button, then choosing User Default in a new model file to apply the settings.

The Model Data format can also be saved to an external file, which can then be shared with different users. The external file can be generated by clicking the Save Control Format button at the bottom of the window. The format can then be applied to other model files by clicking the Load Control Format button.

Show Selected Pipes and Junctions

Specific pipes and junctions can be shown in Model Data. Click the Show Selected Pipes/Jct tab, then click the items in the lists to select the pipe(s) and junction(s) to be shown.

Other Data

When pipes with intermediate elevations are specified, you can specify which pipe data is to be displayed in the Model Data window:

- Show only pipes with intermediate elevations
- Show only non-horizontal pipes
- Show all pipes

In addition, you can specify whether to show Pipe Elevation Data, 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 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

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.
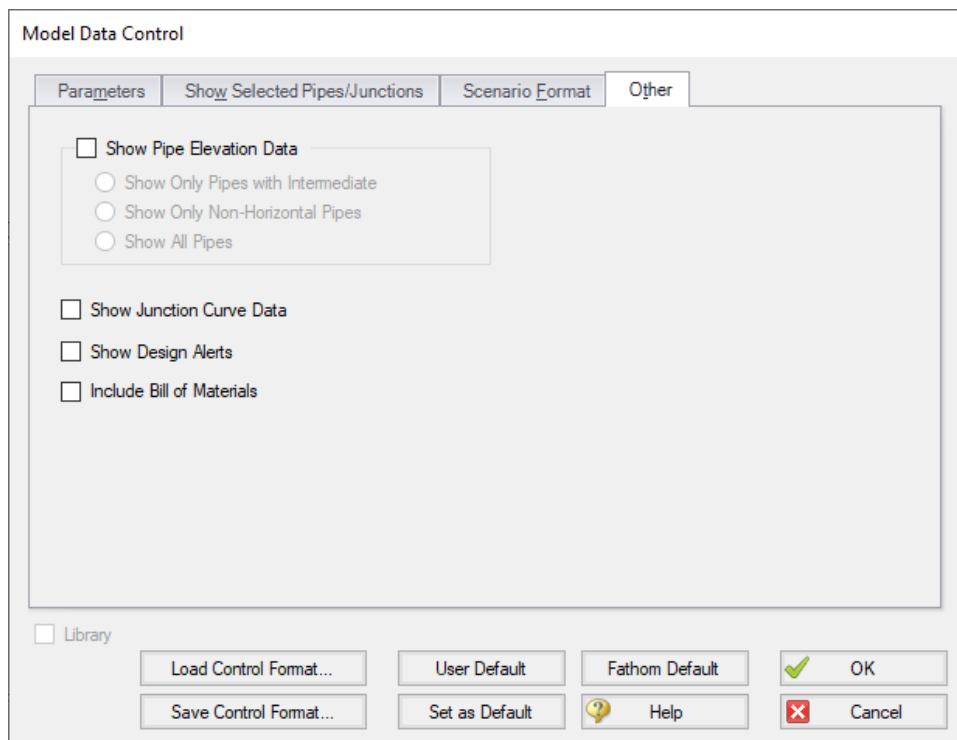
Figure 1: Include Bill of Materials

Related Topics

Output Window

The Output window is the primary vehicle for communicating the results of an analysis in text form (see Figure 1). The Output window follows the same general window organization as the Model Data window.

When a model is submitted to the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system), 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.

The Output Print Content window available on the Tools menu (or Toolbar) lets you specify the content to include in the printed Output report and select the font to be used.

The **Sort** window opened from the Arrange menu (or Toolbar) offers the ability to sort the Pipe Results or Junction Results tables according to the values in any of the columns.

The Transfer Results to Initial 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. To do this double-click the pipe or junction row, which 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.

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.
    - **Create Design Alert** -Create a Design Alert related to the selected object and value.
    - **Copy Selected Cell Data** - Copies the value of the cell to the clipboard.

Output Window Updates

Whenever you change the input model, all output windows are erased. This prevents you from changing your input model, forgetting to rerun it, and mistaking the previous Output window for the current model's results.

Results Diagrams

Results diagrams are available for both pump 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 pump summary tab for pumps or the heat transfer tab for pipes with heat transfer.
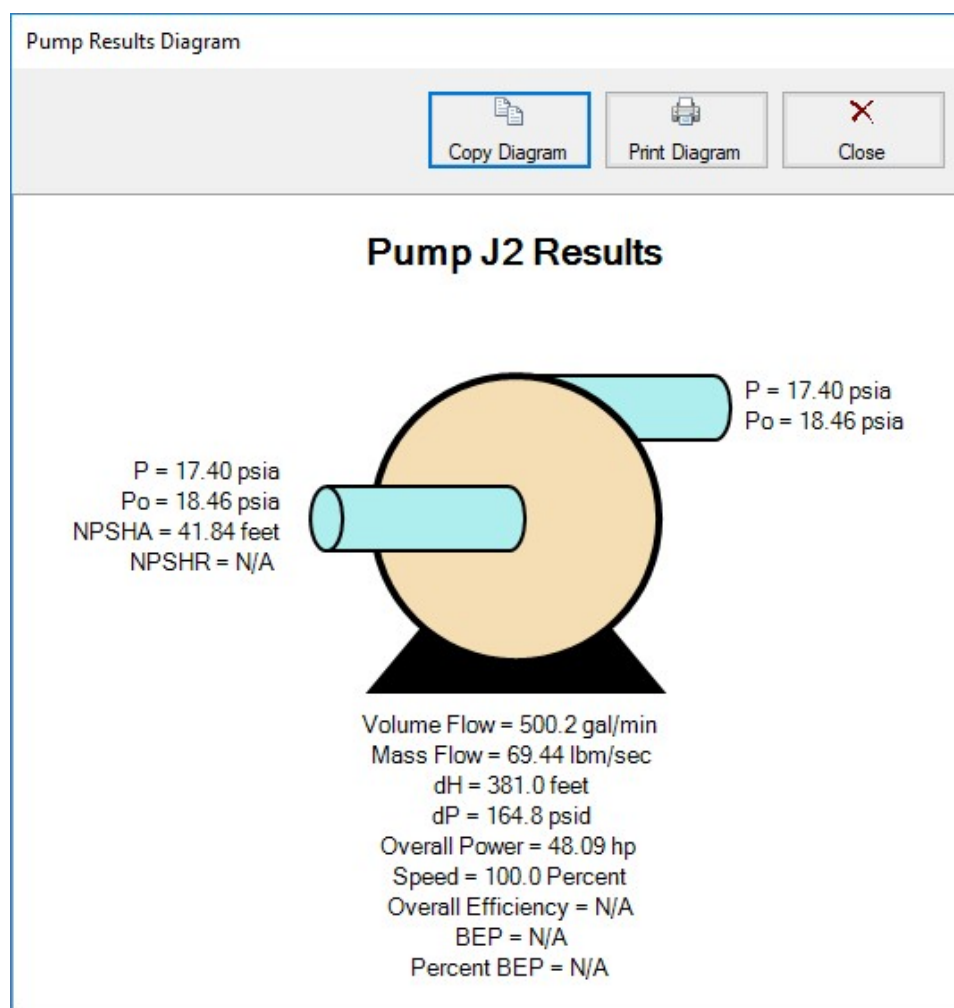
Figure 2: Pump Results Diagrams
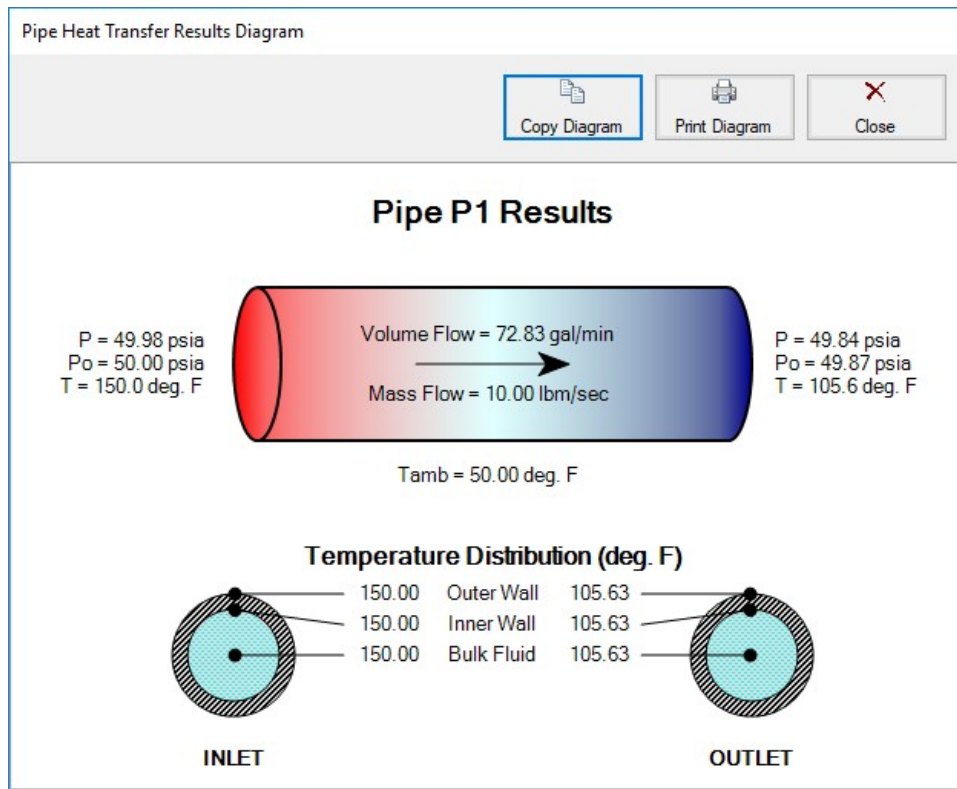
Figure 3: Pipe Heat Transfer Results Diagram

Printing and Exporting

Data in the Output window can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to an Excel spreadsheet
- Saved directly to an Adobe PDF file
- Sent to the printer (see the Printing topic)

The printed and Adobe PDF output follows the same format as the Output window itself.

Related Topics

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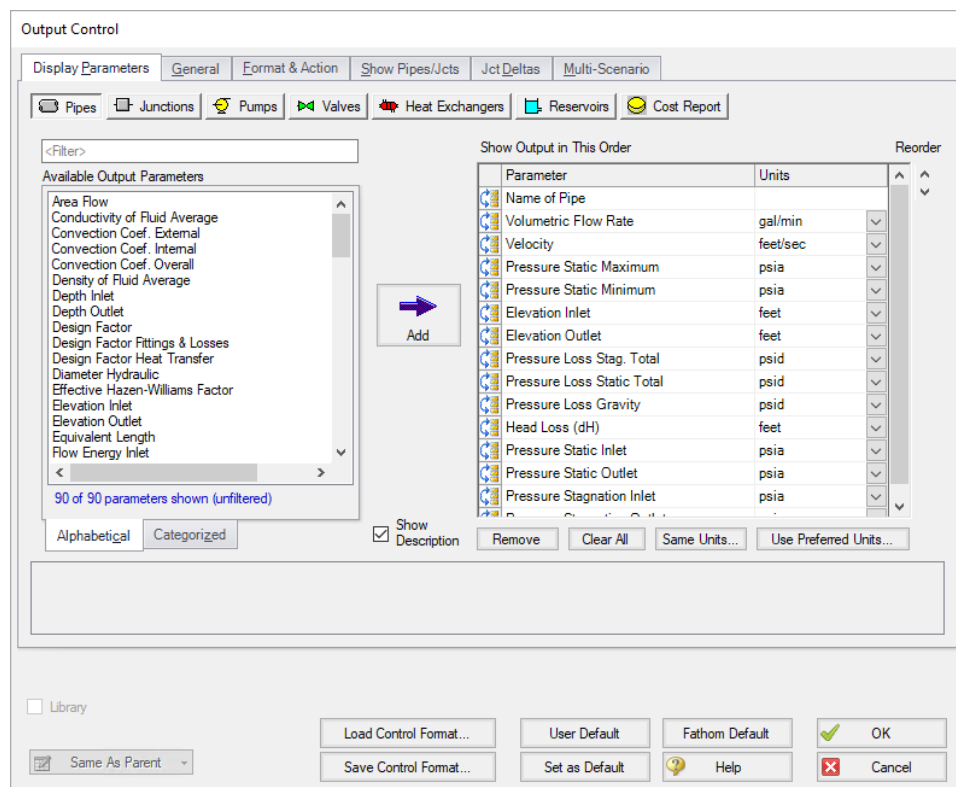
Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe parameters are shown. The Junction and other parameters function the same.

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.

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 input parameters from a library (such as Pipe Nominal Size).

Several Special Summary Reports can be displayed in the Output window. These include the following:

- **Pump Summary**
- **Valve Summary**
- **Heat Exchanger Summary**
- **Reservoir Summary**
- **Pipe Heat Transfer Output**
- **Cost Report**

In addition, if you are using the XTS module there will be transient output reports available for pipes, junctions and many of the summary reports. If you are using the SSL module, there will be a slurry report available and transient slurry report if XTS is also active. 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 as initial guesses, the iteration time for AFT Fathom to arrive at a converged solution is reduced. The initial guess can also be transferred manually by selecting Transfer Results to Initial Guesses on the Edit menu or Toolbar. The model can also be automatically saved after the initial conditions are transferred. The valve state (open, closed or failed) can also be saved along with the initial conditions.

When enabled, the Wrap Table Column headers option makes the output tables wrap the column headers 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.

Time Simulation Formatting

(XTS Module only) XTS output can be displayed in different units of time. In addition, when running transients over hours or days there are formats which display time in formats such as dd:hh:mm:ss.

Show Pipes/Jcts

You can choose which pipes and junctions are to be displayed in the Output window. Clicking a selected item in the list on the Show Pipes/Jcts tab will deselect it. Clicking a deselected item will select it. This feature is especially useful when you are interested only in a certain area of the model.

You can quickly define the output you want by doing the following:

1. Select the pipes and junctions on the Workspace
2. Open the Output Control window and select the Show Pipes/Jcts tab
3. Click the Workspace button for both the pipes and junctions
4. Click the Show button

Junction Deltas

You can have AFT Fathom 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 top left of 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 current scenario to be displayed by clicking the appropriate choice under the Display Current Scenario section. Figure 3 shows the Output window with multiple scenarios displayed.
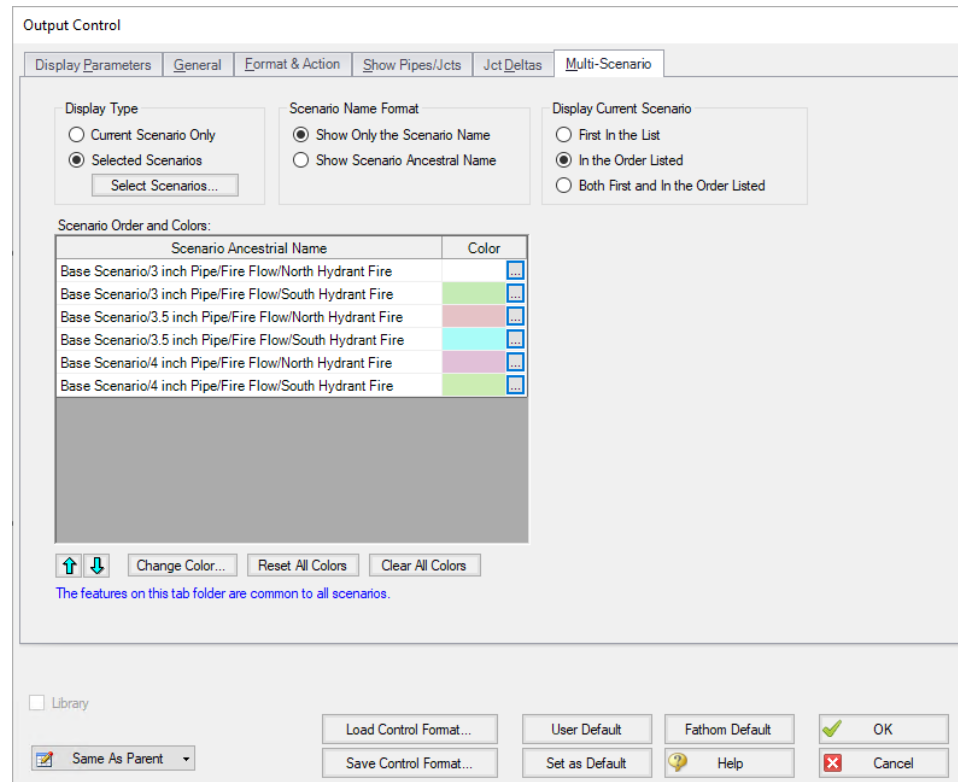
Figure 2: Output Control window selections for multi-scenario output

Figure 3: 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 con-nected. 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. Fathom has built-in default parameters, units and settings which you can choose by clicking the Fathom Default button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of Fathom's defaults). To make your own default, first select the output para-meters, units and settings you would like to use then click the Set As Default button. Your set-tings 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 set-tings 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, or incorporated into a company-wide library, 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 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

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.

Pressure Loss Static Friction Component

Static pressure drop due to the Fittings & Losses entered in the Pipe Properties window. Excludes pressure drop due to pipe wall friction.

Pressure Loss Static Friction Total

Static pressure drop due to friction from the wall and the Fittings & Losses entered in the pipe. This value will be equivalent to the stagnation pressure drop due to friction because the velocity along the pipe is constant.

Pressure Loss Static Friction Wall

Static pressure drop due to friction from the wall. Excludes static pressure drop due to friction from Fittings & Losses in the pipe. This value will be equivalent to the stagnation pressure drop due to wall friction because the velocity along the pipe is constant.

Volumetric Flow Rate Net At Junction

Displays the total volumetric 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 Volumetric 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.

Volumetric Flow Rate Through Junction

Displays the total volumetric 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 "Volumetric Flow Rate Net At Junction" to view flow exiting the system rather than flowing out of the junction into a pipe.

Related Topics

Applied Standards

When an applicable standard is used in a calculation, an additional tab appears in the general area of the Output Window. This tab indicates what standard(s) were used for what calculations in the model.
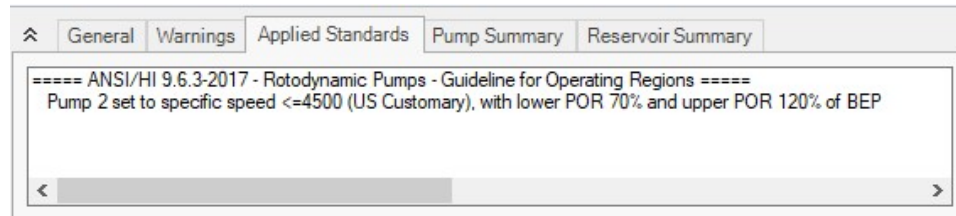
Figure 1: Applied Standards when Pump Viscosity Corrections are used in a model

Related Topics

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 pump vs. system, profile, transient (XTS Module Only), and slurry system (SSL Module Only) information. Users can also control graph content, plotting information for multiple parameters, pipes, paths, or scenarios simultaneously.

Results can be graphed as linear or log versus linear.
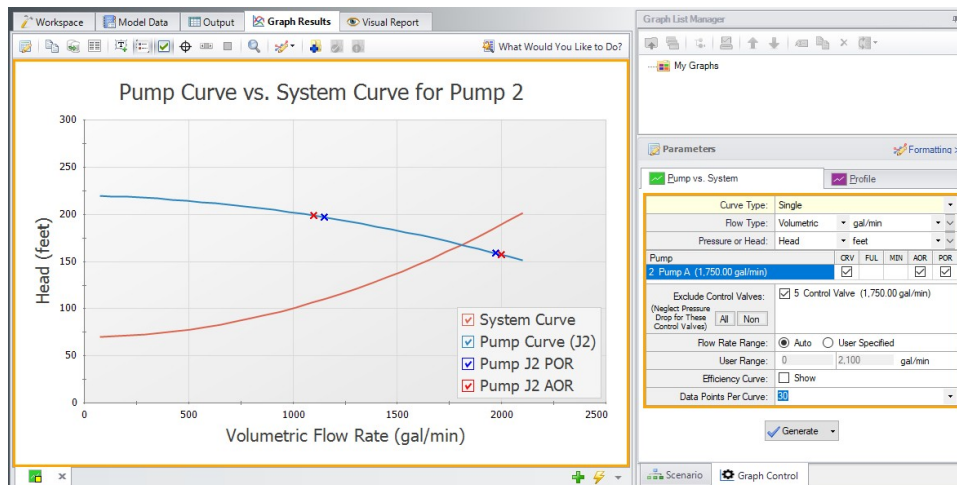
Figure 1: The Graph Results window offers full-featured plot generation

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 graph highlighted in the Graph List with the current graph that is shown.
- **Copy Graph Image** – Copies a .jpeg of the Graph to the clipboard
- **Copy X-Y Data** – Copies the X-Y data 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.
- **Manage Junction Locations** - Allows the user to show/hide the labels for each junction on the profile graph when Show Junction Labels is enabled.
- **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 the 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** – Pump vs. System
- **Profile** – Purple
- **Transient** – Light Blue
- **Slurry System Curve** – Pink
- **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 which Graph tab will be active (shown) based on the Graph Name. The user can also switch tabs by clicking on the desired tab.

Profile 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 command buttons on the Graph Results File window.

The Graph Results window is always erased when you make changes to model input. Rerun the model to graph the new results.

Multiple paths in the same scenario can also be plotted against each other by adding each path to a group, then choosing the "Plot Multiple Paths Using Groups" option as shown in Figure 3.
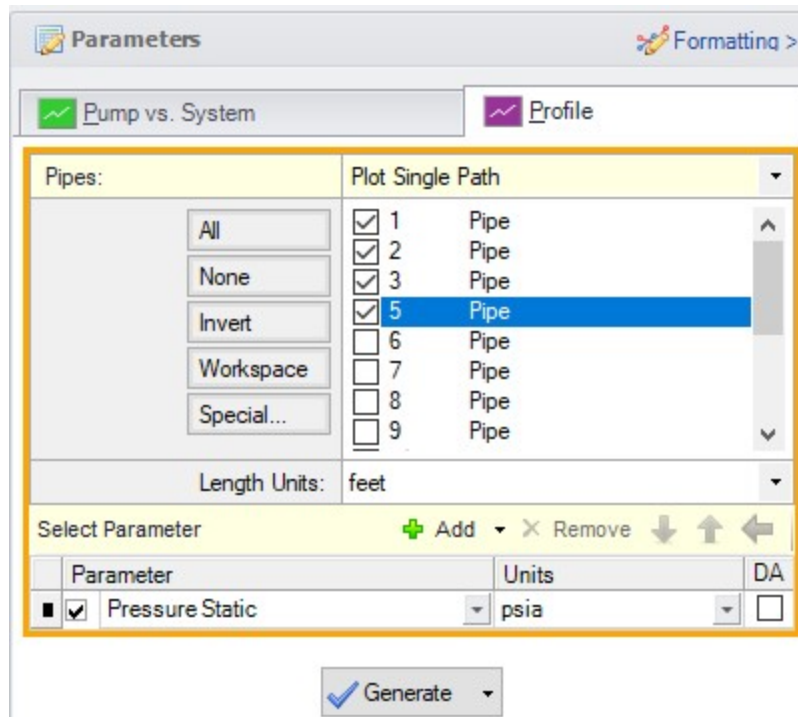
Figure 2: The Select Graph Parameters window controls the Graph Results window
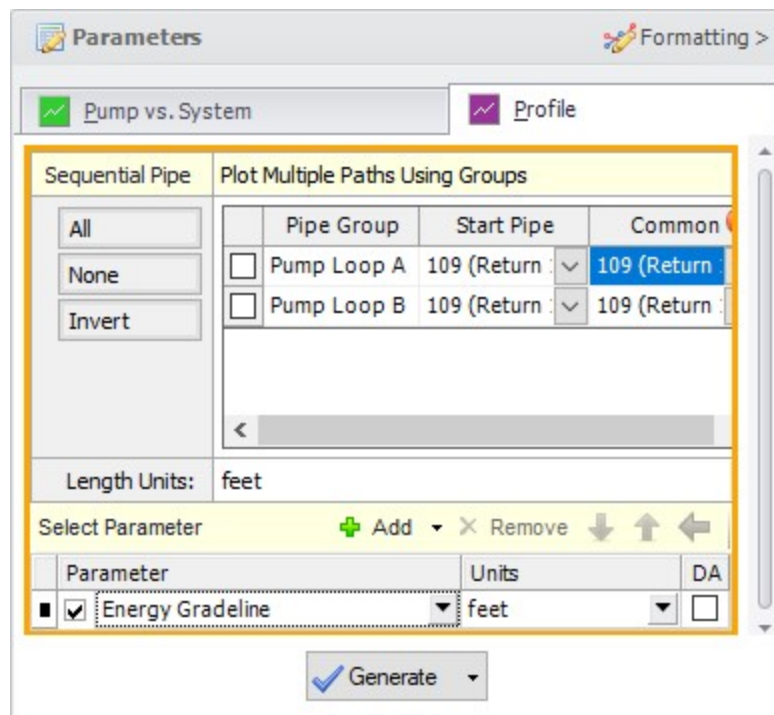
Figure 3: Multiple paths can be cross-plotted using predefined groups

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

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 Fathom Default or User Default buttons at the bottom of the window.

Related Topics

Graph Types

AFT Fathom offers several types of graphs to help users better understand and interpret their data. Each type of graph is given a brief description below.

Pump vs. System Curve

This graphing option allows you to plot a pump curve versus a system curve for a single pump or group of pumps.

Note: The Pump Curve vs. System Curve is only available if there is a pump in the model

You can create pump vs. system curves for complex parallel flow networks. Composite pump curves for pumps in parallel or in series can also be also generated. If you create a composite pump curve, you can also show single pump curves on the same graph. If efficiency data exists you can cross-plot the efficiency curve.

If POR or AOR have been defined in Pump Configuration, these items can be displayed on the Pump vs. System graph.

If control valves are present in the model the control valves will be excluded from the Pump vs. System curve by default. Excluding the control valves will set them to be fully open for all flow rates. This option is useful to prevent convergence issues.

Pump vs. System curves can be displayed in two ways when parallel pumps are selected. One way is the "traditional" method which is how AFT Fathom displayed them in AFT Fathom 6.0 and before. The second method is called the "enhanced" method and it attempts to determine the flow split on the pumps and adjust the system curve accordingly. It also attempts to create a better composite curve when parallel pumps are dissimilar. For more information see Pump vs. System Curve Methods.

Pump vs. System curves for different scenarios can be plotted together on one graph by adding individual pump vs. system curves to a Multi-Scenario Pump Vs. System Folder, then loading them together in a single Multi-Scenario graph. For detailed information on how to do this see Multi-Scenario Pump 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 several 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. To do this change the profile plot from "Plot Single Path" to "Plot Multiple Paths Using Groups" at the top of the parameter selection area. 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 Fathom 12, users can plot information from multiple scenarios in the same graph for profile plots (or Transient plots with the XTS Module). Note Multi-Scenario Pump vs System curves are generated using a different method.

In the Select Graph Parameters window, the button for 'Multiple Scenarios' allows users to select the scenarios in the model they wish to plot for a particular graph. The 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, Fathom 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.
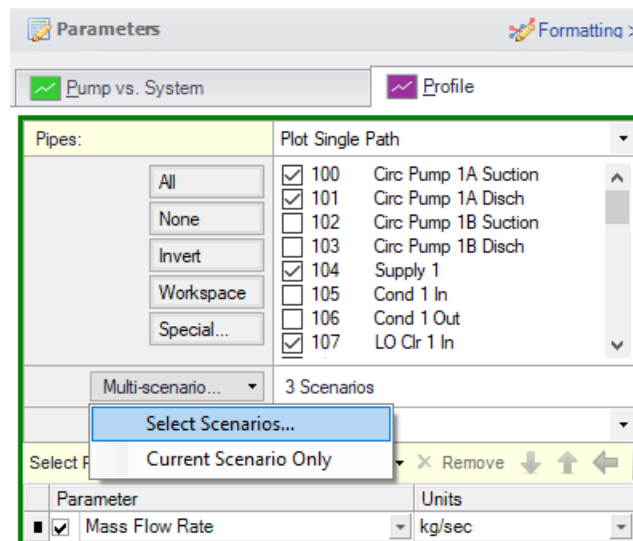
Figure 1: Graph Parameters window showing Multi-scenario selection options

Transient

([XTS Module Only](#)) The Transient allows you to create plots of pipe and junction output with time as the independent parameter. See [Transient Graph Results](#).

The overall maximum and/or minimum values can be plotted. Both of these graph types support [animation](#).

To graph the results at a particular point in time, select the animation option and use the tools provided to display the desired time.

Slurry System Curve

([SSL Module Only](#)) Special system curves can be generated for slurry modeling. See [Graphing Slurry Results](#).

Related Topics

Pump vs. System Curve Methods

When plotting pump and system curves with two or more parallel pumps, AFT Fathom offers two methods: Enhanced and Traditional.

It should be understood by the engineer that it is not possible to create an unambiguous composite pump and system curve for parallel pump systems. The reasons why are discussed here.

First, let's discuss the head curve. A composite pump curve shows a single head curve with values at particular flow rates. However, when there are parallel pumps with unequal head curves or even equal head curves with unequal flow distribution, it is not clear what flow rate to use in the composite head curve. Further, with unequal flow distribution, different heads for each parallel pump can result. But the composite head curve shows only a single head for any given flow. What head 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 head.

Composite system curves have similar issues to composite head curves. The entire system performance is boiled down to a single head vs. flow rate curve. What flow rate is being referred to and what head is being referred to which goes with a given flow rate?

AFT Fathom attempts to create as meaningful and helpful composite pump and system curves as possible, but the fact remains that the curves AFT Fathom creates are ambiguous because of the nature of composite head vs. system curves.

Enhanced

The "enhanced" method was introduced in AFT Fathom 7.

For composite head curves, it assumes that the parallel pump head curves are added to the right with respect to flow rate. No attempt is made to apportion flow splits among the pump curves. With the head curves added to the right, there is a unique head value for every flow rate. In reality, the parallel pumps may not work this way, depending on the flow split among the pumps and whether or not the head curves are identical and whether there are check valves at each pump discharge. This is, however, a reasonable approximation and represents the historical way that parallel pump head 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 pump and then determining the required head from the pump 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 head for each pump at each flow rate and then averages these heads together arithmetically.

If the pump head curves are identical and the flow split among pumps is proportional, this method will give the same results as the Traditional method.

Traditional

The "traditional" method was used in AFT Fathom 5.0 and 6.0. The traditional method assumes that the flow splits equally among all pumps at all flow rates, regardless of whether pump curves are identical or not.

The composite head curve is created by taking the composite flow rate, splitting it equally among each pump, obtaining the head for each pump at the split flow rate, and then taking the arithmetic average of the pump heads.

The composite system curve is obtained by taking the composite flow rate, splitting it equally among each pump, running the system model with these flow rates, and then taking the arithmetic average of the required heads for each pump.

Related Topics

Basic Graph Tutorial

Graphs are powerful analysis tools. AFT Fathom has special tools to create and save Graphs and regenerate them later even if the output data has changed due to model changes. Follow the tutorial below to make and manage graphs from one of the example models.

Create and run a fully defined model. This tutorial uses the Sizing a Pump example model, available in the standard installation of AFT Fathom.

To skip to different sections of this tutorial, use the links below:

Pump vs. System Curves

Using a model with output, navigate to the Graph Results window. On the right in the Quick Access Panel, select the Pump vs. System tab and select 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 the graph will appear on the left.
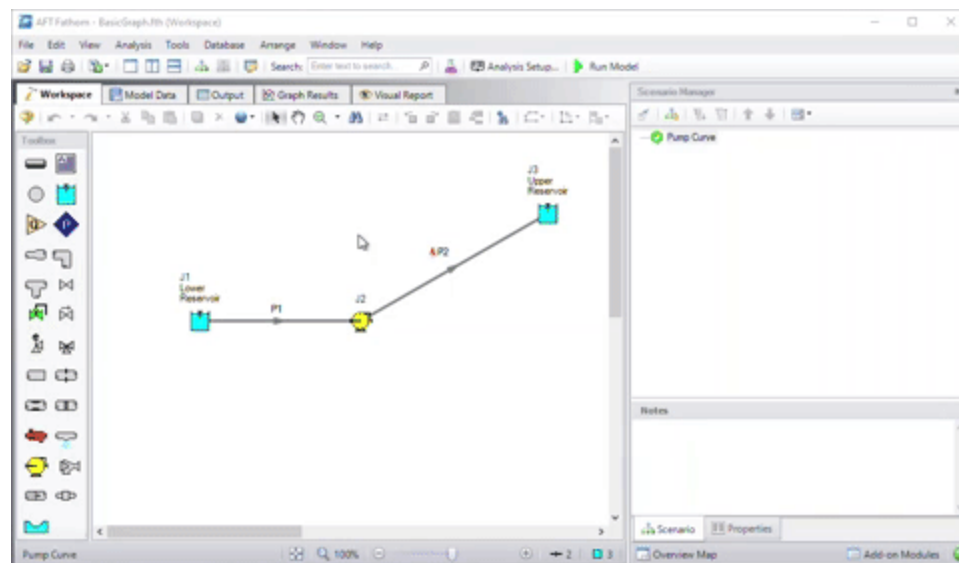
Profile Graphs

Using a model with output, navigate to the Graph Results window. On the right in the Quick Access Panel, click the Profile tab. This graph type displays the selected parameter along the length of the flow path. Select both pipes in the system and change the parameter to Pressure Stagnation. Using the default units of feet for length and psia for pressure, click Generate and the graph will appear on the left.
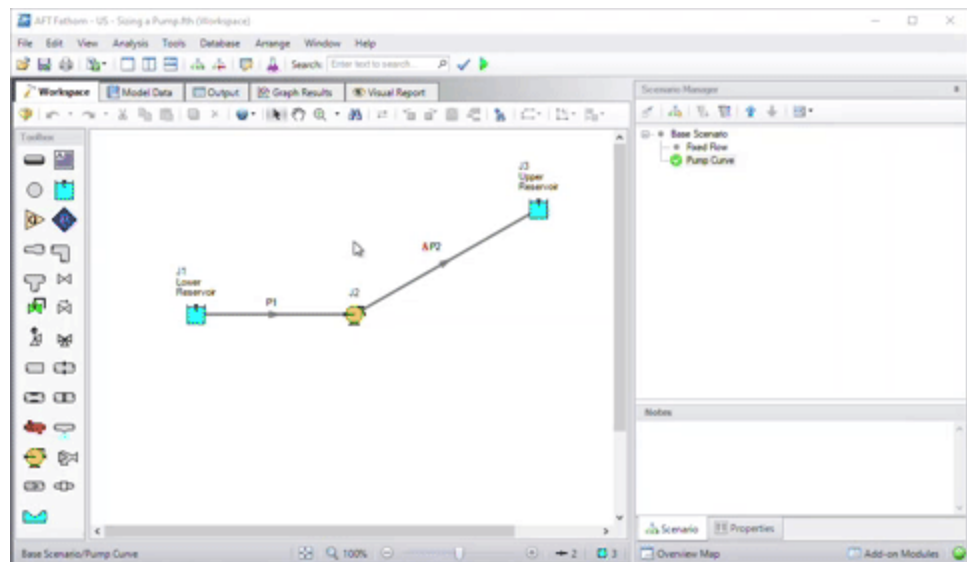
Saving Graphs

After gen-
erating a
graph, save
the graph by
selecting the
Add Graph
To List short-
cut on the
Graph Res-
ults Toolbar.
Give the
graph a
name and it
will appear
under My
Graphs in
the Graph
List Man-
ager on the
right. Easily
regenerate
this graph
with the
same para-
meters and
format if the
output
changes by
double-click-
ing on the
graph name.
The border
of the Graph
has also
changed to
dark green
along with
borders of
the para-
meters and
the name of
the Graph in
the Graph
List Man-
ager. This
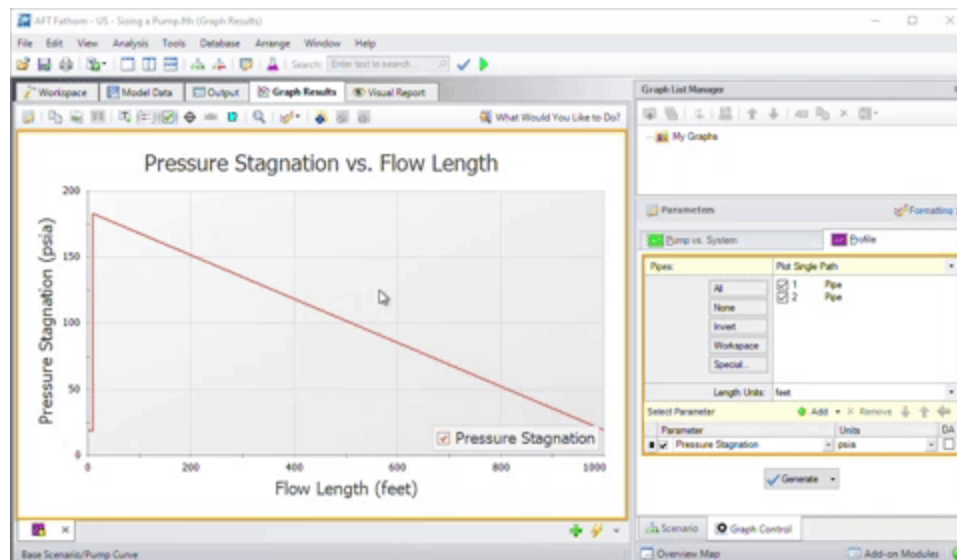
indicates that that Graph is current, saved, and that the displayed parameters are the ones used to generate the Graph.

Creating Multiple Graphs

Create multiple graphs stored on their own tabs by first clicking on the green plus button on the bottom right of the graph window to create a new graph tab. This creates a new graph tab. Generate a graph and then toggle between the graphs by selecting the tabs on the bottom of the graph window. Add as many graphs as desired.
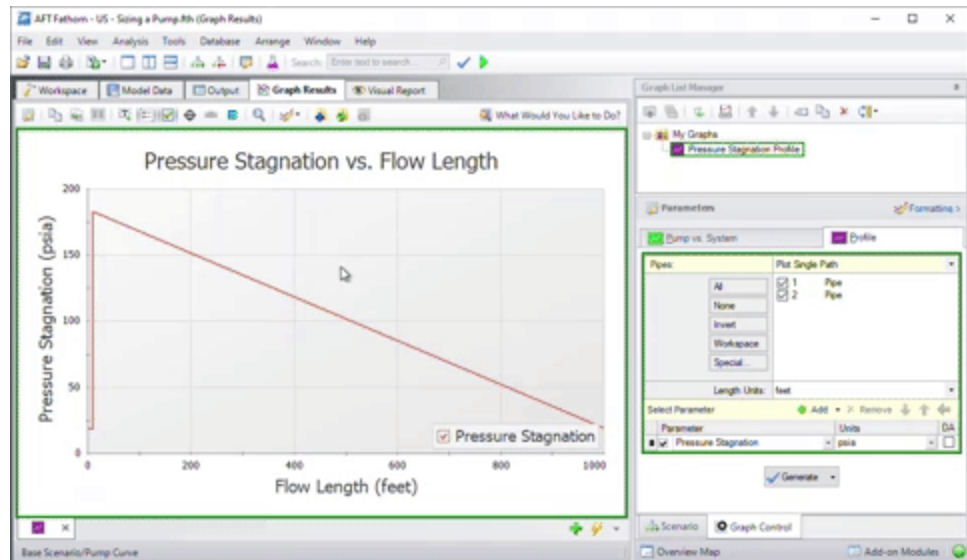
Alternatively, double-click a saved graph in the Graph List Manager in the Quick Access Panel to auto-matically gen-erate that graph on a new tab.
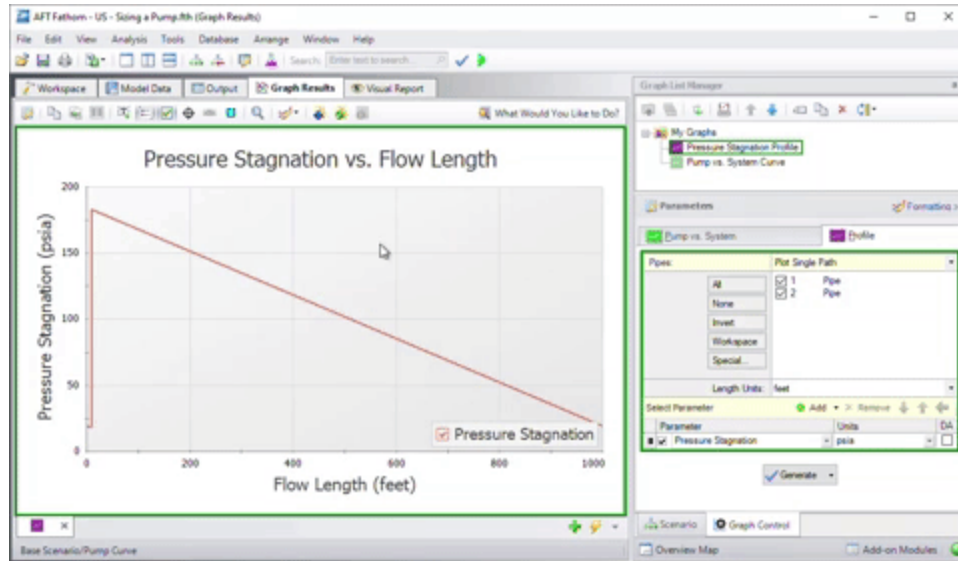
Displaying Multiple Graphs Simultaneously

To view multiple graphs in the same window, right click on a graph tab and select either Two Vertical Tab Groups or Two Horizontal Tab Groups. The graphs will then display next to each other such that both graphs can be viewed sim-ultaneously.
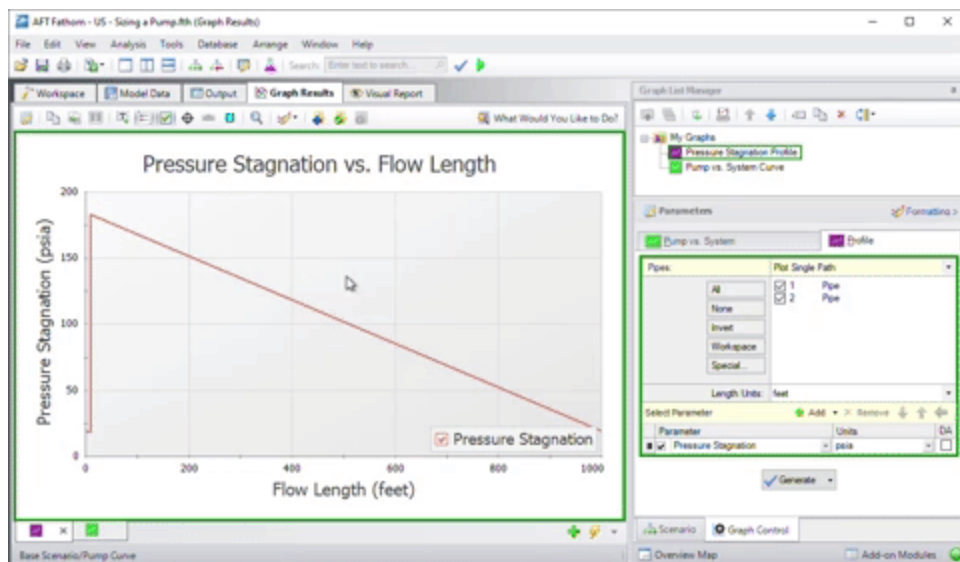
Alternatively, right-click on the My Graphs folder in the Graph List Manager, choose Load Folder Items, then click Same Tab. This will generate all graphs in the My Graphs folder into the *same tab* (as opposed to displaying *multiple tabs* simultaneously as described previously).
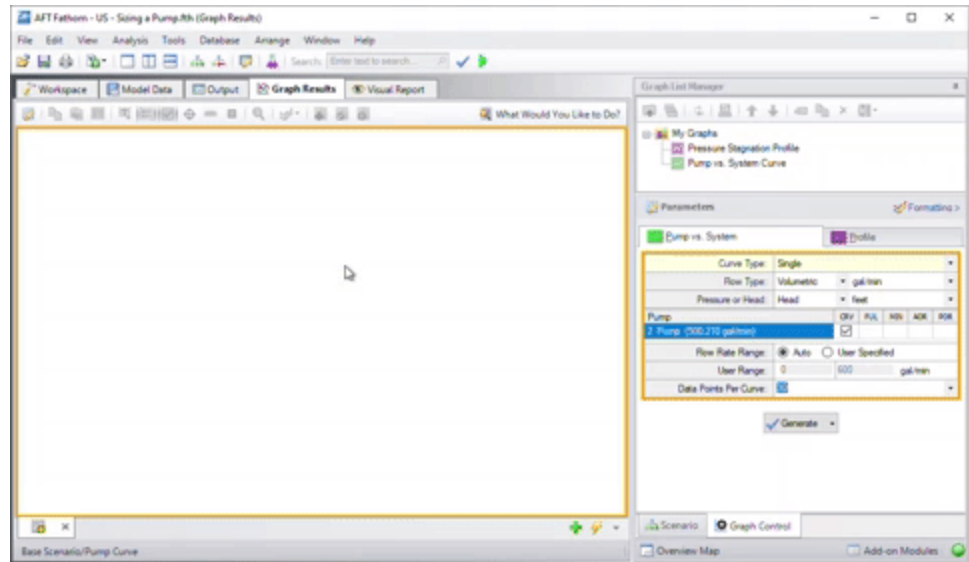
Creating Graphs from the Workspace

Some graphs can also be generated directly from the Workspace after running the model. Generate the Pressure Stagnation vs. Flow Length graph from the Workspace by selecting the flow path on the Workspace (note that you can select multiple pipes by
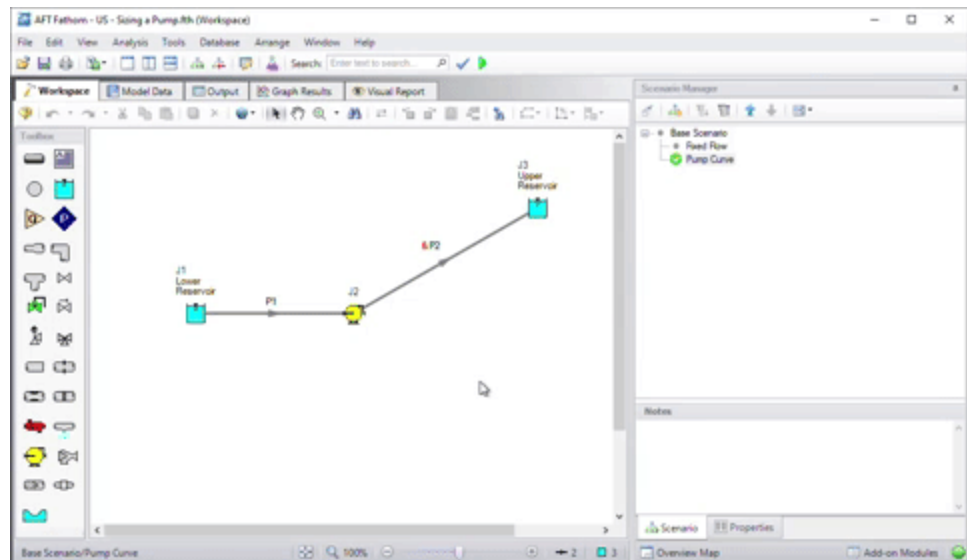
holding the
SHIFT but-
ton on the
keyboard
while select-
ing each
pipe). Right
click on the
Workspace,
choose the
Generate
Profile Graph
for Pipes
option, then
click Pres-
sure Stag-
nation.

Multi-Scenario Pump vs. System Curves Tutorial

When a model file has multiple scenarios that have pumps in them, a graph containing multiple pump curves and/or system curves can be generated for the purpose of comparison. To create a multi-scenario pump vs. system curve graph, two options are available:

- **Standard Method**: Save pump 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 pump vs system curve on an individual scenario basis, but is more time consuming.

- **Batch Run Method**: Create a template pump vs system curve graph, then set up a batch run to automatically generate the pump 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 Pump 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.
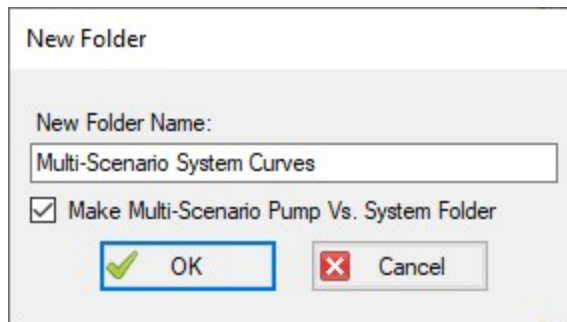
Figure 1: Creating a Multi-Scenario Pump Vs. System Folder

2. Load the first scenario you want to add to the graph, and run the scenario so that Output is present. Generate a Pump vs. System Curve for the scenario.

3. Add the Pump vs. System Curve to the Multi-Scenario Pump 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.
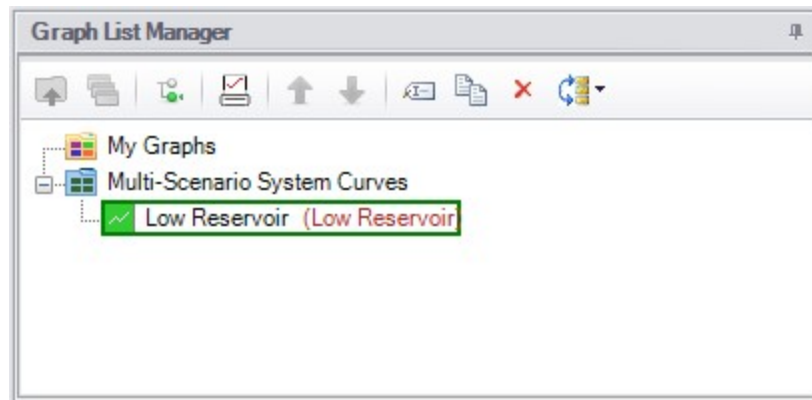
Figure 2: The name of the scenario that each Pump vs. System Curve was created in is displayed in the Graph List Manager

4. Repeat steps 2 and 3 for all scenarios that will be included in the graph.

5. When all of the individual Pump 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.
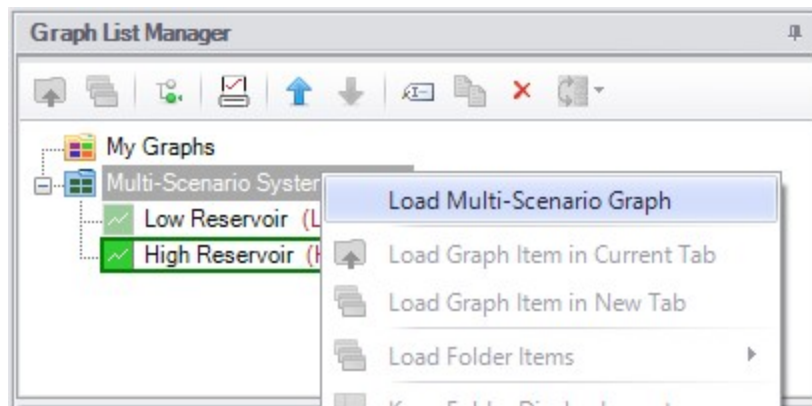
Figure 3: A Multi-Scenario Graph can be loaded from a Multi-Scenario graph folder

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.
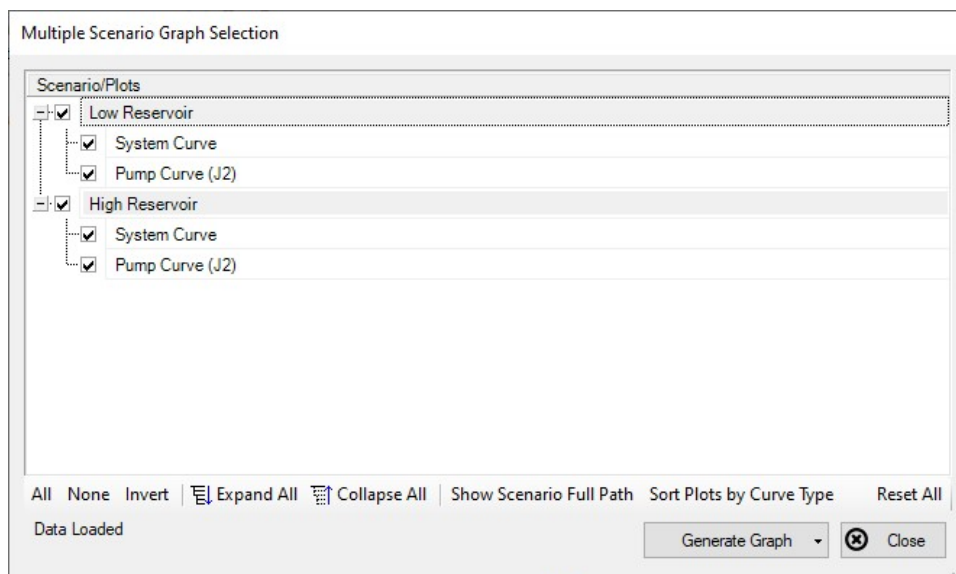
Figure 4: Multiple Scenario Graph Selection

7.   The selected curves will now be displayed on a single graph, as is shown in Figure 5.
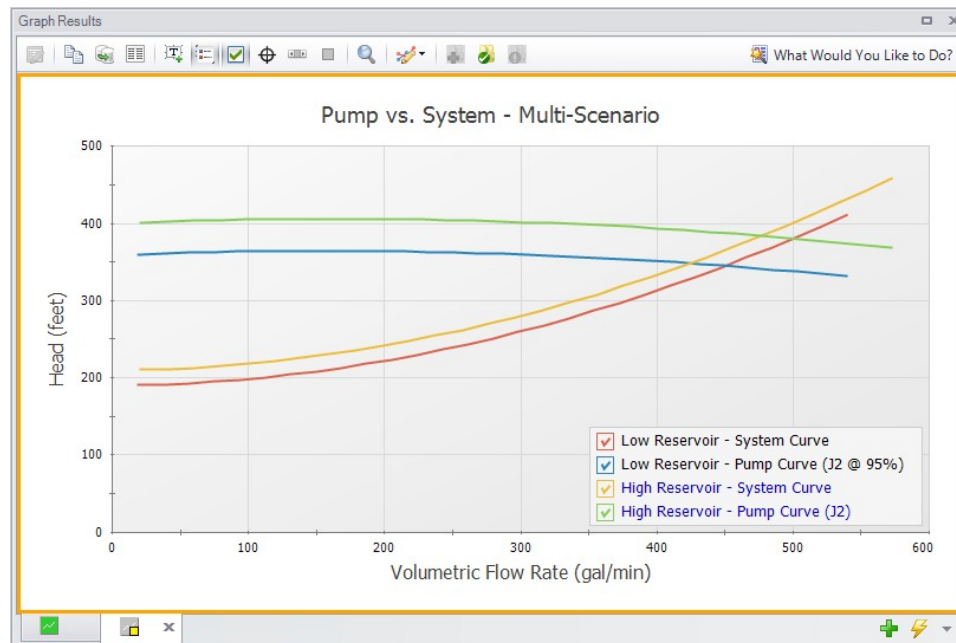
Figure 5: Multi-Scenario Pump vs. System curve

Batch Run Method

To speed up the process outlined above when making Multi-Scenario Pump 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 Pump vs. System curve, generate a single scenario Pump 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 pump 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 Pump vs. System curves, and select the template Pump vs. System curve as shown in Figure 6. If you have multiple templates, all options will be shown for selection.
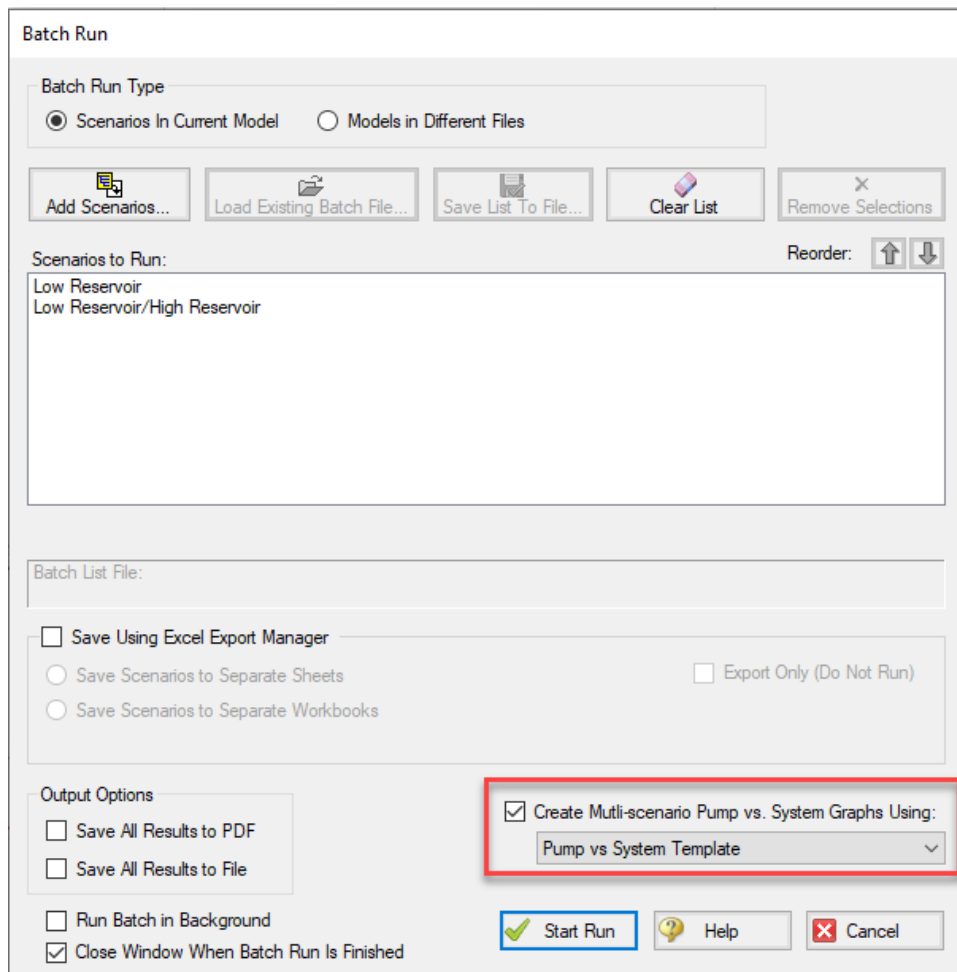
Figure 6: Multi-scenario batch run option enabled in the Batch Run window

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 Pump System Curve folder named with the date and time of the run. Right-click on the folder name and choose "Load Multi-Scenario Graph," as is shown in Figure 7.
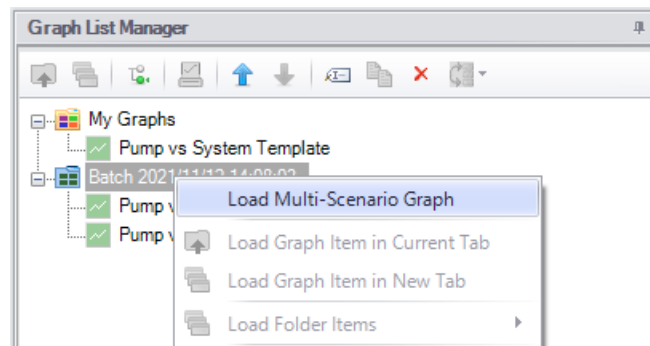
Figure 7: A Multi-Scenario Graph can be loaded from a Multi-Scenario graph folder

5. 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 8.
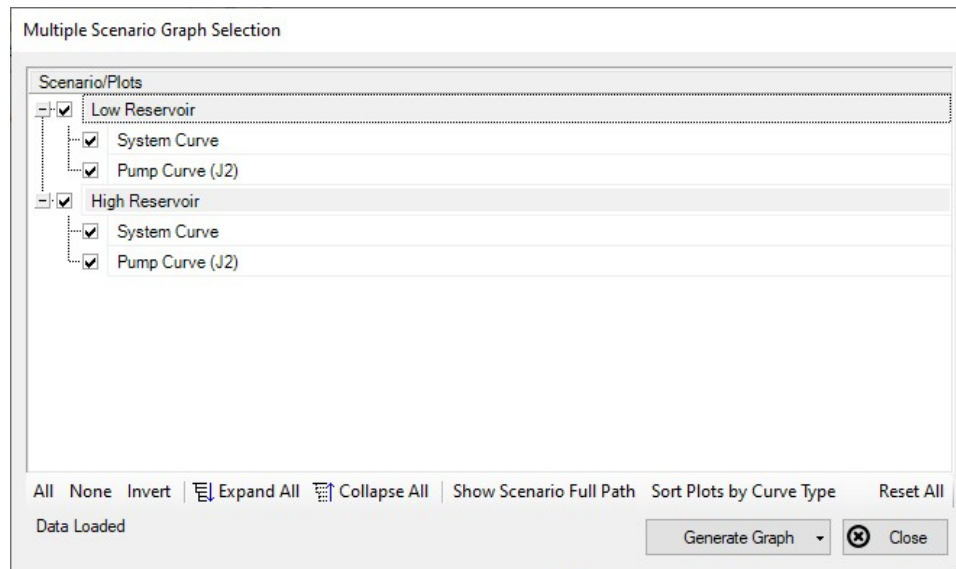
Figure 8: Multiple Scenario Graph Selection

6. The selected curves will now be displayed on a single graph, as is shown in Figure 9.
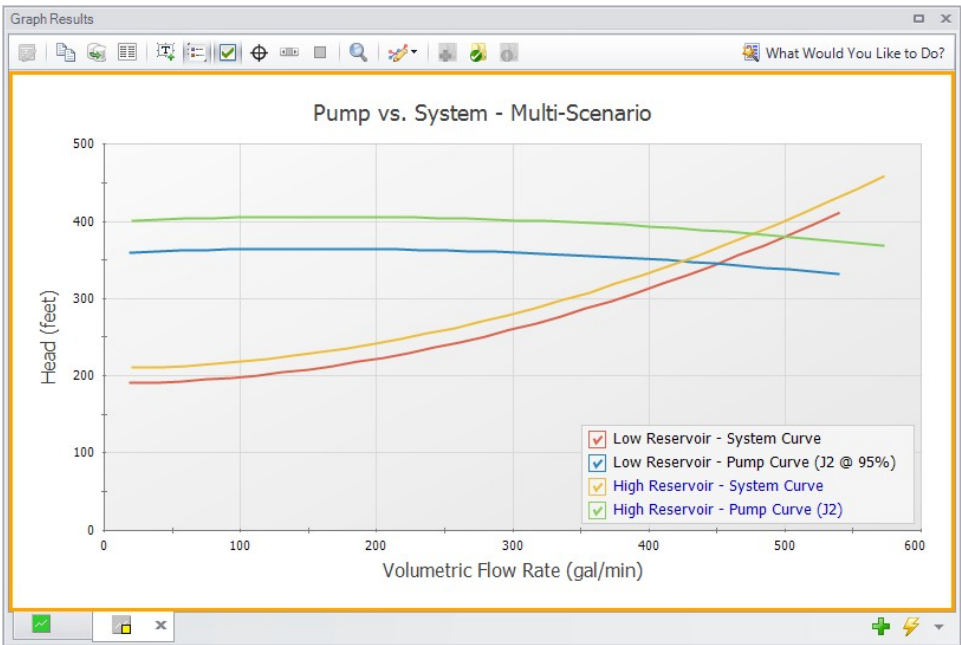
Figure 9: Multi-Scenario Pump vs. System curve

Related Topics

User Options Overview

An array of User Options are provided in a single window that can be found in the <u>Tools</u> 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 Fathom 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

Unit System

You can set the units you prefer to use in AFT Fathom 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 Fathom, first select Preferred Units under the Unit System section in the User Options window under the Tools menu.

AFT Fathom 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/min^2".

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

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, Fathom 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 styles include classic icons and transparent icon. Classic icons are the icons used in earlier versions of Fathom ending at Version 7 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. The 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, Fathom 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. Pipe and junction numbers are defined globally and not per layer or per scenario.

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 within the Workspace group 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

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 Reservoir Pipe Specification box allows you to specify the default option selected for how the height at which a pipe connected to a reservoir is specified. Selecting Pipe Elevation requires you to input the pipe elevation with respect to the reference elevation. Therefore, if a pipe is located 100 feet (30.5 m) from the reference elevation, or elevation= 0 ft (0 m), then you would input 100 ft as the pipe elevation when Pipe Elevation is selected in the Reservoir Properties window. Selecting Pipe Depth requires you to input the pipe elevation with respect to the level of fluid in the tank. Here, if a pipe is located 10 feet from the top of the fluid level in the tank, you would input 10 ft. Note that the same results will be calculated regardless of whether you specify Pipe Elevation or Pipe Depth as long as the height of the pipe is defined at the same height with respect to the system in both cases.

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 Setpoint Not Achievable for more information.

The Default Design Factors box allows you to add a default Design Factor to each junction shown in the list on the Pipes and Junctions window. Setting a default design factor for 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

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 Fathom 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 Files & Auto Save

A Backup File is a copy of a previously saved version of your model file (*.fth) that can function as a complete, standalone model. These files are used as a safety measure to provide you with one or more previous versions of the model in the case that the current model is deleted or damaged.

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

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

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

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

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

Create Recovery File

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

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

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

- **Yes** - This will copy the previously saved model file to a new file with "(Original)" appended to the front, and the Recovery File containing the latest unsaved changes will be opened.

- **No** - The Recovery File will be deleted, and the previously saved model file will be opened. This will open the model file in its last saved state, which may not include the latest unsaved changes if any are stored in the Recovery File.

Encoding

The File Encoding option can be set to "Unicode" in order to use characters outside of the standard English keyboard such as diacritics, cyrillic, etc for text fields such as the pipe/junction name, notes, etc.

Related Topics

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 Fathom.

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

File Import and Export

This section focuses on the various file types that can be imported and exported from AFT Fathom. 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

AFT Fathom 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 to other pipes by a junction. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be combined before the model could be solved. This option attempts to automatically combine pipes in this situation.
  - **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.
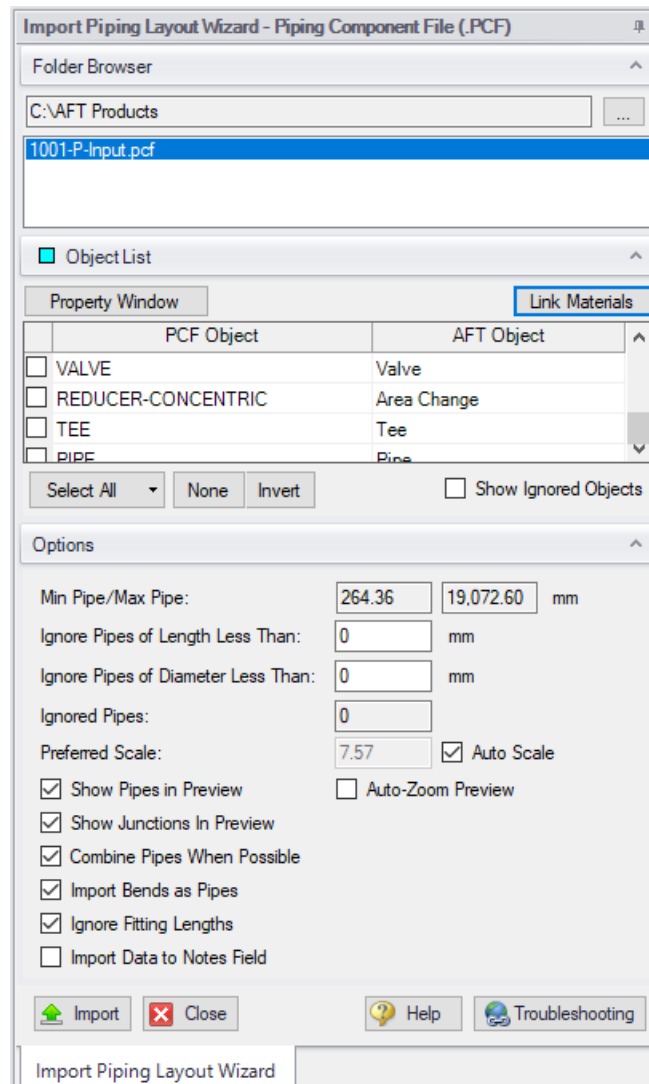
Figure 1: Piping Component File Import
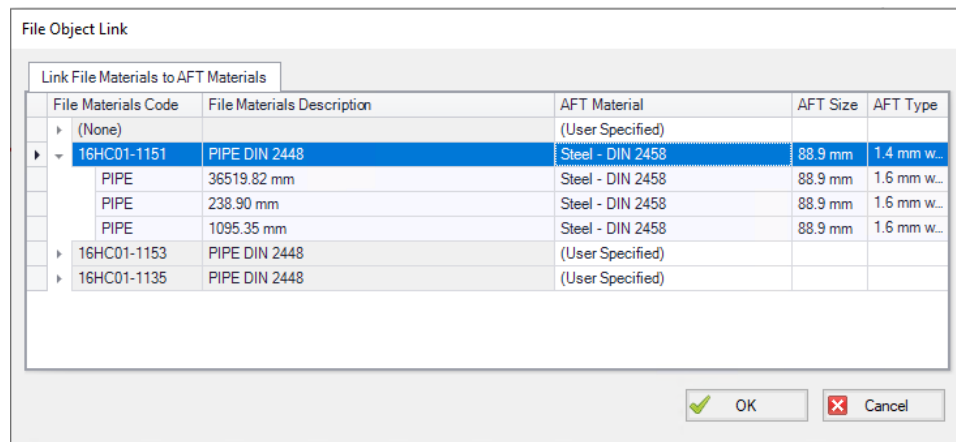
Figure 2: File Object Link window for Piping Component File Import

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 hydraulic model of a system and the import process will not produce a completed model or a perfectly accurate representation of the .pcf model. There will be missing data that must be assigned prior to analysis, such as fluid properties and boundary conditions. However, there are import practices and preparations that can be made to the .pcf file that will make creating model from an imported piping layout go more smoothly.

- **Workspace** - Piping Component Files are a three-dimensional file format, whereas model layouts in the AFT Workspace are a two-dimensional representation of a system. Because the Workspace is a purely visual tool independent of actual system dimensions, the Scale/Flip Workspace feature can be useful for achieving a readable model when working with imported piping layouts.

- **Object Positioning** - It is recommended to have some distance between most objects (this is not relevant to welds, branch/tap connections, gaskets, and supports). This aids in pipe connections, as objects that are very close together may have trouble connecting automatically. This is especially true of non-isometric models. Spacing between objects also streamlines the process of scaling objects on the Workspace after import.

- **Bends** - By default, Import Bends as Pipes is unchecked, and importing bends as bends is recommended for the sake of model readability and a more flexible approach to incorporating bend losses. Bends that have common angles and r/D ratios present in the internal AFT library can be imported as Fittings & Losses automatically when importing bends as pipes, but other bends will need to be added manually as junctions, or have their losses specified using the Additional K Factor option in the Pipe Fittings & Losses window or as user-specified fittings & losses using the Fittings & Losses Library. When bends are imported as bends, they will be treated as smooth bends, the upstream pipe determines the cross-sectional flow area, and the r/D ratio and/or angle is imported from the Piping

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

Note: Bends modeled explicitly can be joined with adjacent pipes as using Join Pipes/Junctions later if desired. Modeling bends as explicit junctions is advised for models containing bends that are not smooth flanged or have an uncommon angle or r/D ratio.

- **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

Importing GIS Shapefiles

Import GIS Shapefile

GIS Shapefiles that adhere to the ESRI standard can be imported into AFT Fathom for ease of model creation. These files have the extension .shp, but there are two other supporting file types that need to be in the same folder as the .shp in order for the import to work correctly. These are .shx and .dbf file types. The option is located under File > Import Piping Layout From > GIS Shapefile (.shp). This opens the "Import Piping Layout Wizard" in the Quick Access Panel that allows the importing of one line .shp that is of pipes and multiple point .shp files for branches, valves, pumps, and reservoirs. 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, Pumps, or Reservoirs.

Scaling

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 Workpace 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.

Mapping

The Data Mapping window allows the user to select what data fields have relevant data for AFT Fathom 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, Hazen-Willams Factor, Explicit Friction Factor, Weymouth, and Panhandle friction models are available.
    - **Inner Diameter**
- Branches can map:
    - **Labels** - Names
    - **Elevation**
- Pumps can map:
    - **Labels** - Names
    - **Elevation**
    - **Assigned Flow** - Imports as a Fixed Flowrate pump
- Reservoirs can map:

- • **Labels** - Names
- • **Elevation** - To the base of the reservoir
- • **Liquid Elevation** - Height from base to liquid surface elevation. Liquid elevation and elevation are summed and input as liquid surface elevation in the generated reservoir junctions.
- Valves can map:
  - • **Labels** - Names
  - • **Elevation**

Related Topics

Importing CAESAR II Neutral Files

AFT Fathom 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 to other pipes by a junction. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be combined before the model could be solved. This option attempts to automatically combine pipes in this situation.

  - **Import Bends as Pipes** - Pipe bends can be imported as junctions, or simply as a visual change in direction of the pipe. Importing bends as pipes is identical to using segments to show piping layout - no additional losses due to direction change are accounted for with this option. To account for losses, additional information will need to be entered using either junctions or fittings & losses. This option is helpful to reduce unnecessary model clutter and input if bend losses are negligible.

  - **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.
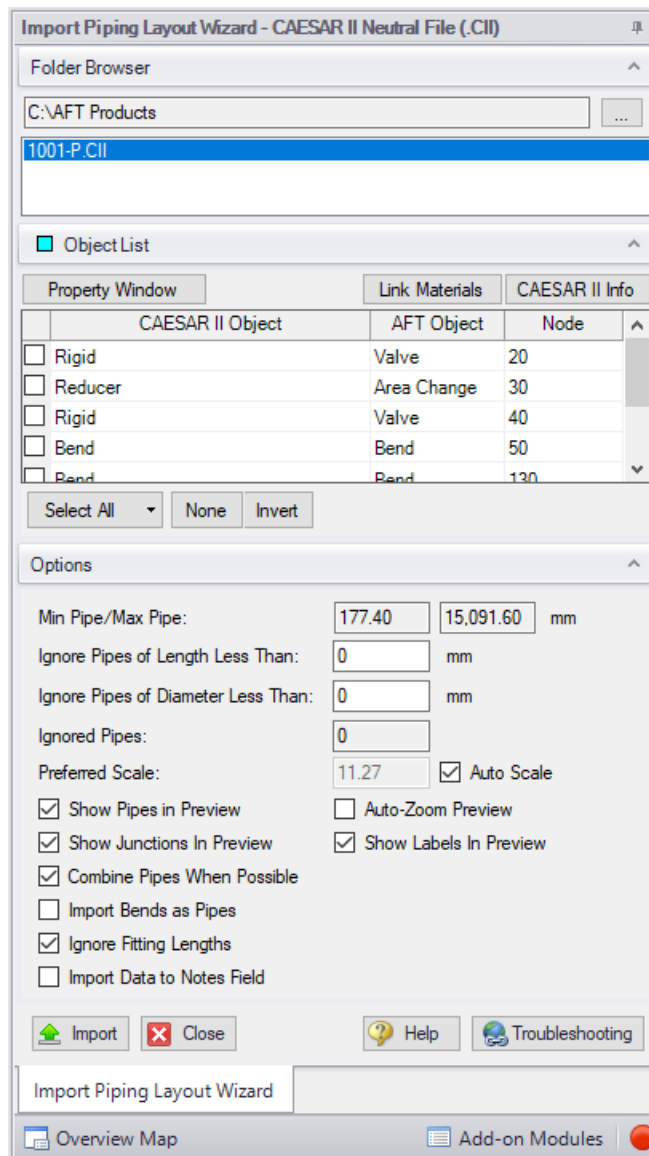
Figure 1: CAESAR II Neutral File Import

Related Topics

Importing and Exporting EPANET Files

AFT Fathom 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.
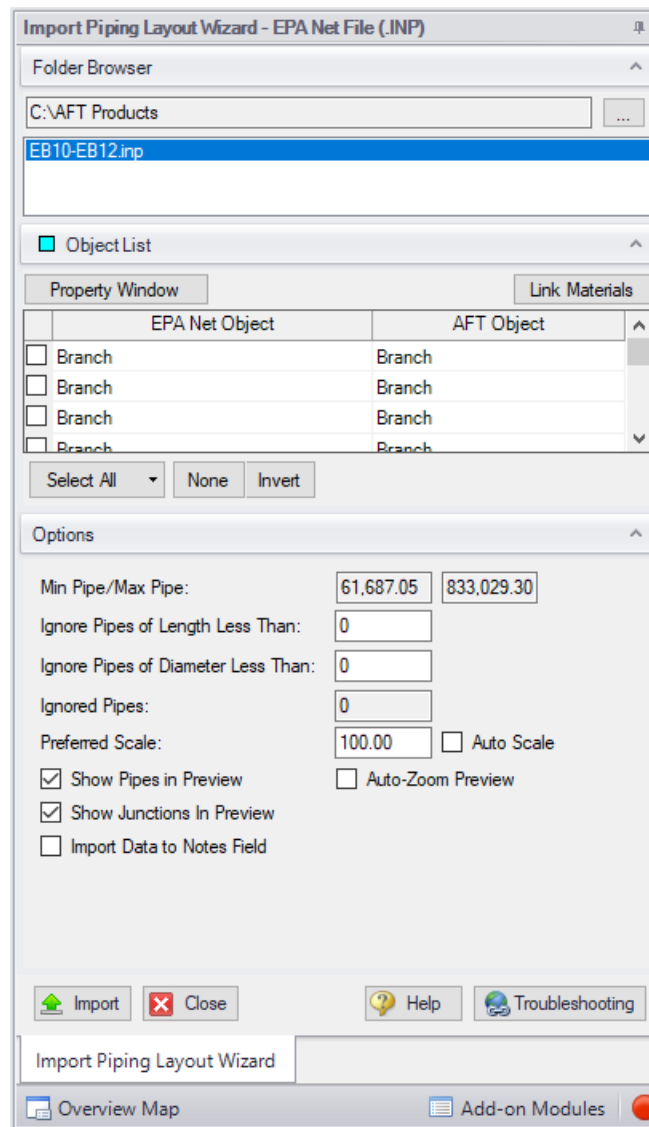
Figure 1: EPANET Import/Export Dialogs

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

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

- **Tanks** - If XTS is on, finite-volume tanks will imported as finite open tanks. Otherwise, they will be imported as normal reservoirs.

- **Reservoirs** - EPANET reservoirs are imported as a reservoir with a surface pressure of 1 atm with liquid elevation imported from EPANET.

- **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 the emitter coefficient imported as a K Fire Sprinkler value with the flow units defined by the base demand units and the pressure component defined by the emitter coefficient units (e.g. flow unit per square root of psi in EPANET). The number of sprinklers per emitter are set to 1 and the exit pressure is set to the atmospheric pressure (zero gauge pressure). The correct atmospheric pressure for the imported system should be set in Environmental Settings in Analysis Setup.

Exporting EPANET Files

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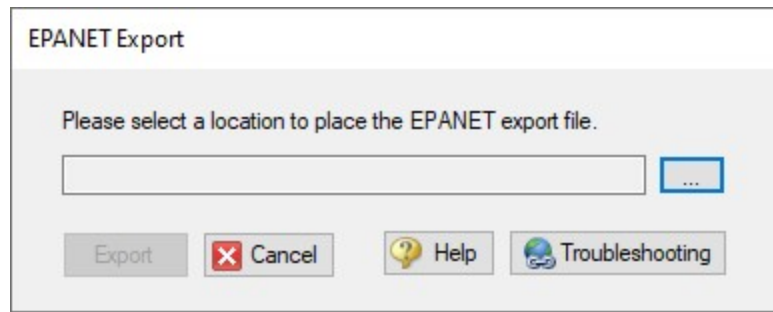
Figure 2: EPANET Import/Export Dialogs

Related Topics

Importing and Exporting from Other AFT Products

Models made in other AFT products such as AFT Impulse can be opened directly in AFT Fathom 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 plot pump vs system curves for a model built in Impulse.

To convert a model from one AFT product to another just browse to the model you would like to convert and open it in the software you are converting the model to. This can be done by changing the file extension selected, as is shown below in Figure 1. Note that though AFT Arrow and AFT xStream are not explicitly listed, models from Arrow and xStream can also be converted by selecting "All" and browsing to the model file.

When the model is opened in AFT Fathom 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 Fathom 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 Fathom. If a junction is defined using a feature not available in AFT Fathom, 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.
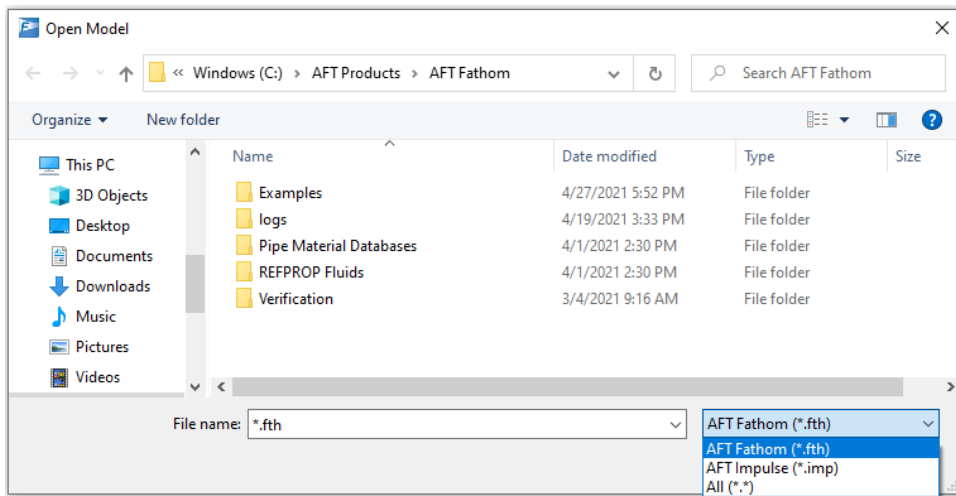
Figure 1: File extensions options for Open File window

Related Topics

Exporting To Excel

AFT Fathom 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.
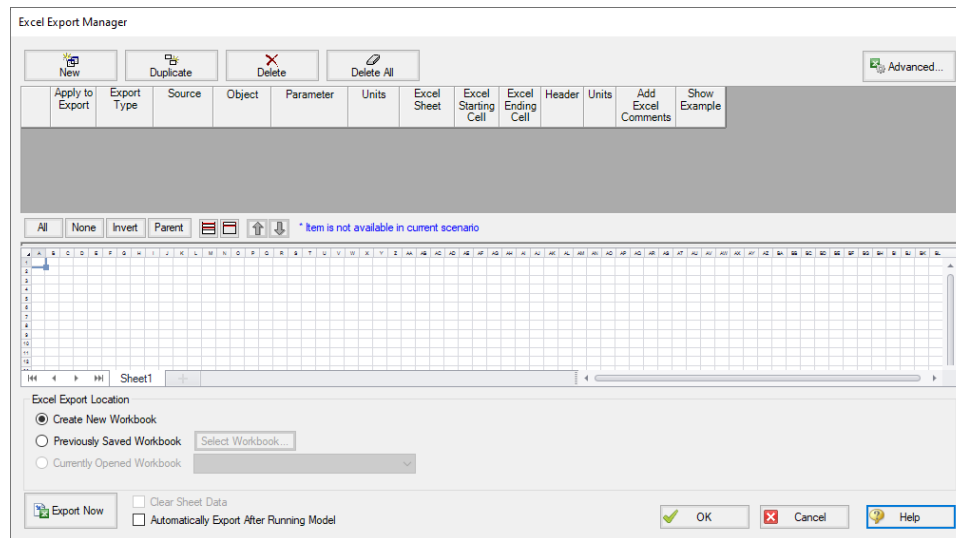
Figure 1: The Excel Export Manager

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.
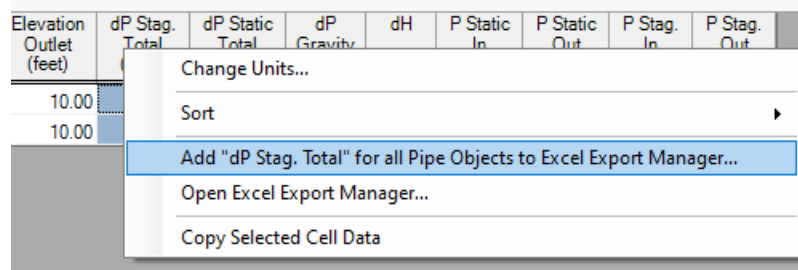
Figure 2: Adding items directly from the Output

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 pump junction in the model, Pump 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.

Figure 3: Example Export

Figure 4: Result of the above example, color added for emphasis

Excel Export Location

After all of the desired data has been added to the Excel Export Manager, the next step is to actually export the data. There are three options available for the export:

- **Create New Workbook** - A new Excel Workbook will be opened with the exported data.
- **Previously Saved Workbook** - Browse to a Workbook saved to file. The Workbook will be opened and the modifications made to the sheets as specified.
- **Currently Opened Workbook** - Select a Workbook that is currently open in Excel. The already open Workbook will be modified.

Note: The Excel Export Manager **does not** save Excel files for non-batch run exports. Any changes made need to be manually saved by the user in Excel.

There are two options available for exporting:

- **Clear Sheet Data** - All of the existing data on the sheets specified in the Excel Sheet column will be cleared. This is especially useful for setting up a new export when many changes are being made.
- **Automatically Export After Running Model** - After the model is run, the Export will be carried out automatically, per the settings defined by the user in the Excel Export Manager.

Advanced Options

Items can be added directly from the Output instead of adding custom items with the New button. There are two behavioral options that can be modified for this action:

- **Open Excel Export Manager** - If enabled, the Excel Export Manager will be opened every time an item is added from the Output. This is useful for double checking the export process completed as expected.
- **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 by using the Export Only (Do Not Run) option. 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
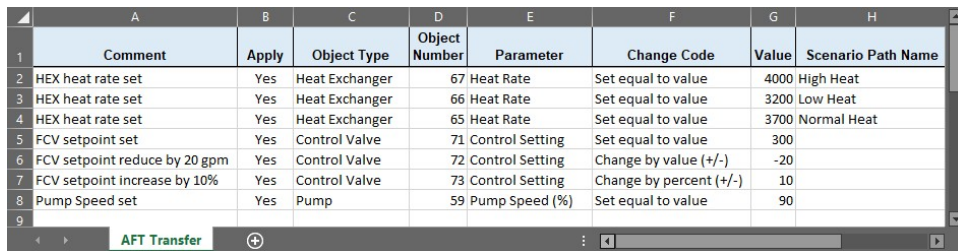
Importing From Excel

A Microsoft Excel spreadsheet can be used to make changes to objects in an AFT Fathom 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 Fathom will search for a worksheet named **AFT Transfer**. There can be other worksheets in the workbook but Fathom 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	67	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	Yes	Heat Exchanger	66	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	Yes	Heat Exchanger	65	Heat Rate	Set equal to value	3700	Normal Heat
5	FCV setpoint set	Yes	Control Valve	71	Control Setting	Set equal to value	300	
6	FCV setpoint reduce by 20 gpm	Yes	Control Valve	72	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increase by 10%	Yes	Control Valve	73	Control Setting	Change by percent (+/-)	10	
8	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	
9								

Figure 1: Sample AFT Transfer Excel Sheet

Row 1 and column A are not read by AFT Fathom - 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 Fathom installation, or can be downloaded here. By default the file path is: C:\AFT Products\AFT Fathom\AFT Fathom Excel Import Workbook.xlsx

Apply Column

This column accepts Yes or No. Only changes marked as Yes will be imported into the model.

Object Type

This column must be a junction name or Pipe. Valid choices can be seen below in Table 1.

Object Number

This corresponds to the junction or pipe number.

Parameter

The specific parameter to be changed. These parameters vary based on the Object Name selected. Each parameter has a type (C/S/I) associated with it, as is explained in the next section.

Table 1: Object Types, Associated Parameters, and Change Codes

Pipe	Area Change	Assigned Flow	Assigned Pressure	Bend
S Additional K Factor	**S** Design Factor	**S** Design Factor	**S** Design Factor	**S** Design Factor
S Ambient Fluid Velocity	**S** Initial Guess Pressure	**S** Elevation	**S** Elevation	**S** Initial Guess Pressure
S Ambient Temperature	**S** Initial Guess Temperature	**S** Flow Rate	**C** Junction Name	**S** Initial Guess Temperature
S Buried Pipe Depth	**S** Inlet Elevation	**S** Initial Guess Pressure	**S** Pressure	**S** Inlet Elevation
S Convection Coefficient External	**C** Junction Name	**C** Junction Name	**I** Sizing/Cost Setting	**C** Junction Name
S Convection Coefficient Internal	**S** Outlet Elevation	**I** Sizing/Cost Setting	**S** Temperature	**S** Outlet Elevation
S Cylindrical Annulus Inner Pipe OD	**I** Sizing/Cost Setting	**I** Special Condition		**I** Sizing/Cost Setting
S Cylindrical Annulus Outer Pipe OD	**S** User Specified K	**S** Temperature		**S** User Specified K
S Design Factor - Pipe Fittings & Losses				
S Design Factor - Pipe Friction				
S Design Factor - Pipe Heat Transfer				
S External Heat Flux/Tracing Flux				
S External Heat Rate				
S ID Reduction				

- 483 -

- 484 -

(Scaling)				
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)				
S Non-cylindrical Pipe Flow Area				
S Non-cylindrical Pipe Wetted Perimeter				
C Pipe Name				
S Rectangular Duct Height				
S Rectangular Duct Width				
S Roughness Value (for unspecified friction models)				
I Sizing/Cost Setting				

S Soil Surface Temperature **I** Special Condition				
Branch **S** Design Factor **S** Elevation **S** Initial Guess Pressure **S** Initial Guess Temperature **C** Junction Name **I** Sizing/Cost Setting **S** Source/Sink Flow Rate **S** Source/Sink Temperature **I** Special Condition	**Check Valve** **S** Design Factor **S** Forward Velocity to Close Valve **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** Loss Value **S** Outlet Elevation **I** Sizing/Cost Setting **I** Special Condition	**Control Valve** **S** Design Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** Loss When Fully Open **S** Outlet Elevation **S** Setpoint **I** Sizing/Cost Setting **I** Special Condition	**Dead End** **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Elevation **C** Junction Name **I** Sizing/Cost Setting	**General Component** **S** Design Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** K Factor **S** Outlet Elevation **I** Sizing/Cost Setting **I** Special Condition
Heat Exchanger **S** Controlled Outlet Temperature **S** Design Factor - Friction Loss **S** Design Factor - Heat Transfer	**Jet Pump** **S** Design Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elev-	**Orifice** **S** Area/Diameter **S** Design Factor **S** Exit Pressure **S** Exit Temperature	**Pump** **S** Control Setpoint **S** Delta Pressure to Re-Open Combined Check Valve **S** Design Factor	**Relief Valve** **S** Blowdown Pressure **S** Design Factor **S** Exit Head/Pressure **S** Exit Temperature

S Heat Flow Into System	ation	**S** Initial Guess Pressure	**S** Fixed Flow Rate	**S** Initial Guess Pressure
S Heat Transfer Area	**C** Junction Name	**S** Initial Guess Temperature	**S** Fixed Speed (%)	**S** Initial Guess Temperature
S Initial Guess Pressure	**S** Outlet Elevation	**S** Inlet Elevation	**S** Forward Velocity to Close Combined Check Valve	**S** Inlet Elevation
S Initial Guess Temperature	**I** Sizing /Cost Setting	**C** Junction Name		**C** Junction Name
S Inlet Elevation	**I** Special Condition	**S** Loss Value	**S** Initial Guess Pressure	**S** Loss Value
C Junction Name		**S** Outlet Elevation	**S** Initial Guess Temperature	**S** Outlet Elevation
S K Factor		**I** Sizing/Cost Setting	**S** Inlet Elevation	**S** Overpressure
S Outlet Elevation			**C** Junction Name	**S** Remote Pressure
S Overall Heat Transfer Coefficient			**S** Liquid Depth	**S** Set Pressure
S Secondary Fluid Flow Rate			**S** Liquid Surface Pressure/HGL	**I** Sizing/Cost Setting
S Secondary Fluid Inlet Temperature			**S** Outlet Elevation	**I** Special Condition
S Secondary Fluid Specific Heat			**I** Sizing/Cost Setting	
I Sizing/Cost Setting			**I** Special Condition	
I Special Condition			**S** Submerged Suction Pressure	
			S Submerged Suction Temperature	
			S Trimmed Impeller Ratio as Percent	
Reservoir	**Screen**	**Spray Discharge**	**Three Way Valve**	**Tee or Wye**
S Cross-Sec-	**S** Design			**S** Design

tional Area **S** Design Factor **C** Junction Name **S** Liquid Surface Elevation **S** Liquid Surface Pressure **S** Liquid Temperature **I** Sizing/Cost Setting **I** Surface Elevation Type **S** Tank Bottom Elevation	Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** K Factor **S** Outlet Elevation **I** Sizing/Cost Setting	**S** Cd (Discharge Coefficient) **S** Design Factor **S** Discharge Flow Area **S** Elevation **S** Exit Pressure **S** Exit Temperature **S** Initial Guess Pressure **S** Initial Guess Temperature **C** Junction Name **I** Number of Holes **I** Sizing/Cost Setting **I** Special Condition	**S** Actual Percent Open **S** Design Factor **S** Elevation **S** Initial Guess Pressure **S** Initial Guess Temperature **C** Junction Name **I** Sizing/Cost Setting	Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Elevation **C** Junction Name **I** Sizing/Cost Setting
Valve **S** Design Factor **S** Exit Pressure **S** Exit Temperature **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation	**Venturi** **S** Design Factor **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** Loss Value	**Volume Balance** **S** Initial Guess Pressure **S** Initial Guess Temperature **S** Elevation **C** Junction Name **I** Sizing/Cost Setting	**Weir** **S** Channel Width **S** Downstream Bottom Elevation **S** Downstream Cross-Sectional Area **S** Height of Weir **S** Initial Guess Pressure	

C Junction Name **S** Loss Value **S** Open Percentage **S** Outlet Elevation **S** Restricted Flow Area **I** Sizing/Cost Setting **I** Special Condition	**S** Outlet Elevation **I** Sizing/Cost Setting		**S** Initial Guess Temperature **S** Inlet Elevation **C** Junction Name **S** Notch Width/Angle **S** Outlet Elevation **I** Sizing/Cost Setting **I** Special Condition **S** Surface Pressure **S** Upstream Bottom Elevation **S** Upstream Cross-Sectional Area **I** Weir Type	

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

Junc-tion	Special Condi-tion	Value
Assigne-d Flow	None	0
	Closed	1
Branch	None	0
	Closed	1
Check Valve	None	0
	Closed	1
	Open	2
Control Valve	None	0
	Closed	1
	Fully Open - No Con-trol	2
General Com-ponent	None	0
	Closed	1
Heat Exchang-er	None	0
	Ignore Heat Trans-fer	2
Jet Pump	None	0
	Closed	1
Pump	None	0
	Pump Off No Flow	1
	Pump Off With Flow	2

Junc-tion	Special Condi-tion	Valu-e
	Throug-h	
Relief Valve	None Failed Open Ignore Relief Valve	0 2 5
Spray Dis-charge	None Closed	0 1
Valve	None Closed	0 1
Weir	None Ignore Weir	0 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 Fathom 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 valie. 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 Edit to an arbitrary value before the changes are made from Excel.

Related Topics

Special Modeling Features

This section is dedicated to unique standard based outputs as well as the designation of design alerts. Topics include discussions on using the submergence parameter to prevent vortexing, and utilizing the design alerts feature in the Workspace.

Contents

Submergence

Submergence is a parameter used to determine the amount of liquid needed above a pipe supply from a tank or reservoir to ensure smooth flow and avoid vortexing. AFT Fathom offers a pipe [output parameter](#) for submergence and also submergence margin, which is the difference between the submergence and the actual pipe depth.

Note: Pipe output parameters for submergence include options for Submergence Inlet and Submergence Outlet. Like all pipe output parameters, the Inlet and Outlet reference the inlet and outlet of the pipe. They do not describe if flow is entering or exiting the tank.

The equation used in AFT Fathom for submergence is given by [ANSI/HI Standard 9.8.7 1998](#), p. 32:

$$S = D(1 + 2.3\ Fr)$$

Where *S* is the submergence and *Fr* is the Froude number. Note that the required submergence can be modified by using devices such as baffles in which case the above equation would not apply.

Related Topics

Design Alerts

Design Alerts are a powerful feature that can help speed up the modeling process and help verify that the model is behaving as expected. The creation of a Design Alert does not constrain or affect the operation of the model in any way. However, it automatically creates an output alert any time the design limit is violated. One common use case might be defining a maximum allowable pressure for the pipes in the model. While the output can be reviewed manually for these violations, this quickly becomes tedious as the model grows in size. Creating a Design Alert instead allows the engineer to spend less time reviewing output tables as any violation will be immediately identified.

The Design Alert Manager allows the user to create or modify Design Alerts. It can be accessed from the Tools Menu or in the Design Alerts tab in pipe and junction properties windows.

Design Alerts can also be created directly from the output, or can be imported/exported for use in other models.
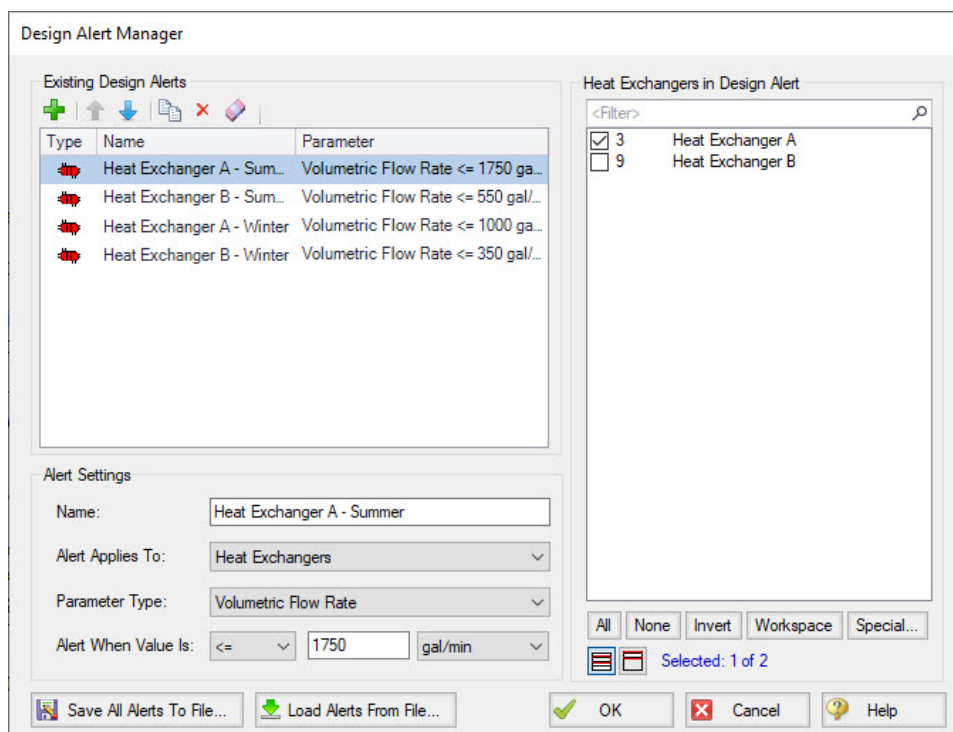
Figure 1: Design Alert Manager

Creating and Applying Design Alerts

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

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

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 >= when the value shouldn't rise above the specified value (i.e. temperature shouldn't exceed a maximum value)
   2. Use <= 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.
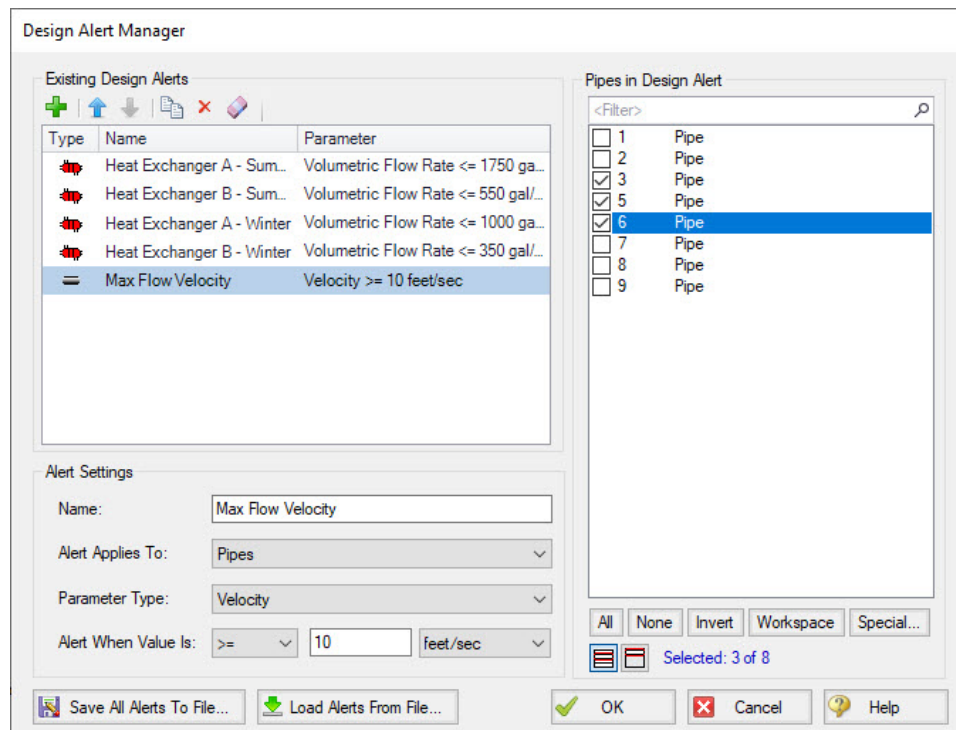
Figure 2: Creating a Design Alert

Table 1: Available Design Alerts

Pipes	General Junctions	Pump	Valves	Heat Exchangers	Reservoirs
Mass Flow Rate Pressure Gradient Stag. Total Pressure Stagnation Pressure Static Submergence Inlet Margin Submergence Outlet Margin Temperature Stagnation Vapor Pressure Margin Velocity Volumetric Flow Rate	Gradeline Energy Inlet (EGL) Gradeline Energy Outlet (EGL) Gradeline Hydraulic Inlet (HGL) Gradeline Hydraulic Outlet (HGL) Pressure loss Stagnation Total Pressure Loss Static Total Pressure Stagnation Inlet Pressure Stagnation Outlet Pressure Static Inlet Pressure Static Outlet	Head Rise Mass Flow Rate NPSH Margin Absolute NPSHA Percent of BEP Power Overall Pressure Discharge Stagnation Pressure Discharge Static Pressure Rise Stagnation Pressure Suction Stagnation Pressure Suction Static Speed Volumetric Flow Rate	Cv Flow Area Head Loss K Kv Mass Flow Rate Open Percentage Pressure Drop Stagnation Pressure Static Inlet Pressure Static Outlet Volumetric Flow Rate	Head Loss Heat Rate Log Mean Temperature Difference Mass Flow Rate Pressure Drop Stagnation Temperature Inlet Temperature Outlet Volumetric Flow Rate	Liquid Height Liquid Surface Elevation Mass of Liquid in Reservoir Volume of Liquid in Reservoir

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 on the Design Alerts tab in the general output section, 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.
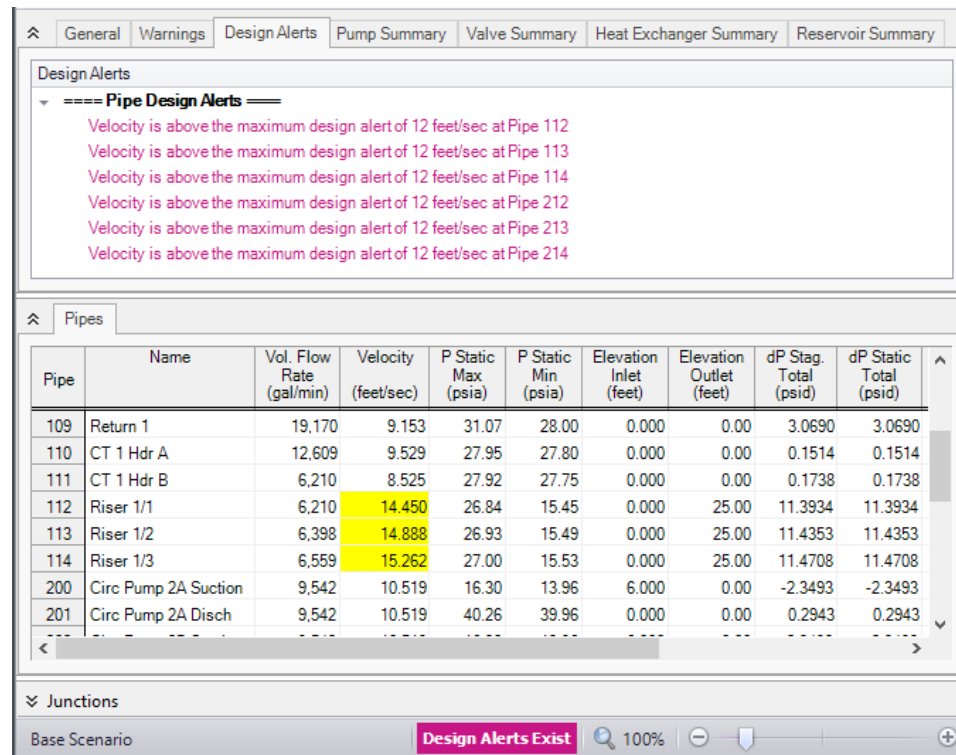
Figure 3: Design Alert Violations in Output

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 descendent scenarios. For more information about scenarios, please read the Scenario Manager help topic.

Related Topics

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...

ow	Velocity	P Static Max	P Static Min	Elevation Inlet	Elevation Outlet	dP Stag. Total	dP Static Total	dP Gravity
n)	(feet/sec)	(psia)	(psia)	(feet)	(feet)	(psid)	(psid)	(psid)
0.0	9.643	22.11	18.41	10.00	0	-3.7043	-3.7043	-4.338
5.0	4.821	102.54	102.29	0.00	0	0.2580	0.2580	0.000
5.3	7.264	73.75						00
5.0	4.821	21.50						00
1.7	7.249	101.17						00
5.5	9.665	72.76						00
8.3	6.533	100.47						00
0.2	6.555	75.31						00
5.0	4.821	22.58						00
0.0	9.643	101.49	100.91	0.00	0	0.5768	0.5768	0.000
5.0	4.821	102.47	102.21	0.00	0	0.2580	0.2580	0.000

Context menu overlay:

Add P2 "P Static Max" to Excel Export Manager...

Open Excel Export Manager...

Add Note...

Create Design Alert...

Copy Selected Cell Data

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 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.
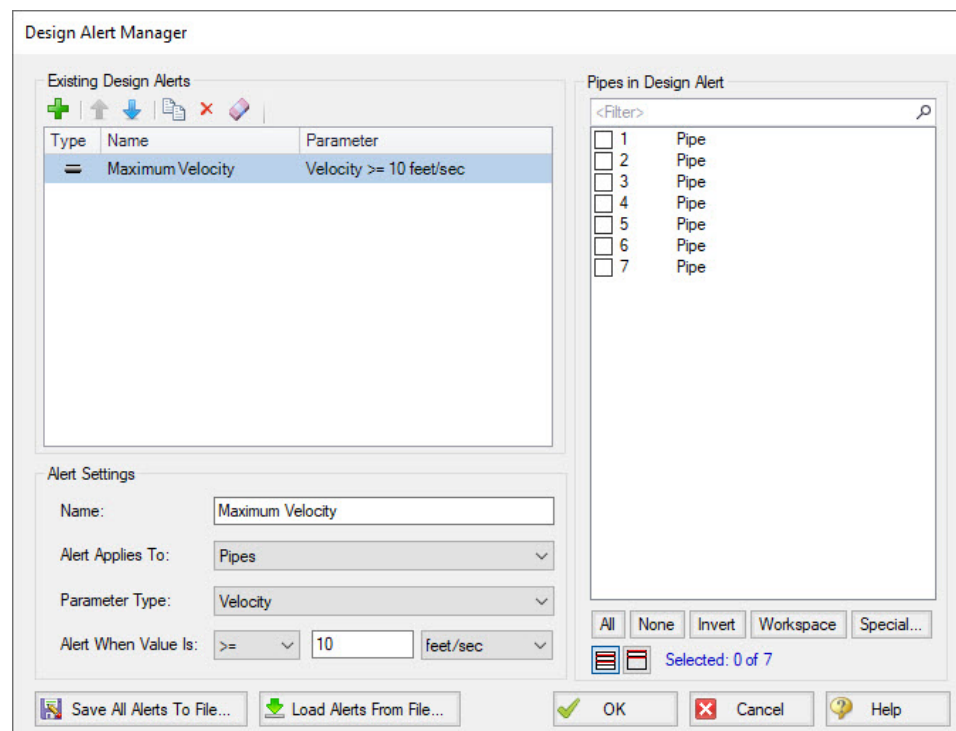
Figure 1: A design alert for velocity to be exported

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

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 on 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 identified both as the scenario with a green check mark next to it and in AFT Fathom's Status Bar). At the bottom, there is a place where you can record notes related to that scenario (propagates to children).

Those attributes whose value comes from a different scenario are linked, while those values coming from the selected scenario are not linked. Linked attributes will inherit changes made in the same object and attribute of the linked ancestor. This inheritance will occur over as many generations as the link exists.

Figure 1: Scenario Manager on Quick Access Panel

Related Topics

Creating, Organizing, and Editing Scenarios

Scenarios are referred to as either a parent or child, plus the Base Scenario, which may be thought of as the root scenario. Upon opening a new model file, only a Base Scenario will be present. To create a new scenario below the Base Scenario, right-click the Base Scenario and select Create Child.

Any scenario can be made current by double-clicking on it.

Figure 1 illustrates a model with a Base Scenario with two children, "English Units" and "Metric Units". These have three children, '3 inch Pipe', '3.5 inch Pipe' and '4 inch Pipe', representing three different sized pipe runs. Each of these has one child scenario "Fire Flow" and each of these, in turn, has two children, 'North Hydrant Fire' and 'South Hydrant Fire', for the two operating cases.
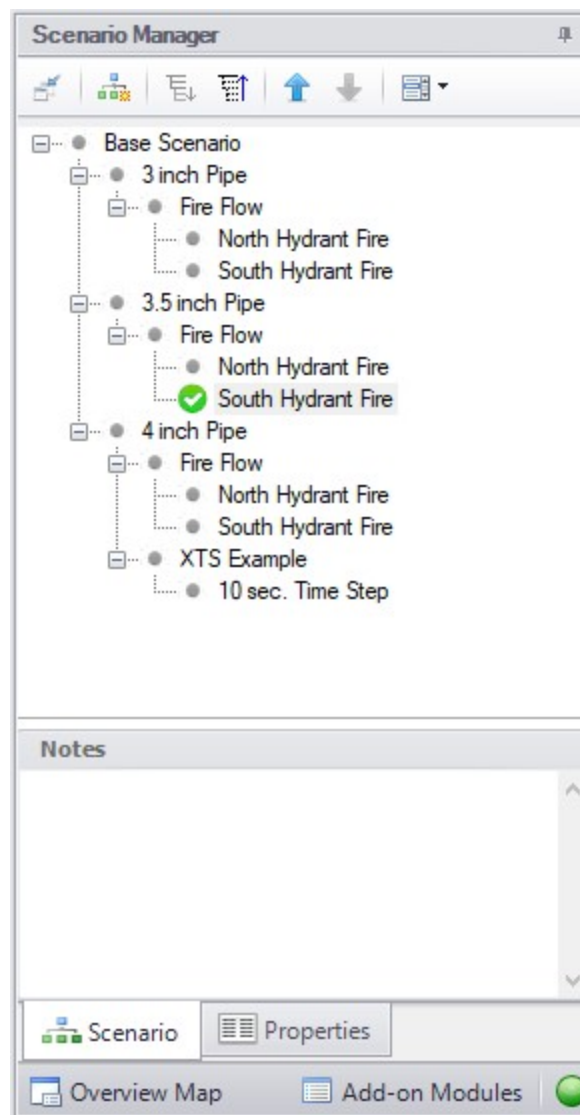
Figure 1: Scenario Manager Window

'English Units' and 'Metric Units' were created by first selecting the Base Scenario and Creating a Child.'3 inch Pipe', '3.5 inch Pipe' and '4 inch Pipe' were created by first selecting the 'English Units' Scenario then Creating Children. 'Fire Flow' was created by clicking on '3 inch Pipe' and creating the child. 'North hydrant open' and 'South hydrant open' were, in turn, created by selecting each of the above children scenarios for the three different pipe run sizes, then Creating Children.

Note: Alternatively, one could have made the 'English Units' scenario current, created the '3 inch Pipe' child, then selected it and created the 'North Hydrant Fire' and 'South Hydrant Fire' scenarios below it, then, with '3 inch Pipe' still selected, selected Clone With Children twice, then rename the new clones '3.5 inch Pipe' and '4 inch Pipe'.

The 'North Hydrant Fire' and 'South Hydrant' scenarios may be thought of as great great 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.fth :: 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 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 Fathom'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 compared to 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 Fathom 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 and the Common Toolbar.

Related Topics

Modifying Individual Scenarios

When first created, a scenario is identical to its parent. To modify or run a scenario, it must first be made in the current Workspace scenario by selecting it 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 Fathom's Status Bar.) Once a scenario is the current scenario, changes may be made to differentiate it from its parent using any of AFT Fathom'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, viscosity model, heat transfer, and laminar and non-Newtonian corrections

- Steady Solution Control - Solution method, tolerances, relaxation

- Output Control - Output window content and formatting

- Cost Settings - Application of cost data

- Transient Control (XTS Module Only) - Start and stop times for transient simulation

- 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.

- Library Manager - The connected library content can be varied between scenarios

Model Data Control and User Options apply to all scenarios.

Related Topics

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, grand-children, 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 Scenarios enabled shows how the data changes between scenarios.

Scenario Logic Examples

For many users, it is easiest to grasp Scenario Manager when it is explained how the coding logic is actually implemented. **Blank fields for children, grandchildren, etc., mean to look to the parent for the data**. The Base Scenario never has blank fields (Table 1). Data only passes downwards, never upwards.

Table 1: Blank fields for Child #1 and Grandchild #1 mean that the data is to come from the parent. If the Base scenario data is changed, all descendants are changed.

	Diameter	**Length**
Base	3	25
Child #1	--	--
Grandchild #1	--	--

If a child scenario does not have a blank field, then data for that property is initiated at that scenario level (see Table 2).

Table 2: Child #1 does not have a blank field, so 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 vale as the parent, the inheritance link is re-established (see Table 5). Its descendant's link is also re-established.

Table 5: If the Diameter in Child #1 is changed to be the same as the Base, it will be "blanked out" the next time the scenario is loaded and the link re-established. And so will Grandchild #1, if its Diameter is also the same

	Diameter	Length
Base	3	25
Child #1	3	--
Grandchild #1	3	--

	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

Table 6 shows data for two pipe properties across three scenarios. The data that would be used when each scenario is loaded is as follows:

- Base scenario
    - Diameter = 3
    - Length = 25
    - Changes to Diameter will not pass downwards
    - Changes to Length will affect only Child #1.
- Child #1 scenario
    - Diameter = 2
    - Length = 25
    - Changes to Base Diameter will not affect Diameter
    - Changes to Base Length will affect Length
- Grandchild #1 scenario
    - Diameter = 2
    - Length = 15
    - Changes to Base Diameter will not affect Diameter
    - Changes to Child #1 Diameter will affect Diameter
    - Changes to Base Length or Child #1 Length will not affect Length

Table 6: Input data for these properties explained in the text.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	15

Re-Establishing Broken Scenario Links

A link may be re-established by returning the attribute to the same value as that of its parent. This can be done manually by entering the value or selecting the Copy Data From Pipe list and selecting the Parent Pipe Data option. Within other data windows, it is done by selecting the Same As Parent option in Output Control, Analysis Setup, 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

Running Models in Batch Mode

AFT Fathom 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. From the File menu, click Start Batch Run.
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 click 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 button. 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. 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.

6. Use the Output Options in the lower left to save the output data to a file and/or send it to a printer or Adobe PDF file. All reports will use the specified font.

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 Fathom 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 Fathom then opens each scenario or model file in sequence, automatically runs each scenario/model that has Analysis Setup fully defined, and sends results to the specified destination. All pertinent information that is generated during the batch run (such as error messages) are automatically displayed when the final scenario/model is finished. Output files generated from each scenario or model run is automatically saved before moving to the next run.

The entire batch run can be canceled at any time by clicking the Cancel button in the Solution Progress window. Output from scenarios that have already been run is still retained when the Batch Run is canceled.

Note: You can view output from multiple scenarios concurrently in the Output window.

Related Topics

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

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 Comparison Tool OverviewScenario Manager

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

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

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.

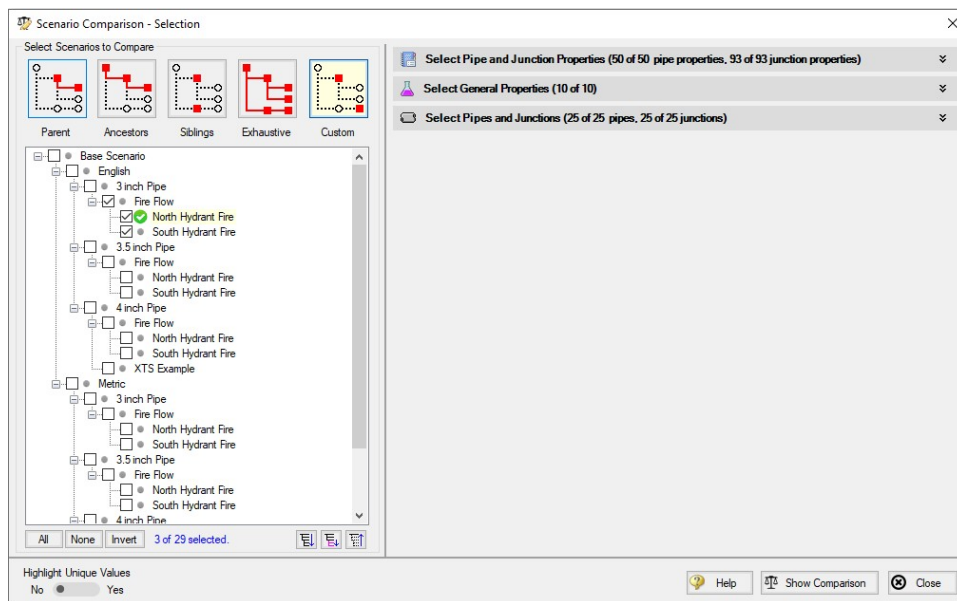*Figure 1: Scenario Comparison window options to Modify Comparison*

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 from 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 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 from and pasted to a separate file or worksheet using the Copy button at the bottom of the window.

Figure 1: Default Scenario Comparison Report view

Filtering Options

When a large comparison is run, typically many of the objects/parameters in the model will be unchanged across scenarios, which may result in a large number of items with no differences. To help condense the Scenario Comparison Grid, several toggles are available at the top of the Item and Parameter columns which allow the user to show/hide rows in the grid. 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.

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 North Hydrant scenario for this example.

Related Topics

Cost Calculation

AFT Fathom 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 pumps in a system, and AFT Fathom makes measuring this value easy.

In addition to energy usage, AFT Fathom 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 the system life and energy cost in the Cost Settings panel.
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 correctly.
8. Run the model and view the cost totals with the Cost Report.

Related Topics

Enabling Cost Calculation

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

The first step in any cost calculation is to enable it in the Cost Settings panel. The following options are available:

- **Do not calculate** - Selected by default.
- **Pump Energy Only** - Only calculate the costs associated with the power consumption of pumps. The energy cost and system life must be entered in Cost Settings.
- **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 costs calculated. This allows only analyzing the cost of certain parts of the model - useful if only one pump 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 below. 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 pumps, so that pump 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.
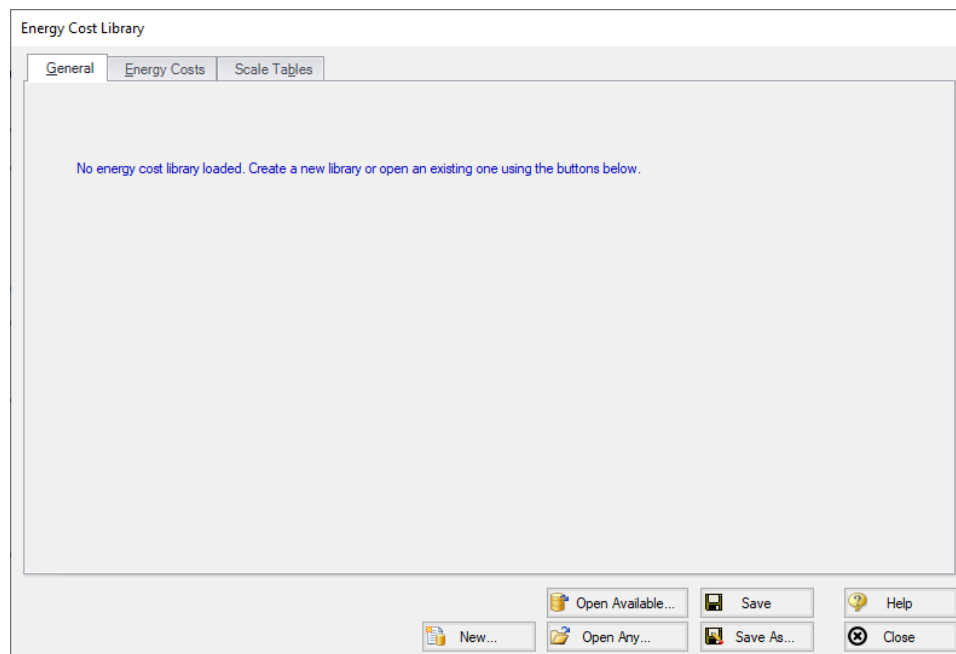
Figure 1: Energy Cost Library window

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.
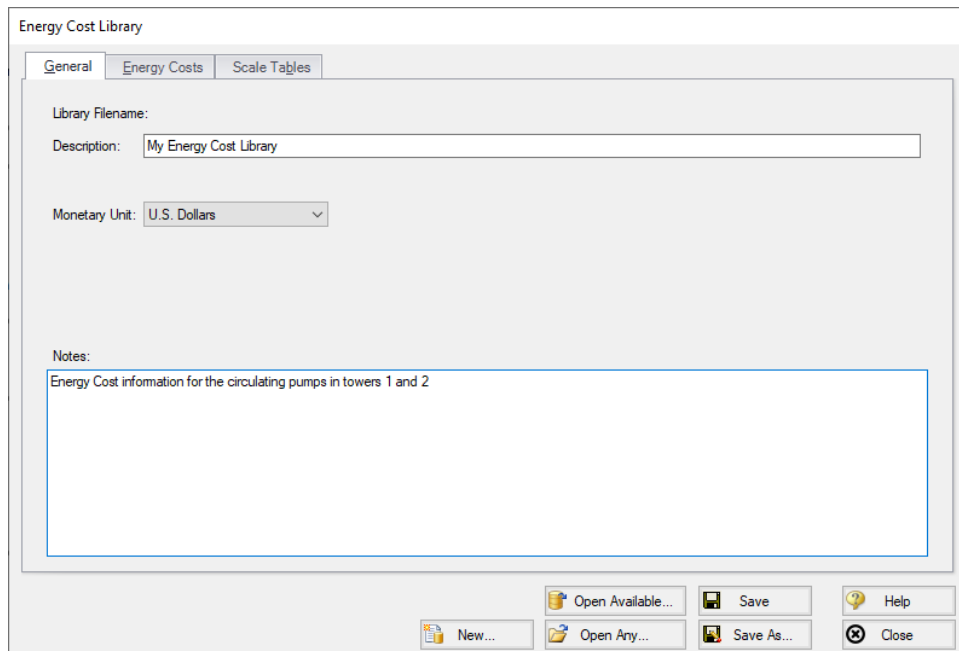
Figure 2: New Energy Cost Library

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.
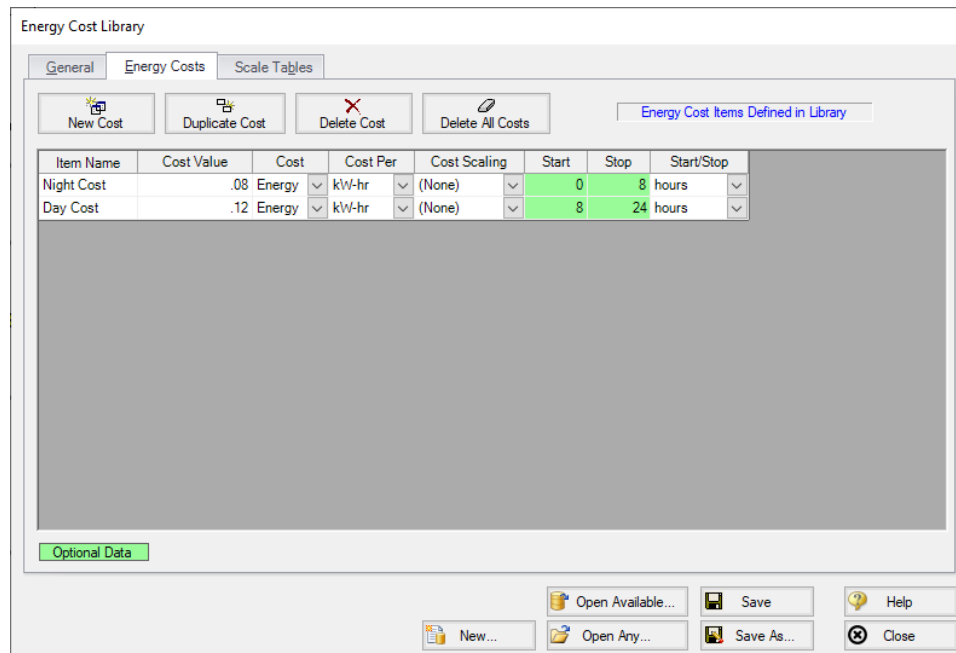
Figure 3: Energy Costs

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 the Scale Tables topic for more information.

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.

The 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 pump energy when the pump 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.
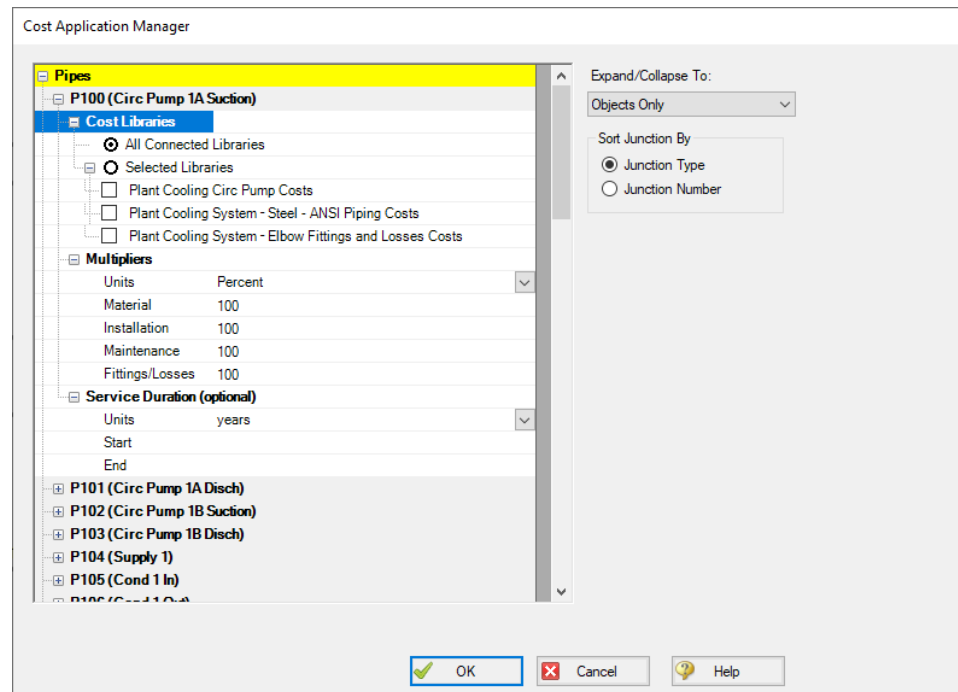
Figure 1: Cost Application Manager

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, pump.dat, exists with a specific pump "PumpCo Model 100A".

Two cost libraries exist based on pump.dat:

- maxcost.cst ("PumpCo Model 100A" installation cost = $1000)
- bestcost.cst ("PumpCo Model 100A" installation cost = $750)

If both of these libraries are connected, and All Connected Libraries are applied to the pump, AFT Fathom will interpret the cost for "PumpCo 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. For example, one library may contain costs of the pump and its casing, while another may contain cost for the motor. A third might contain cost for a variable speed drive. When managed in this way, you want the costs to be summed.

Related Topics

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.
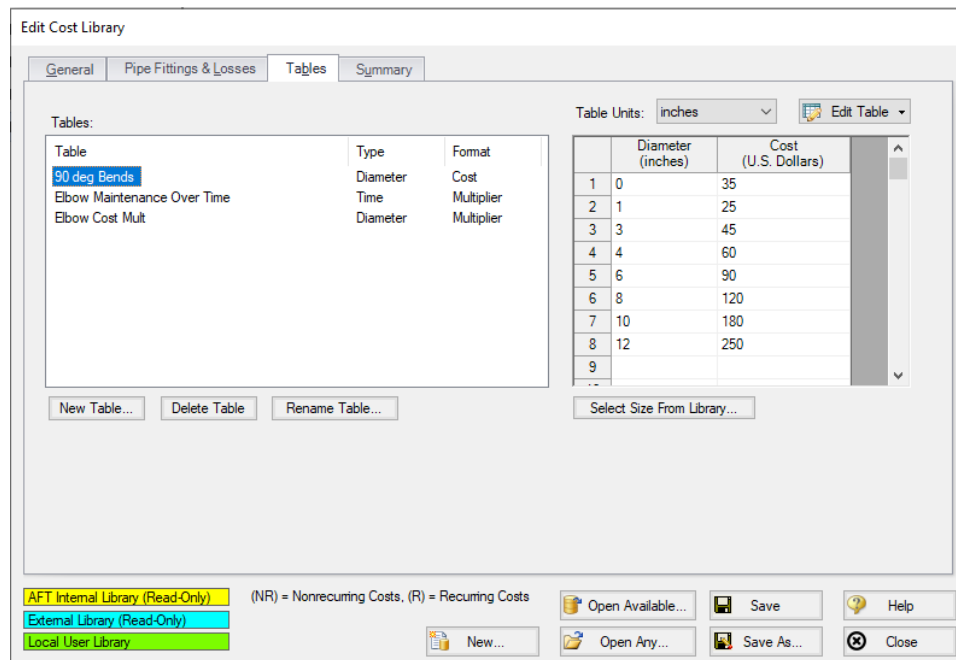
Figure 1: Scale table example of a table of costs (Diameter inches vs U.S. Dollars)

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 a cost item. 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			ow 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, like in Figure 3. 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

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 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 excel. 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.

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 except the volume balance.

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:

- **Pumps** - Cost can be entered per item (pump) or per Power. This will allow an approximate scaling for the size of the pump 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-pump 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 Fathom 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 Libraries

AFT Fathom can track user-defined costs such as maintenance, installation and material cost with a Cost Library.

Note that AFT Fathom 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 Fathom 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.
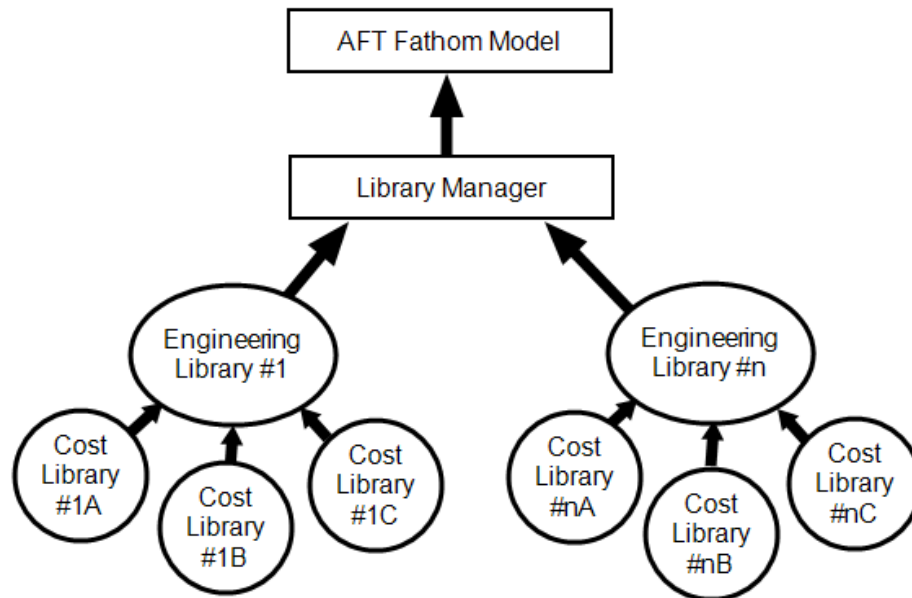
Figure 1: Relationship between Cost and Engineering Libraries

Like Engineering Libraries, Cost Libraries must be connected to be used. See Library Manager 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.
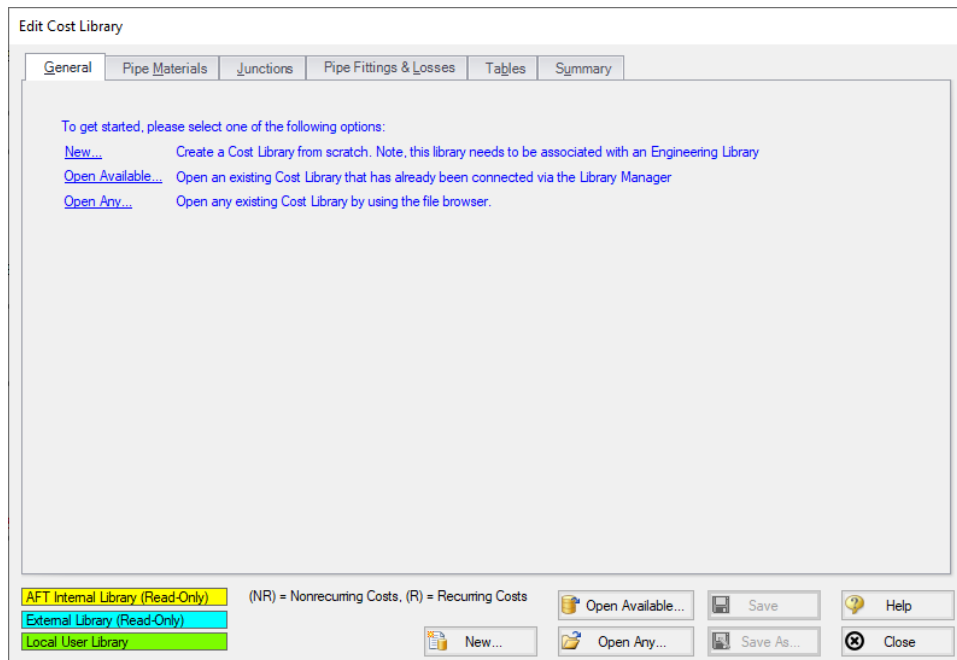
Figure 2: General tab of Edit Cost Library Window

Clicking New brings up the following window:
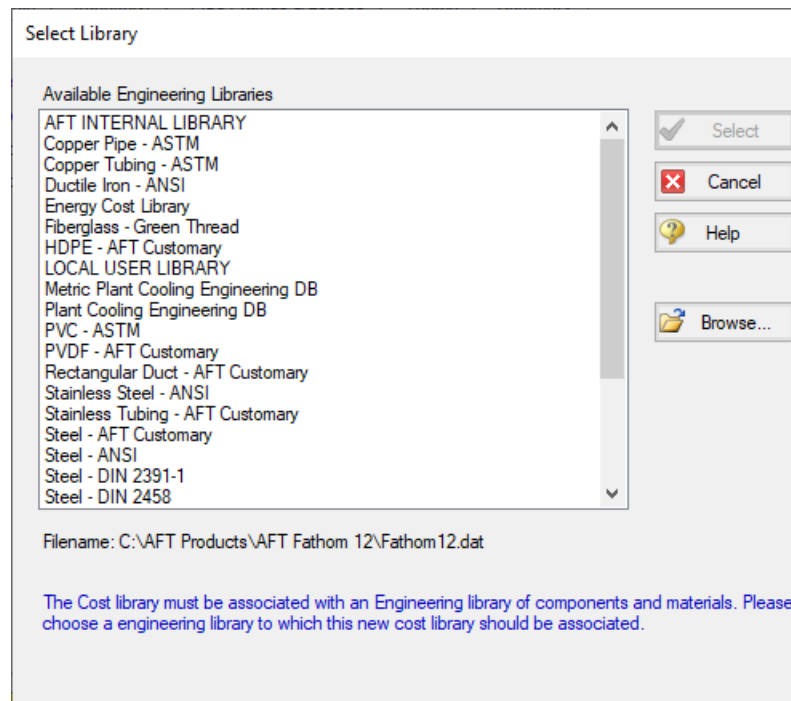
Figure 3: Select Library Window

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: Edit 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.
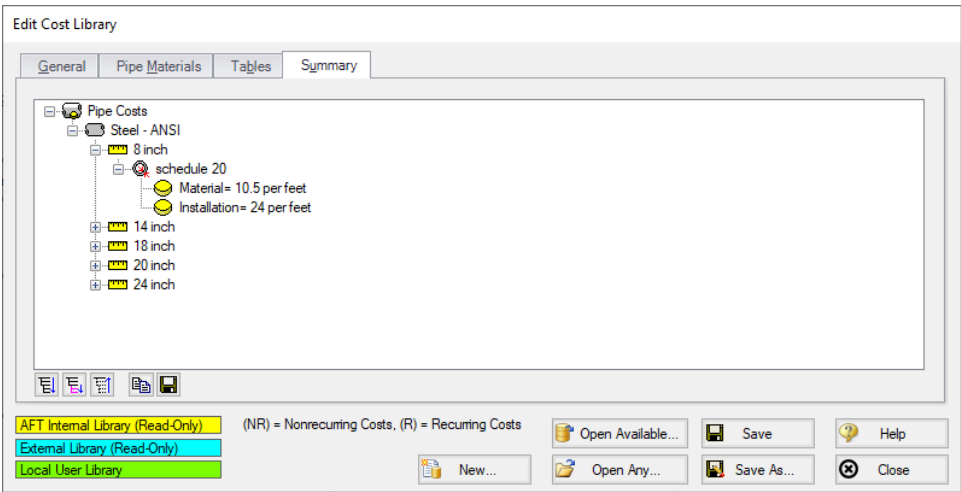
Figure 5: Cost Summary

Related Topics

Cost 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.
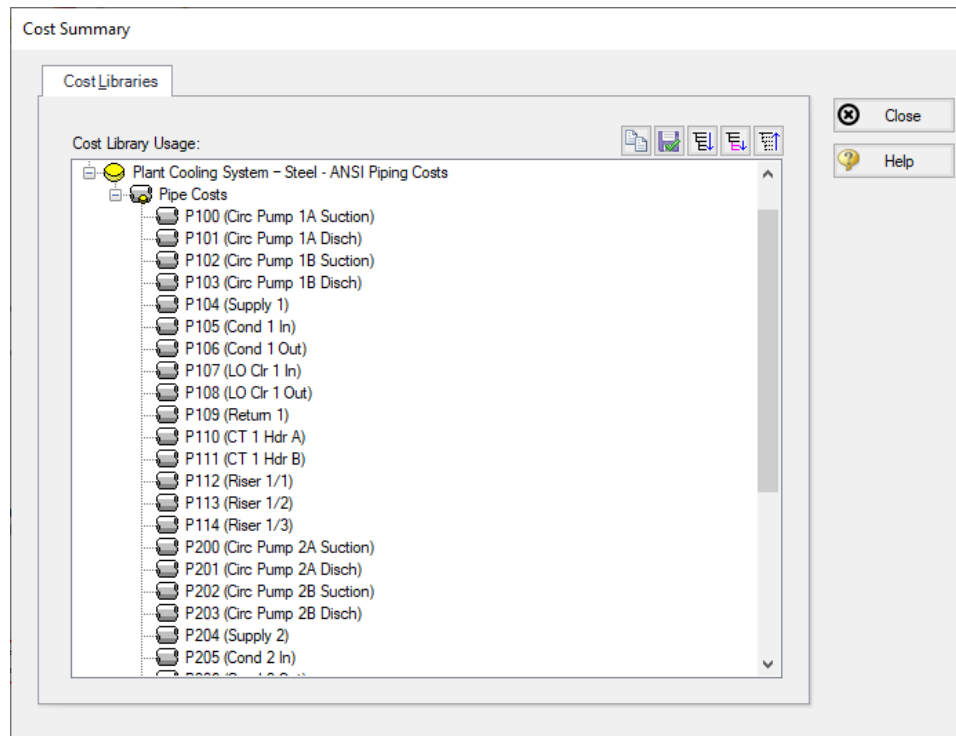
Figure 1: Cost Summary Window

Related Topics

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 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. You can also edit the number of decimal

places shown, or adjust the costs to be truncated to be shown in terms of hundreds, thousands, etc. instead of showing the full number.

Figure 1: The Cost Report is displayed in the General section of the Output window

Using the Library Sources Tables

AFT Fathom 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.

Figure 2: Library Sources table for pipes

Related Topics

Add-on Modules

AFT Fathom has four optional add-on modules which extend its modeling capabilities into other areas. The modules allow multivariable goal seeking and control simulation, settling slurry modeling, 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 four AFT Fathom add-on modules. An additional license to the base AFT Fathom 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.
- SSL (Settling Slurry) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.
- 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

GSC - Goal Seek & Control

What is the GSC Module?

An optional add-on module to AFT Fathom 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 like that used in the AFT Fathom ANS Module. The optimization engine employed by the AFT Fathom GSC module uses state-

of-the-art optimization technology licensed from Vanderplaats Research and Development, the leading company in optimization technology. 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 Fathom is performed by the Hydraulic Solver. In standard AFT Fathom 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.
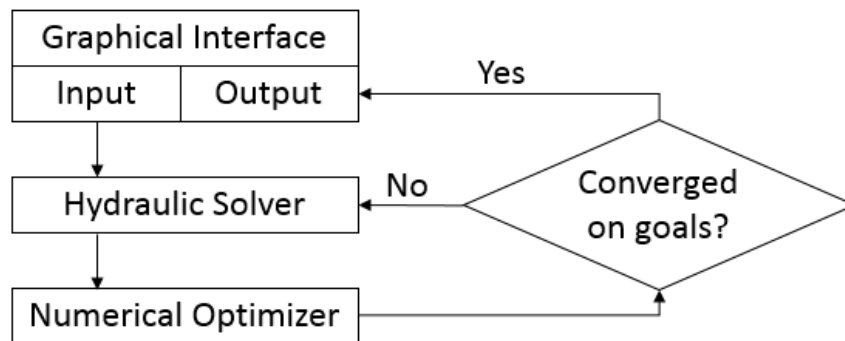
Figure 1: The GSC module flowchart shows how the Hydraulic Solver is called repeatedly in an iterative loop

Using the GSC Module

The user has the option of activating or not activating the GSC module when AFT Fathom first loads. After AFT Fathom is loaded, the GSC module can be activated or deactivated for use from the Modules panel in the Analysis Setup window or the Tools menu. Whether or not GSC is activated impacts the View menu, Analysis menu, Model Data window, Solution Progress window, and Output window as is discussed in the Modules panel topic.

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

GSC in Model Data

(GSC Module Only) 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.
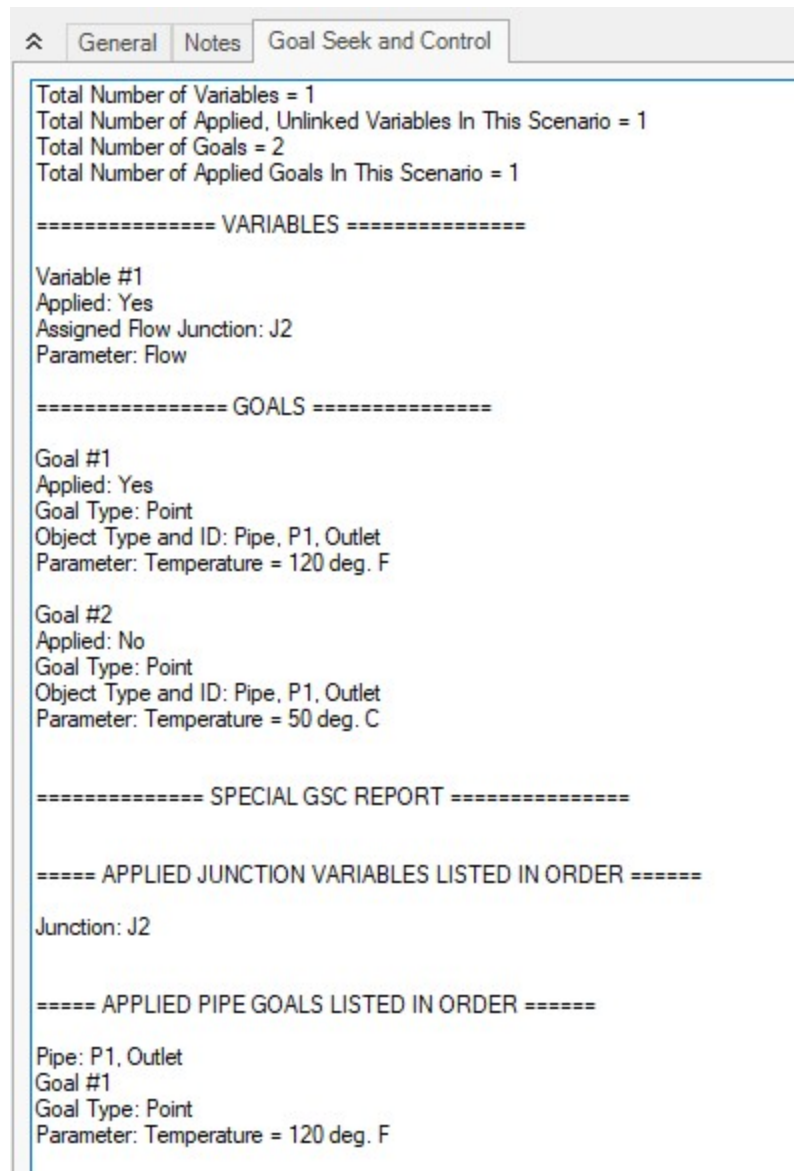
Figure 1: Model Data window shows the variable and goal input data

Related Topics

GSC in Solution Progress

(GSC Module Only) When a model is run with GSC active, an additional line will be shown in the Solution Progress window to indicate the progress that the GSC module has made towards reaching the specified goal. There are three fields displayed, as can be seen in Figure 1 below.

The "Best (Lowest)" and "Current" fields indicate the relative difference between the GSC iterations and the goal. The "Current" field shows the difference for the current iteration, and the "Best" field tracks the difference for the iteration that had the smallest difference from the goal. Both fields start at 100, as the initial iteration is the farthest from the goal. Both fields will decrease in magnitude as GSC proceeds closer to reaching the goal value. If these numbers are not significantly changing when the model is run, this is an indication that GSC may not be able to reach the goal with the current model settings. See the Goal Seek and Control Troubleshooting topic for more information on resolving this.

The field on the far right of the GSC Calls to Solver line tracks the total iterations performed by GSC. For every GSC iteration, GSC makes a call to the hydraulic solver in Fathom to determine the hydraulic solution with the adjusted value of the GSC variable. Thus, this value will always be smaller than the flow and head iterations.

Figure 1: The Solution Progress window has an additional line to show information on the GSC Calls to Solver when the Fathom model is run with GSC enabled

Related Topics

GSC in Output

(GSC Module Only) 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).
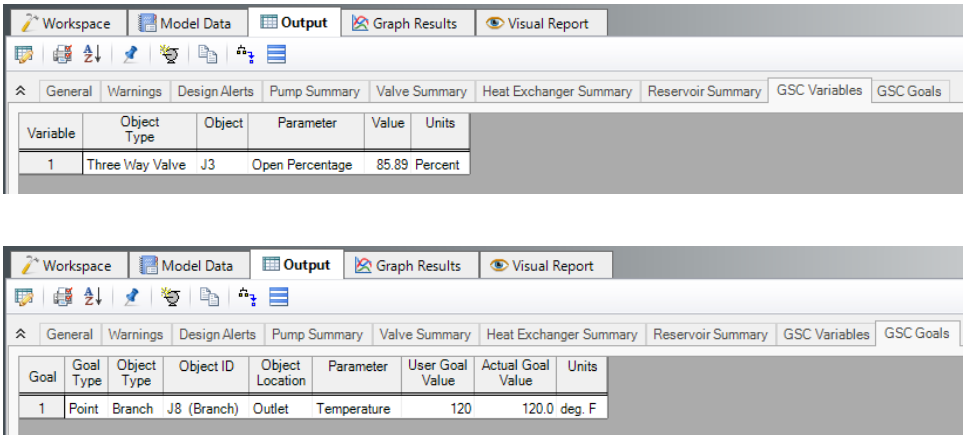
Figure 1: Calculated GSC Variables table (top) shows values required to achieve goals. GSC Goals table (bottom) shows user's goals and actual values achieved (these should match).

Related Topics

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

Changing Input Values to GSC Results

(GSC Module Only) 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 Guesses and generating a Disconnected Scenario.

Transferring Results to Initial Guesses

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, the calculated results, or 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, rerun the model, and obtain the same results as before. The options to transfer the calculated results will transfer the pipe and junction results to the initial guesses field, similar to what would be done if the GSC module were not enabled.

Generating a Disconnected Scenario

Generating a disconnected scenario 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 Troubleshooting

(GSC Module Only) See the below topics for more information on troubleshooting issues with the GSC module:

- 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
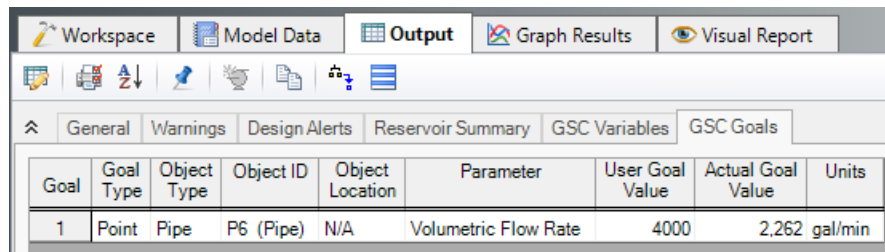
When Goals Cannot Be Achieved

(GSC Module Only) 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.
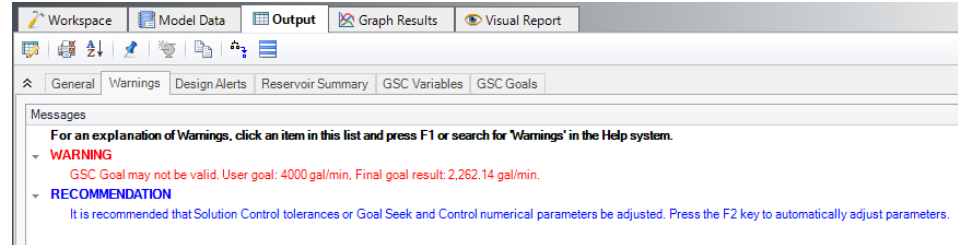
Figure 1: Warning display in Output window when a goal cannot be met

Physically Unrealistic Goals

It may be possible that a goal is not achievable because it is not physically possible. For example, a pump may be too small to satisfy a flow rate 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 area that can be changed in such cases: the hydraulic solution tolerances in the Steady Solution Control group and Numerical Controls panel in Analysis Setup.

In the next few sections recommendations on how to manually change these parameters will be given. However, before making a manual change it is worthwhile to try the automated parameter adjustment feature.

Figure 1, bottom picture, shows the warning when a goal is not met, and also shows a recommendation and suggest 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. Accepting the default values is typically a good starting point. Click OK, and rerun the model.

The purpose of the window shown in Figure 2 is to simplify making common changes to resolve goal seeking problems.
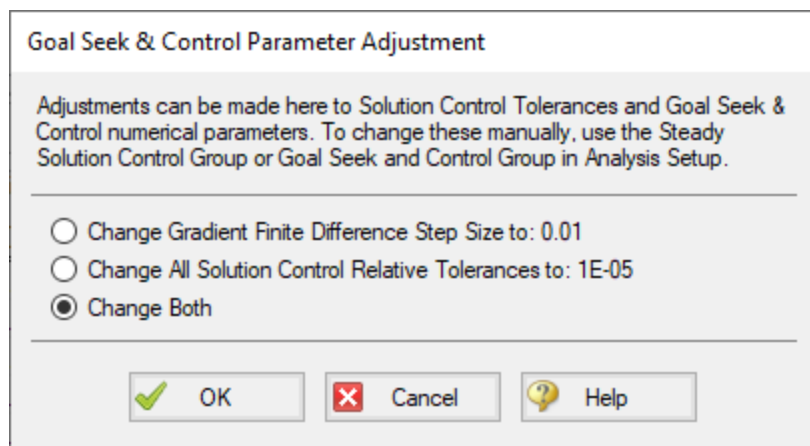
Figure 2: When goal seeking fails, pressing the F2 key will show this window which allows the user to automatically implement recommend numerical parameter adjustments

Caution: The user should be very cautious in manually changing parameters in the Steady Solution Control Group and Numerical Controls panel. Making changes without a good understanding of the parameters involved can lead to incorrect results.

- **Changing Tolerance on Tolerance panel** - By default AFT Fathom 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 Iterations panel for a more in-depth discussion.
- **Changing Goal Seeking Numerical Controls** - 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.
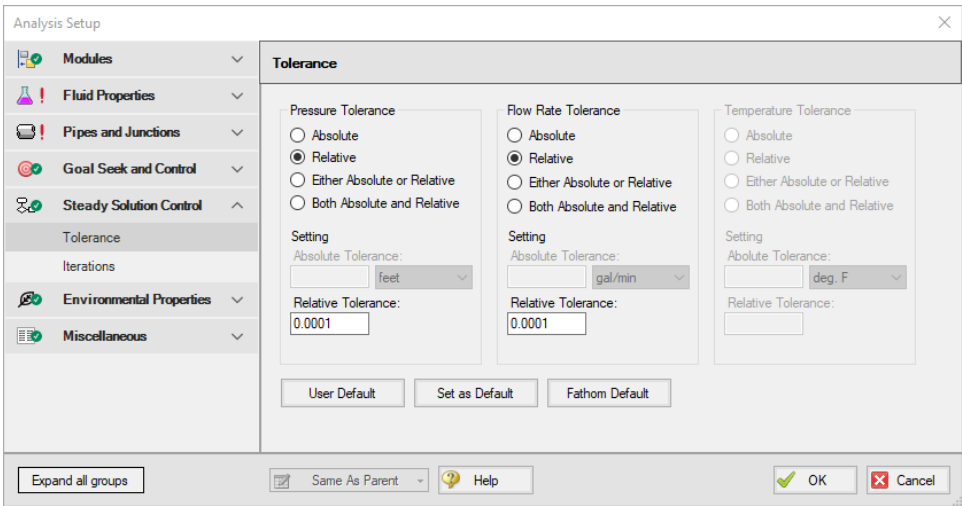
Figure 3: Reducing tolerance may resolve goal seeking difficulties
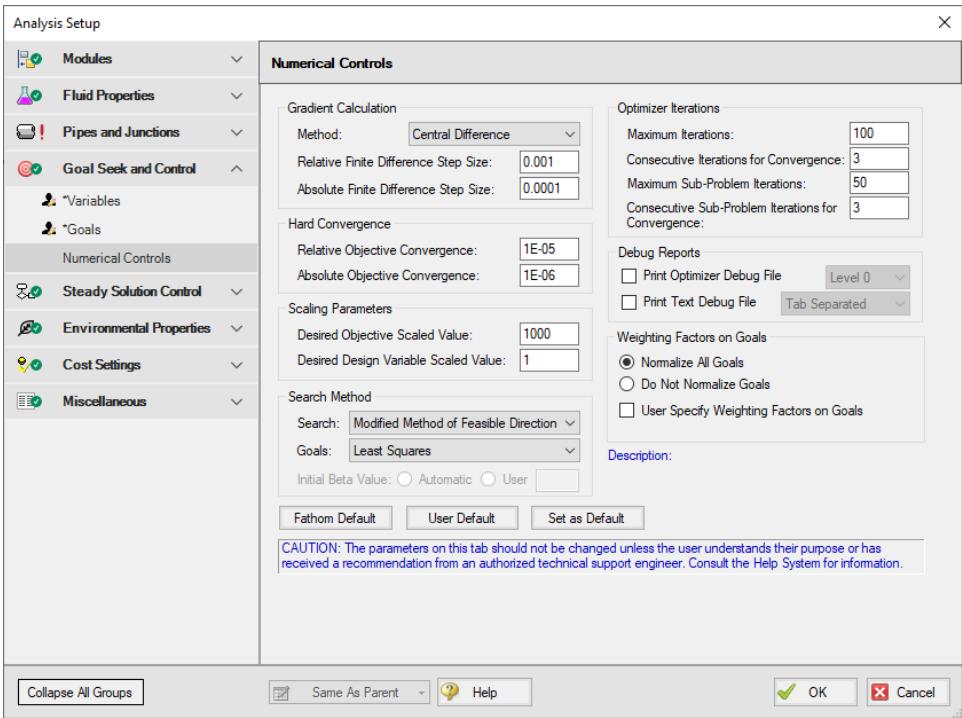
Figure 4: Changing certain numerical control parameters in the Numerical Controls panel in the Goal Seek and Control group may resolve goal seeking difficulties

Related Topics

What To Do if the Solver Gets Stuck in GSC

(GSC Module Only) 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 Fathom 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 Fathom 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 on the Iterations panel in Analysis Setup). It is perfectly fine for one to wait until this limit is reached and allow the AFT Fathom 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 Fathom to more quickly conclude the model that 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 Iterations panel, 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

GSC Adjustments

(GSC Module Only) When a GSC run results in the goals not being met, the user may want to adjust GSC Numerical Controls or 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 Numerical Controls panel, Tolerance panel, and/or Iterations panel.

Related Topics

GSC Inconsistencies

If an object in the Variables panel in Analysis Setup 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 Variables panel, or by changing the specified junction to be defined using the specified variable. Note the properties window for the junction can be opened directly by double-clicking the far left column of the row on the Variables table.

Related Topics

SSL - Settling Slurry

What is the SSL Module?

An optional add-on module to AFT Fathom which allows users to simulate settling slurries.

Settling slurries are fluids which carry solids of relatively large size which cannot be suspended in the liquid under all conditions. At sufficiently low velocities the solid particles will settle to the bottom of the pipe and may ultimately plug the pipe completely. It is important to predict the beha-

vior of such flows to avoid settling conditions and economically move the fluid and solids through the pipe.

The SSL module allows users to predict the behavior of settling slurries within the framework of the powerful AFT Fathom software. The SSL module allows users to specify solid particle properties, distribution and concentration. State-of-the-art correlations are then used to predict the flow, pressure drop and reduced pump performance when pumping solids.

Special output reports and system curve generation are available to assist the user with understanding the hydraulic behavior of the slurry.

The SSL module can work in conjunction with the other AFT Fathom modules. The XTS module allows users to model slurry flows of constant concentration which vary over time. The GSC module allows users to vary input parameters automatically to achieve desired output parameters.

How Does the SSL Module Work?

The SSL module offers additional input windows where users can enter the properties of the solid particles. The SSL module takes the properties of the solid particles and carrier fluid (which is often water but can be another fluid) and calculates the hydraulics of the piping and pumps using widely accepted correlations.

Predicting the flow of settling slurries depends heavily on correlations based on experiment. Such predictions are inherently less accurate than standard clear fluid pipe flow.

Using the SSL Module

The user has the option of activating or not activating the SSL module when AFT Fathom first loads. After AFT Fathom is loaded, the SSL module can be activated or deactivated for use from the Modules panel in the Analysis Setup window or the Tools menu. Whether or not SSL is activated impacts the Analysis menu, Library menu and Output window as is discussed in the Modules panel topic.

If the SSL module is active, the user can still run models without slurries. This is selected under Slurry Modeling on the Analysis menu. Hence there are three possibilities for the SSL module.

- SSL is not active
- SSL is active and slurry calculation is disabled
- SSL is active and settling slurry calculation is performed

Related Topics

Slurry Calculation Method Overview

(SSL Module Only)

Several different slurry pipe calculation methods are offered, each with their own assumptions and requirements for input data. The type of slurry model is selected on the Slurry Definition panel.

Wilson, Addie, Clift

The Wilson, Addie, Clift slurry methodology is taken from Wilson, et al, 2006. There are three Slurry Calculation Methods – Minimal, Simplified, and Detailed. These require an increasing level of knowledge and data about the slurry being modeled. Each of the methods are described with repeated information as there are many calculations that are common among the three.

- Minimal Slurry Calculation Method
- Simplified Slurry Calculation Method
- Detailed Slurry Calculation Method

4-Component

The 4-Component Slurry Model is a holistic approach to characterizing how slurry flows function by considering the combined effects of four slurry flow regimes.

Wasp-Durand

The Wasp-Durand Slurry Model is a two-layer slurry approach with heterogeneous (also referred to as the Durand flow) and homogenous fluid components (also referred to as the vehicle component) typically used for coal slurry pipeline applications.

Related Topics

Minimal Slurry Calculation Method

(SSL Module Only) The Minimal Slurry Calculation method was developed for fully-stratified flow, but can be used as a conservative method for two-layer heterogeneous flow. The following data is required for the Minimal Calculation Method:

- Temperature
- Solids concentration
- Density
- Sliding friction coefficient, μ_s

Engineering Assumptions

See the SSL Engineering Assumptions topic.

Horizontal Pipes

The pressure drop in a given by the relationship given by Wilson, et al, 2006, on page 116. As described in the discussion in the book, the coefficient 0.55 is based on sand or gravel sliding in steel pipe which has a coefficient of sliding friction, μ_s of 0.40. The equation then is modified with the ratio of the value entered on the Slurry Definition panel and becomes the following:

$$\frac{i_m - i_f}{S_m - S_f} = \left(\frac{V_m}{0.55\,V_{sm}}\right)^{-0.25}\left(\frac{\mu_s}{0.40}\right) \tag{1}$$

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. Note that when the Minimal input option is selected in Analysis Setup on the Solids Definition panel there is no value entered for the d_{50} or d_{85} particle sizes and therefore the M exponent, V_{50} and other parameters cannot be determined.

Related Topics

Simplified Slurry Calculation Method

(SSL Module Only) The Simplified Slurry Calculation Method was developed for two-layer heterogeneous slurry flow with a newtonian carrier fluid. This method is similar to the Detailed Slurry Calculation Method, but uses a simpler method to estimate V_{50}. The following data is required for the Simplified Calculation Method:

- Temperature
- Solids concentration
- Density
- d_{50} particle size
- M exponent value

Engineering Assumptions

See the SSL Engineering Assumptions topic.

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by Wilson, et al. 2006, on page 130

$$\frac{i_m - i_f}{S_m - S_f} = 0.22 \left(\frac{V_m}{V_{50}}\right)^{-M} \tag{1}$$

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. By adding in the base fluid pressure loss, the full head gradient is given by Wilson, et al. 2006, on page 134 (note that Wilson, et al, assume the fluid is water where here the fluid is generalized)

$$i_m = \frac{f_f}{2\,g\,D} V_m^2 + 0.22 \left(S_s - S_f\right) C_{vd} \left(\frac{V_m}{V_{50}}\right)^{-M} \tag{2}$$

Noting that:

$$\left(S_s - S_f\right) C_{vd} = S_m - S_f$$

we get the following solution for the total head gradient:

$$i_m = \frac{f_f}{2\,g\,D} V_m^2 + 0.22\,(S_m - S_f) \left(\frac{V_m}{V_{50}} \right)^{-M} \tag{3}$$

Calculating the V_{50} Value

Equation 3 above needs the velocity at which 50% of the solids are suspended by the fluid, V_{50}. An estimation shown in Wilson, et al, 2006, on page 141, is used.

$$V_{50} \approx 3.93\, d_{50}^{0.35} \left(\frac{S_s - 1}{1.65} \right)^{0.45} \nu_r^{-0.25} \tag{4}$$

The relative viscosity ν_r is the ratio of the actual fluid viscosity to that of water at 20 degrees C (i.e. 1.0016 centipoise).

Related Topics

Detailed Slurry Calculation Method

(SSL Module Only) The Detailed Slurry Calculation Method was developed for two-layer heterogeneous slurry flow with a newtonian carrier fluid. This method is similar to the Simplified Slurry Calculation Method, but includes more sophisticated methods to calculate M and V_{50}, as is discussed below. The following data is required for the Detailed Calculation Method:

- Temperature
- Solids concentration
- Density
- d_{50} particle size
- d_{85} particle size if the M exponent it so be calculated
- M exponent value if no d_{85} particle size is available
- Terminal velocity parameter, using either the Shape Factor (K) or V_t / V_{ts}

Engineering Assumptions

See the SSL Engineering Assumptions topic.

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by Wilson, et al, 2006, on page 130

$$\frac{i_m - i_f}{S_m - S_f} = 0.22 \left(\frac{V_m}{V_{50}}\right)^{-M}$$

(1)

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. By adding in the base fluid pressure loss the full head gradient is given by Wilson, et al, 2006, on page 134 (note that Wilson, et al, assume the fluid is water where here the fluid is generalized)

$$i_m = \frac{f_f}{2\,g\,D} V_m^2 + 0.22\,(S_s - S_f)\,C_{vd} \left(\frac{V_m}{V_{50}}\right)^{-M}$$

(2)

Noting that:

$$(S_s - S_f)\,C_{vd} = S_m - S_f$$

We get the following solution for the total head gradient:

$$i_m = \frac{f_f}{2\,g\,D} V_m^2 + 0.22\,(S_m - S_f) \left(\frac{V_m}{V_{50}}\right)^{-M}$$

(3)

Calculating the V_{50} Value

Equations 2 and 3 above need the velocity at which 50% of the solids are suspended by the fluid, V_{50}. This is discussed by Wilson, et al, 2006, with a functional relationship given on page 130.

$$V_{50} = w_{50} \sqrt{\frac{8}{f_f}} \cosh\left(60\,d_{50}/D\right)$$

(4)

Where:

- w_{50} is the particle associated velocity
- d_{50} is the particle diameter
- D is the pipe inner diameter

W_{50} used in Equation 4 above, is the particle associated velocity for the median particle size, and is defined in equation 5 below. The 3rd edition of Wilson (Wilson, et al, 2006, from which the equations for the rest of this discussion are taken) defines w using Figure 6.5 on page 136 of the text. The plot of w vs. particle size is based on an empirically-derived equation in the 2nd edition of the text (Wilson, et al, 1996), equation 6.12 found on page 134. The SSL module uses that equation (Equation 5 below) in its solution methodology.

$$w_{50} = 0.9v_t + 2.7\left[\frac{(\rho_s - \rho_f)\, g\mu}{\rho_f^2}\right]^{1/3} \tag{5}$$

Where:

- v_t is the particle settling velocity for the particle size in question
- ρ_s is the slurry density
- ρ_f is the fluid density

Calculating the M Exponent Value

If both the d_{50} and d_{85} values are entered, the M exponent value can be calculated using the method shown in Wilson, et al, 2006, on page 140.

$$M = \left(0.25 + 13\, \sigma_s^2\right)^{-0.50} \tag{6}$$

Where:

$$\sigma_s = \log\left[\frac{w_{85}\, cosh(60\, d_{85}/D)}{w_{50}\, cosh(60\, d_{50}/D)}\right] \tag{7}$$

When using these calculations M is limited to between 0.25 and 1.7.

Related Topics

4-Component Slurry Model

(SSL Module Only) The 4-Component Slurry Model is a holistic approach to characterizing how slurry flows function by considering the combined effects of four slurry flow regimes. The model strongly references and utilizes the method of the same name outlined by GIW Industries Inc.

The 4-Component Model can be selected on the Slurry Model dropdown in the Slurry Definition panel of Analysis Setup. Optionally, a Solids Library can be created to define the slurry properties.

Engineering Assumptions

See the SSL Engineering Assumptions topic.

The Four Slurry Regimes

The four slurry flow regimes are:

- Carrier Fluid
- Pseudo-homogeneous
- Heterogeneous
- Fully Stratified

The Carrier Fluid regime considers particles smaller than 40 μm. These solid particles mix with liquid in such a way that mixture viscosity is affected.

The Pseudo-homogeneous regime considers particles larger than 40 μm and smaller than 200 μm. Due to the turbulence, the solid particles are supported within the fluid. The mixture's viscosity is not affected by this regime.

The Heterogeneous regime considers particles larger than 200 μm and smaller than 1.5% of the pipe's diameter (μm). Particles of this size begin to settle creating a heterogeneous mixture where the solids are found in greater concentration at the bottom of the pipe. Turbulence in the fluid and contact with the pipe wall support these solids from settling out of the fluid.

The Fully Stratified regime considers particles larger than 1.5% of the pipe's diameter (μm). Solids are primarily supported by the pipe wall and will slide along the bottom of the pipe. A sliding bed friction model is used for this flow regime.

Carrier Fluid
particles <40 μm

Pseudo-homogeneous
40 μm < particles < 200 μm

Heterogeneous
200 μm < particles < 0.015D

Fully Stratified
0.015D < particles

The 4-Component Slurry model offers a comprehensive and advanced approach to slurry modeling, enabling more accurate and detailed analysis of slurry behavior in various particle size distributions.

The model was developed by industry-leading experts and is respected for its accuracy in part because the individual components are based on well established sub models.

Figure 1: The 4-Component Slurry Model considers combined effects from four slurry regimes

The water pressure gradient i_m (in meters of water head per meter of pipe) from frictional losses is the sum of i_f, the pressure gradient of the carrier fluid component, with the excess pressure gradients of the pseudo-homogeneous component Δi_p, the heterogeneous component Δi_h, and the fully stratified component Δi_s. The slurry pressure gradient j_m is related to i_m by the specific gravity of the total slurry mixture S_m.

$$i_m = i_f + \Delta i_p + \Delta i_h + \Delta i_s$$

$$j_m = \frac{i_m}{S_m}$$

Defining and Using a 4-Component Slurry Model

To begin defining and using the 4-Component Slurry Model, enable the Settling Slurry Module with the radio button set to Settling. Define the Carrier Fluid on the Carrier Fluid panel. On the Slurry Definition panel, set the Slurry Model dropdown to Settling Slurry - 4-Component Model. Navigate to Solids Definition panel and select one of the Solids Specifications options. If a Solids library is attached, it can likewise be selected from the dropdown. Density, d100 (the Maximum Solid Diameter), and a Particle Size Distribution are required to be filled out. Once the Solids Definition panel is completely defined, navigate to the Slurry Definition panel. Enter an Amount Solids Added and Sliding Friction Coefficient in the required fields. With all of the required information entered, the 4-Component Slurry Model can now be used.

4-Component SSL Pump De-Rating is available for use with 4-Component slurries.

Related Topics

Wasp-Durand Heterogeneous Slurry Model

(SSL Module Only) The Wasp-Durand Slurry Model is a two-layer slurry approach with heterogeneous (also referred to as the Durand flow) and homogenous fluid components (also referred to as the vehicle component). This method started with the heterogeneous Durand and Condolios model in the 1950s, was expanded by Wasp et al. in the 1960s and 1970s, and has continued to evolve based on the research of Liu Dezhong, Fei Xiangjun, Richard W. Hanks and others. This method is used most often with long distance coal pipelines, particularly for coal slurries and slurries with relatively fine particles.

The homogenous component incorporates the viscosity effects of fine particles, similar to the 4-Component Slurry Model, but there is no hard size boundary on which particles can affect the viscosity but finer particles have a greater impact. Similarly, particles of any size may also contribute to the heterogeneous component depending on the degree of stratification for their size group, making Wasp-Durand a model well-suited for finer slurries that may encounter modeling limitations when using Wilson, Addie, Clift or 4-Component slurry models.

Engineering Assumptions

See the SSL Engineering Assumptions topic.

Definition of Solids

Use of Wasp-Durand requires a Granulometric Spread Distribution. This particle size distribution may be defined in either the Solids Library or the Solids Definition Panel of the Analysis Setup Window. A Granulometric Spread Distribution is a histogram-like distribution defined in terms of the fraction of total solids in each size group and the mean particle diameter of each size group.

Stratification in Wasp-Durand is calculated per size group as a ratio of the concentration of solids near the top of the pipe to the concentration of solids towards the middle of the pipe, as shown in Figure 1.
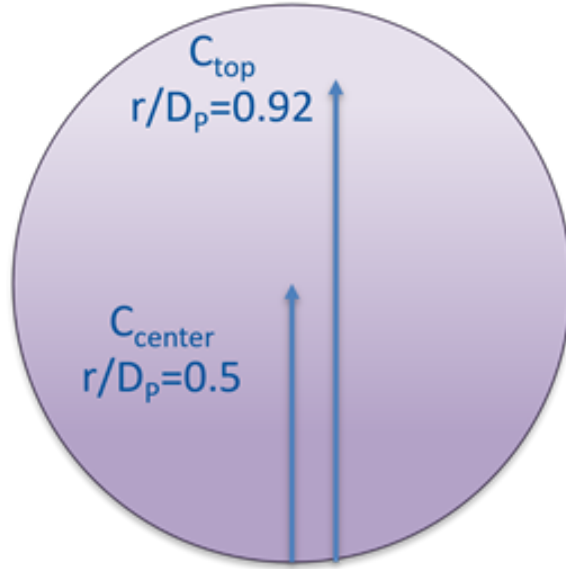
Figure 1: A cross-sectional illustration of the locations within a pipe where Wasp-Durand top and center solids concentrations are defined

Different industries and authors may use different notations for the Wasp-Durand stratification term, but any of the forms shown in Equation 1 are equivalent.

$$\frac{C_v}{C_{vA}} = \frac{C_{top}}{C_{center}} = \frac{C}{C_A}$$

(1)

Wasp-Durand is essentially a modular approach, and can be customized using the Wasp-Durand Options area of the Slurry Definition panel.

Deposition Velocity

The deposition velocity is the velocity where solids of a given size have come out of the slurry forming a stationary bed on the bottom of the pipe. For Wasp-Durand, the selection of heterogeneous calculation method determines which method is used for the deposition velocity calculation. The Liu Dezhong deposition velocity calculation is shown in Equations 2 and 3.

$$V_{sm} = 9.5\left[gD\left(\frac{\rho_s}{\rho_h} - 1\right)V_t\right]^{1/3} C_V^{1/m}\left(\frac{V_{tw}}{V_t}\right)^{1/6}$$

(2)

$$m = \begin{cases} 4.2857\frac{C_{top}}{C_{center}} + 2.5714 & \text{if } \frac{C_{top}}{C_{center}} < 0.8 \\ 6 & \text{if } \frac{C_{top}}{C_{center}} \geq 0.8 \end{cases}$$

(3)

Where ρ_s is the solid density, ρ_h is the homogenous component density, C_V is the volumetric concentration of solids, V_t is the weighted average of sedimentation/terminal velocities of each particle size group in the homogeneous/vehicle fluid, and V_{tw} is the weighted average of particle sedimentation velocities in a pure carrier fluid (i.e. neglecting viscosity and density effect of fine solids). The m exponent is constant for quasi-homogeneous flow, and variable for more heterogeneous flow regimes.

This version of the Liu Dezhong deposition velocity calculation with the m exponent is from the Chinese national slurry transport code, and differs from the original form in which m is always 6.

Note: The sedimentation velocity terms are calculated using whichever method is specified as the Sedimentation Velocity Model.

The Wasp and Fei Xiangjun equations for deposition velocity are shown in Equations 4 and 5, respectively.

$$V_{sm} = 3.113\, C_V^{0.1858} \left[2gD \left(\frac{\rho_s - \rho_m}{\rho_m} \right) \right]^{1/2} \left(\frac{d_{85}}{D} \right)^{1/6} \tag{4}$$

$$V_{sm} = \frac{2.26}{\sqrt{f_h}} \left[gD \left(\frac{\rho_s - \rho_m}{\rho_m} \right) \right]^{0.5} C_V^{1/4} \left(\frac{d_{90}}{D} \right)^{1/3} \tag{5}$$

Where ρ_m is the overall slurry density and f_h is the homogeneous Darcy friction factor. Different particle diameters are used by different sources for the Wasp deposition velocity calculation, and we follow Liu Dezhong's convention of using d_{85}.

Sedimentation Velocity

The sedimentation velocity is per particle size group and is the terminal particle free-fall velocity that occurs when the drag, gravity, and buoyancy forces on a particle are balanced. Both the Ruby & Zanke and Liu Dezhong approaches can be used across a wide range of particle sizes and flow conditions, including laminar, transition, and turbulent flow. Equations 6 and 7 show the Ruby & Zanke (1977) approach for terminal settling velocity.

$$V_{t,i} = \frac{10\, v_h}{d_i} \left[\sqrt{1 + \frac{\rho_s - \rho_h}{\rho_h} \frac{g\, d_i^3}{100\, v_h^2}} - 1 \right] \tag{6}$$

$$v_h = \frac{\mu_h}{\rho_h} \tag{7}$$

Where $V_{t,i}$ is the terminal settling velocity for each particle size group i and μ_h is the dynamic viscosity of the homogeneous component.

The Liu Dezhong sedimentation velocity calculation is shown in Equations 8-11. The particle size number for each particle size group $N_{d,i}$ and μ_f is the clear fluid viscosity. The homogeneous Bingham stiffness factor μ_{ph} is calculated as shown in Equations 22 and 23.

$$V_{t,i} = \left[g\left(\frac{\rho_s}{\rho_h} - 1\right) \frac{\mu_{ph}}{\rho_h} \right]^{1/3} N_{\omega,i} \tag{8}$$

$$N_{\omega,i} = \frac{20.5209}{N_{d,i}} \left[\left(1 + 0.105587812 \, N_{d,i}^{1.5}\right)^{0.5} - 1 \right]^2 \tag{9}$$

The weighted average sedimentation velocity is shown in Equation 10. The decimal solid fraction in each band from the Granulometric Spread Distribution is w_i.

$$V_t = \sum w_i V_{t,i} \tag{10}$$

Heterogeneous Gradient

The total slurry gradient takes the form shown in Equation 11, where i_1 is the homogenous gradient and Δi_2 is the heterogeneous contribution. The homogeneous gradient will depend on which slurry viscosity model is applied and whether or not Hanks' Bingham plastic is utilized, while the heterogeneous component is calculated with the method specified by the user in the Slurry Definition panel.

$$i = i_1 + \Delta i_2 \tag{11}$$

The Wasp-Durand heterogeneous gradient is shown in Equations 12-14, where C_{1V} is the volumetric concentration of the homogeneous portion, C_{2V} is the volumetric concentration of the heterogeneous portion, and C_{1D} is the settlement resistance coefficient of particles in the homogenous fluid.

$$\Delta i_2 = K \, i_1 \left[\frac{gD\left(\frac{\rho_s}{\rho_h} - 1\right)}{V^2} \right]^{1.5} \sum (C_{2V})_i (C_{1D})_i^{-0.75} \tag{12}$$

$$(C_{1D})_i = \frac{4gd_i \left(\frac{\rho_s}{\rho_h} - 1\right)}{3V_{t,i}^2} \tag{13}$$

$$K = \begin{cases} 150 & \text{if } C_{1V} > C_{2V} \\ 82 & \text{if } C_{1V} < C_{2V} \end{cases} \tag{14}$$

The heterogeneous gradient method for Liu Dezhong in defined in Equations 15 and 16, while Equation 17 defines the Fei Xiangjun approach.

$$\Delta i_2 = \frac{k}{V}\left[\frac{\rho_s}{\rho_h} - 1\right]\sum (C_{2V})_i V_{t,i} \tag{15}$$

$$k = 15\frac{C_{top}}{C_{center}} + 1.2 \tag{16}$$

$$\Delta i_2 = \frac{4.07\,V_t C_V}{V}\left[\frac{\rho_s}{\rho_h} - 1\right] \tag{17}$$

Apparent Viscosity

The option to Apply Apparent Slurry Properties allows the viscosity of the homogenous portion to be impacted by the presence of solid particles. These can be Newtonian viscosities, or a non-Newtonian approximation if the Hanks' Bingham Plastic option is enabled. This is described in the next section.

The Newtonian Fei Xiangjun model for apparent viscosity is defined in Equation 18, where C_{Vm} is the maximum volumetric concentration.

$$\mu_h = \mu_f\left(1 - \frac{C_V}{C_{Vm}}\right)^{-2} \tag{18}$$

The Thomas (1965) model for apparent dynamic viscosity of the homogenous component is shown in Equation 19.

$$\mu_h = 1 + 2.5C_V + 10.05C_V^2 + 0.00273e^{16.6C_V} \tag{19}$$

Hanks' Bingham Plastic

The Hanks' Bingham Plastic Approach was developed by Richard W. Hanks. This is a two-layer model that assumes slug or plug flow in an annular distribution, with a constant velocity slug surrounded by a non-Newtonian boundary layer in which frictional losses are concentrated. The general expression for Bingham Plastic shear stress is shown in Equation 20, where τ_0 is the

yield stress and μ_{ph} is the effective Bingham stiffness factor of the Wasp-Durand vehicle component.

$$\tau = \tau_0 + \mu_{ph}\frac{du}{dy} \tag{20}$$

In the context of the Wasp-Durand approach, Hanks' Bingham Plastic is used to calculate the friction from the vehicle opponent and allows treating this component as apparently non-Newtonian. The yield stress may be user-specified, or calculated using the Fei Xiangjun method in Equation 21.

$$\tau_0 = 0.098 \exp\left(B\frac{C_V - C_{V0}}{C_{Vm}} + 1.5\right) \tag{21}$$

Where C_{V0} is the boundary concentration between the Newtonian and non-Newtonian viscosity regimes and C_{Vm} is the maximum volumetric concentration. B is an empirical coefficient equal to 8.45 for mineral slurries and 6.87 for coal slurries. The yield stress is a fluid property, and may be calculated in the Analysis Setup Window without running the solver.

Fei Xiangjun also defines a method for calculating the Bingham stiffness factor, shown in Equations 22 and 23, which is used when Fei Xiangjun is selected as the apparent viscosity model and Hanks' Bingham Plastic is enabled in Wasp-Durand options.

$$\mu_{ph} = \mu_f \left(1 - K\frac{C_V}{C_{Vm}}\right)^{-2.5} \tag{22}$$

$$K = 1 + 2\left(\frac{C_V}{C_{Vm}}\right)^{0.3}\left(1 - \frac{C_V}{C_{Vm}}\right)^4 \tag{23}$$

The clear fluid friction factor is μ_f.

Related Topics

Vertical Pipe Theory For Slurries

(SSL Module Only) Flow in vertical pipes offer a special-case flow regime. The basic equation for flow rising in a vertical pipe is given in Wilson, et al, 2006, on page 178.

$$\frac{dP}{dz} = \rho_w\, g\, [1 + (S_s - 1)\, C_{vi}] + \frac{\rho\, g\, f\, V_m^2}{2\, g\, D} \tag{1}$$

It is important to note that this is the total pressure drop per unit length whereas the horizontal pipe equations have components for the pure fluid and the solids separately.

Noting that for water, for which Equation 1 is based, we can substitute:

$$S_m = 1 + (S_S - 1)\, C_v$$

and obtain the following basic relationship:

$$\frac{dP}{dz} = \rho_w\, g\, S_m + \frac{\rho\, g\, f\, V_m^2}{2\, g\, D} \tag{2}$$

Wilson, et al, states that for particles smaller than 150 microns, the density in the final term should be the mixture density:

$$\rho_m = \rho_w\, S_m$$

Substituting this in to Equation 2 we obtain:

$$\frac{dP}{dz} = \rho_w\, g\, S_m \left(1 + \frac{f_w\, V_m^2}{2\, g\, D}\right) \tag{3}$$

The friction factor f could be found with homogenous methods, but it is often reasonable to use the friction factor for a water flow f_w with the same mean velocity. For larger particles Wilson, et al, suggests that the density term should be that of water which, when substituted in to Equation 2 yields:

$$\frac{dP}{dz} = \rho_w\, g\, \left(S_m + \frac{f_w\, V_m^2}{2\, g\, D}\right) \tag{4}$$

Related Topics

Sloped Pipe Theory For Slurries

([SSL Module Only](#))

Wilson, Addie, Clift Settling Slurry

Sloped pipe calculations are a combination of [horizontal head loss](#) and a [vertical component](#). A widely used formula is given in [Wilson, et al, 2006](#), on page 183. This is used for all [Wilson, Addie, Clift](#) slurry models.

$$\Delta i\,(\theta) = \Delta i\,(0)\cos\theta + (S_s - 1)C_{vd}\sin\theta \tag{1}$$

Where $\Delta i(\theta)$ is simply the excess pressure gradient $(i_m - i_w)$ for a pipe at an angle θ.

The term $\Delta i(\theta)$ is the solids effect for a horizontal pipe and is calculated using the equations described in that section above. It is modified by $cos(\theta)$ to reflect the pipe inclination from horizontal.

The Deposition Velocity, V_{sm}, is also modified based on pipe angle. The following is added to the value calculated for horizontal pipes.

$$\Delta_D\sqrt{2\,g\,D(S_s - 1)} \tag{2}$$

Where Δ_D is determined from the graph in Wilson, et al, 2006, on page 181.

4-Component Settling Slurry

The sloped pipe corrections used for the [4-Component Slurry Model](#) developed by [GIW Industries Inc](#) are similar to the Wilson, Addie, Clift approaches shown above, but have been adapted for use with multiple components and the graphical method for Δ_D has been replaced with an analytical expression. Settling velocity corrections are shown in Equations 3 and 4.

$$V_{sm,\theta} = V_{sm,horizontal} + \Delta_D\sqrt{2gD(S_s/S_f - 1)} \tag{3}$$

$$\Delta_D = 0.75\,\theta - \frac{0.50\,(0.6366\,\theta)^2}{1 - 0.6366\,\theta} \quad for\ \theta > 0$$

$$\tag{4}$$

$$\Delta_D = 0.75\,\theta - \frac{0.02\,(2.29\,\theta)^2}{1 - 2.29\,\theta} \quad for\ \theta < 0$$

Where D is the pipe diameter in meters, θ is the pipe angle in radians, S_s is the solid specific gravity and S_f is the specific gravity of the carrier portion of the fluid which is affected by particles smaller than 40 microns. A plot of Δ_D is shown in Figure 1.
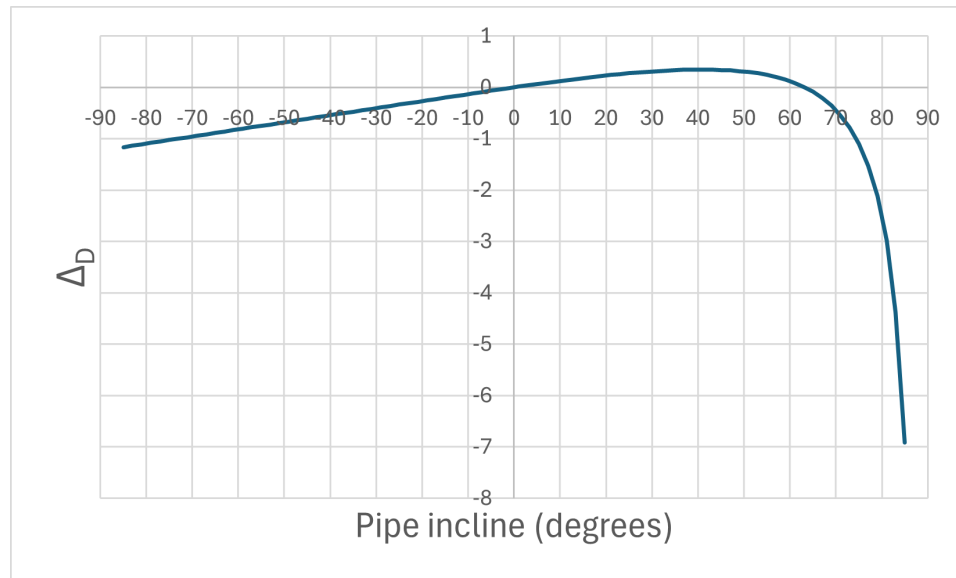
Figure 1: A plot of the analytical form of delta D used for 4-Component slope corrections

The corrected pressure gradient is shown in Equation 5. Note that the solutions to the heterogeneous gradient component Δi_h and the stratified gradient component Δi_s are affected by the change in deposition velocity, and therefore differ from their horizontal values.

$$i_m = i_f + \Delta i_p + (\Delta i_h + \Delta i_s) \cos \theta$$

(5)

Wasp-Durand Heterogeneous Model

The Wasp-Durand Heterogeneous Slurry Model does not account for inclined pipes or vertical conveying effects. For this reason, a caution message will be displayed for pipes with an angle greater than 8 degrees, and a warning message will be shown for pipes angled beyond 18 degrees.

Related Topics

SSL Nomenclature

([SSL Module Only])

C_E	derated pump multiplier on efficiency
C_H	derated pump multiplier on head
C_m	mass concentration of solids
C_{vd}	volume concentration of solids, delivered
C_{vi}	volume concentration of solids, in situ
C_{vb}	volume concentration of solids in sliding bed (bed concentration)
d_{50}	mass-median particle diameter
d_{85}	diameter for which 85% by mass of the particles are finer
d_{100}	max particle diameter size (100% by mass of the particles are finer)
D	inner diameter of pipe
f	friction factor (Darcy)
f_f	friction factor when calculating i_{fg}
f_{turb}	turbulent friction factor
f_w	friction factor of water
g	acceleration due to gravity
i	head gradient
i_f	head gradient contribution from fluid
i_m	value of i for flow in terms of water density
i_s	head gradient contribution from solids
i_w	head gradient contribution from fluid when fluid is water
j_m	head gradient in terms of slurry density
K	loss factor
K	Power Law constant
L	pipe length
L_{eq}	equivalent pipe length of fittings
M	an exponent in the stratification ratio equation

ρ_w	density of water
S_s	relative density of solids in slurry
S_f	relative density of fluids in slurry
S_m	relative density of slurry
μ_s	coefficient of sliding bed friction
V_{50}	value of V_m at which 50% of the solids are suspended in the fluid
V_m	mean velocity of mixture
V_{sm}	deposition velocity, velocity at which particles have all settled out
V_t	terminal velocity of a solid particle in the slurry carrier fluid
V_{ts}	terminal velocity of a perfectly spherical solid particle in the slurry carrier fluid

SSL Engineering Assumptions

(SSL Module Only)

Calculations done with the SSL module are based on the following assumptions:

Applicable to All Slurry Methods

- Slurry models are empirical models developed from steady state flow data.
- Solid particles are moving with the fluid. Stationary beds cannot be modeled. Sliding beds can be modeled with caution.
- Dynamic mixing of slurries with different solid concentrations is not modeled.
- Heat transfer is not modeled.
- The settling of slurries is modeled in pipes only, and settling is not modeled within junctions or components.

Wilson, Addie, Clift Slurry Model

- Base liquid behaves in a Newtonian viscosity manner.
- Slurry viscosity is equivalent to the viscosity of the carrier fluid.
- Particles below 150 µm are treated as part of the carrier fluid.
- Minimal Slurry Calculation Method assumes all particles are in a sliding bed and none are suspended by turbulence.
- Simplified and Detailed methods assume a sliding friction coefficient of 0.45.
- The velocity used to calculate the friction factor in the deposition velocity limiting equation is approximated as the current pipe velocity.

4-Component Slurry Model

- Carrier fluid component behaves in a Newtonian viscosity manner.
- Only particles below 40 μm affect carrier fluid viscosity.
- The deposition velocity nomograph is assumed to follow the same behavior as the Wilson, Addie, Clift nomograph.
- The limiting equation for deposition velocity is not used.

Wasp-Durand Slurry Model

- Carrier fluid viscosity is assumed to be Newtonian unless the Hank's Bingham Plastic option is used.
- Particle distribution along the vertical axis of the pipe cross-section is assumed to be continuous. Due to this, the heterogeneous gradient will be underestimated in systems with significant sliding beds.
- All pipes are assumed to be horizontal.
- All particle size groups are assumed to contribute to both the homogeneous and heterogeneous flow components with larger particles contributing more to the heterogeneous (Durand) portion and smaller particles contributing more to the homogeneous (vehicle) portion.

SSL Flow Rates

(SSL Module Only) When specifying or referring to slurry flow rate, three different meanings of flow rate are possible and two are commonly used. The definitions for flow rate are as follows:

- **Slurry** - Combined flow rate of the fluid and solids. This is the most common flow rate definition.
- **Solids** - Flow rate of just the solids in the slurry. This is often used because the purpose of the slurry is frequently to convey solids and this flowrate is of the greatest importance to the engineer.
- **Clear Liquid** - Flow rate of the carrier fluid. This is not commonly used.

The three flow rate definitions are of course interrelated. The slurry flow rate is the sum of the other two. As shown in the slurry output section, all three flow rates can be displayed in the Output window.

The three flow rate definitions can be specified in three places in AFT Fathom. Two are on the Pump Properties Window and one is on the Assigned Flow Properties window.

An example of one of the three locations is shown in Figure 1 in the Pump Properties Window. When the SSL module is in use there is a dropdown list adjacent to the flowrate specification fields (as shown in Figure 1) that allows one to choose one of the three. The default selection is the flow rate of slurry.

Note that anywhere in AFT Fathom that specifies or displays flow rate information and does not offer the qualifier of slurry/solids/clear liquid, the flow rate of slurry is assumed.
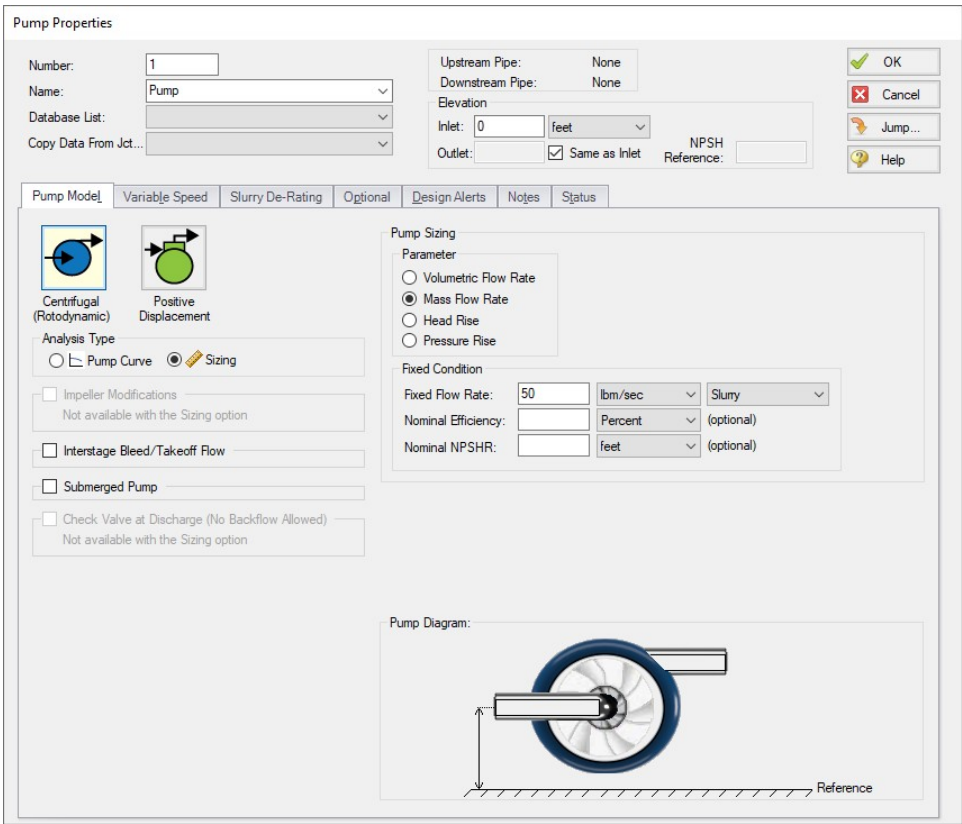
Figure 1: Pump Properties Window with fixed mass flow rate of slurry specified

Related Topics

Estimating M From Recorded Data

(SSL Module Only) There is an option on the Solids Definition panel of the Analysis Setup window and Solids Library to estimate *M* from recorded data. Choose the button for *M* Curve Fit. This is based on discussion in Wilson, et al, 2006, on pages 128-136.

Figure 1 shows the *M* Parameter Curve Fit window. Data for velocity and relative excess pressure gradient are entered and a linear curve fit is generated on a log-log plot. The *M* factor is displayed at the top. Data that does not fit the curve can be disregarded by the user by deselecting the checkbox in the table.

M values are not allowed to be greater than 1.7 or less than 0.25 based on recommendations in Wilson, et al.
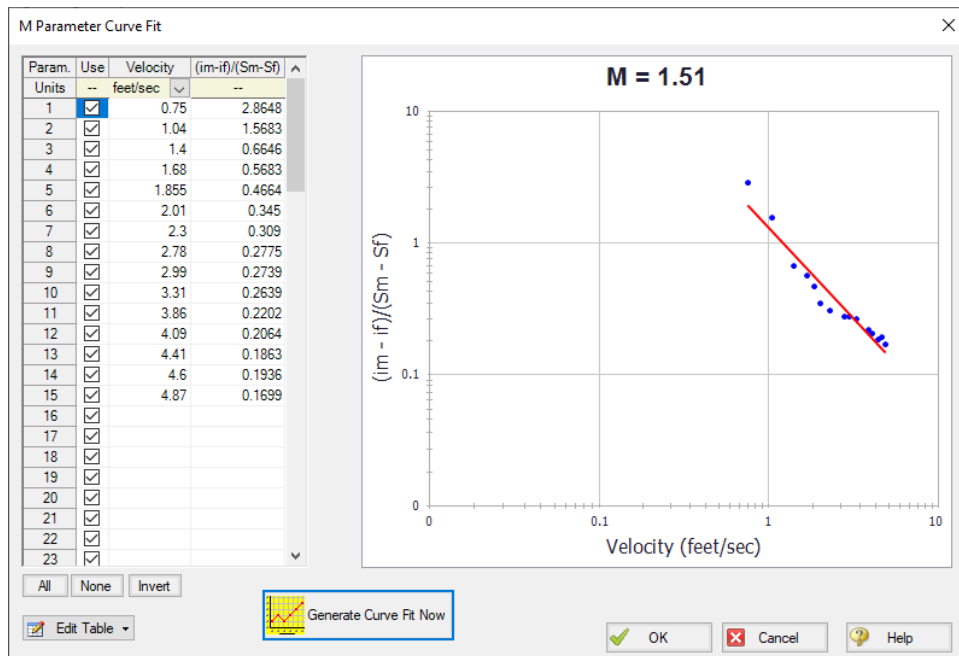
Figure 1: M curve fit window based on recorded data

Related Topics

Solids Library

(SSL Module Only) Users can create Solids Library entries which retain data for specific solids profiles. The Solids Library window (Figure 1) can be accessed from the Library menu or from Analysis Setup on the Solids Definition panel.

This allows you to view, add, edit and delete solids in the library. The physical properties are entered in the Change Solids Physical Properties window. Enter the name of the solids data and the temperature units.

Often the physical properties of the solids do not vary appreciably with temperature or the variation is unknown. In this case select Use Constant Properties (Figure 2). The temperature units are ignored but one must still be selected. The density is always required. The specific heat and thermal conductivity are optional and are only used when heat transfer is modeled.

If one or more properties vary with temperature, uncheck the Use Constant Properties box. The property values as a function of temperature can then be entered in the table on the Solids Property Data tab (Figure 2). If a property does not vary with temperature, simply enter the same value for each temperature listed in the table.
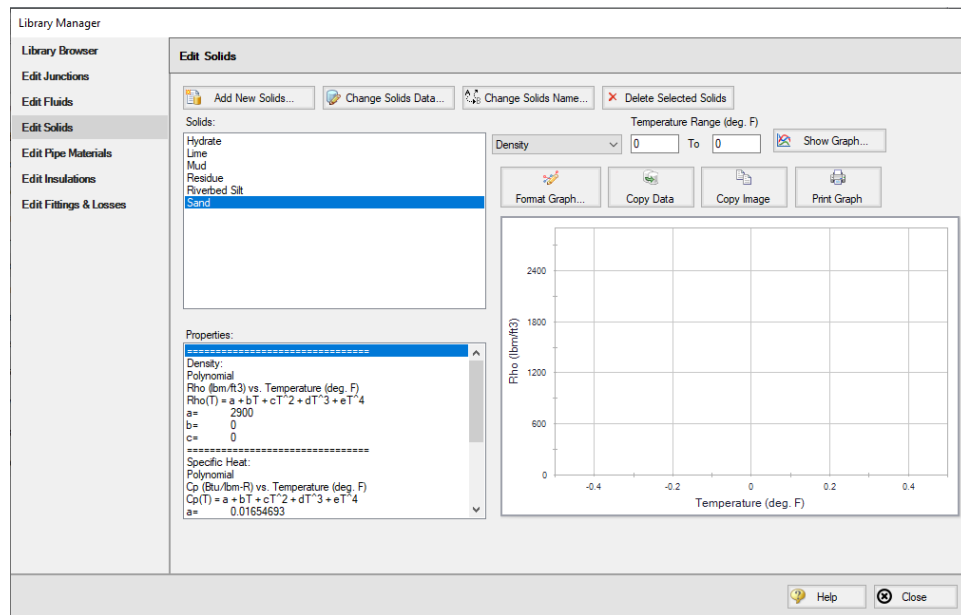
Figure 1: Solids Library showing example of user-entered solids data

Figure 2: Change Solids Physical Properties window showing tab to specify thermophysical properties of solids (available when "Use Constant Properties" is unchecked)

Other Solids Parameters

The Terminal Velocity Parameter and the M exponent are entered on the Other Solids Parameters tab (Figure 3). The particle size distribution is also entered on this tab.

Similar to direct entry on the Solids Definition panel, the user can allow AFT Fathom to estimate the M factor from recorded data or can input an M value directly. If M is calculated, values for both d_{50} and d_{85} must be entered.

The Terminal Velocity Parameter can be entered as either a Shape Factor (K) or the ratio of the particle's terminal velocity (V_t) to that of a sphere (V_{ts}). This ratio is designated by z or X_i

Typical shape factors are given in Wilson, et al, 2006, on page 46 where $K = 0.26$ for sand. To ignore this effect select $V_t / V_{ts} = 1$.

Particle Size Distribution

The particle size parameters d_{50} and d_{85} can be entered directly or statistically calculated based on particle size distribution data.

With the Use Particle Size Distribution option checked, you can enter the percent of the solids that remained on the mesh as observed from test data for each Tyler Mesh Size. The d_{50} and d_{85} parameters will automatically be calculated (Figure 3).

Figure 3: Solids Library showing the definition of particle size distribution

Related Topics

SSL Pump De-Rating

(SSL Module Only) When centrifugal pumps are pumping slurries the solids in the slurry can cause decreased pump performance. This is known as de-rating, as the process involves reducing the pump head curves provided by manufacturers which are based on clear water.

The SSL module offers three methods to estimate pump de-rating. The first is based on a standard published by the Hydraulic Institute in 2016. In AFT Fathom it is called the "ANSI/HI Standard 12.1-12.6-2016" method. The second is based on the Warman method. The third allows the user to define a CH, CQ and CE rather than having them calculated.

When the SSL module is used, a new tab appears on the Pump Properties window called Slurry De-Rating (Figure 1). Here the user can choose the de-rating method. The de-rating factors are

shown in the Output window Pump Summary as CH (for head correction) and CE (for efficiency correction). Figure 2 shows the Pump Summary with correction factors displayed. If you do not see them in your Output you can add them to the Pump Summary in the Output Control window opened from the Tools menu.

Figure 1: Slurry De-Rating tab on Pump Properties Window lets user estimate reduced pump performance due to solids

Figure 2: Pump de-rating factors CH and CE are shown in Output window Pump Summary

Theory of Pump De-Rating

The head reduction for slurry pumps is based on ANSI/HI Standard 12.1-12.6-2016, page 52, Figure 12.3.2.7 (not shown here). Correction factors for solids density, solids concentration and percent of fine particles are applied per the standard. This figure and methodology is the same as found in Wilson, et al. 2006 on page 243.

The Warman Method is in use in industry but is not widely available. The method used in AFT Fathom is based on a white paper by Walker, C. I., Undated, page 2, and uses a correction factor for impeller size. C_m is the solids mass concentration of the slurry in terms of percent mass.

$$CH = 1 - \left[0.000385 \, (S_s - S_f) \left(1 + \frac{4}{S_s} \right) \right] C_m \ln \left(\frac{d_{50}}{0.0227} \right)$$

The 4-Component Model option is a modified form of ANSI/HI Standard 12.1-12.6-2016 by GIW Industries Inc. intended for use with the 4-Component Slurry Model. The head reduction is given by the following equation.

$$r_h = \sqrt{(r_{h,f})^2 + (r_{h,p} + r_{h,h} + r_{h,s})^2}$$

Where r_h is the total percent head reduction factor (i.e. 1 percent de-rate is 1.0) and $r_{h,f}$, $r_{h,p}$, $r_{h,h}$, and $r_{h,s}$ are the contributions of the carrier fluid viscosity, pseudo-homogeneous fraction, heterogeneous fraction, and fully stratified fraction, respectively.

For viscosity below 20 centipoise, GIW guidelines state that $r_{h,f}$ can be assumed to be zero. Above 20 centipoise, ANSI/HI 9.6.7-2021 should be used to calculate this term.

Note: The Viscosity Correction term $r_{h,f}$ is not automatically applied when pump slurry de-rating is in use. To incorporate this term, Use Viscosity Correction must be enabled in the Optional tab. That option alone determines whether $r_{h,f}$ is calculated and the recommended 20 centipoise threshold is not used.

The equations for the other components take the following general form where D_2 is pump diameter in meters, D_{REF} is 1.0 m, S_S is the solid specific gravity, S_x is the mixture specific gravity for each component, C_v is the volumetric concentration of solids, and X_x is the fraction of solids associated with each component. $S1$ is equal to 0.5 times D_2/D_{REF} and $S2$ is 0.4.

$$r_{h,x} = N" \; 8(D_{REF}/D_2)^{S1} \, (d_{50x})^{S2} \, \frac{S_s-S_x}{1.65} \, \frac{X_x \, C_v}{0.15}$$

This results in the following three equations:

$$r_{h,p} = A" \; 8(D_{REF}/D_2)^{S1} \, (d_{50p})^{S2} \, \frac{S_s-S_f}{1.65} \, \frac{X_p \, C_v}{0.15}$$

$$r_{h,h} = B" \; 8(D_{REF}/D_2)^{S1} \, (d_{50h})^{S2} \, \frac{S_s-S_{fp}}{1.65} \, \frac{X_h \, C_v}{0.15}$$

$$r_{h,s} = C" \; 8(D_{REF}/D_2)^{S1} \, (d_{50s})^{S2} \, \frac{S_s-S_{fhp}}{1.65} \, \frac{X_s \, C_v}{0.15}$$

Where $A"$, $B"$, and $C"$ are empirical parameters and d_{50p}, d_{50h}, and d_{50s} are the average particle sizes in mm of the pseudo-homogenous, heterogeneous, and fully stratified fractions, respectively. CH is determined from r_h with the following expression.

$$CH = 1 - \frac{r_h}{100}$$

Related Topics

SSL in Output

([SSL Module Only](#))

Selecting Slurry Output Data

The specific slurry data to be shown in the Output window is specified in the [Output Control window](#). Select the Slurry button on the Display Parameters tab to see the available parameters (Figure 1).
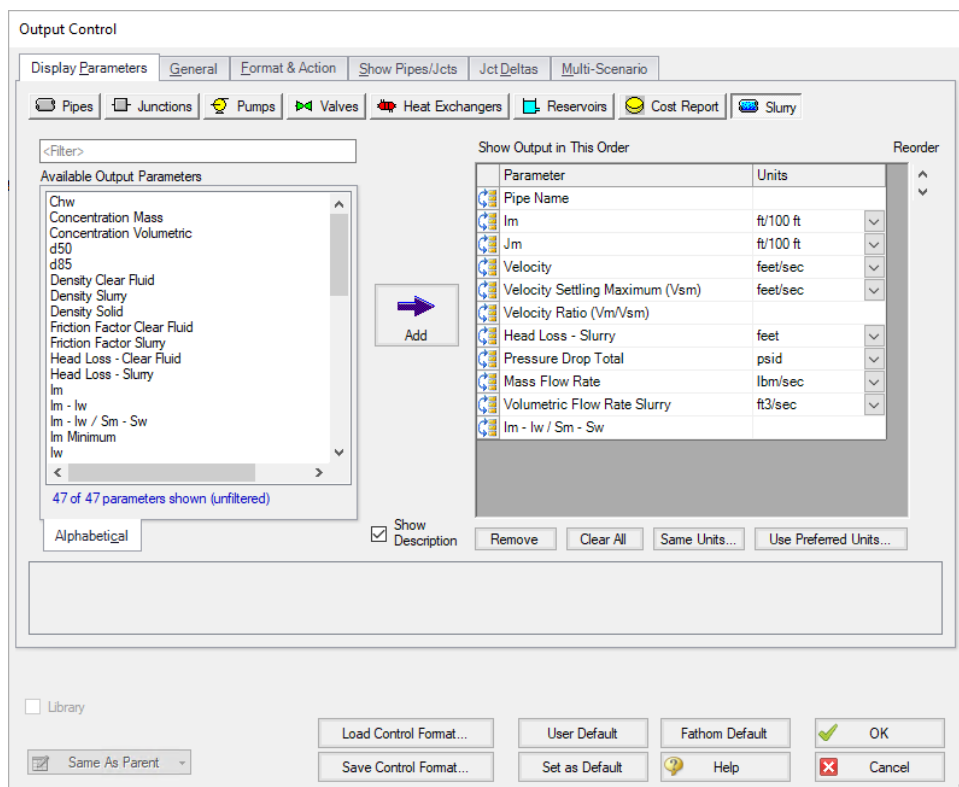
Figure 1: The Output Control window has a slurry button to allow choice of Output table parameters

Viewing Slurry Output

When an AFT Fathom model is run there is a special output table for the SSL module labeled Slurry that shows the results in the Pipes section (Figure 2).

Figure 2: Output window showing Slurry table in Pipes section

Related Topics

SSL in Graph Results

(SSL Module Only) Slurry system curves are common way of interpreting slurry behavior. Such system curves can be specified in the Graph Control tab of the Quick Access Panel.

When the SSL module is in use a new tab appears on the right called Slurry System Curve (Figure 1). Here, one can select the type of curve to generate.

Note: All pipes must be of the same diameter to use the Slurry System Curve.

Options to include pump curves (if a pump curve is used) and water curve cross plots are available.

The system curve is displayed in the Graph Results window (Figure 2).

Equations for Slurry System Curve

Fathom uses the equations below as are given in Wilson et. al (2006) on page 320 to calculate the velocity (V) and head gradient (j) for the slurry system curve.

$$V_x = \left[\left[0.22\, V_{50}{}^M\right] 2gD/f_w\right]^{1/(M+2)}$$

$$j_x = \left[f_w/2gD\right]^{M/(M+2)} \left[0.22\, V_{50}{}^M\right]^{2/(M+2)}$$

where V_{50} is the value of V_m at which 50% of solid are suspended in fluid, V_m is the mean velocity of the mixture, and f_w is the friction factor of water.

Since the head gradient depends on the velocity, all pipes must be the same diameter to produce a meaningful curve.
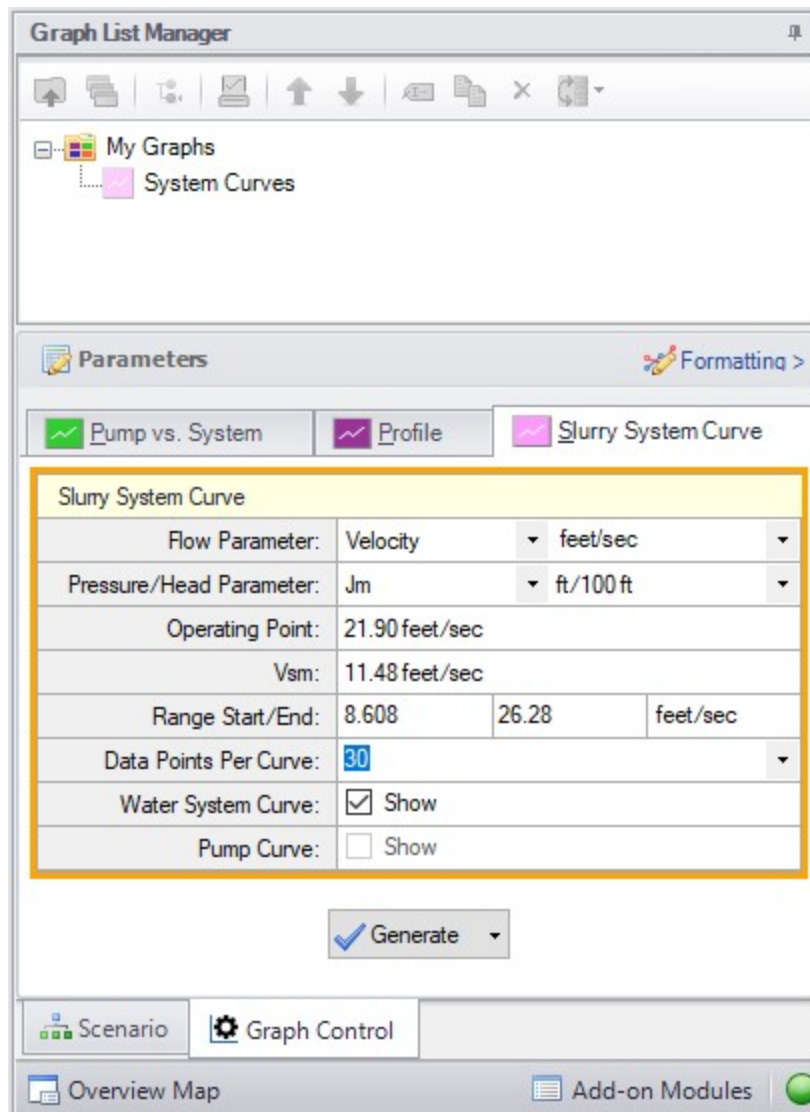
Figure 1: Slurry System Curve tab in the Quick Access Panel allows system curves to be created
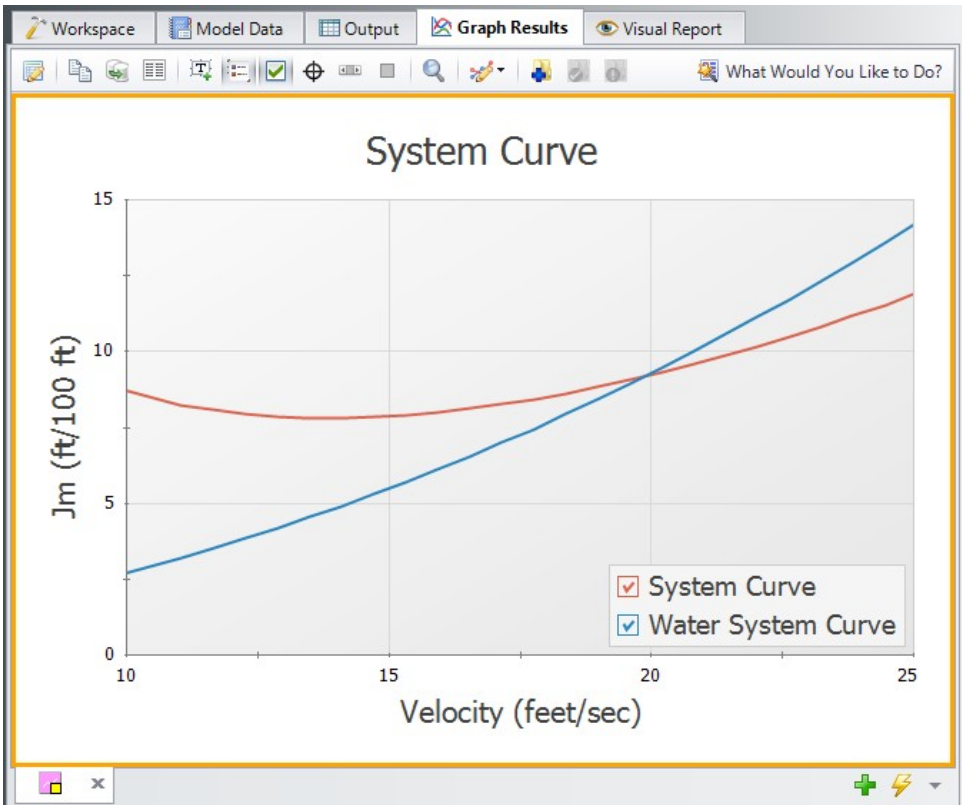
Figure 2: Graph Results window shows slurry system curve when using the SSL module

Related Topics

SSL Initial Guesses for Pipe Velocity

(SSL Module Only) AFT Fathom uses iterative methods to solve for the equations of pipe flow. This requires initial guesses for pipe flowrate. Such guesses are optional for the user and can be entered on the Pipe Properties Window Optional tab. If the user does not provide an initial guess then AFT Fathom makes an initial guess itself.

Typical settling slurry applications operate at a higher velocity than non-slurry flows and AFT Fathom will converge more reliably on a result if the initial guess at velocity reflects this higher velocity.

By default the initial guess flowrate for all pipes is based on a velocity of two times V_{SM}. This can be modified by the user in the Advanced Parameters panel or overridden directly by entering an initial guess for the pipes manually.

Related Topics

SSL Troubleshooting

([SSL Module Only](#)) Figure 1 shows a system curve for a settling slurry. One of the distinct aspects of settling slurry behavior is the minimum point of the system curve. This means that for a given j_m (or, in general, head difference) there are two potential flowrates, as shown in Figure 1 below. This has the potential for unstable numerical results.

It is recommended that the system operate with a velocity ratio (V_m/V_{sm}) somewhat greater than 1.0 in all pipes so that solids will not settle out of the carrier fluid and form a stationary bed (which is not modeled using the SSL module) or plugging the pipe.

Often it is common practice to operate at or above the minimum i_m/j_m point to achieve a desirable velocity ratio, and to avoid higher head losses that can occur to the left of the i_m/j_m minimum. For example, in the below figure the minimum i_m occurs at about 5.25 m/s, so a mean velocity of 5.25 m/s or greater would be favorable. Model convergence may be difficult the closer the velocity ratio is to 1.0. Note that head loss curves like the one shown below will vary at different values of pipe diameter and solids concentration.
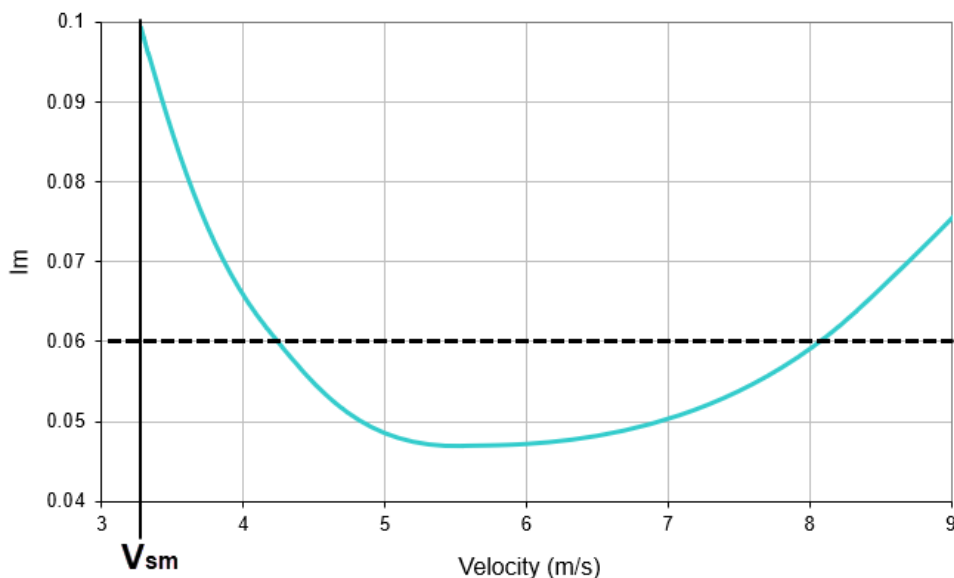
Figure 1: i_m vs mean velocity for a slurry system at a specific slurry concentration and pipe diameter.

Related Topics

Tips When Using GSC with SSL Module

The GSC module in AFT Fathom can be used in conjunction with the SSL Module. Similar considerations apply as are discussed in SSL troubleshooting. If one is looking for a goals but the system is not a fixed flow type system then the goal seeker can become unstable.

The following tips are the same as given for general GSC troubleshooting but can be even more important when modeling slurries:

- Use bounds on design variables to keep the search within certain limits
- Vary the design variable starting point. If the simulation is started far away from the actual value then GSC may not be able to find the solution
- Manually modify some of the Numerical Control parameters in the Goal Seek & Control Group. Specifically, increasing the Relative Finite Difference Step Size can help the goal seeker obtain better gradients and provide more reliability

Related Topics

SSL Pipe Slope

(SSL Module Only) Pipes with different slopes will have different pressure drops for slurry flow when using the Wilson Addie Clift or 4-Component settling slurry models because of the way solids are carried horizontally, vertically or in between. AFT Fathom determines the pipe slope based on the pipe length and connected junction elevations.

Pipe slope corrections are not applicable to the Wasp-Durand Heterogeneous Slurry Model.

The pipe angle can be added to the Slurry output in the Output Control table.

Related Topics

XTS - Extended Time Simulation

What is the XTS module?

An optional add-on module to AFT Fathom 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 Fathom 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 pumps 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 draining and filling can be simulated. Tanks can be open or pressurized, with the gas pressure automatically calculated as the liquid level changes.

The XTS module is a powerful tool which extends AFT Fathom'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 Fathom 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 Fathom first loads. After AFT Fathom 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

- **Reservoirs -**
  - Reservoirs can be modeled as finite tanks with user specified tank geometry.
  - Finite reservoirs with known geometry allows AFT Fathom to model liquid level changes over time to simulate tank filling and draining and its impact on the flow in the system.
  - Finite reservoirs can be open or enclosed. If enclosed, then surface gas pressure changes are automatically updated.
- **Pump transients -**
  - Pumps can be started or stopped based on time or event transients
  - Pump and Pump as Turbine speed can be changed with time
  - Pump flow or head can be changes with time
  - Controlled pump setpoint can be changed with time
  - Pump as Turbine Load Rejection
- **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 varied based on time or event transients
  - Assigned Pressure junctions can have pressure varied based on time or event transients
  - Reservoir junctions can have liquid level or surface pressure varied based on time or event transients
  - Three-way valve junctions can have valve position varied based on time or event transients

Related Topics

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 steady models with XTS or transient models.

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.
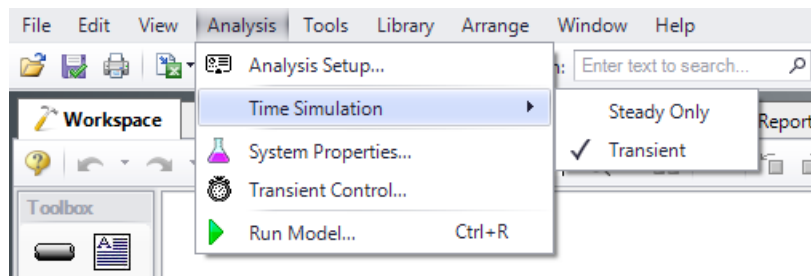
Figure 1: Time Simulation option in the Analysis menu

Related Topics

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

Junction Type	Type of Transient
Assigned Pressure	Pressure
Branch	Flowrate source or sink
Control Valve	Setpoint
Pump	Speed, flowrate, head rise, control set-point
Reservoir	Liquid level, surface pressure
Spray Discharge	Flow area
Three-Way Valve	Position
Valve	K, C_v, or open percentage

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 Cv 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 Cv 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 case. 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. Because the transient data is constant 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 filled up and thus reached a certain level. 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 level, the valve never closes.

Related Topics

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.
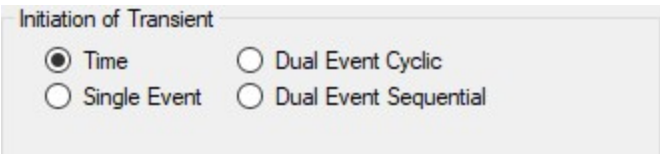
Figure 1: Type of transient initiation

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.
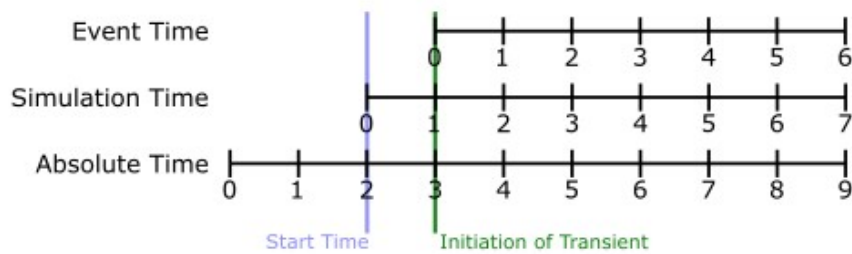
Figure 2: Time basis

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 remain open until (and if) the pressure at the inlet of P101 exceeds 100 psia. 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 **	Kv	Reservoir Liquid Height
Check Valve State	Mass Flow Rate in Pipe	Reservoir Liquid Volume
Control Valve State	Pressure Stag. Diff. at Junction	Reservoir Surface Elevation
Cv	Pressure Stag. Diff. Between Pipes	Reservoir Surface Pressure
EGL at Pipe	Pressure Stagnation at Pipe	Spray Discharge Outflow
EGL Diff. at Junction	Pressure Static at Pipe	Time Absolute
EGL Diff, Between Pipes	Pressure Static Diff. at Junction	Time Difference
HGL at Pipe	Pressure Static Diff. Between Pipes	Valve Open Percent
HGL Diff. at Junction	Pump Speed	Velocity in Pipe
HGL Diff. Between Pipes	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 Fathom 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 Topics

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.
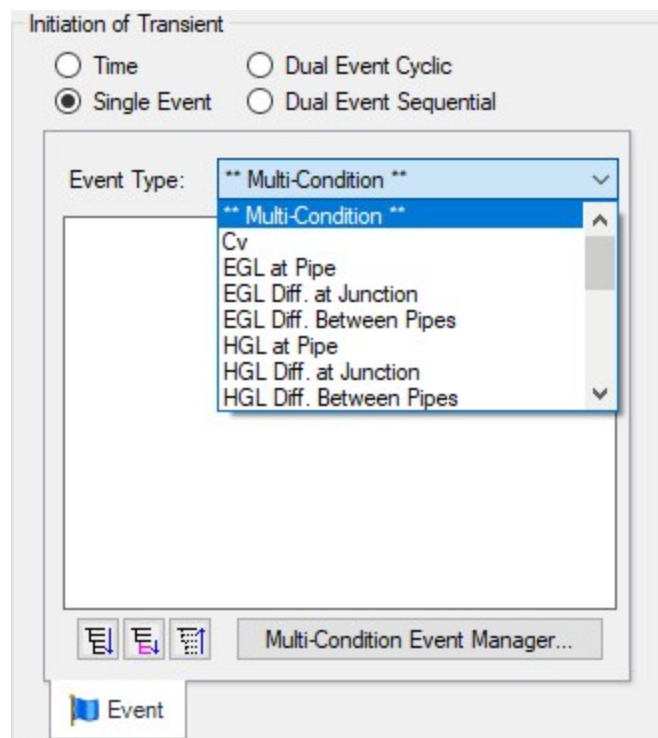
Figure 1: Selecting Multi-Condition Event Type

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.
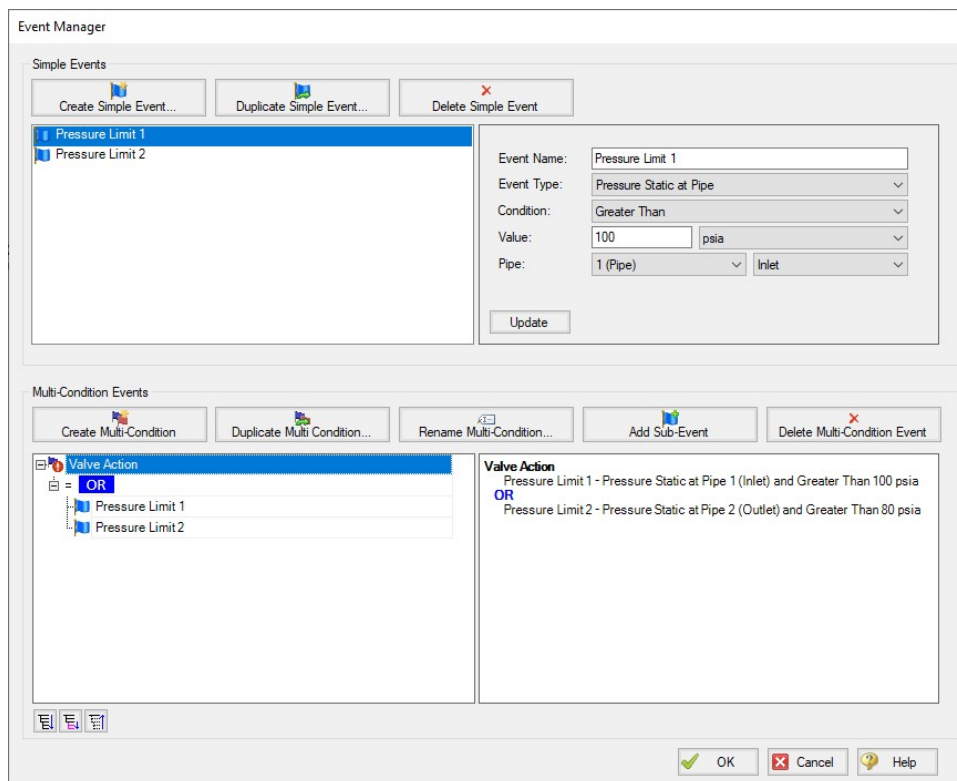
Figure 2: The Event Manager

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:
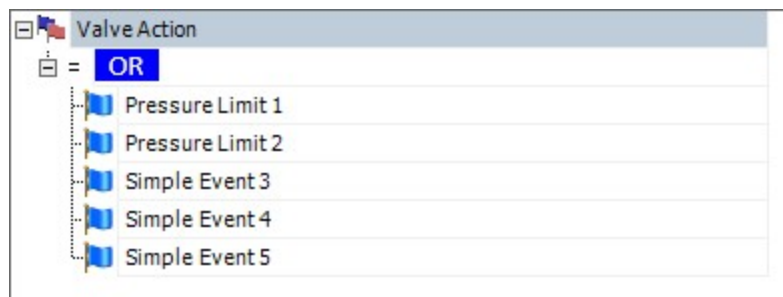
Figure 4: Multiple Events

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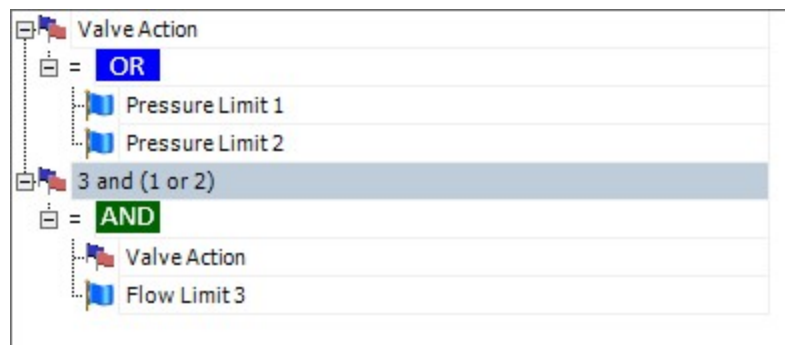
Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))

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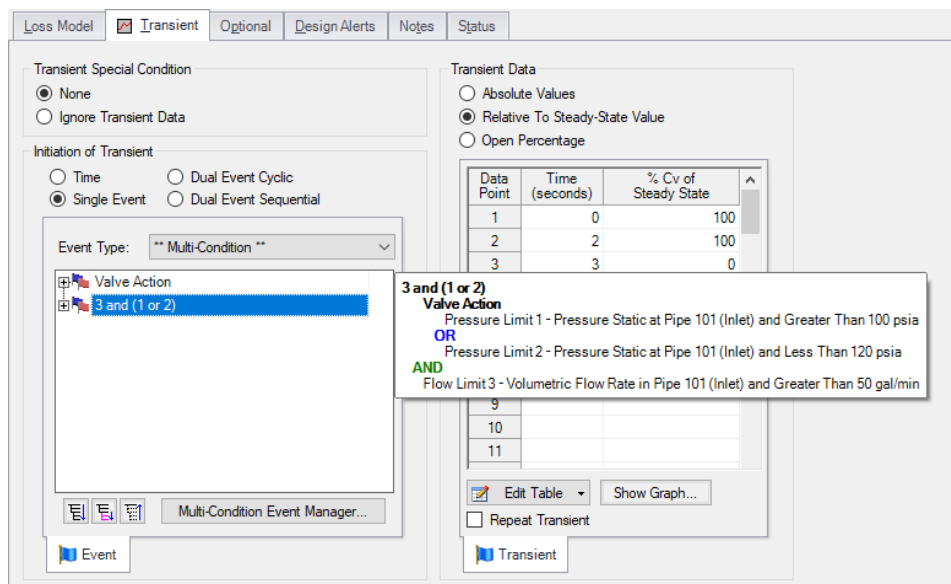
Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event.

Related Topics

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.

Pump Special Conditions

Pump Special Conditions are slightly more complicated than for valves. In fact, pump junctions have two types of Special Conditions. The first is to turn the pump off and have no flow through it. This is called "**Pump Off No Flow**". The second type of Special Condition turns the pump off but allows flow to go through the pump. This is called "**Pump 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 pumps and the second for centrifugal pumps.

Here's why. When a positive displacement pump 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 pump. 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 pump is turned off and has a pressure difference across it, in the absence of other valving which prevents flow the pump 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 pump. One would use the second Special Condition, and, in conjunction with a pump 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 have no transient initiation at that 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.

Related Topics

Finite and Infinite Reservoirs

Without the XTS Module, reservoirs are infinite. With the XTS module, reservoirs can be infinite or finite.

- **Infinite reservoirs** - infinite reservoirs refer to a massive body of fluid whose surface level does not change appreciably as a result of liquid inflow or outflow during the time frame of the simulation. An example is a large lake or the ocean.

- **Finite reservoirs** - (XTS Module Only) finite reservoirs (also called finite tanks) refer to a body of fluid which is small enough that its surface level changes significantly during the time frame of the simulation as a result of liquid inflow or outflow. An example is a tank which drains as the simulation progresses.

  - Finite reservoirs can be **open** or **closed**. Finite open reservoirs are open to the atmosphere or some other fixed surface pressure. Typically open reservoirs have a constant surface pressure with time, although users have the latitude to pre-specify time varying surface pressure.

  - Finite closed reservoirs are closed tanks in which, as the liquid level changes, the gas above the liquid expands or contracts and hence its pressure changes according to the gas thermodynamic laws. For closed reservoirs the user specifies the ini-

tial liquid level and gas pressure, but during the simulation all gas pressure and liquid levels are calculated by AFT Fathom.

Related Topics

Entering Reservoir Volume Data

(XTS Module Only) When running AFT Fathom in Steady Only mode (or without the XTS module), each reservoir junction is considered an infinite reservoir. This means that the liquid heights and surface pressures entered are constant. When running transients, reservoirs can be either *infinite* or *finite*. Similar to Steady Only mode, an infinite reservoir maintains its level during the transient. The differences are discussed below and summarized in Table 1.

Table 1: Various possible inputs with infinite and finite Reservoir junctions

Tank Model		Tank Volume Needed	Surface Level/Pressure Required?	Surface Level Over Time	Surface Pressure Over Time
Infinite		No	Both	Constant or User Specified	Constant or User Specified
Finite	Open	Yes	One or Both	Calculated	Constant or User Specified
	Closed	Yes	One or Both	Calculated	Calculated from PV^n

Infinite Reservoirs

Infinite reservoirs refer to a massive body of fluid whose surface level does not change appreciably as a result of liquid inflow or outflow during the time frame of the simulation. An example is a large lake or the ocean. Figure 1 shows a Reservoir junction modeled as infinite.

Finite Reservoirs

Finite reservoirs (also called finite tanks) refer to a body of fluid which is small enough that its surface level changes significantly during the time frame of the simulation as a result of liquid inflow or outflow. An example is a tank which drains as the simulation progresses. Tanks can have

constant cross-section (e.g., a pure vertical cylinder) or tank cross-section can change with height. When the cross-section changes, the tank volume at each height data point must be specified.

Finite tanks can further be broken down into open or closed tanks.

- **Finite open tanks -** Finite open tanks are open to the atmosphere or some other fixed surface pressure. Typically open tanks have a constant surface pressure with time, although users have the latitude to pre-specify time varying surface pressure. Figure 2 shows a Reservoir junction modeled as a finite open tank.

- **Finite closed tanks -** Finite closed tanks are tanks in which, as the liquid level changes, the gas above the liquid expands or contracts and hence its pressure changes according to the gas thermodynamic laws. For closed tanks the user specifies the initial liquid level and gas pressure, but during the simulation all gas pressure and liquid levels are calculated by AFT Fathom. Figure 3 shows a Reservoir junction modeled as a finite closed tank.

Known Parameters Initially

In some cases the liquid level may not be known initially when modeling finite tanks. An example would be a vertical standpipe. At time zero the net flow at the standpipe would be balanced, and the liquid surface would find its own level (this would equal the local hydraulic gradeline). Hence the initial liquid level would be calculated based on equilibrium.

However, under transient conditions the liquid level will change, and the liquid volume in the standpipe can significantly affect pipeline behavior.

This is the purpose of the features in the Known Parameters Initially area (see Figure 2). By default, both the initial liquid level and initial surface pressure are assumed known. If you want to model one of them as unknown initially, select the desired parameter from the drop-down. If the surface pressure is selected as unknown initially, it functions similarly to the unknown liquid level except initial equilibrium surface pressure would be calculated.

This feature only affects the calculation at the initial "zero" time step. For all other time steps, the junction behaves the same as a junction in which the liquid level and surface pressure are known initially.

Figure 1: Reservoir junction modeled as infinite

Figure 2: Reservoir junction modeled as finite open

Figure 3: Reservoir junction modeled as finite closed

Related Topics

Entering Reservoir Transient Data

([XTS Module Only](#))

Infinite Reservoirs

For an infinite reservoir, the liquid level or surface pressure can be varied over time according to a user specified transient profile (see Table 1 in the [Entering Reservoir Volume Data](#) topic). This is input on the Transient tab (see Figure 1).

Figure 1: Infinite reservoirs can have user specified time varying liquid level or surface pressure

Finite Open Tanks

For a finite open tank, the surface pressure can be varied over time according to a user specified transient profile. This is input on the Transient tab (Figure 2).

Finite Closed Tanks

For a finite closed tank with a maximum allowable pressure (this is a relief pressure), the relief pressure can be varied over time according to a user specified transient profile. This is input on the Transient tab.
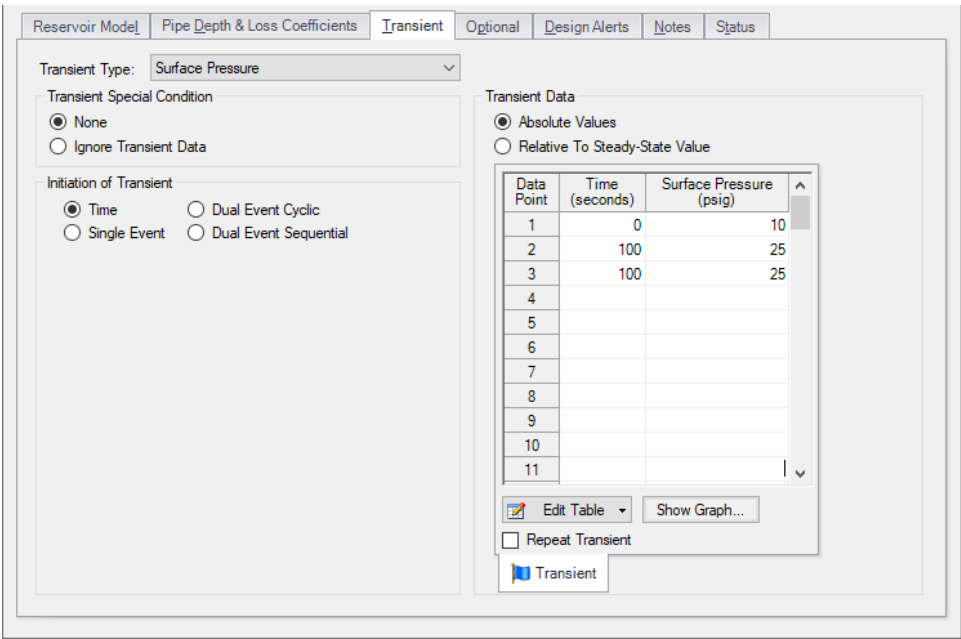
Figure 2: Finite tanks can have user specified time varying surface pressure

Related Topics

Finite Tank Overflow & Draining

([XTS Module Only](#))

Finite Tank Overflow

As the transient progresses finite tanks can fill up. If the tank is open, then it is possible for the liquid to reach the top. How does AFT Fathom handle this?

When a finite tank liquid level reaches the top, it is assumed that the liquid spills over the top. Hence the liquid height is maintained at the top of the tank. Note that this causes a loss of mass from the system model.

If the tank is closed, the gas pressure will increase as the tank fills. Thus the tank cannot overflow. If the tank has a maximum pressure (which represents a relief line) then it is possible for the liquid mass to be lost as it would ostensibly flow out the relief line. The liquid level in the tank would then be maintained at the top of the tank.

Finite Tank Drained

When a tank drains to the bottom, it cannot supply any more liquid to the connected pipes. The system behavior after a tank has drained cannot be accurately modeled by AFT Fathom. In a real system, and in the absence of a valve to stop the flow, a drained tank which continues to flow would result in the connected pipes themselves draining and a gas/liquid interface moving down the pipes. AFT Fathom assumes all pipes are liquid full, and cannot model draining pipes.

Since it is not desirable in most applications to have the pipes drain, this limitation is not a significant issue. The user can, for example, use a valve in the pipe which closes when the tank drains. This would use an event transient.

Since AFT Fathom cannot model a tank after it has drained, what does it do? Rather than halt the transient run, which would not be very helpful to the user, AFT Fathom assumes the connecting pipe "turns off" when it no longer has liquid to supply it. To implement this, AFT Fathom sets the pipe's Special Condition so that it turns the pipe off.

Finite Tank Pipes Uncovered

AFT Fathom allows multiple pipes to connect to a reservoir, all at different elevations. As a tank drains, pipes may be uncovered. AFT Fathom handles this in the same manner as described above for completely drained tank.

If a pipe is uncovered, then there are two possibilities:

- **Liquid flows from the pipe and into the tank** – AFT Fathom can model this case. The liquid free falls to the liquid surface, and appropriate boundary conditions are used to solve the system.

- **Liquid tries to flow from the tank and into the pipe** – AFT Fathom cannot model this case. Since the pipe is not covered by liquid, then the tank cannot supply liquid to the pipe and in reality the pipe itself would drain. AFT Fathom does the same thing as described above for a completely drained tank – it sets the pipe flow to zero.

Important: When a tank drains completely or sufficiently to uncover a pipe, AFT Fathom cannot adequately model the system behavior after this point unless a valve is used to stop the flow into the pipe. AFT Fathom will warn the user when this happens.

Related Topics

Finite Reservoir Maximum and Minimum Pressures

(XTS Module Only) A finite closed tank allows the user to specify maximum and minimum pressure. This is the pressure of the gas in the tank. The purpose of maximum and minimum

pressure is to simulate relief and vent systems in the tank which represent over and under-pressurization.

Related Topics

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.
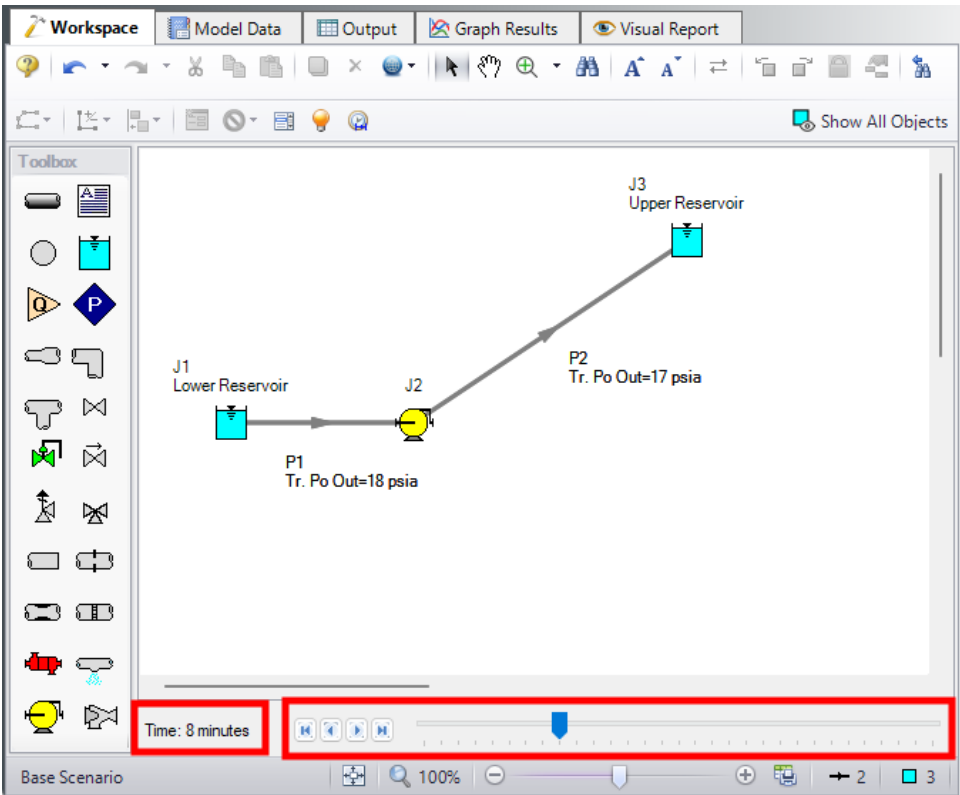
Figure 1: The Workspace has additional time adjustment controls when the XTS Module is in Transient mode

Related Topics

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.
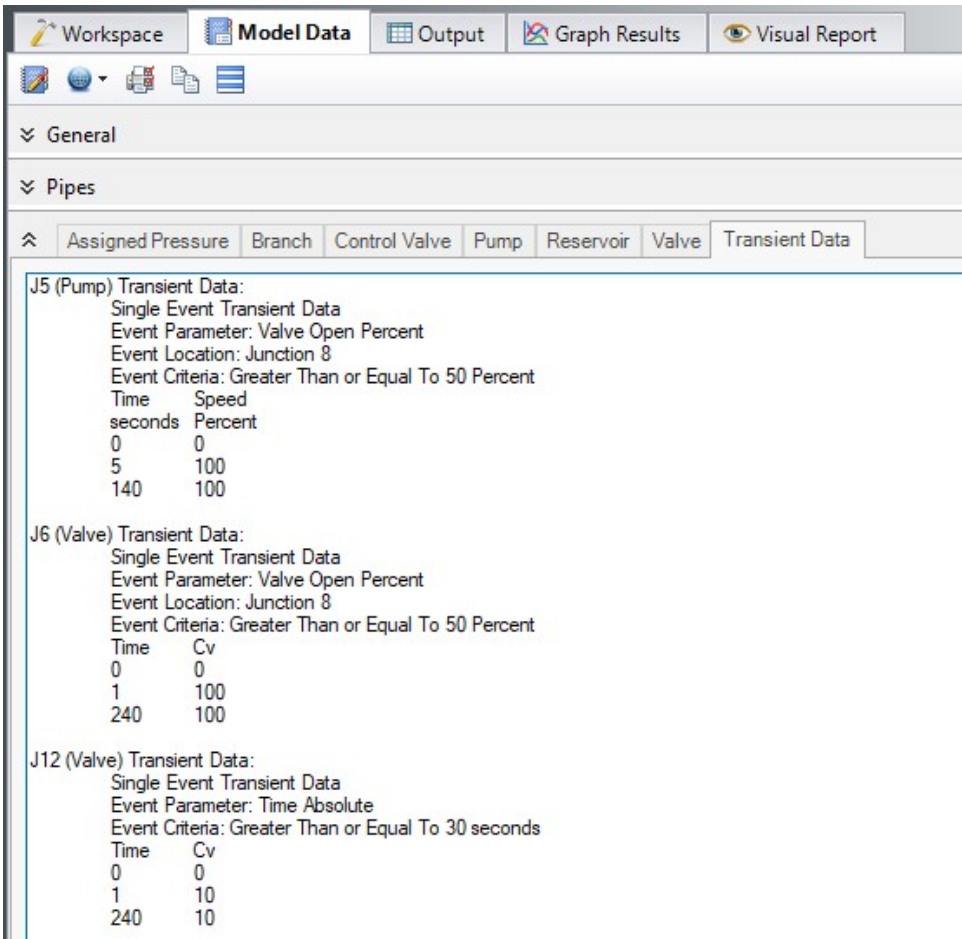
Figure 1: Transient input data is displayed in the Junction section of the Model Data window

Related Topics

XTS in Output

(XTS Module Only) Transient simulations create more output data than steady-state runs. This 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 minutes into the simulation. This can be seen at the bottom

left of Figure 1. The forward/reverse buttons and the slider at the bottom allow changing 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 reservoir, pipe and pipetabs 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 middle 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, pipe 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, - minutes 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 Figure 1 (in the General Section, see Pump, Valve and Reservoir Summary) have a parallel 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 in Graph Results

(XTS Module Only) Transient data can graphed in the Graph Results window in two general ways: Profile graphs and transient graphs.

Pump vs. System curves can still be generated, but only when the Time Simulation is set to Steady Only. See the bottom of this topic.

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.
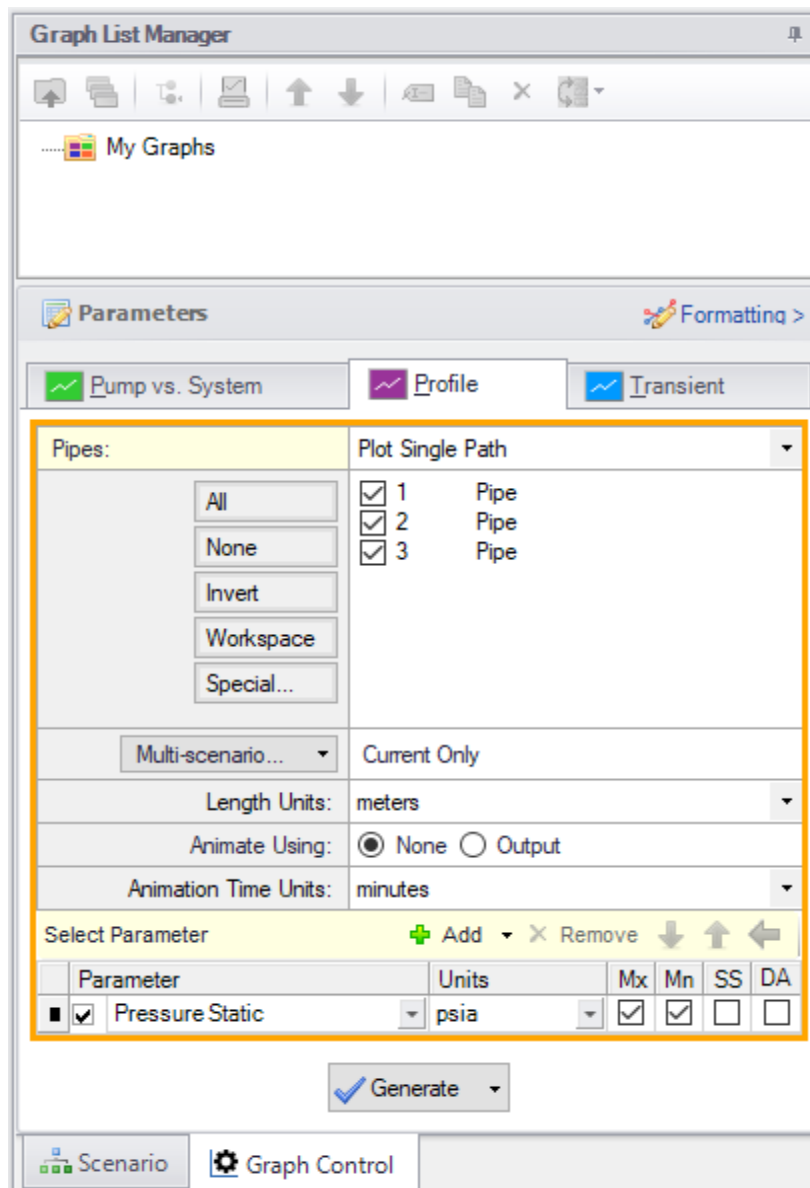
Figure 1: The Graphing panel showing a profile plot for a pipe sequence for maximum and minimum static pressure is to be generated
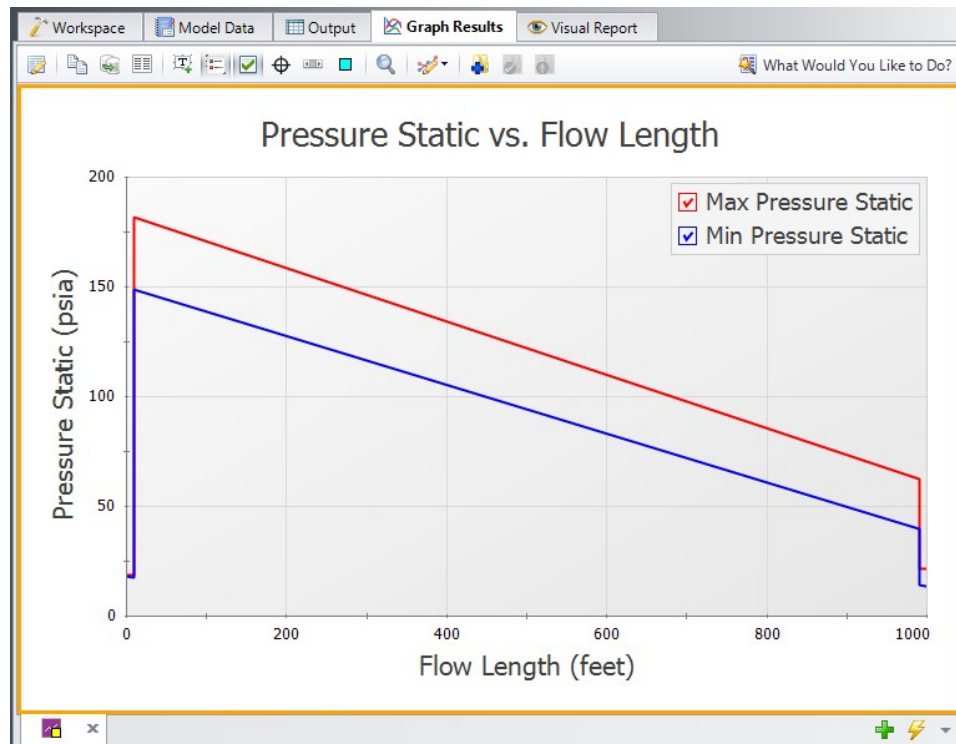
Figure 2: Transient max/min profile plot for pressure as specified in Figure 1

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 (rather than distance as in profile graphs) as shown in Figure 2.
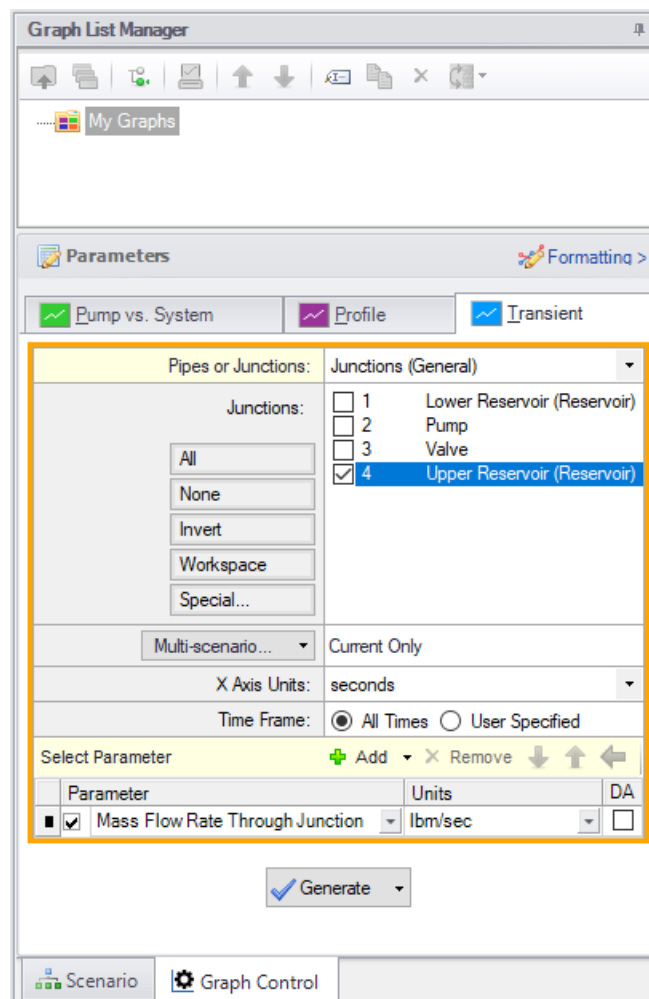
Figure 3: Setting up a Transient graph
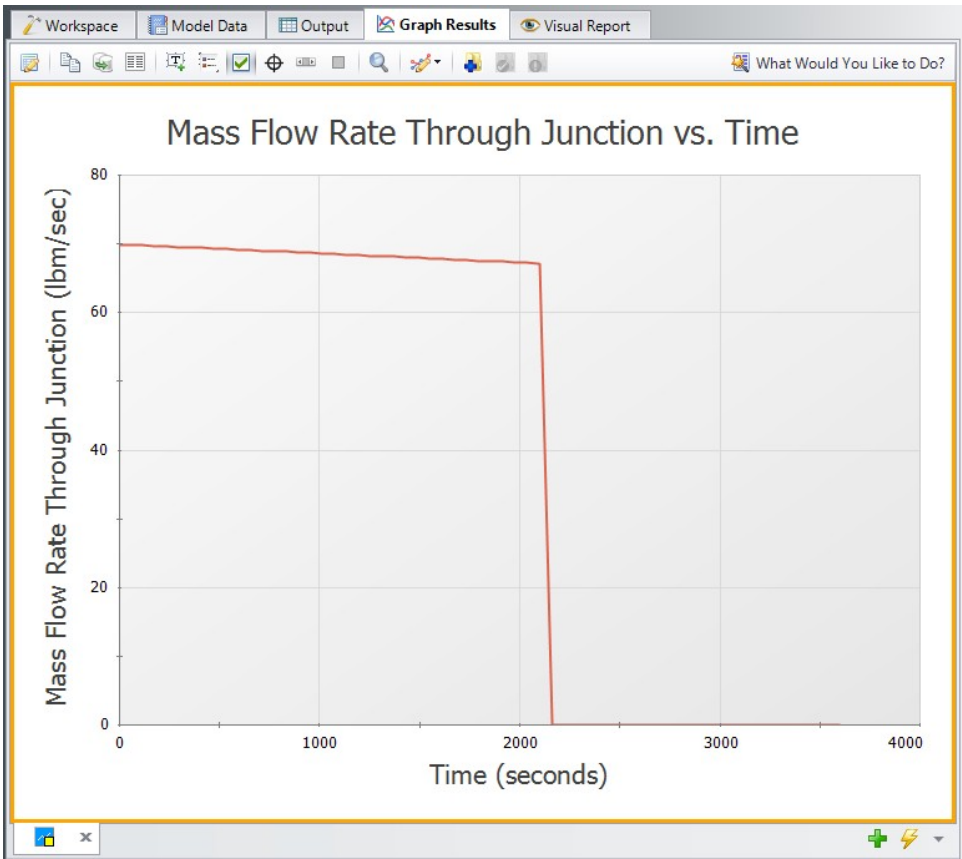
Figure 4: Transient graphs show how parameters vary with time

Pump vs. System Curves

Pump vs. System Curve plots can only be created for a steady analysis. To perform a steady analysis change the Time Simulation to "Steady Only" from the Analysis menu.

To create a pump vs. system curve for a time step other than the initial time step, create a child scenario, change the Time Simulation to "Steady Only," and adjust the model file inputs to reflect the time step you want to create a pump vs. system curve for.

Related Topics

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 or use the time slider controls (to the right of the Stop button) 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).
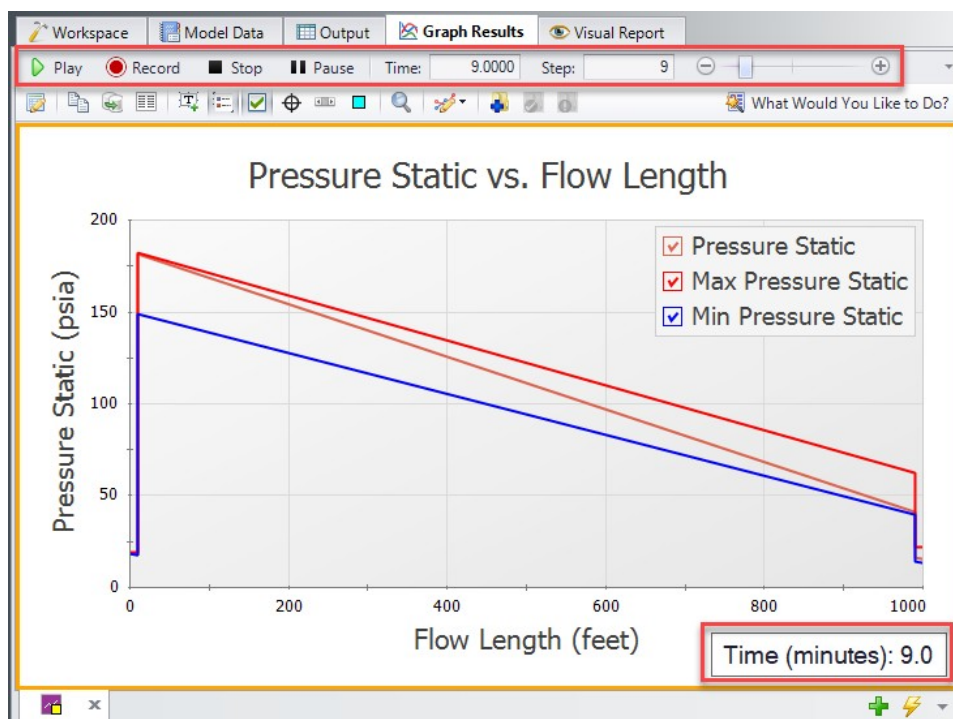
Figure 1: The animation options display additional controls
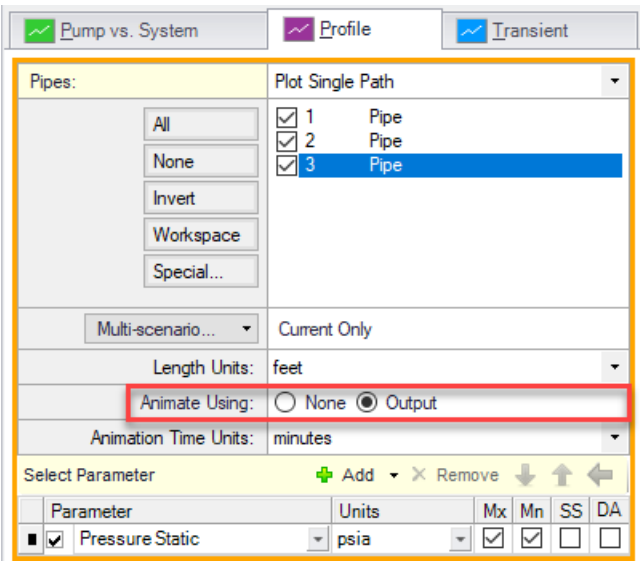
Figure 2: Enabling Animation in Graph Control

Related Topics

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).
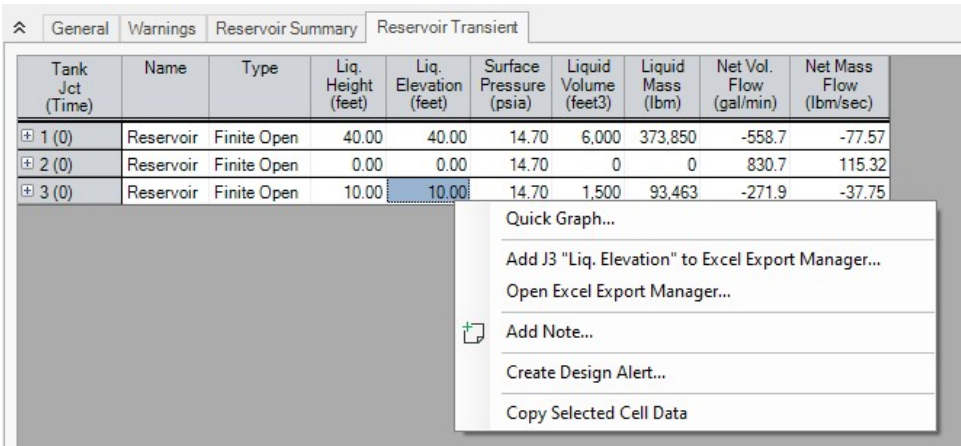
Figure 1: Quick Graph on the Output window is selected by clicking the with the right mouse button in a column of transient data
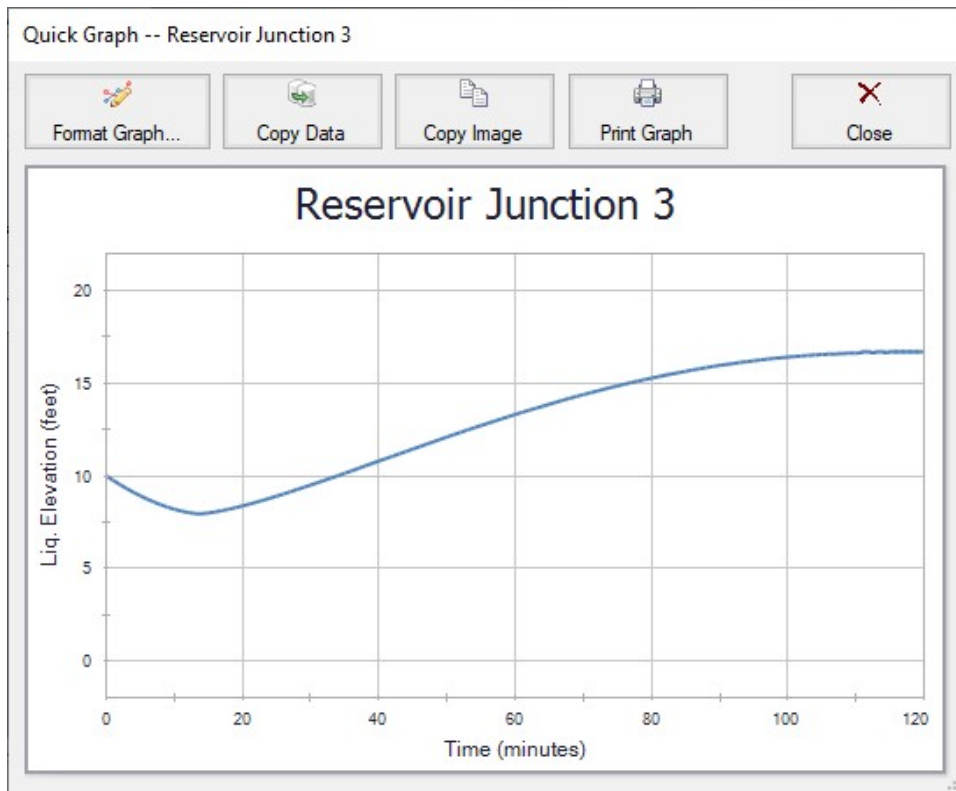
Figure 2: Quick Graph window is opened from Output window as shown in Figure 1

Related Topics

ANS - Automated Network Sizing

What is the Automated Network Sizing (ANS) module?

Sizing a single pipe or a single pump 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 <u>not</u> 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 Fathom 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 Fathom 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

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 Fathom 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 Fathom already automates the "*Specify Model -> Analyze*" portion of the process. AFT Fathom 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

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 and fluid levels 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.
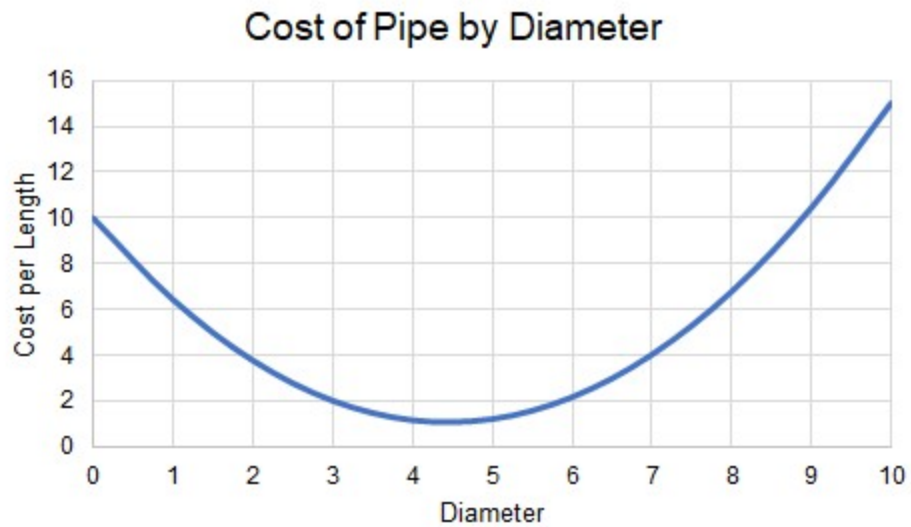
Figure 1: Cost for different diameters of specialized piping

Clearly, there is a minimum cost between diameters "4" and "5."
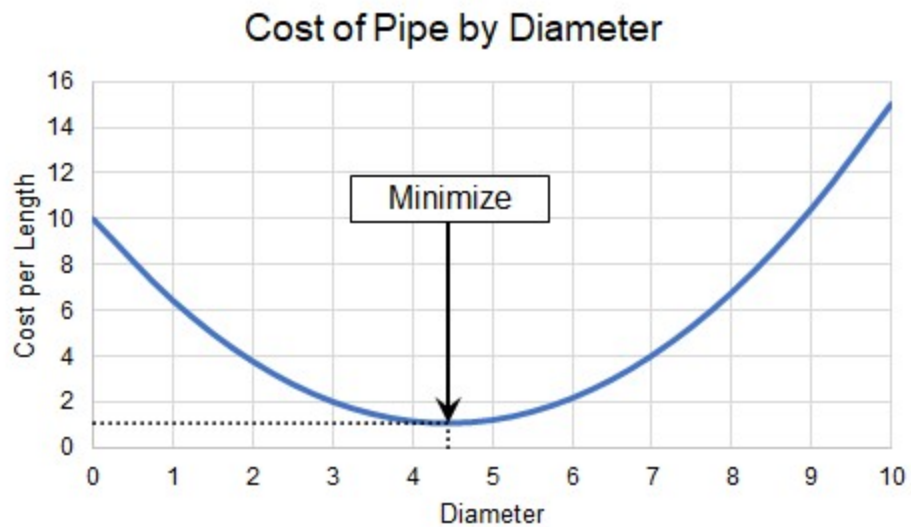
Figure 2: Locating the Diameter with minimum cost

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

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 from the Objective topic, 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 Rate vs Diameter
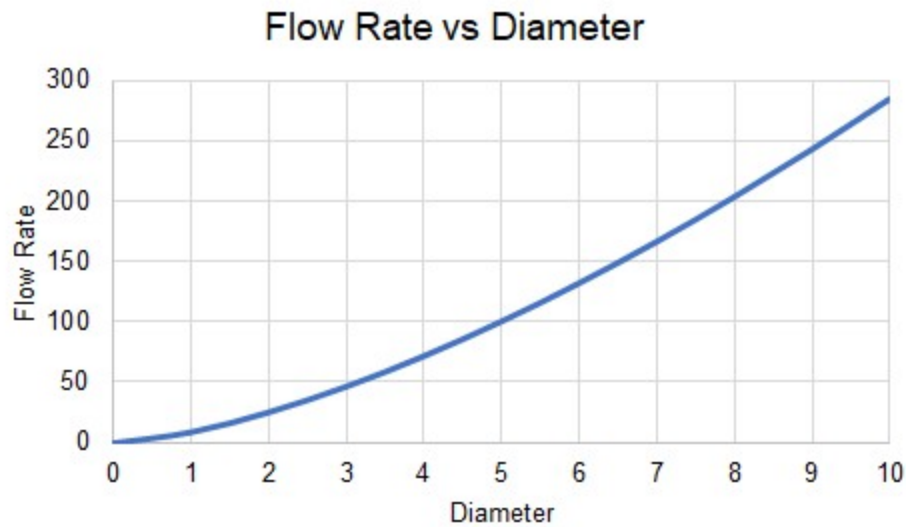
Figure 1: Flow Rates for different diameters, showing the minimum required for our pipe

Our pipe has a Design Requirement that flow rate is at least "150", which indicates a minimum diameter between "6" and "7" (Figure 2).

Flow Rate vs Diameter
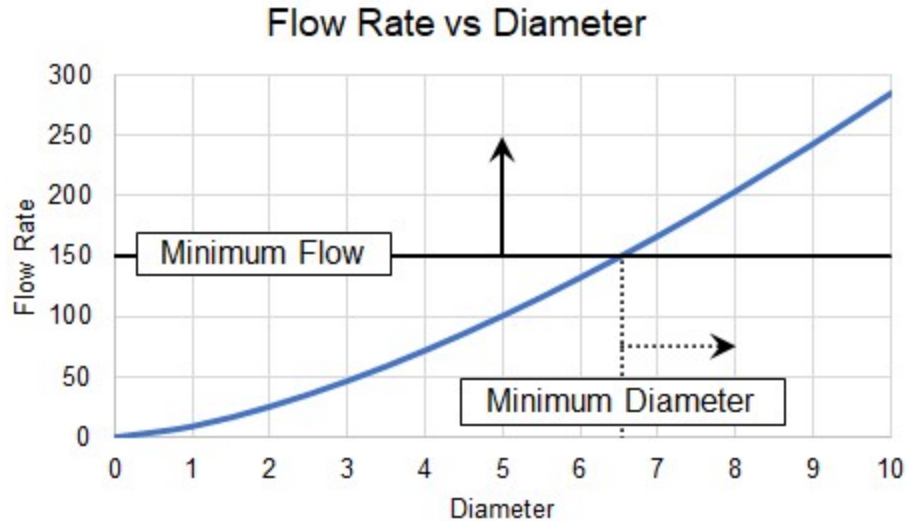
Figure 2: Minimum diameter indirectly imposed by flow rate requirement

Adding Design Requirements may influence the sizing results as this requirement makes certain solutions Infeasible.

Related Topics

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 (Figure 1). The minimum flow rate Design Requirement requires a minimum diameter.

Cost of Pipe by Diameter
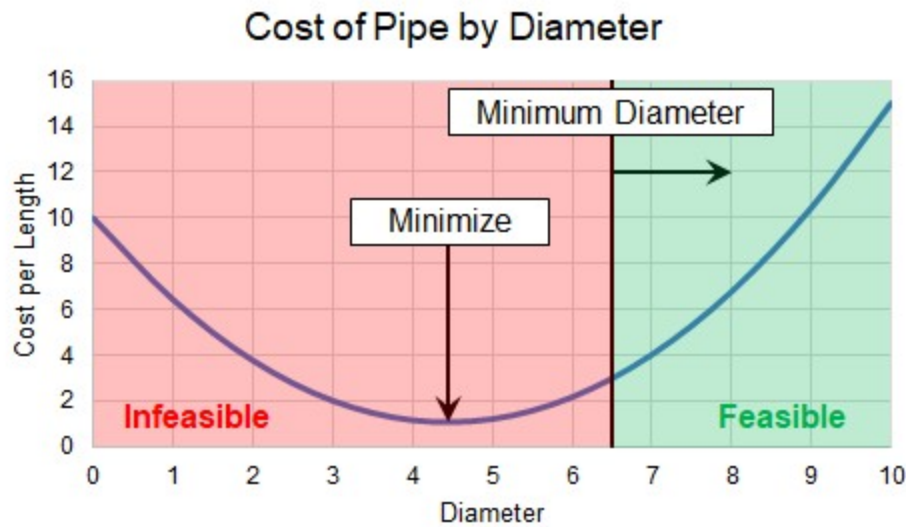
Figure 1: Certain diameters are infeasible when a Design Requirement is added

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).

Cost of Pipe by Diameter
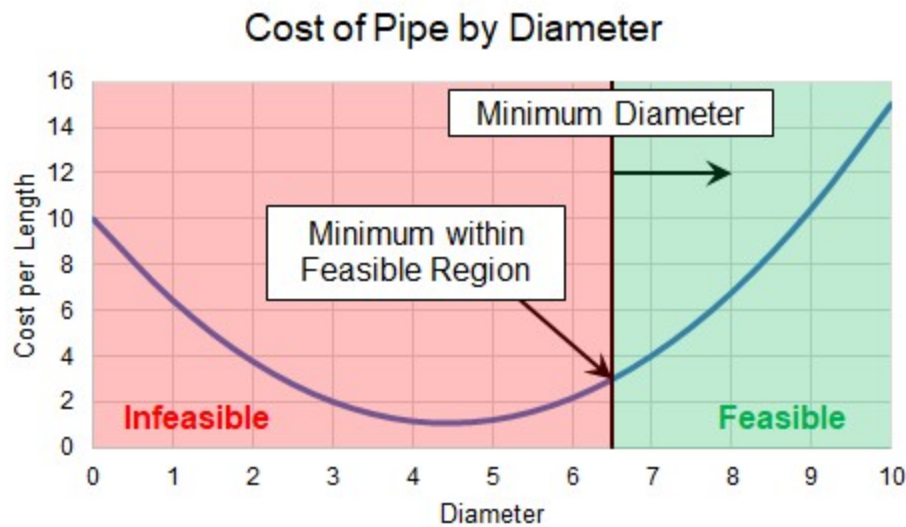
Figure 2: Minimum Feasible Cost

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

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).
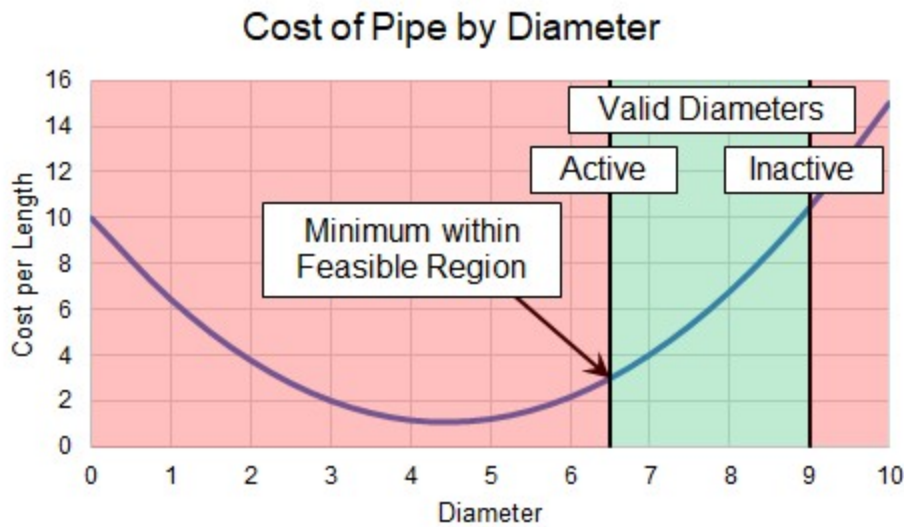
Figure 1: Multiple Design Requirements

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

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 at 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).
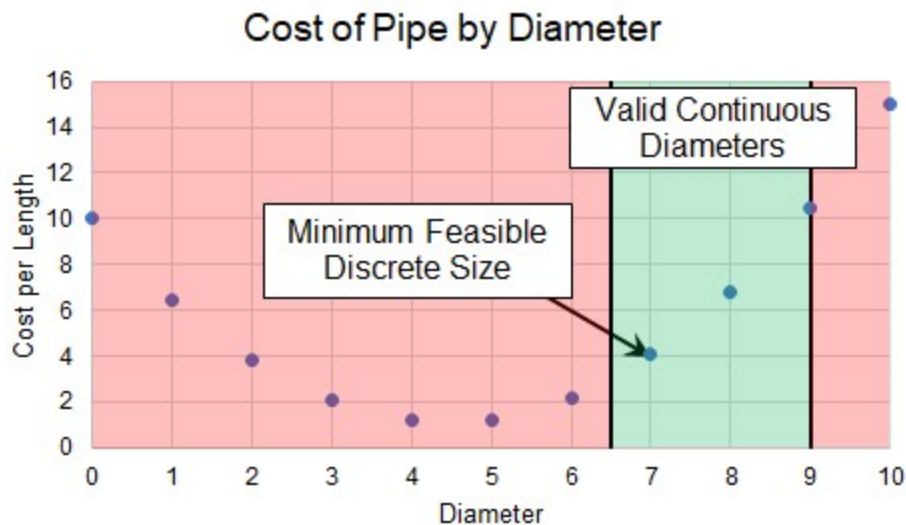
Figure 1: Minimum Feasible Discrete Size

The minimum flow Design Requirement is still be 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

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

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

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 (Figure 1). 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 pumps.

Note: Costs must be defined in Cost Libraries, which are available as part of the base Application.
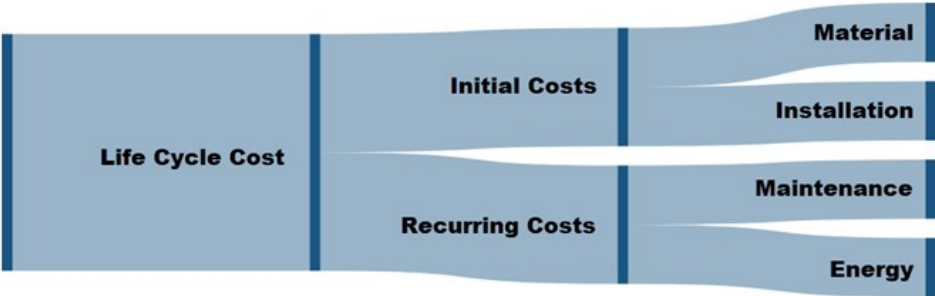
Figure 1: Components of Life Cycle Cost

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.
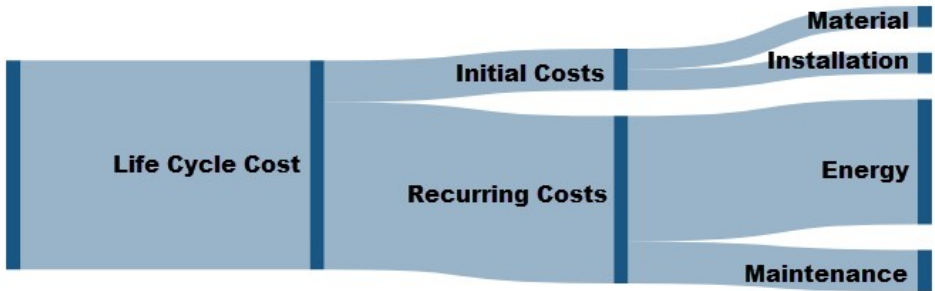
Figure 2: Potential distribution of costs for a long System Life

Related Topics

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 Window

The Sizing Window is a sixth Primary Window that appears when the ANS module is activated and enabled (Figure 1).
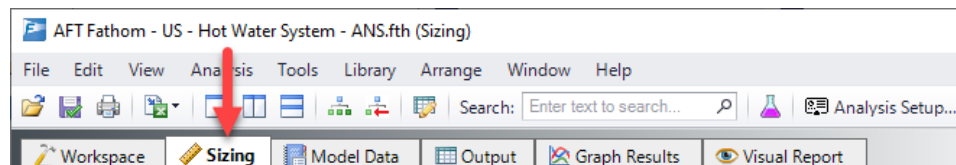
Figure 1: The Sizing Window

The Sizing window contains all of the information required to define a sizing analysis. Other areas of AFT Fathom 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).

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 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).
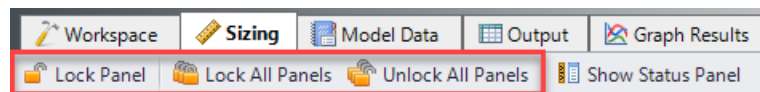
Figure 3: Locking panels

Related Topics

Sizing Objective Panel

Sizing Options

Figure 1: The three sizing ANS sizing methodologies are shown

- **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 on the Fluid panel of Analysis Setup.
- **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 were monetary cost of the fluid system is not the primary concern.

Other Options
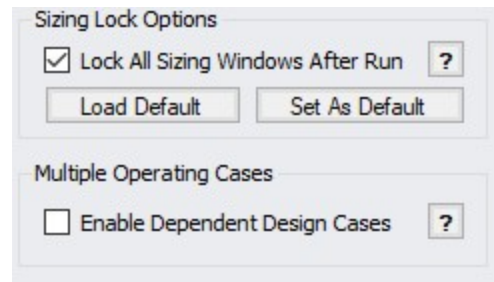
Figure 2: The additional options in the Sizing window are shown

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 (Figure 2). 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.

Figure 3: The various life cycle costs that can be considered are shown

- **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 (Figure 4).

Figure 4: Additional required information when recurring costs are included

- **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

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 (Figure 1).

- **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.

Figure 1: The options for including pipes in both sizing and cost calculation. Additionally, the options to create custom size groups is shown.

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(Figure 2) available elsewhere in AFT Fathom.
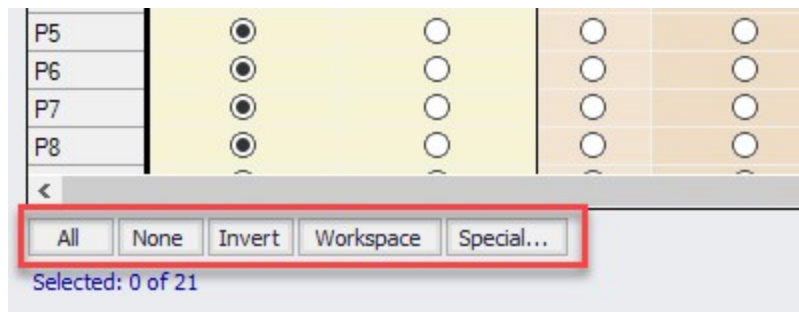
Figure 2: Selection options available for the Size/Cost Assignments panel

Selected rows can have their settings changed all at once with the buttons just above the table (Figure 3).
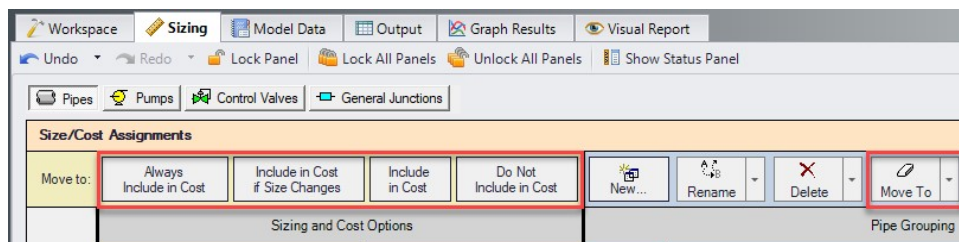
Figure 3: The buttons above the table can be used to change selected rows all at once

Note the indication of the total number of Independent Pipe Sizes in the lower right (Figure 4). This quantity has important implications on run time and what Search Method will be appropriate for the model.
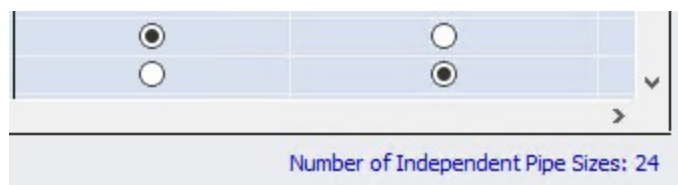
Figure 4: The Number of Independent Pipe Sizes in the model are shown which has implications on runtime

Pumps, Control Valves, and Junctions

- **Cost Options** - This section of the table controls sizing and cost calculation (Figure 5).
    - **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.

∧ Pumps	Cost Options		
	Include in Cost Report and Sizing	Include in Cost Report Only	Do Not Include Cost
J3 (Pump#1)	◉	○	○
J6 (Pump#2)	○	◉	○

Figure 5: Cost options available for Pumps, Control Valves, and Junctions

Maximum Cost Groups

Pumps and Control valves can optionally be grouped together in maximum cost groups (Figure 6). 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 pumps installed in parallel, with only one operating and the other functioning as a spare. Clearly, the cost of purchasing and installing two identical pumps will be identical regardless of how they end up being operated. The automated sizing calculation will base the cost of the pump 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 pump is "zero" - no pump at all! ANS must be told that the spare pump is the same size as the running pump. 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
●	○	●	○	●	○
●	○	●	○	●	○
●	○	●	○	●	○

Figure 6: The Maximum Cost Groups feature is shown allowing for items with identical costs to be grouped together

Related Topics

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.
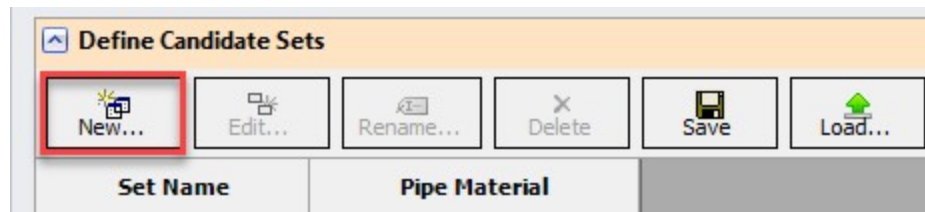
Figure 1: The new button allows for the creation of new Candidate Sets

Clicking New will request a name for the new Candidate Set (Figure 1). 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 (Figure 2). It would be common to select commercially available Sizes of a certain Type, but any selection can be made.
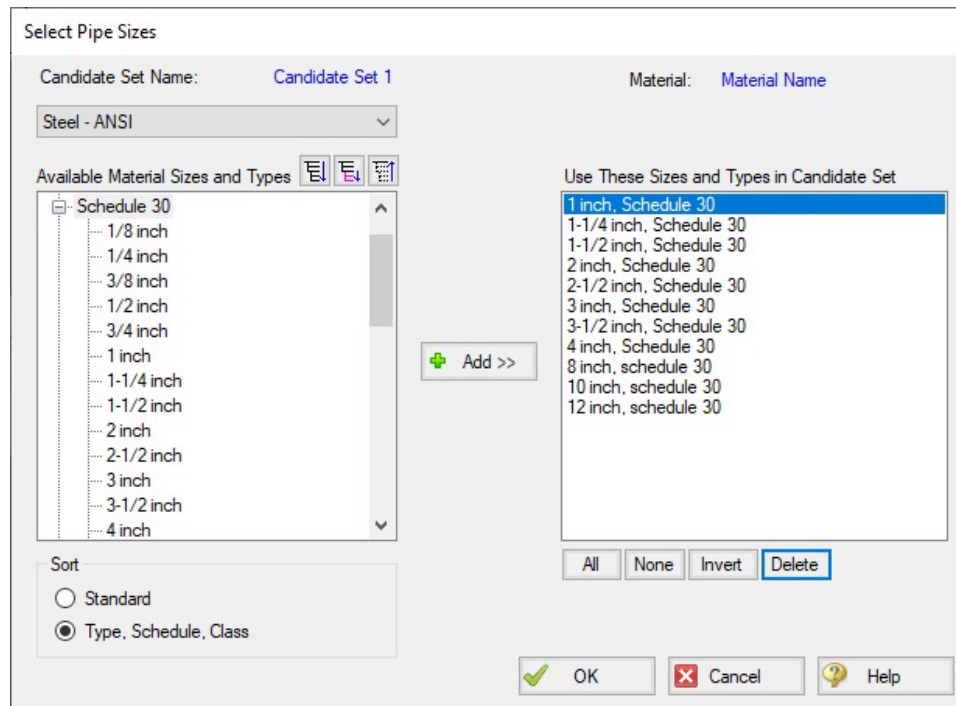
Figure 2: The Select Pipe Sizes window allows for the sizes to be considered for sizing to be specified
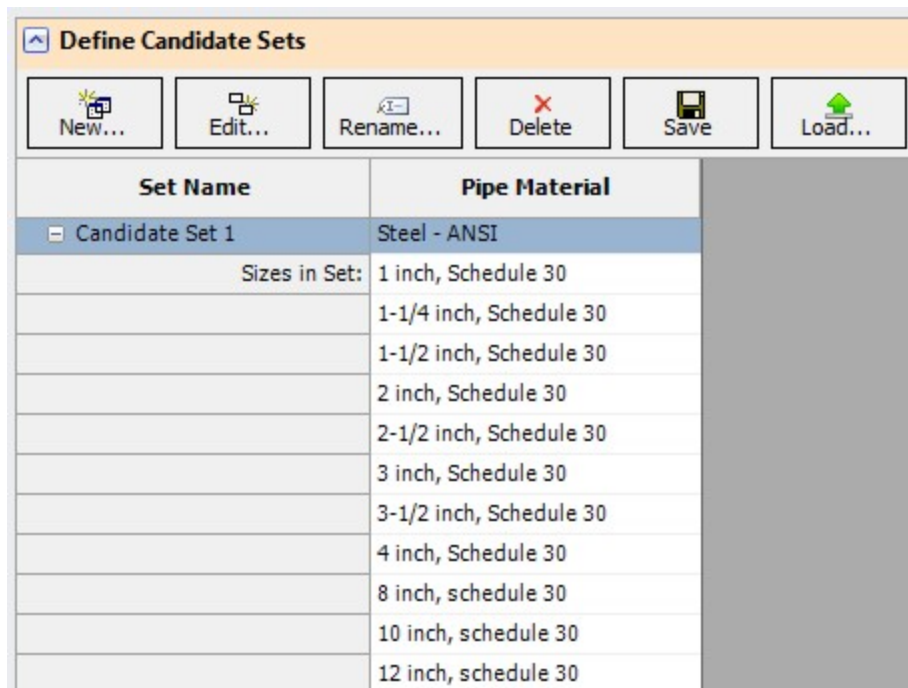
Figure 3: A fully defined candidate set

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 (Figure 4). 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.
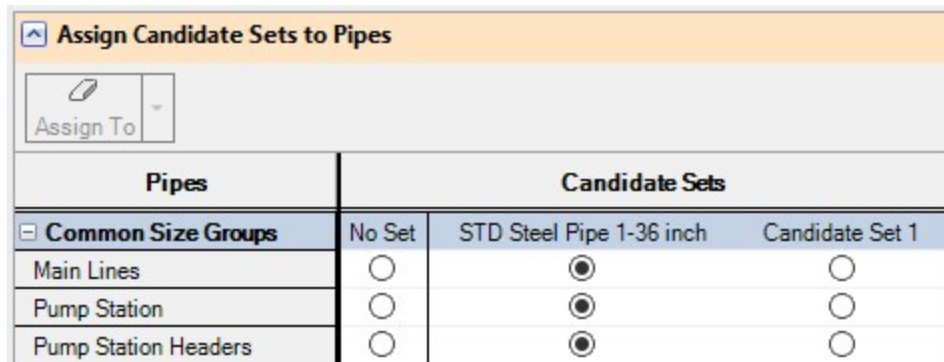
Figure 4: Size Groups can be assigned Candidate Sets to limit the sizing options

Related Topics

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 (Figure 1). In both cases, individual Design Requirements are defined with a Name, Parameter, Location, and limiting Value.
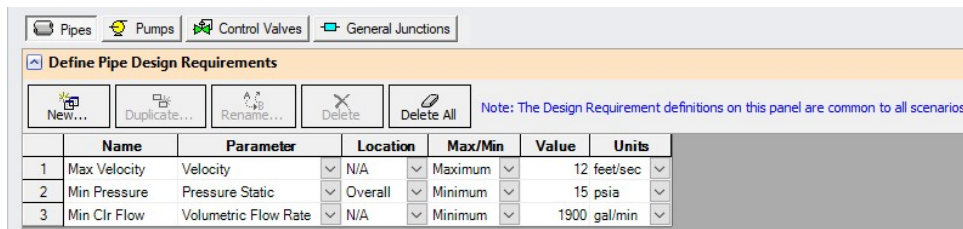
Figure 1: Defined Design Requirements in the Define Pipe Design Requirements

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 (Figure 2). If this is not desired, the group can be expanded and individual selections can be made.
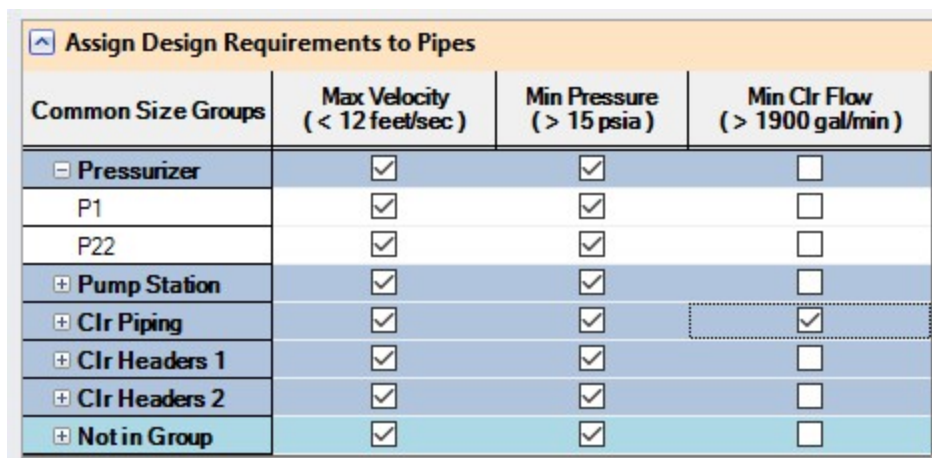
Figure 2: Created Design Requirements can be assigned to multiple objects

Related Topics

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 Cooling System - Creating Libraries - ANS 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 (Figure 1). 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.

Figure 1: Candidate sets can have multiple Cost Libraries assigned to them

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 pump 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.

Pumps	Multipliers (percent)				Service Duration (years)			
	Material	Installation	Maintenance	Operation	Full	Partial	Start	End
J4 (Pump C)	100	100	100	100	●	○	0	20
J7 (Pump B)	100	100	100	100	●	○	0	20
J10 (Pump A)	100	100	70	100	○	●	0	5

Figure 2: The Multipliers and Service Duration fields allow for modification of the library cost

Related Topics

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 pump 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 bio-logical 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

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.
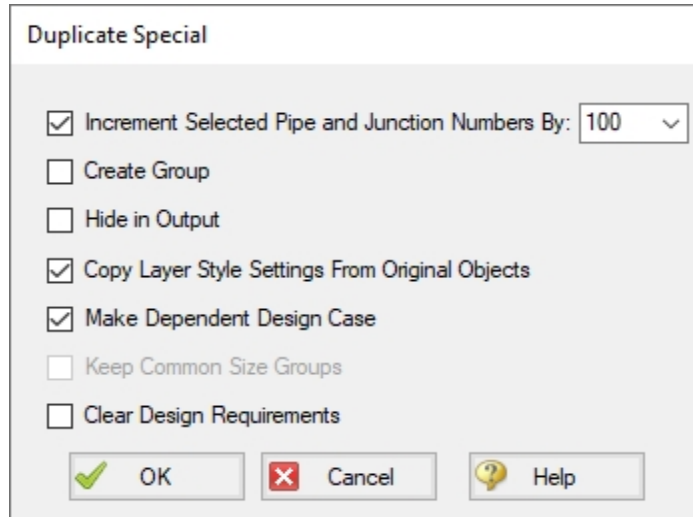
Figure 2: The Duplicate Special window can be used to create Dependent Design Cases

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 liquid levels can change, pumps 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 (Figure 2) shows a review of how the objects are connected:

Dependent Design Cases

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.

Pipes	DDC Pipe?	Assigned Design Requirements
P1	NO	Max Vel
P101	YES (P1)	Max Vel
P2	NO	Max Vel
P102	YES (P2)	Max Vel

Figure 3: The Dependent Design Cases panel

Related Topics

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 visu-

alize. 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.
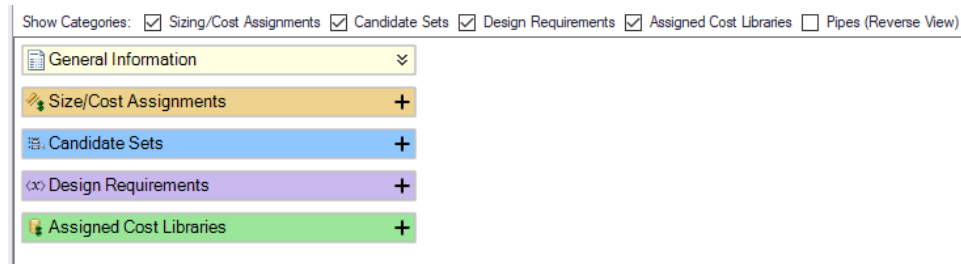
Figure 1: The Sizing Summary panel on launch

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.
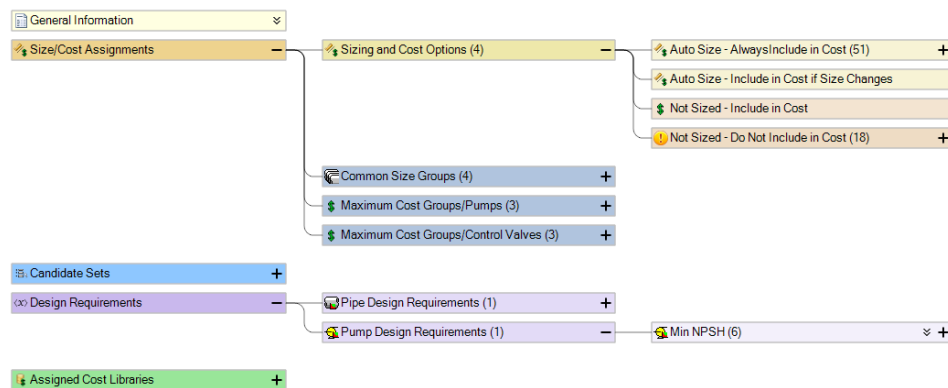
Figure 2: The Sizing Summary panel with options expanded

Individual objects can be expanded, showing detailed related information about them.
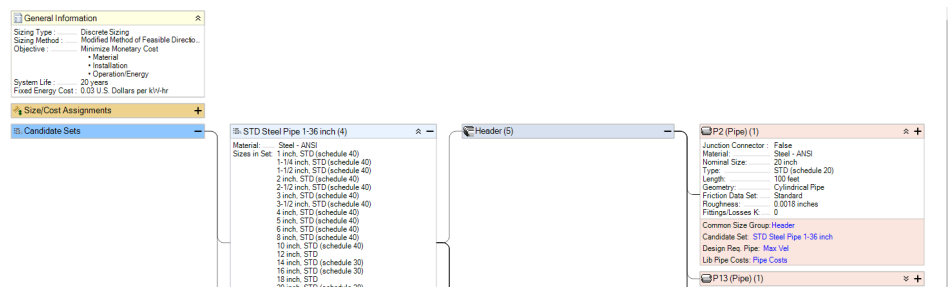
Figure 3: The Sizing Summary panel can show detailed related information

Clicking on a related category for a specific object will indicate the associated node(s).
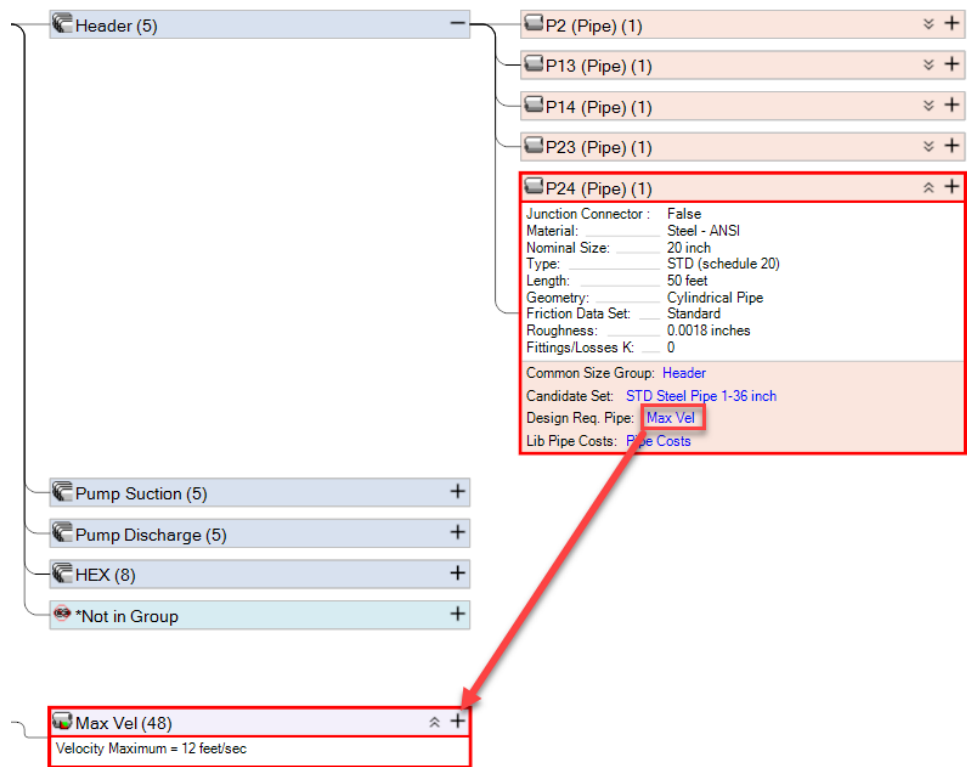
Figure 4: Associated nodes can be determined by clicking on a related category

Related Topics

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 Fathom 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 form the Other Actions window to assist with convergence.

Selecting Solution Control allows the Solution Control options to be adjusted before calculations are resumed. This is not recommended unless you have a good understanding of the parameters in the Steady Solution Control Group, 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.

Figure 1: The Solution Progress window has an additional line to show information on the ANS Calls to Solver when the Fathom 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.

Figure 1: The sizing tab of the output window shows the pipe sizes indicated by the ANS module

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.

Figure 2: Active Design Requirements in the Output window are indicated by blue highlighting

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 <u>disconnected scenario</u> 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.

Output Control

Items included in the Cost Report can be changed by adding/removing items from the Cost Report tab on the Display Parameters tab of Output Control. Similarly, the items shown in the Sizing tab can be controlled from the Pipe Sizing tab.

Workspace Integration

Modifying Sizing from the Workspace

Right-clicking anywhere on the Workspace allows the selection of existing <u>Common Size Groups</u>:
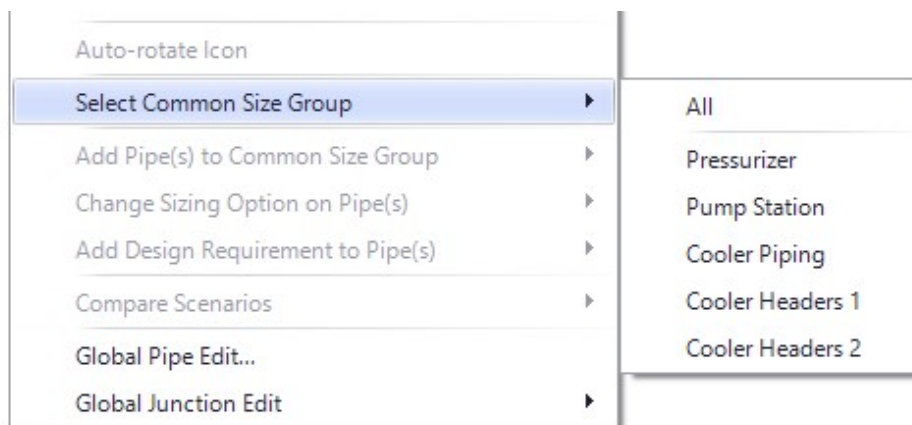
Figure 1: Objects can be added to Common Size Groups from the right-click menu

Selected pipes can be moved to a new or existing Common Size Group:
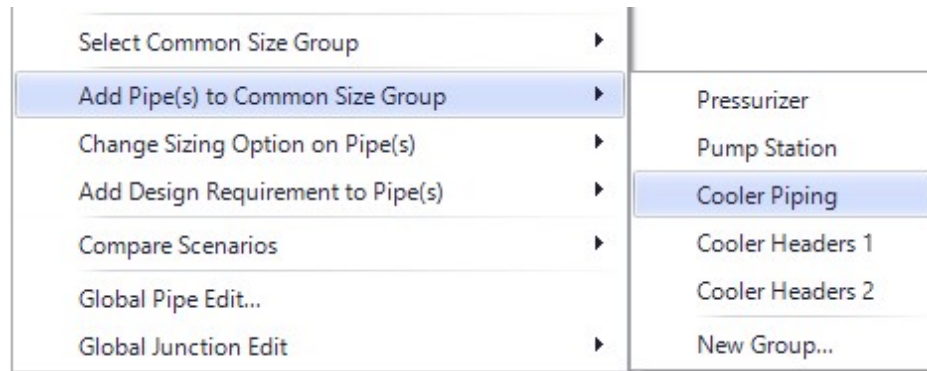
Figure 2: Pipes can be added to a previously created Common Size Group

Changes will be previewed before they are applied:

Figure 3: Preview of changes to Common Size Groups

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.
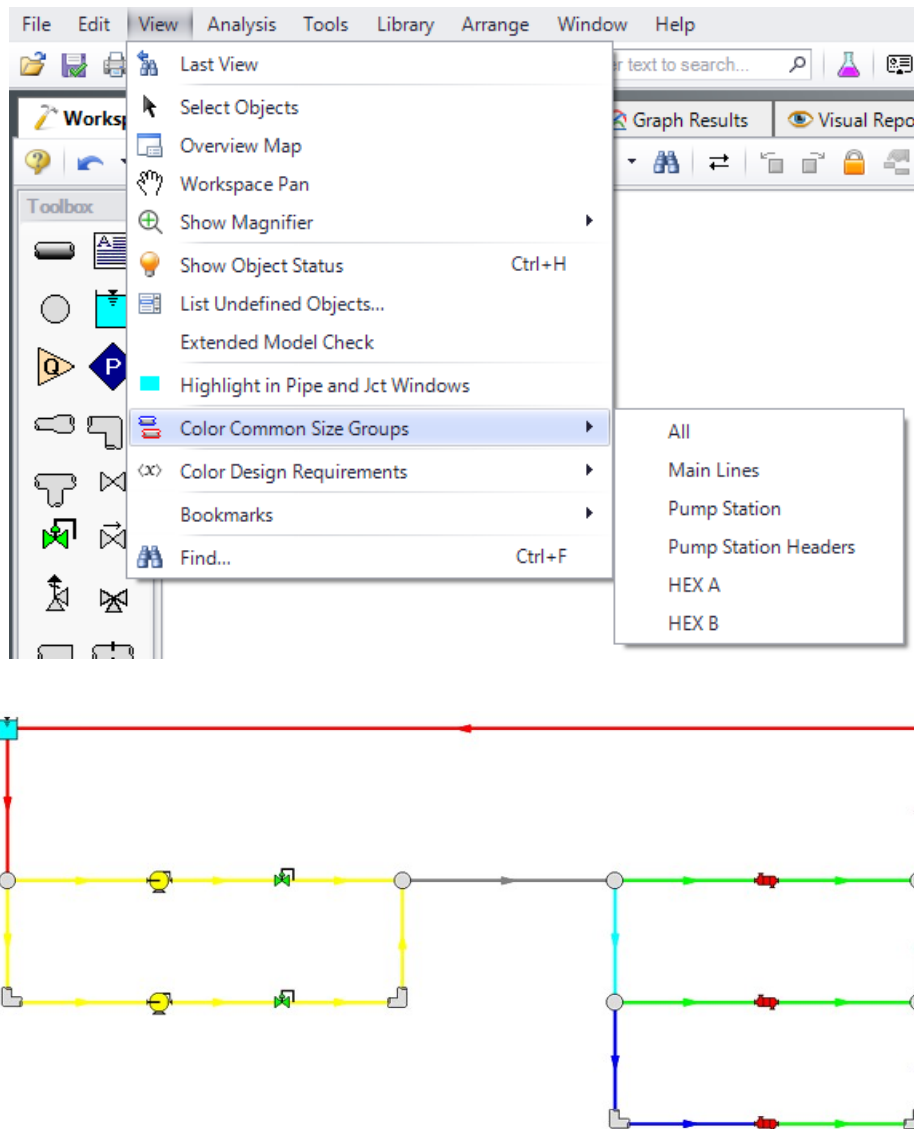
Figure 4: Common Size Groups organized by color

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.

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. This can be done by inputting the Diameter and K-factor columns, specifying one of the rows as a multiplier of one, then using the Calculate Multipliers button below the table.

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 bet-

ter 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 pump 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 difficultly. 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 Pumps

While pumps 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 pump requires a Cost Library and understanding of standard Pump Sizing.

At the most basic, pump sizing is simply selecting a pump that is capable of meeting pressure and flow requirements. Too often this involves selecting a pumps that is larger than required, under the assumption that a bigger pump will provide for operational margin. Unfortunately, over-sizing a pump 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 pumps is built into AFT Fathom. By using the Pump Sizing option, AFT Fathom will indicate the amount of head required to deliver the specified flow. It is up to the user to find a pump that operates at its BEP at the calculated flow and head.

The best size of pump to use when the pipe sizes are allowed to change as well is harder to handle, and requires iteration. See Sizing Pipes and Pumps Concurrently.

> **Note**: Certain pump configurations require special attention in ANS. See Special Concerns For Sizing Pumps.

Sizing Control Valves

Control Valve sizing is similar - AFT Fathom already has all of the tools needed to size a Control Valve for a specific set of pipes. AFT Fathom 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 pumps, 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 Pumps

There are several special concerns with sizing pumps. For a general overview of the process of sizing pumps see: Sizing Pipes and Pumps Concurrently.

End of Curve Flow Rate

Centrifugal pumps typically have an end of curve, or runout, flow rate. They cannot operate reliably past this point. This can be entered in Pump Configuration, but this will not actually prevent the model from running the pump 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 pump.

Partial Service Duration

Without other input, ANS will assume that any pump included in the sizing operates constantly.

If this is not the case, then either an appropriate Service Duration or Multiplier should be defined in Assign Cost Libraries.

Variable Speed Pumps with Partial Control Applicability

ANS can size pumps with a user defined fixed speed other than 100%, or a controlled pump that modifies speed to meet a flow or pressure setpoint. However, pressure-type variable speed controlled pumps 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 pump to change from a fixed 100% speed pump (non-controlling pressure) to a variable speed pump (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 pump as fixed 100% speed.

Either an Always Controlling pump 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 pump flow rate requirements, the ideal pump curve may be unrealistic.

For example, one operating case may require 500 gal/min, and another may require 750 gal/min. It is desired to select the ideal pump. After running ANS, it is determined that the pump should generate 50 feet of head in the 500 gal/min case and 70 feet in the 750 gal/min case. To meet this requirement, we need a pump curve that increases from 50 feet of head to 70 feet with a large increase in flow. This is an unrealistic demand.

To prevent this, a head rise Design Requirement can be added to maintain the high flow head at 70-90% of the low flow head. This will effectively force the sizing to consider only realistically shaped pump curves.

Related Topics

Sizing Pipes and Pumps Concurrently

The cost of a pumped piping system clearly depends on both the cost of the pipes, and the pump. It is tempting to select pumps and pipes in stages, but this is unlikely to produce the lowest cost solution. Selecting a smaller pump 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 pump's BEP is considered the ideal place to run a pump as the efficiency is highest, and thus cost is lowest. Similarly, any piping system will have a lowest cost design.

Whether a pump is running at BEP, however, depends on the piping system, and the lowest cost piping system depends on the pumps it contains. What may be the lowest cost piping system *for a given pump*, may not me the lowest *possible* cost option. It is possible that a cheaper pump could be selected with the correct piping adjustments while still maintaining all Design Requirements.

The point at which no adjustments to pump or system will result in lower cost is known as the **Optimal Pumping System Operating Point (OPSOP)**. One way to determine this point is to exhaustively size the system for different potential sizes of pumps. This is inefficient, but can be used to construct a visual representation of the OPSOP as shown below.
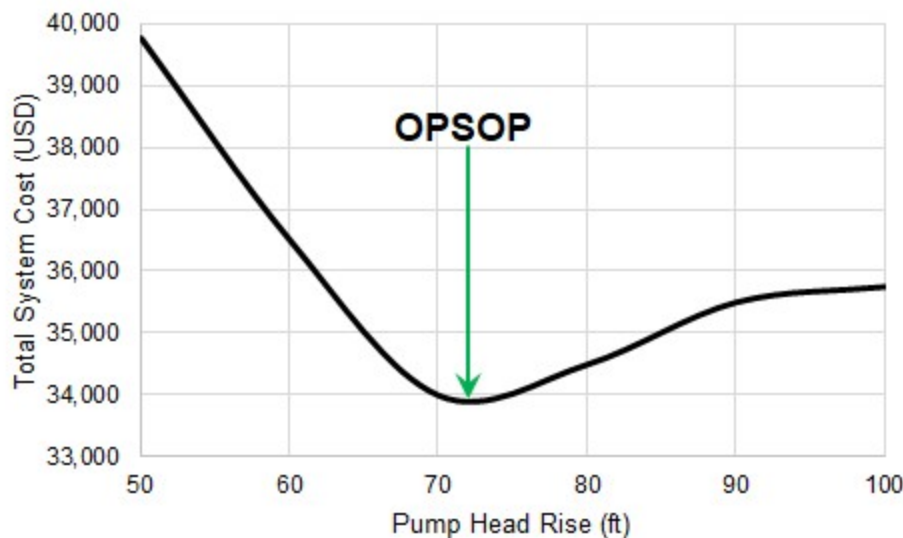
Figure 1: The OPSOP of a given system. Note that <u>every point is an ideal sizing solution for the given pump size</u>

If the sizing process is approached in stages, rather than concurrently, it is easy to get a result that says, for example, a 60 foot head rise pump is required. The **best case** system for the 60

foot head rise pump has a minimum cost of about $36,500. However, allowing the pump to change as well can result in a system more than 7% cheaper.

Process Outline

Find the Approximate Pump Size

1. Model the pump with the Pump Sizing option, fixing the flow to the desired amount. AFT Fathom will determine the required head rise.
    *   Include an estimate of Nominal Efficiency.
    *   Optionally, include Nominal NPSHR.
2. Add the pump to an Engineering Library.
    *   Attach a Cost Library to the Engineering Library if one does not already exist.
    *   Create a Power vs. Cost Scale Table for the pump with estimated cost values.
3. Select the appropriate options in the Sizing Window.
    *   The cost for the pump will only be included in the Objective if it is specified to be included in Size/Cost Assignments.
    *   If there are several pumps that will ultimately be of the same type, they should be placed into a Maximum Cost Group.
4. Run ANS.
    *   Investigate the resulting pump size.
    *   Update the Nominal Efficiency and NPSHR based on the result. Also update the cost information if necessary. If the pump accounts for a large portion of the overall cost, accurate values here are more important.
    *   Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Pumps

1. Locate accurate cost information and pump curves for the candidates.
    *   Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate pump.

Estimating Pump Costs

Without knowing specifically what type or size of pump is going to be included in the system, the pump behavior must be approximated. The behavior is approximated with the Pump Sizing option, indicating a design flow. We also need to approximate cost. Because energy cost is often the largest component of a pump's overall cost, and this cost is directly related to the pump's power, it makes sense to correlate other pump costs to power as well.
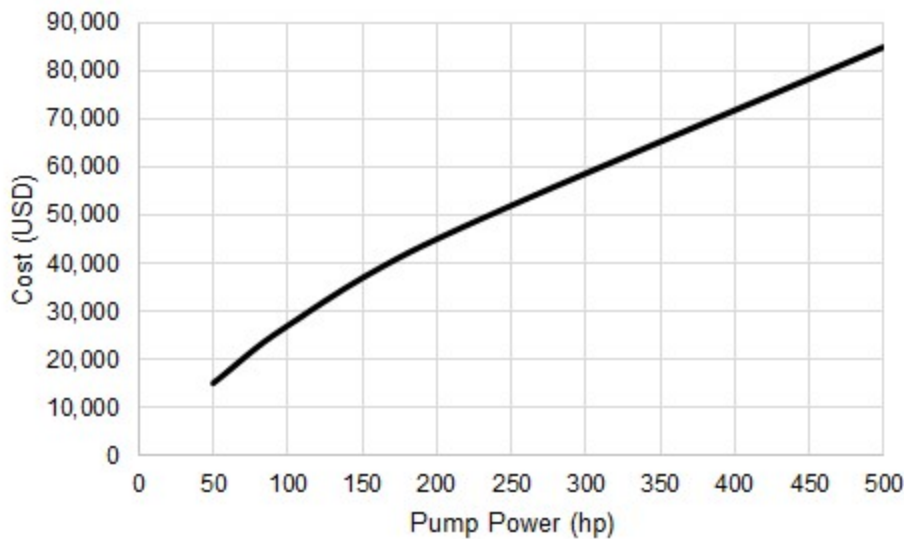
Figure 2 - Total Initial Costs for a series of pumps

Keep in mind that such a comparison is only valid for the selected range of pumps. Different designs of pumps, or pumps from different manufacturers may follow different relationships. One manufacturer may offer cheaper low-power pumps, while their competitor offers cheaper high-power pumps. This cost information is represented in ANS as a Power vs Cost Scale Table. This requires the creation of a Cost Library including the pump as a Custom Component.

Because we do not know what specific pump 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 pump runs at some **Nominal Efficiency**. AFT Fathom will calculate the fluid power required at any given flow and head rise. However, a pump or motor is not perfectly efficient and will therefore require more power than the fluid power. This affects the pump 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 pump size will minimize the initial pump 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 pump correct as this can affect the best sizing. The nominal efficiency will require more power and hence a more expensive pump - 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 pumps 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 pump. Without cost information on the pump, 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 head rise at the pump, and hence higher cost. By including pump cost information, the balance between pump costs and piping costs can be found.

However, this is still an **estimate** because we are not modeling a real pump. Fixing the flow and nominal efficiency are good design tools but it is important to represent a real pump curve not only to account for real effects, but also to ensure that the desired pump can actually be acquired.

If the pump 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 Pump

Now that there is a good idea of the size and cost of pump we are looking for, we can select a real pump we may want to use.

It will rarely be clear which pump is the best choice. First, the exact operating point determined from the approximate solution is likely unavailable in a commercially available pump. Second, different pumps have different costs - even if a pump with the exactly correct specifications is available, it may be much more expensive than a pump that is slightly mismatched. This cost difference could mean the cheaper pump is better overall, despite not appearing to be the ideal choice.

The best course is to select several candidate pumps, and size the system independently with each candidate pump. It is important that accurate pump curves and cost information be included for each candidate, as this no longer meant to be an estimation. After testing each candidate pump, the system with the lowest overall cost can be selected.

If the sizing results differ significantly in this stage this could indicate that the pump is poorly matched, that there was an error in the process, or that the system is dominated by pump costs and therefore very sensitive to the estimations.

Related Topics

Sizing Pipes and Control Valves Concurrently

Process Outline

Find the Approximate Control Valve Size

1. Model the Control Valve with the appropriate settings. AFT Fathom will determine the required loss.
2. Add the Control Valve to an Engineering Library.
   - Attach a Cost Library to the Engineering Library if one does not already exist.
   - 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.
   - The cost for the control valve will only be included in the Objective if it is specified to be included in Size/Cost Assignments.
   - 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.
   - Investigate the resulting control valve size.
   - 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.
   - Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Control Valves

1. Locate accurate loss and cost information for the candidates.
   - 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 Pump modeled as a fixed flow. This situation has no unique solution, as discussed in the Sizing Pumps 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 pump 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 Fathom is no different in this regard - it is possible to specify a system that would require physically unrealistic behavior. For example, the liquid pressures are allowed to drop below vapor pressure. 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 Fathom 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 pressure Design Requirement to prevent the vapor pressure issue.

How does this relate to Control Valves? Control Valves in AFT Fathom 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 pump - providing "free" pressure to the system in order to meet the setpoint. As the cost of this "pump" is not considered, the sizing process is trivial - we can simply select the smallest pipe size available, as our "pump" 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 operated 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 pumps 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 pump 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 Pipes, Pumps, and Control Valves Concurrently

Many systems contain both pumps and control valves, which makes the sizing process somewhat more complex.

Like sizing pipes and pumps 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

Find the Approximate Pump and Control Valve sizes

1. Model the pump(s) with the Pump Sizing option, fixing the flow to the desired amount.
2. Model the Control Valve(s) with the appropriate settings.
    - If the control valve(s) are Flow Controls downstream of the pump(s) to be sized, temporarily set them to be Pressure Drop Control Valve with a setpoint equal to the minimum required pressure drop.
    - Any non-Pressure Drop control valves will require a Design Requirement of minimum dP/dH or maximum Cv/Open Percentage.
3. Add the pump(s) and control valve(s) to one or several Engineering Libraries.
    - Attach a Cost Library to the Engineering Library if one does not already exist.
    - Create a Power vs. Cost Scale Table for the pump with estimated cost values.
    - Create a Cv vs. Cost Scale Table for the control valve with estimated cost values.
4. Select the appropriate options in the Sizing Window.

- The cost for the pump(s) and control valve(s) will only be included in the Objective if it is specified to be included in Size/Cost Assignments.
- If there are several pumps or control valves that will ultimately be of the same type, they should be placed into Maximum Cost Groups.

5. Run ANS.

- Investigate the resulting pump size.
- Update the Nominal Efficiency and NPSHR based on the result. Also update the cost information if necessary. If the pump accounts for a large portion of the overall cost, accurate values here are more important.
- Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Pumps

1. Locate accurate cost information and pump curves for the candidates.
   - Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate pump.

Select the two or three best candidate pumps and compare candidate Control Valves.

1. If necessary, change Pressure Drop Control Valves back to Flow Control Valves.
   - This will require a Design Requirement of minimum dP/dH or maximum Cv/Open Percentage.
2. Locate accurate loss and cost information for the candidate control valves.
   - Incorporate these values into Engineering and Cost Libraries.
3. Compare the sizing results for each candidate control valve, under each candidate pump.

Related Topics

Sizing Duct Systems

A Pipe Material 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 Pipe Material 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 3 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

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.**

- 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.
- 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

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 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 Fathom 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*0.0001 = 0.1. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.

- **Sequential Unconstrained Parameters**
  - **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

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 Fathom 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*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 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

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 Fathom 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*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

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

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.

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

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, pumps, 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 gal/min. The design is considered acceptable if the flow ends up as 120 gal/min. On the other hand, using a boundary of 100 gal/min **forces the model always to maintain 100 gal/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

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**. Pumps, 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.
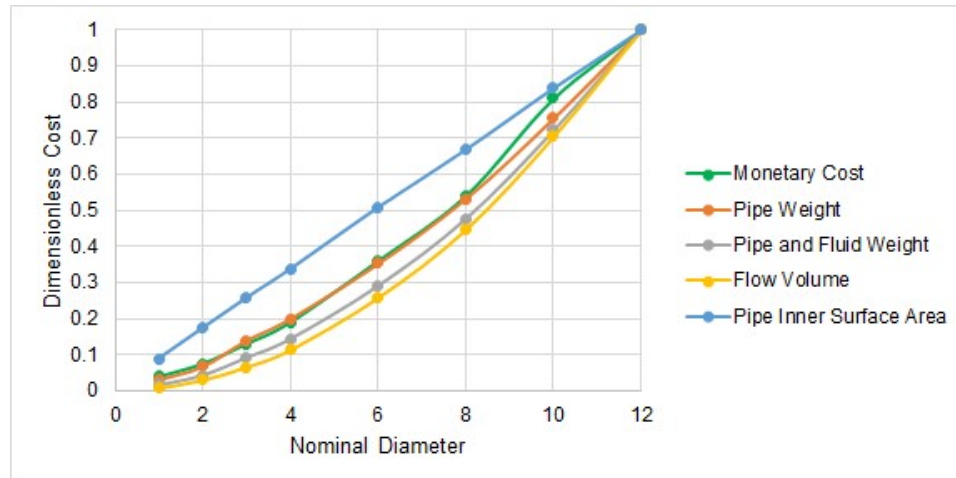
Figure 1: Comparison of Dimensionless Costs for Schedule 40 ANSI Steel water pipe. RS Means monetary cost closely follows pipe weight in this example.

Related Topics

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 reasonable 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.

$$Unique\ Hydraulic\ Models = (Candidate\ Set\ Count)^{Number\ of\ Independently\ Sized\ Pipes}$$

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

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:
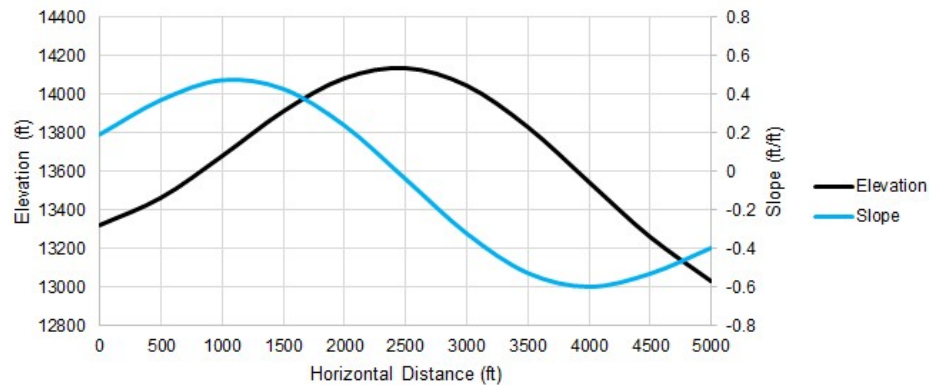
Figure 1: One of many possible elevation profiles of a mountain peak. The slope can be calculated at any point.

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 an employ an intelligent gradient based search method. Starting at a horizontal distance of zero, we can measure an elevation and slope:
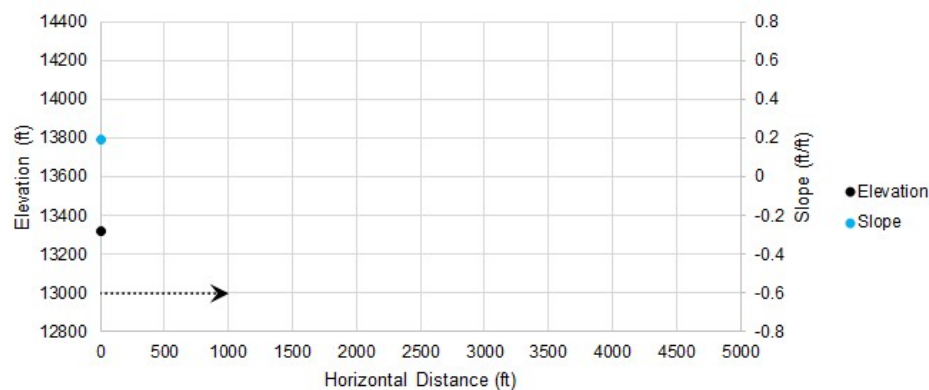
Figure 2: Starting a gradient based search

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:
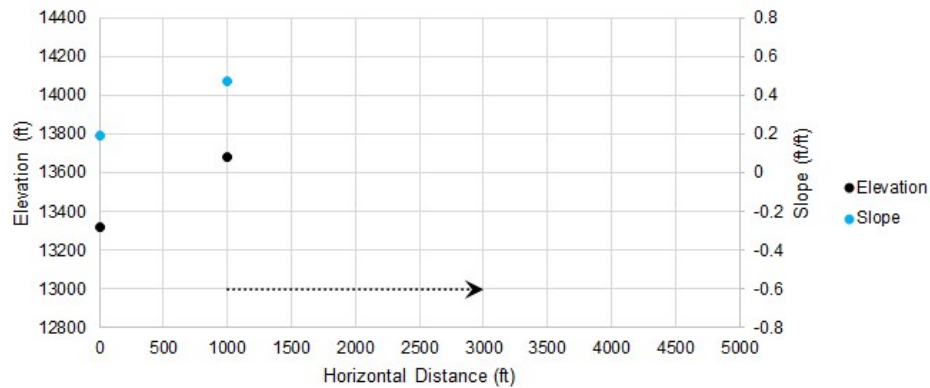
Figure 3: Iteration number 2

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.
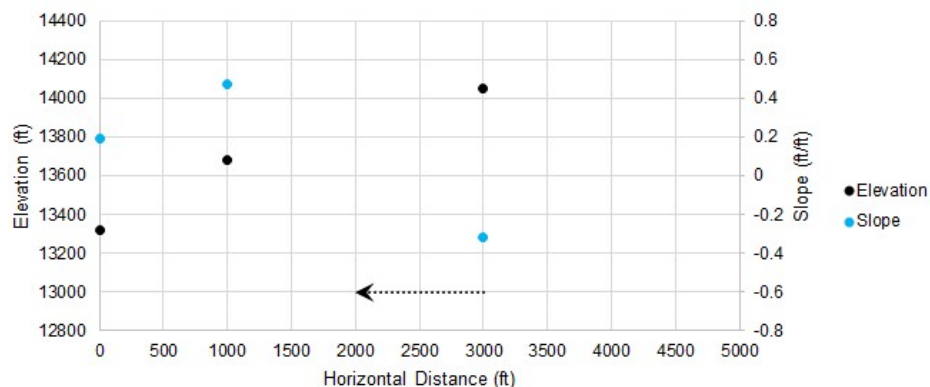
Figure 4: Iteration number 3

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.
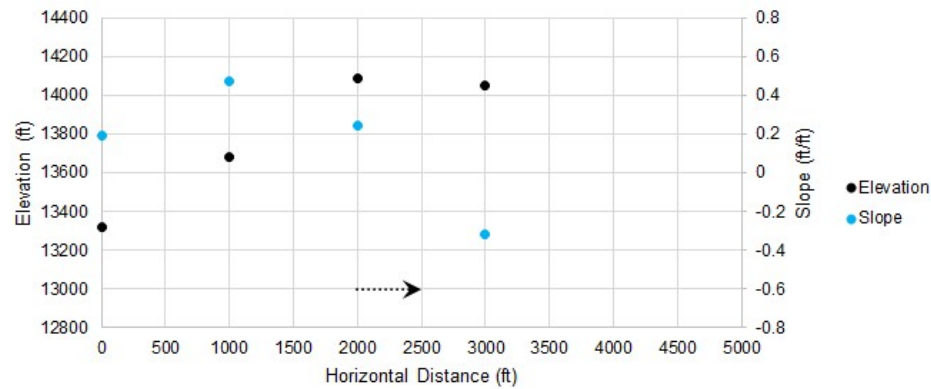
Figure 5: Iteration number 4

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.
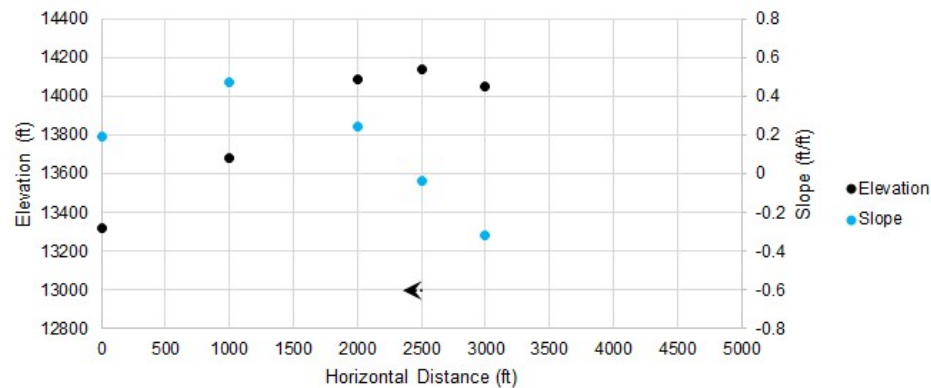
Figure 6: Iteration number 5

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

Estimating the Gradient

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.
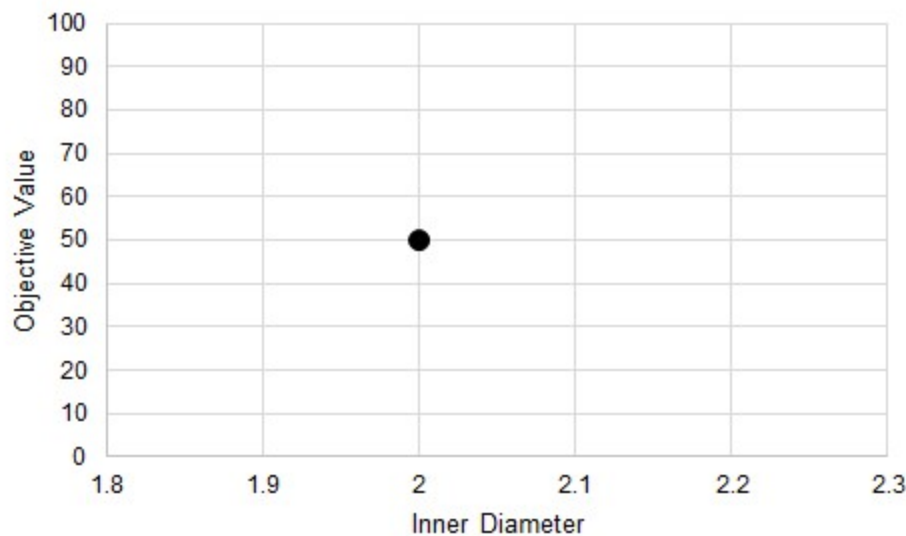
Figure 1: An initial test. It is desired to know the gradient at this point.

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.
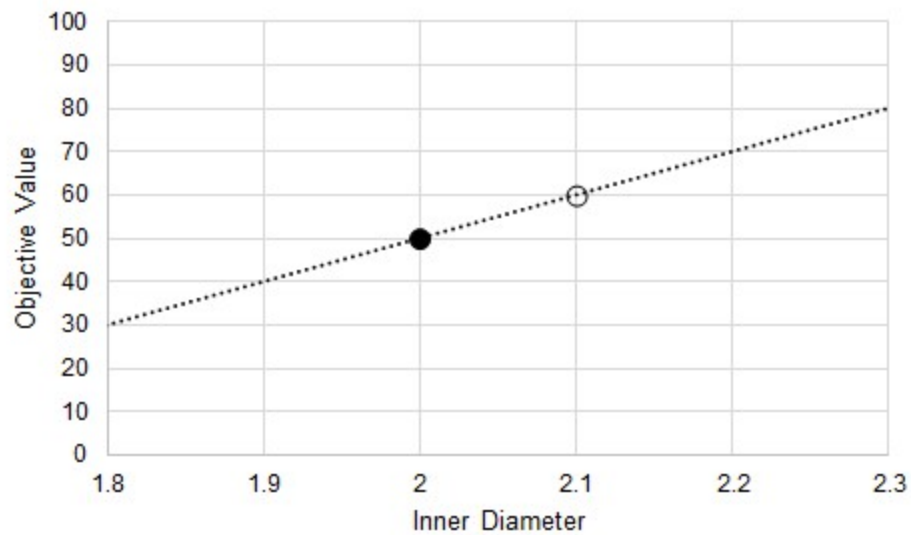
Figure 2: A perturbation. This allows the calculation of an average gradient as an approximation of the gradient at the initial point.

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**.
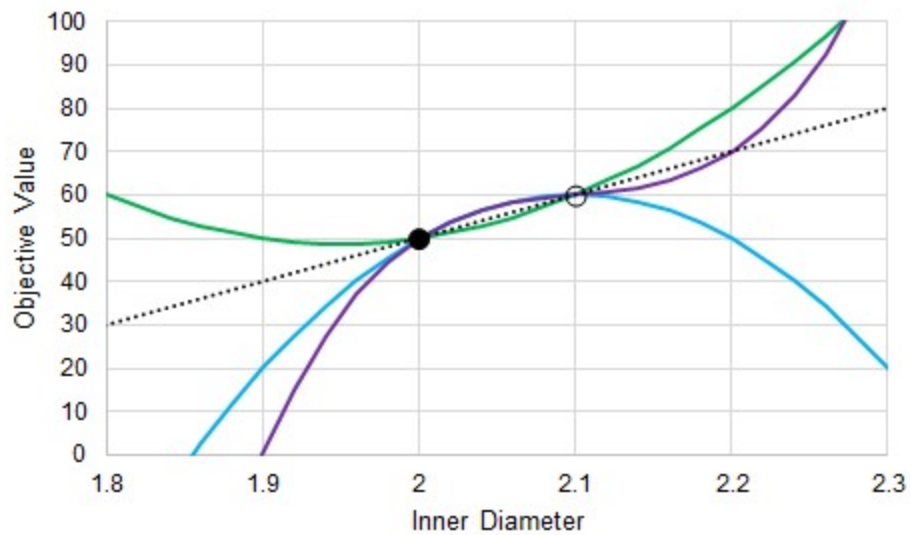
Figure 3: The actual gradient depends on the function. Many possible functions pass through two test points.

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.
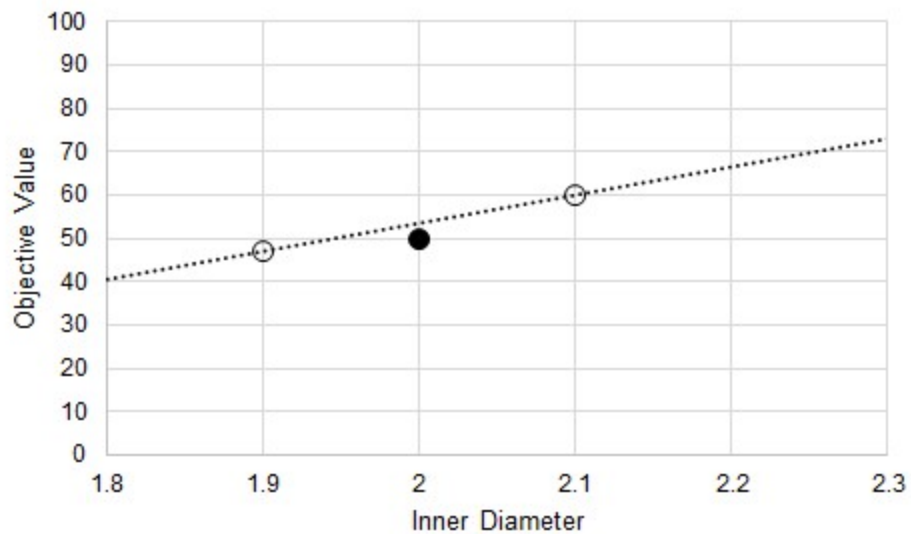
Figure 4: Central Difference estimate of gradient at the initial point

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.
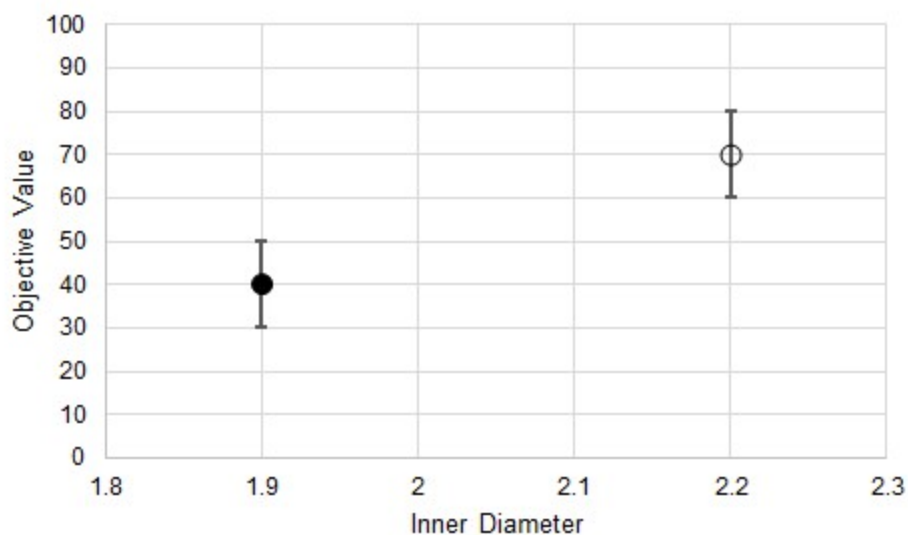
Figure 5: Uncertainty in the Objective Value due to iterative tolerance

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.
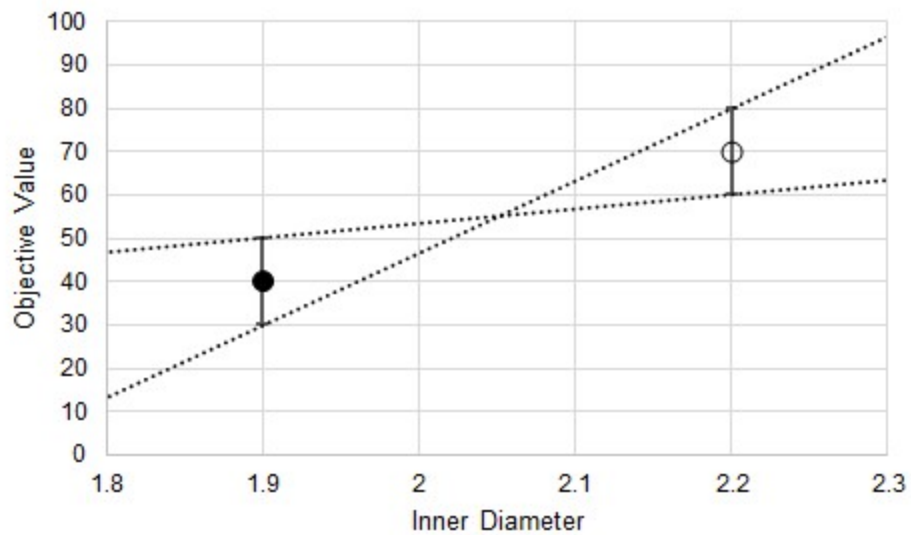
Figure 6: Uncertainty in the estimated gradient

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.
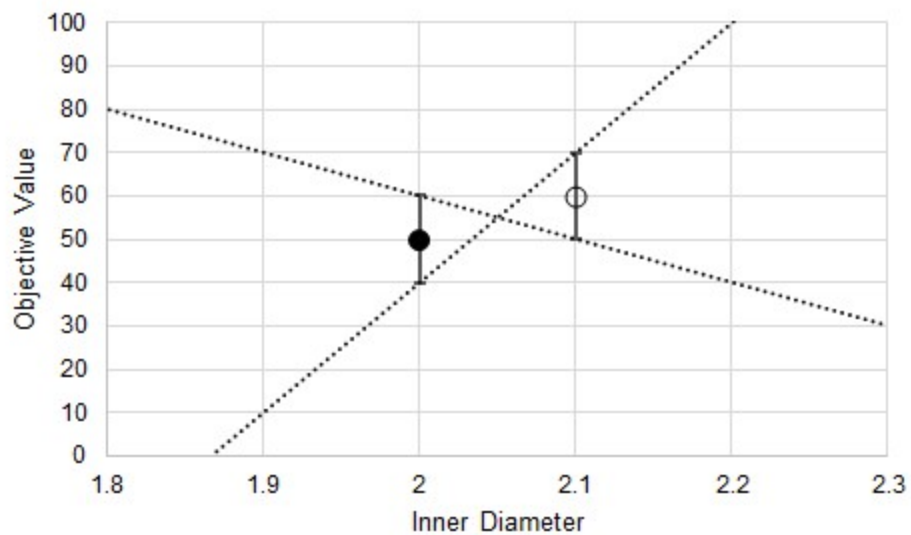
Figure 7: Substantially increased uncertainty in the estimated gradient from decreasing the step size

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

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 psi 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.
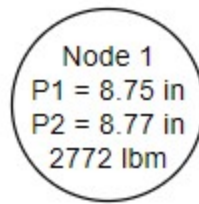
Figure 1: The first node showing the Continuous Optimum

From this first **node**, we want to divide the solution space on our first variable - the diameter of pipe 1. The pipe size cannot be 8.75 inches, it can only be a discrete size, in our case an integer value. Therefore, we **branch** the into two sub-problems with pipe 1 restricted to less than 8 or more than 9 inches (Figure 2). We have also established a **bound** on the problem - the solution will never have a feasible weight lower than 2772 lbm.
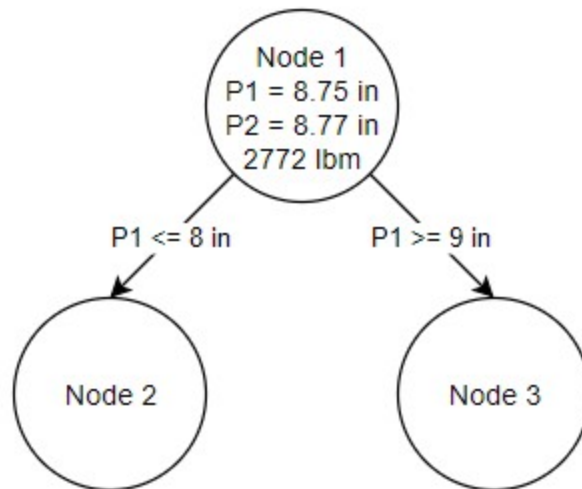
Figure 2: Two Sub problems are created from the Continuous Optimum in Node 1

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.
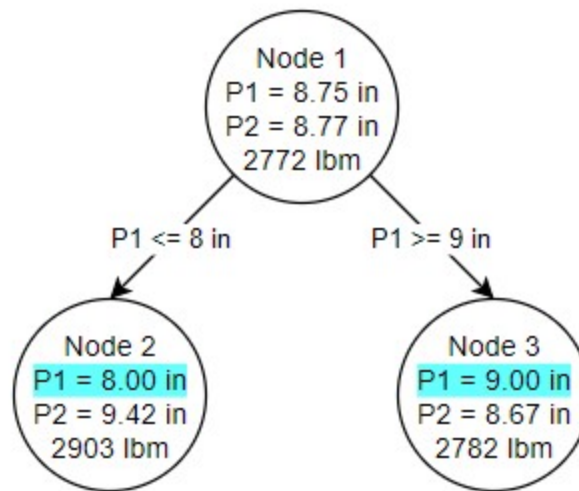
Figure 3: The conditions of Nodes 2 and 3 are shown, as well as the sizing arrived at in the two nodes

Due to the nature of this particular problem, the ideal pipe sizes for pipe 1 in both nodes 2 and 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 (Figure 4). This time, however, we place a restriction on our second pipe's diameter.
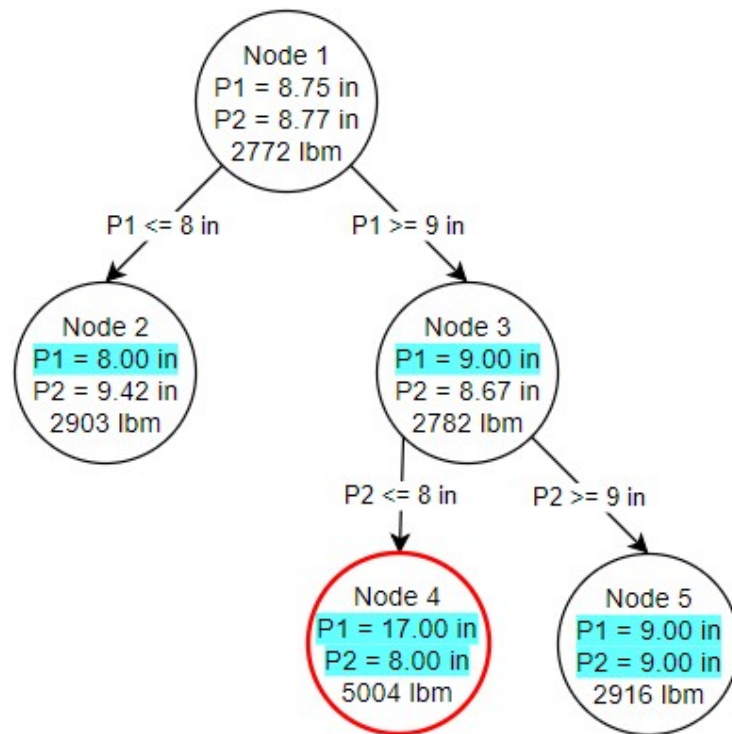
Figure 4: Node 4 produces an infeasible solution while Node 5 produces a feasible solution based on the branching criteria set from Node 3

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 (Figure 5).
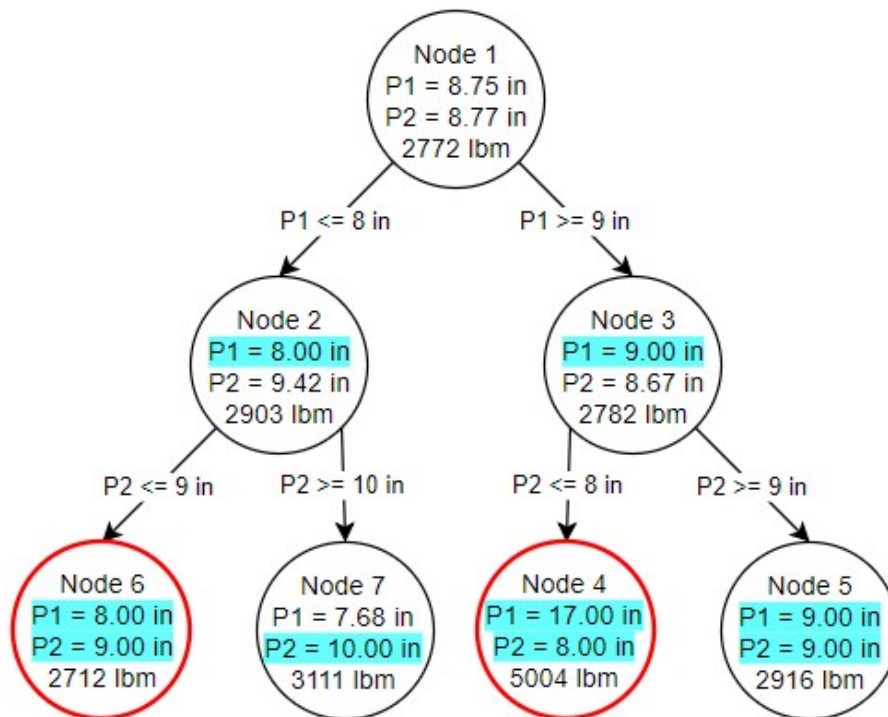
Figure 5: Branching occurs from Node 2 which produces one feasible (Node 7) and one infeasible (Node 6) solution

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.
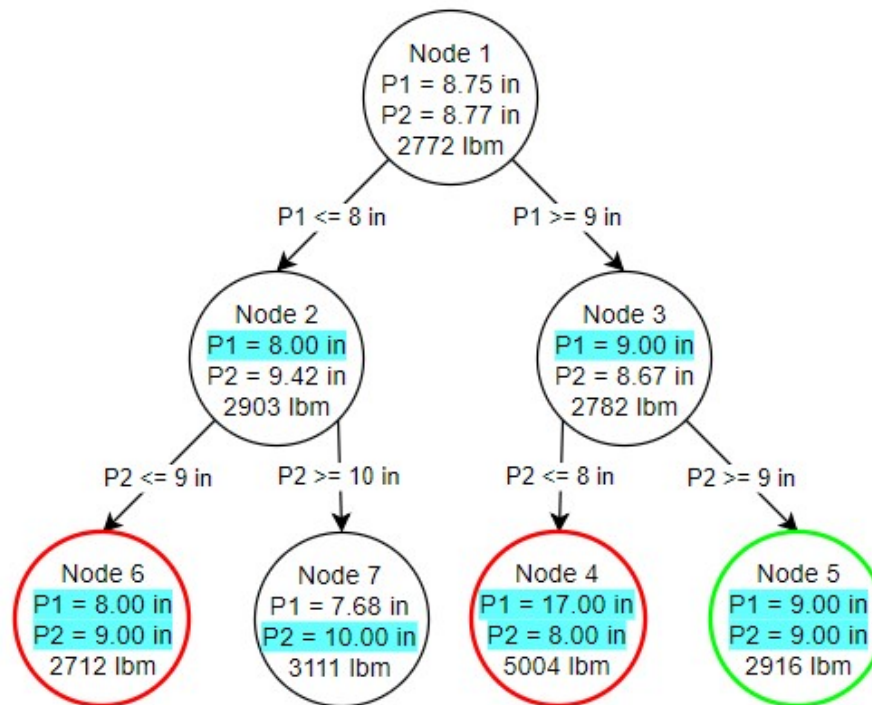
Figure 6: Results of the Branch and Bound method

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 7 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

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 the Volume Balance. 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** - Fathom**#**.dat (where # is the corresponding version number).
- **Location** - Fathom installation folder, default is C:\AFT Products\AFT Fathom **#**.
- **Limitations** - Cannot be transferred to other users, local use only.

Local User Library

- **Purpose** - Stores user-specified data for most component types.
- **Filename** - Fth_User**#**.DAT (where # is the corresponding version number).
- **Location** - C:\Users**username**\AppData\Roaming\Applied Flow Technology\AFT Fathom**#**.
- **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

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

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 Fathom:

- Engineering Libraries - Contain all component types except cost information
- Energy Cost Libraries - Contains energy cost data associated with components in existing engineering libraries
- Cost Libraries - 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 Fathom. 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 FATHOM LOCAL USER LIBRARY may not be removed and will always appear in the list of available libraries. External shared libraries will also always appear in the available library list.
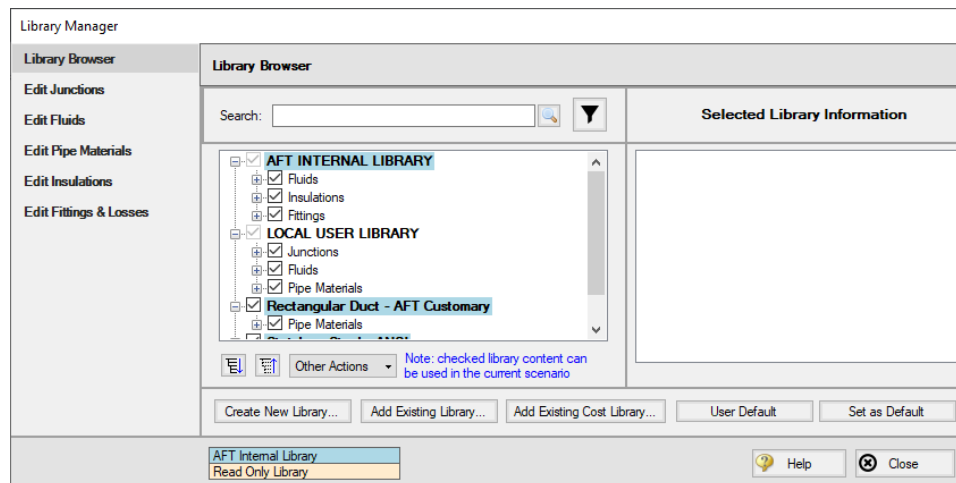
Figure 1: Library Browser in the Library Manager

Connecting and Disconnecting Libraries

In order to use the components from a library in the model the library must be **connected**. A library is connected to the model when it is shown with a check mark in the Library Browser. All libraries are automatically connected when they are first added to the Library Manager.

Sections or components in a library can also be connected/disconnected on an individual basis if the library as a whole is connected.

It may be desired to disconnect a library if you do not want the components from that library to be available for use in the current model, but you do want the library to be available for use in other models.

Clicking on Set As Default will establish the list of connected libraries as the default list, while clicking on User Default will set the connections to the default list.

AFT Internal Pipe Material libraries can be connected or disconnected as a group from the Other Actions button under the available libraries list.

Editing Library Contents

Existing content can be moved to/from, copied to/from, or deleted from available libraries in the Library Browser.

To edit an item right-click on the item in the Library Browser, then click the desired action from the right-click menu. Note that moving, copying, and deleting can be done to a library as a whole, or can be done for just a section or single component in a library. Individuals components can also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.

Note: Editing Cost libraries must be done separately in the Cost Library or Energy Cost Library windows, accessed from the Library Menu.
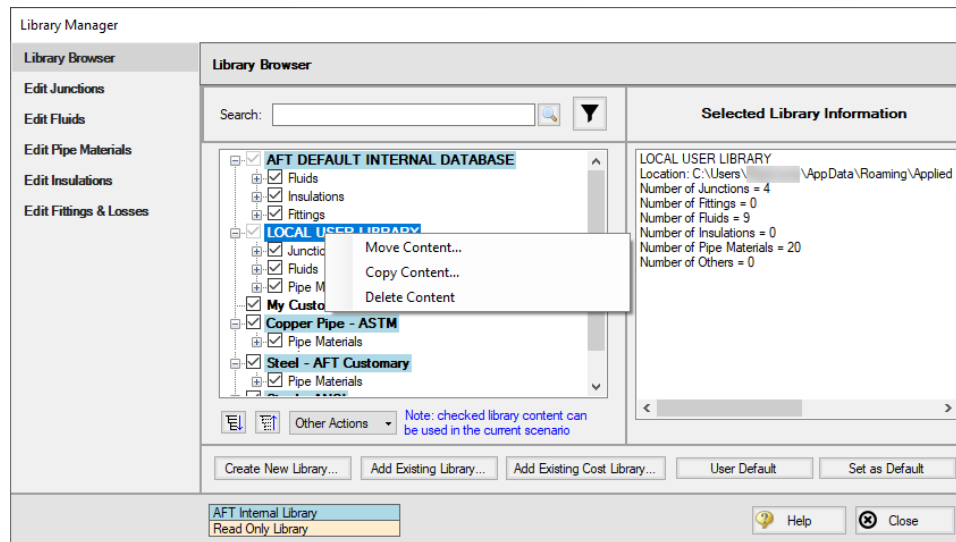
Figure 2: Library contents can be transferred or deleted from the right-click menu

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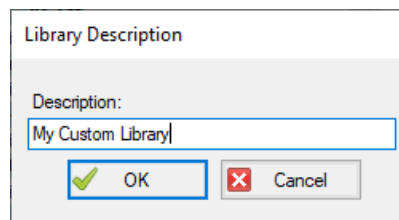
Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected

Related Topics

Junction Library

Adding Custom Junction Data to the Component Library

To simplify modeling and eliminate repetitive data entry, AFT Fathom allows you to save fre-quently-used junction data to a custom library. Once saved, the custom equipment data can be selected from the Library Jct dropdown 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 drop-down.

AFT Fathom accepts custom data for all junction types except the Volume Balance.

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

Fluid Library

AFT Fathom 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. Raw data can be provided for the variation of density, dynamic viscosity, specific heat, thermal conductivity, and vapor pressure as they depend on temperature or solids concentration. This window provides the tools to perform polynomial curve fits of fluid data, or the raw data can be interpolated.

Required Data:

- Density
- Dynamic Viscosity

Optional Data:

- Specific Heat (required for heat transfer capabilities)
- Thermal Conductivity (required for heat transfer capabilities)
- Reference Temperature/Enthalpy point (required for heat transfer capabilities) - Note that the reference temperature/enthalpy do not need to be exact values since only changes in the parameters are used
- Vapor Pressure (required if checks and warnings for cavitation are desired)

The Fluid Library window also allows you to open the Change Fluid Data window which is essentially the same window as the Add New Fluid window. For easy editing, data for the fluid you wish to change is entered for you in the appropriate areas. The original curve fit data you entered is also displayed.

The Add New Fluid/Change Fluid Data windows offer cross-plotting of your custom curves against the curve fit data points. You can also import data from text files into these windows.

Related Topics

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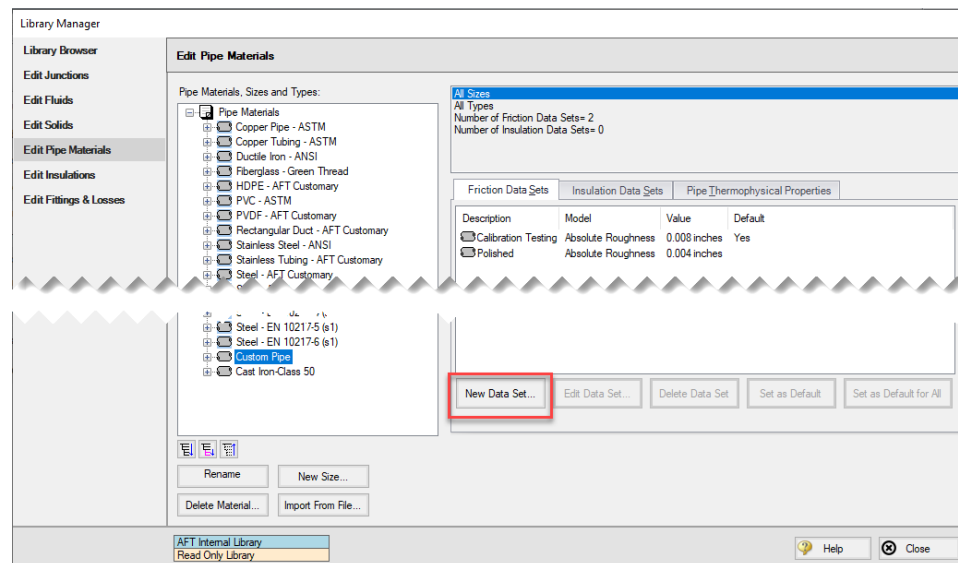
Figure 1: Friction Data Set tab on the Edit Pipe Materials panel

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 with Heat Transfer enabled.

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 pipes 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

Insulation Library

The Edit Insulations panel is accessed from the Library menu. This panel allows you to view the insulation data provided with AFT Fathom, 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

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 Fathom, and to add, edit or delete user defined Fitting & Losses in the AFT Default Internal Library.

Fittings & Losses can either be added by using the entry form in the window, or by importing a text file as described below. To enter a new Fitting/Loss item in the Edit Fittings & Losses panel click Create New Loss Item. Define the applicable category, type, geometry and loss model information then click Accept. The user-defined item will appear in the Fittings & Losses on the left under the relevant category/type with the source shown in parentheses as "U".

Custom fittings & losses are saved to the local user library. From there the user can move them to other libraries (see Library Manager).

Import From File

Fittings & Losses can be imported from a *.CSV type file, which can be created from any spreadsheet-type software, such as Excel.

The entries in the file should be in the order as shown in Figure 1, but with no headers in the workbook. If a header row or a blank row is used, Fathom will present a warning, but give the user the option to still import the valid rows.

	A	B	C	D	E	F	G	H
1	Category	Type	Sub Type (Optional)	Geometry (Optional)	Geometry Value (Optional)	Loss Model	Loss Value	Reference/Notes (Optional)
2	Elbow/Bend	Mitre	General	Angle (degrees)	45	K (constant)	5	

Figure 1: Example of import file setup for a custom elbow/bend. Headers should not be present in the file to import.

Valid options for each of the columns are listed below, with detailed options given in Table 1.

- **Category** - Must match one of the values from Table 1. "Other" may be used if none of the specific fitting/loss categories fit.

- **Type** - This field is required, any name/value may be entered. Types already existing in the library are shown in Table 1, and may be used if desired.

- **Sub Type** - This field is optional. A sub-type may be chosen from Table 1, or any other name/value may be entered.

- **Geometry** - This field is optional. A geometry type from Table 1 must be used if a geometry is entered. If the Geometry field is left blank, the Geometry Value will be ignored.

- **Geometry Value** - This field is only required if a Geometry type has been entered. Any positive numerical value may be entered.

- **Loss Model** - This field is required. The Loss Model must be either "K (constant)" or "K / fT (Crane)".

- **Loss Value** - This field is required. Any positive numerical value may be entered for the loss value.

- **Reference/Notes** - This field is optional. Any information may be entered here as needed.

Table 1: Valid Entries for each field. The Geometry Value and Loss Value fields will accept any numerical entry. The Reference/Notes field will accept any entry.

Category	Type	Sub Type	Geometry	Loss Model
Elbow/Bend	Standard Threaded	General	Angle (degrees) Pipe Size (nominal) None	K (constant) k / fT (Crane)
	Mitre	General		
	Smooth Flanged	r/D=1		
		r/D=1.5		
		r/D=2		
		r/D=3		
		r/D=4		
		r/D=6		
		r/D=8		
		r/D=10		
		r/D=12		
		r/D=14		
		r/D=16		
		r/D=20		
	3-K (Darby)	90 deg		
		45 deg		
		180 deg		
Valve	Angle	Type 1	Angle (degrees) Open Percentage L/D Pipe Size (nominal) None	K (constant) k / fT (Crane)
		Type 2		
		Direct flow		
		General		
	Ball	None		
	Butterfly	Plane disk - AR=0.92		
		Plane disk		

		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		
	Cylindrical	Type C		
		General		
	Gate	Rect. Tube		
		Wedge/Plug		
		Wedge		
		Disk w/o recess		
		Disk with recess		
		Disk		
		Wedge - Full Open 10 deg.		
	Globe	Wedge - Full Open 20 deg.		
		Sluice		
		Disk with recess		
		90 degree wall		
		45 degree wall		
	Plug	Standard 45 deg. wall		
	Poppet	Standard 90 deg. wall		
		Standard		

	Three-way	Y Type		
		Straight-way		
		W/D=.035		
		W/D=.07		
	3-K (Darby)	No Seat Width		
		Through flow		
		Branch flow		
		Angle		
		Globe		
		Plug		
		Gate		
		Ball		
		Diaphragm		
Check Valve	Swing	90 deg. seat	Angle (degrees) None	K (constant) k / fT (Crane)
		45 deg. seat		
	Lift	90 deg. wall		
		45 deg. wall		
	Tilting disc	General		
	Stop-check	Globe 90 deg.		
		Globe 45 deg.		
		Globe thru flow		
	3-K (Darby)	Swing Check		
		Lift Check		
Orifice	Sharp	None	Area Ratio None	K (constant) k / fT (Crane)
	Long	None		
Entrance/Exit	Entrance	Sharp Reentrant Rounded	R/D None	

		Flush		
	Exit	None		
Other	Diff Flowmtr	Orifice Plate	Area Ratio None	K (constant) k / fT (Crane)
		Nozzle		
		Venturi 7 deg.		
		Venturi 3 deg.		
		Dall Tube		
	Honeycomb cell	L/D=2		
		L/D=4		
		L/D=6		
		L/D=8		
		L/D=10		
		L/D=12		
	Screen	Perforated Plate		
		Circular wire		
		Silk thread		
		Round wire or netting		
	Tee	Through flow (dead branch)		
		Branch flow (dead run)		
Flange	Steel	150 lb. 300 lb.	None	N/A

Related Topics

Network Library Overview

Custom libraries contain custom data generated by the user, including fluid properties, pipe materials, and data for junctions types such as pumps and valves.

All custom data created by the user is kept on the local computer in a file called FTH_USER11.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

Benefits of Shared Libraries

When the Fathom 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 Fathom itself, so it is guarded data.

Once the data is placed on a network, users can connect to the library in the Library Manager. Users can connect to as many libraries as they want.

If the user connects to a library with these files and selects to be connected to any of these control files, the model will inherit the control formatting specified in the control files. This means that

special report layouts, special graphical representations and colors, and other special controls can be decided upon by the library constructor, and those control features can be loaded into local models by end users. This is highly useful for creating customized reports for certain customers or project requirements.

By way of example, assume the Output Control content was specified as coming from the "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

Creating an Enterprise-Wide Network Library System

Overview

Users of AFT Fathom 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 Fathom by creating a file named DATABASE.LIB in the FATHOM 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 Fathom uses to find these libraries. An External Library is any library file that is created by the user and listed in the FTH_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 Fathom 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 Fathom 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
Fathom Internal Libraries		
FATHOM#.DAT FTH_USER#.DAT	AFT Default Internal Library AFT Fathom Local User Library	AFT Fathom # folder, where # is the version number AppData folder
External Shared Libraries (DATABASE.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in DATABASE.LIB
External Libraries (FTH_DBUSER.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in FTH_DBUSER.LIB

Creating Library Files

The first thing to do is create the files that contain the information to be shared. These files will be referred to as library files and can contain information about pipe materials, components (e.g. pumps, valves, etc.), fluids, etc. The following are several ways to create library files:

- Use the Create New Library button in the Library Manager, as described previously. Data can be taken from the current model or other libraries.
- Copy and paste sections from one file into another file using a text editor.

This discussion will focus on the first and second methods. Let's assume that the person responsible for the network library has already created a new pipe material. Also, assume that the goal is to create a network-wide library to contain this material.

The information for the material is copied from the Local User Library (Fth_User#.DAT, where # is the corresponding version number) to a new file using the Library Broswer 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 Fathom 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 Fathom. 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 Fathom directory on the network. However, since Fathom 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 FTH_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 Fathom folder (e.g. AFT Products\AFT Fathom\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 Fathom. 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 Fathom 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 (FTH_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 Fathom will write the full path of the library to the External Library listing (FTH_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 Fathom 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 Fathom\Libraries\filename.dat). You can automatically have AFT Fathom 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 Fathom folder where the executable is located.

When the pipe materials library was added it became an External Library listed in FTH_DBUSER.LIB. Now we need to move this listing from the local machine to the AFT Fathom 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 Fathom 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 Fathom, click Set As Default. To apply these settings to a model made previously, open the model, go to Library Manager, click User Default and, if you have set the default, the connections and setting will now be applied to the model.

Sharing Model Formats/Preferences

Note that the Library Manager used to provide the option to include formats/preference settings such as the Output Control 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

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

The model file may have a junction which is derived from a library, but the referenced library is not currently connected in the Library Manager. For example, a valve junction 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:

- 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.

The valve junction in the model does in fact have all of the required data needed to run the model; however, when it is not found in the library, you will be offered several options:

- **Add To Library** - The junction(s) will be automatically added to the [Local User Library](#), from which the contents can be moved to a different library if desired. If the model file is saved, the junction(s) will now associate the library connection with the Local User Library, and not the original library which is currently unavailable.
- **Disconnect** - The model will disconnect the junction from the library.
- **Keep Name** - The junction will keep it's original name, remember what library to which it was originally connected, and connect to that library again when it is made available. If, however, you try to open the junction Properties window, it will become disconnected from the library.

Whatever choice you make, the model will retain all of its engineering data and yield the same results as it did originally.

2. Item in Model is Different from Library

The model file may have a junction which is derived from a library, and the referenced library is connected, but for some reason the data is not the same. For example, you may have a valve junction with a Cv vs Open % curve for the Rocky Mountain Valve Company's Model 100A in your model, but it does not match the curve data for this item in your library. This could be for several reasons:

- When the model was built, the valve was connected to the Rocky Mountain Valve Company (i.e., RMVC) Model 100A library item. But since that time, the RMVC 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 valves to the library values.
- The library may have been changed by an unauthorized individual, and while it was originally accurate, now the data is no longer valid. In this case, it would be best to **not update** the valve in your model, and to inquire into the cause of the inaccurate data within your organization.

Upon loading the model you will be offered several options:

- **Update Junction** - This will actually change the model's input data, such that it is replaced with the data currently in the connected library, and it might yield different results for the model.
- **Update the library (if it is local) to match the model's items** - This will change the data in the library, but will leave the model's input data as it was. The model will yield the same results as it did previously, but the library item will be updated.
- **Disconnect** - This will disconnect the junction from the library, leave the model as it was, and it will yield the same results.

Related Topics

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 Fathom tries to open a model that requires certain libraries, and those libraries have not been connected already, AFT Fathom looks for them in the locations specified in the User Options. In addition, AFT Fathom looks in the model's folder. If AFT Fathom finds the libraries in one of these locations, it automatically adds it to the list of connected libraries in the Library Manager. If it cannot find a library, it informs you once the model is loaded.
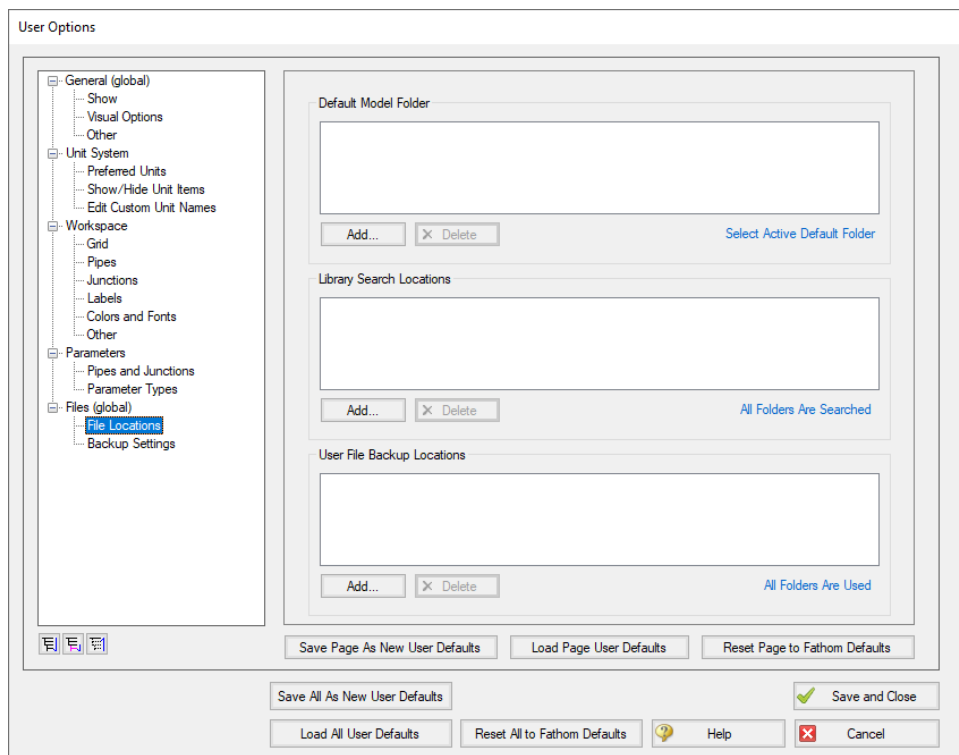
Figure 1: Library Search Locations can be defined in the User Options window

Related Topics

Theory and References

This section focuses on the fundamental theories that are used in AFT Fathom's solution method. Additionally, this chapter gives information on the source material for the theories used. Topics include discussions on the theories that comprise the network solution, detailed information on theory, and information on references used in the development of AFT Fathom.

Content

Nomenclature

A	cross-sectional flow area of a pipe
c_p	specific heat, constant pressure
C_D	discharge coefficient
C_V	valve coefficient
CHW	Hazen-Williams factor
D	diameter of a pipe
f	friction factor
f_T	turbulent friction factor (used by Crane, 1988)
f_L	laminar friction factor
f_{Tr}	transition friction factor
F	error value in mass balance
Fr	Froude number
g	gravitational constant
h	convection coefficient

h	enthalpy
H	head
HGL	hydraulic gradeline (same as head)
J	Jacobian matrix
k	thermal conductivity
K	loss factor
K	constant for Power Law fluid
$K_{sprinkler}$	fire sprinkler loss factor
L	length of a pipe
$\dot{m}$	mass flow rate
n	general constant
n	constant for Power Law fluid
Nu	Nusselt Number
P	pressure, static
P_h	heated perimeter
P_o	pressure, stagnation
P_w	wetted perimeter
q	heat rate to a pipe
Q	volumetric flow rate
q''	heat flux
r	radius
r	relaxation
R, R'	resistance
R_c	coefficient of rigidity
Re	Reynolds Number
R_h	Hydraulic radius (equal to radius divided by 2)
s	pump speed
S	Submergence
S	Hydraulic gradeline gradient
S_y	yield stress

T	temperature
U	overall heat transfer coefficient
V	velocity
x	distance along pipe centerline
z	elevation
a, *m*, *q*	angle
b	diameter ratio
e	roughness
n	kinematic viscosity
ρ	density
μ	dynamic viscosity

Subscripts

i	junction at which solution is sought
j	junctions with pipes connecting to junction i
o	stagnation
∞	infinity, far away, ambient

Network Solution Methodology

AFT Fathom makes use of standard matrix solution techniques (Jeppson 1976). The method is known as the H-Equation method, where *H,* the piezometric head, is solved for at each junction by forcing continuity of flow through each connecting pipe. Simultaneously, the head loss across each pipe is updated based on the flow balance information. The flow rate and head are solved in an inner-outer loop algorithm, where the flow is guessed, the head loss is calculated consistent with that guess, and the flow is updated according to the new pressure drop information. The Newton-Raphson method is employed to refine each successive solution, resulting in a sparse square matrix that is solved during each solution pass.

The concepts of pressure and hydraulic gradeline (*HGL*, also called piezometric head) are related but use different frameworks for considering pipe system behavior. The *HGL* includes both the static and elevational effects of pressure. The relationship between the two is given by Equation 1:

$$HGL = \frac{P}{\rho \, g} + z \tag{1}$$

where *z* is the elevation and *P* is the static pressure. Each concept offers certain advantages.

The solution technique makes use of the continuity and one-dimensional momentum equations. AFT Fathom uses pressure and mass flow rate as the unknown parameters because they are more fundamental in nature than the head and volumetric flow rate used by Jeppson and thus extend more naturally to mechanical and chemical systems.

In the following discussion, subscripts denote values at junctions. Thus, P_i represents the static pressure at junction i. Double subscripts denote values along pipes connecting two junctions. Thus, $\dot{m}_{ij}$ represents the mass flow rate in the pipe connecting junctions i and j.

Application of the law of mass conservation to each junction yields:

$$\sum_{j=1}^{n} \dot{m}_{ij} = 0 \tag{2}$$

where n is the number of pipes connected to junction i. Equation 2 states that the net mass flow rate into each junction must sum to zero.

The basic equation for pipe pressure drop due to friction can be expressed with the Darcy-Weisbach equation:

$$\Delta P_f = f \frac{L}{D} \left(\frac{1}{2} \rho V^2 \right) \tag{3}$$

where ΔP_f is the frictional pressure loss. The total pressure change between junctions is given by the momentum equation in the form of the Bernoulli equation:

$$P_1 + \frac{1}{2} \rho V_1^2 + \rho g z_1 = P_2 + \frac{1}{2} \rho V_2^2 + \rho g z_2 + \Delta P_f \tag{4}$$

The static (normal), P, and stagnation (pressure if all kinetic energy were converted isentropically into pressure), P_o, pressures are related as follows:

$$P_o = P + \frac{1}{2} \rho V^2 \tag{5}$$

Substituting Equation 5 into 4

$$P_{o,1} + \rho g z_1 = P_{o,2} + \rho g z_2 + \Delta P_f$$

Solving for the frictional pressure drop for a constant area pipes yields:

$$P_{o,i} - P_{o,j} + \rho g (z_i - z_j) = \Delta P_f \tag{6}$$

where i and j denote upstream and downstream junction values.

The definition of mass flow rate is:

$$\dot{m} = \rho \, A \, V \tag{7}$$

Combining Equations 3 and 6 and substituting for velocity, V, using Equation 7 gives the mass flow for each pipe:

$$\left[\frac{P_{o,i} - P_{o,j} + \rho \, g \, (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = \dot{m}_{ij} \tag{8}$$

where R_{ij}' is the effective flow resistance in the pipe and the subscript ij refers to the pipe connecting junctions i and j.

$$R_{ij}' = \left(\frac{f_{ij} \, L_{ij}}{D_{ij}} + K_{ij} \right) \frac{1}{2 \, \rho \, A_{ij}^2} \tag{9}$$

Note that R' is not the same as R, which is the resistance used in the Output Control window and is expressed by the following equation:

$$\Delta H = R \, Q^2$$

Substituting Equation 8 into Equation 2 results in the equation to be applied to each junction i:

$$\sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho \, g \, (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = 0 \tag{10}$$

where n is the number of pipes connected to junction i.

To be completely general, Equation 10 should be written for junction i:

$$\sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho \, g \, (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = \dot{m}_{i,Applied} \tag{11}$$

to allow for application of boundary condition flow rates to a boundary junction node.

Equation 11 as applied to each junction in the network represents the system of equations that need to be solved to determine the stagnation pressure at each junction. To solve this system, the Newton-Raphson method is employed. In the Newton-Raphson method, new values for each unknown are calculated based on the previous value and a correction that uses the first derivative of the function.

In this instance the function that Newton-Raphson is applied to would be of the form:

$$F_i = \sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho\, g\,(z_i - z_j)}{R_{ij'}} \right]^{\frac{1}{2}} - \dot{m}_{i,Applied} \tag{12}$$

The method involves finding all the junction stagnation pressures, $P_{o,i}$, that cause all of the F_i to go to zero, thus satisfying Equation 11 at all junctions.

When applied to a system of equations, the Jacobian matrix contains all the required derivative information to employ the Newton-Raphson technique. The Jacobian, J_F, is given by:

$$J_F = \begin{bmatrix} \dfrac{\partial F_1}{\partial P_{o,1}} & \dfrac{\partial F_1}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_1}{\partial P_{o,n}} \\[2mm] \dfrac{\partial F_2}{\partial P_{o,1}} & \dfrac{\partial F_2}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_2}{\partial P_{o,n}} \\[2mm] \vdots & \vdots & \ddots & \vdots \\[2mm] \dfrac{\partial F_n}{\partial P_{o,1}} & \dfrac{\partial F_n}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_n}{\partial P_{o,n}} \end{bmatrix}$$

The column matrix $\vec{P}_o$ contains the stagnation pressure at each junction, and column matrix $\vec{F}$ contains the F values (Equation 12) at each junction. The updated solutions for $\vec{P}_o$ are obtained from the following Newton-Raphson equation:

$$\vec{P}_{o,new} = \vec{P}_{o,old} - J_f^{-1}\,\vec{F} \tag{13}$$

Traditional implementations of the Newton-Raphson method require initial guesses at the solution before the Solver can begin. This often results in the situation where you have to know the solution ahead of time in order to solve for it. To avoid placing this burden on the user, AFT Fathom uses a proprietary method for generating first guesses. This method is usually sufficient to get the solution going in the right direction, especially for systems that don't have highly non-linear features.

However, there are times when AFT Fathom's first guess does not lead to a converged solution. In these cases, AFT Fathom provides you with the option of specifying your own initial guesses of flow and pressure.

Related Topics

have meaning for zero g systems..

Continuity Equation at a Branch

By applying the continuity equation at branch *i*, we loop over all connecting pipes and get:

$$\sum_{j=1}^{n} \dot{m}_{ij} = 0$$

where $\dot{m}_{ij}$ is the flow rate in the pipe connecting junctions *i* and *j*.

Related Topics

Energy Equation at a Branch

By applying the energy equation at a branch *i*, we loop over all connecting pipes and get

$$\sum_{j=1}^{n} \dot{m}_{ij}\, h_{ij} = 0$$

where h_{ij} is the enthalpy at the connection point of pipe and junction.

Related Topics

Network Implementation

By applying the Continuity Equation at a Branch about junction *i* and branches *j*, we get

$$\sum_{j=1}^{n} \dot{m}_{ij} = 0 \tag{1}$$

By applying the Energy Equation at a Branch about junction i and branches j, we get

$$\sum_{j=1}^{n} \dot{m}_{ij}\, h_{ij} = 0$$

(2)

To obtain a valid solution in a network, the stagnation pressure at each branch point will be equal and the mass flow and energy flow will sum to zero at the branch. Although the stagnation pressures at the pipe endpoints will equal that at the connecting junction, this is not true for enthalpy (and temperature). The 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 enthalpy will change to the mixture enthalpy, or the average enthalpy of all inflowing pipes.

If only one pipe flows into the branch, the branch enthalpy will equal the enthalpy coming out of the pipe. But if more than one pipe flows into the branch, the enthalpy will reach an averaged condition. It is this averaged enthalpy that results from mixing of two or more streams at different enthalpies. This enthalpy is supplied to all outflowing pipes. The preceding discussion in depicted in Figure 1 and follows equation 3.

$$h_{mix} = \frac{\sum_{j=1}^{n} \dot{m}_j^+\, h_j}{\sum_{j=1}^{n} \dot{m}_j^+}$$

(3)

where the + superscript indicates the summation applies only to positive inflows into the junction.

To provide converged solutions in the shortest time possible, matrix solution techniques are required. Matrix techniques allow all pipes to be solved together, which offers speed and the best convergence characteristics.
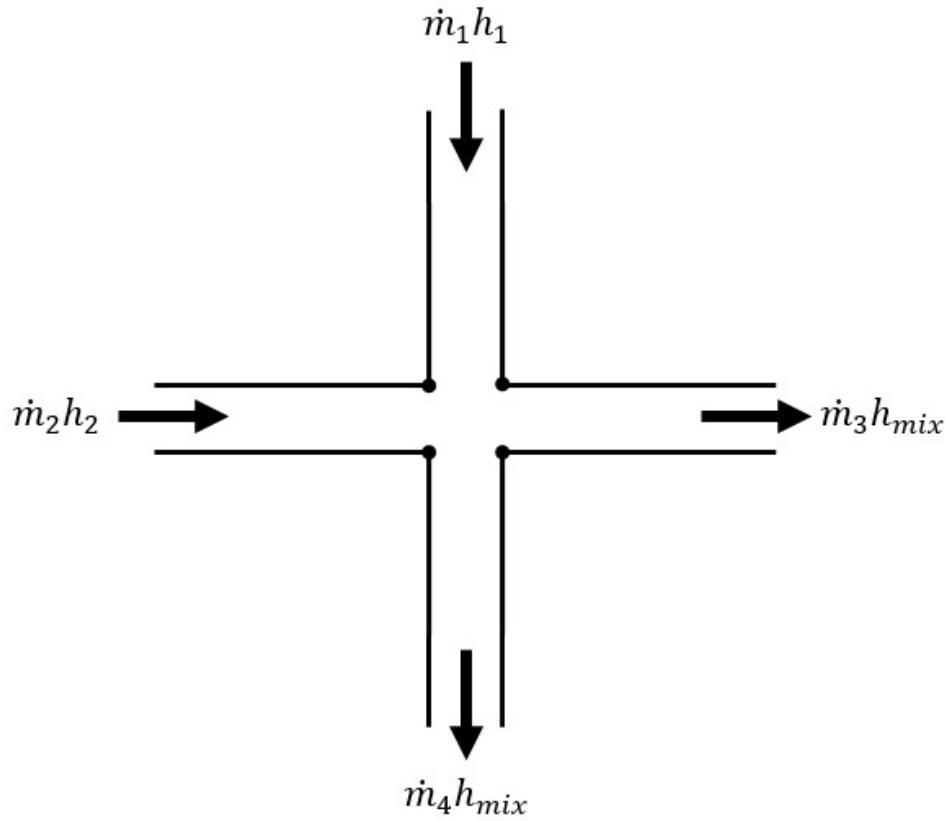
Figure 1: Flow weighted mixing and energy balance

Energy Balances with AFT Standard fluids

The AFT Standard fluids do not have enthalpy data, so Fathom uses the Reference Enthalpy Method, described in the following equation, to calculate enthalpy:

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

Therefore Equation 3 can be expressed as the following:

$$h_{mix} = \frac{\sum_{j=1}^{n} m_j^+ [href + c_{p,avg,j}(Tj - Tref)]}{\sum_{j=1}^{n} m_j^+} \tag{4}$$

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.

Chempak and NIST REFPROP fluids have enthalpy data and Equation 3 can be used directly.

Related Topics

Verifying Network Solutions

As discussed previously, AFT Fathom uses a Newton-Raphson matrix method to obtain a system level mass balance and energy. When the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) is finished, the Continuity Equation and Energy Equation are satisfied at all branching sections. Once solved, it is a simple matter to go back and sum up all mass flows and energy at each branch to verify that these equations are satisfied. In fact, AFT Fathom does just that.

After the Solver converges, AFT Fathom loops over each junction and adds up the inflow and outflow of mass and energy and compares the sum to the solution tolerances. If it appears that any junction is out of tolerance, a warning is given in the output.

In the Solution Balance Summary table AFT Fathom provides additional information on the balance at each junction. This table can be displayed by selecting the option in the Output Control window General tab. Figure 1 shows an example of the Solution Balance Summary table.

Figure 1: Solution Balance Summary table shown in the Output window offers a balance report of all junctions in the window

Related Topics

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/API 610 11th Edition - Centrifugal Pumps for Petroleum, Petrochemical and Natural Gas Industries*, 2010, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA

3. *ANSI/HI Standard 6.1-6.5-2015*, American National Standard for Reciprocating Power Pumps for Nomenclature, Definitions, Application, and Operation, 2015, published by American National Standards Institute, 6 Campus Drive, Parsippany, NJ, USA.

4. *ANSI/HI Standard 9.6.3-2017*, Rotodynamic Pumps - Guidelines for Operating Regions, 2017, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA

5. *ANSI/HI Standard 9.6.7-2021*, Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance, 2021, published by Hydraulic Institute, 300 Interpace Parkway, Building A, 3rd Floor, Parsippany, New Jersey, USA.

6. *ANSI/HI Standard 9.8-1998*, American National Standard for Pump Intake Design, 1998, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA

7. *ANSI/HI Standard 12.1-12.6-2016*, Rotodynamic (Centrifugal) Slurry Pumps, 2016, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA

8. 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.

9. Bloch, H. P., & Geitner, F. K. (1994). *An Introduction to Machinery Reliability Assessment*. Houston, Texas: Gulf Publishing Co.

10. Brock, Pat D., *Fire Protection Hydraulics and Water Supply Analysis*, Third Edition, Fire Protection Publications Oklahoma State University, Copyright 2012 by the Board of Regents, Oklahoma State University.

11. Crane Co., *Flow of Fluids Through Valves, Fittings, and Pipe*, Technical Paper No. 410, Crane Co., Joliet, IL, 1988.

12. Darby, R., *Chemical Engineering Fluid Mechanics*, 2nd edition, Marcel Dekker, New York, NY, 2001.

13. Dezhong, L., *Flow Status Identification and Hydraulic Calculation of Tailings Slurry Pipeline*, China Minmetals, MCC China ENFI, pp 71-83, No Date.

14. Durand, R., & Condolios, E., "Etude experimentale du refoulement des materieaux en conduites enparticulier des produits de dragage et des schlamms". *Deuxiemes Journees de l'Hydraulique*, 27-55, 1952.

15. Eckert, E.R.G., and Drake, R., *Analysis of Heat and Mass Transfer*, McGraw-Hill, New York, NY, 1972.

16. Garcia, E. and Steffe, J., "Comparison of Friction Factor Equations for Non-Newtonian Fluids in Pipe Flow," *Journal of Food Process Engineering*, Vol. 9, pp. 93-120, April 1986. [Online]. Available: https://doi.org/10.1111/j.1745-4530.1986.tb00120.x

17. Hanks, R.W. and Dadia, B.H., Theoretical analysis of the turbulent flow of non-newtonian slurries in pipes, *AIChE J.*, 17: 554-557, 1971. [Online]. Available: https://doi.org/10.1002/aic.690170314.

18. Idelchik, I. E., *Handbook of Hydraulic Resistance*, 4th edition, Begell House, Redding, CT, 2007.

19. Ingersoll-Dresser Pumps, *Cameron Hydraulic Data*, Liberty Corner, NJ, 1995.

20. Ito, H., "Friction factors for pipe flow," *Journal of Basic Engineering*, Vol. 81, pp. 123-126, 1959.

21. Jeppson, R. W., *Analysis of Flow in Pipe Networks*, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.

22. Karassik I., Krutzsch W., Fraser W. and Messina J., *Pump Handbook*, 2nd edition, McGraw-Hill, New York, NY, 1986.

23. Miller, D. S., *Internal Flow Systems,* 2nd edition, Gulf Publishing Company, Houston, TX, 1990.

24. Mullineux, G., "Non-linear least squares fitting of coefficients in the Herschel-Bulkley model," *Applied Mathematics Modeling*, Vol. 32, Issue 12, pp. 2538-2551, Dec. 2008. [Online]. Available: https://www.sciencedirect.com/science/article/pii/S0307904X07002235

25. Nayyar, M. L., *Piping Handbook*, 7th edition, McGraw-Hill, New York, NY, 2000.

26. NFPA 13, *Installation of Sprinkler Systems*, 1999 Edition, page 13-139.

27. NFPA 15, *Standard for Water Spray Fixed Systems for Fire Protection*, 2012 Edition.

28. *Pipe Line Rules of Thumb Handbook*, Gulf Publishing, Houston, TX.

29. TAPPI, *Generalized method for determining the pipe friction loss of flowing pulp suspensions*, TAPPI, TIS 0410-14, Atlanta, GA, 1988.

30. Val-Matic Valve and Manufacturing Corp., *Surge Control in Pumping Systems*, Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.

31. Visintainer, Robert. "GIW 4-Component Model Summary 2022-09-03." GIW Industries Inc., 2022. [Online]. Available: https://www.ksb.com/en-us/services/service-for-slurry-pumps/hydraulic-lab

32. Walker, C. I., *The effect of settling slurries on pump performance*, undated, Warman International.

33. Wasp, E. J., Kenny, J. P., & Gandhi, R. L., *Solid liquid flow slurry pipeline transportation*, Transactions Technical Publications, 1977.

34. Wilson, K.C., G.R. Addie, A. Sellgren and R. Clift, *Slurry Transport Using Centrifugal Pumps*, 3rd Ed., published by Springer, 2006.

35. Wilson, K.C., "*Effect of Solids Concentration on Deposit Velocity*", Journal of Pipelines, 5, 1986, pp. 251-257.

36. D. Wood, T.W Walters, "Operational problems in pumping non-settling slurries resolved using an improved laminar pipe fitting loss model" in *28th Intern. Pump Users Symp.*, Houston Texas, Turbomachinery Laboratory, Texas A&M University, 2012

37. Xiangjun, Fei, *Hydraulics of Slurry and Granular Material Transportation*, Tsinghua University Chuxin Society, 1994.

38. Zipparro, V. J., and H, Hasen, editors, *Davis' Handbook of Applied Hydraulics*, Fourth Edition, McGraw-Hill. New York, NY, 1993.

Chempak Technical Information - Detailed Discussion

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 sub-stances, 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

$$w = -\log_{10}(Pvr \text{ at } Tr = 0.7) - 1$$

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 Tc, Pc, Vc and Tf where no experimental data or other predictive method was available.

$$Tc = Tb/(0.584 + 0.965 \, Sum(Dt) - Sum(Dt)2)$$

$$Pc = 1/(0.113 + 0.0032 \, na - Sum(Dp))2$$

$$Vc = 17.5 + Sum(Dv)$$

$$Tf = 122 + Sum(Df)$$

where na 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,

$$Vb = a \, Vcn$$

$$a = 0.285$$

$$n = 1.048$$

8.3 Liquid Specific Volume

Liquid specific volume rises slowly and approximately linearly with rise in temperature to about Tr = 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,

$$Vs/V^* = Vr(O)(1 - wsrkVr(1))$$

$$Vr(O) = Sum\{an(1 - Trn/3)\} \qquad 0.25 < Tr < 0.95$$

$$Vr(1) = Sum\{bnTrn/(Tr - 1.00001)\} \qquad 0.25 < Tr < 1.0$$

$$aO = 1$$

$$a1 = -1.52816$$

a2 = 1.43907

a3 = -0.81446

a4 = 0.190454

bO = -0.296123

b1 = 0.386914

b2 = -0.0427258

b3 = -.0480645

V*, wsrk and Tc 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 (Vref at Tref) is available then

Zra = (PcVref / RTc)n

n = 1/(1 + (1 - Tref / Tc)m)

m = 2/7 or other empirical constant

The saturated specific volume is given by,

Vs = VrefZrax

with

x = -(1 - Tref / Tc)m + (1 - T/Tc)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 Zra. 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 > Ps + 0.1 Pc

The correction is derived from the equation of state as follows,

VL = VLs - VLs,es + VL,es

VL = specific volume at T and P

VLs = VL at T and Ps from methods of this section

VLs,es = VL at T and Ps from the equation of state

VL,es = VL 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

$(CpL - Cpo)/R = 1.45 + 0.45/X + 0.25w(17.11 + 25.2X0.333/Tr + 1.742/X)$

$X = 1 - Tr$

w = acentric factor

This method is generally applicable to the range from Tf to values approaching Tc. Note that CpL approaches infinity as T approaches Tc.

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

$CpL = a + bT + cT2$

The accuracy is usually better than 5%. The method cannot deal with double bonds and is not applicable for Tr > 0.75

$a = Sum\{an\}$

$b = Sum\{bn\}$

$c = Sum\{cn\}$

The group contributions are available in Missenard. See also Reid at 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.

$CpLs = CpL$ at Ts and Ps determined by the methods of this section.

Cpo = ideal gas specific heat at Ts

$CpL = CpL$ at Ts and P > Ps

The equations of state give estimates of

$Ds = (CpLs - Cpo)es$ at Ts and Ps

$D = (CpL - Cpo)es$ at Ts and P

The corrected value of the liquid specific heat is

$CpL = CpLs + D - Ds$

The correction is not applied when T is close to Tc

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 Tr = 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 Tf to Tb

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 Tf to Tb

· No explicit treatment for heterocyclics or esters (apart from acetates).

The Morris method takes the following form

$$\ln(v/v^*) = 2.3026 \, J(1/T_r - 1)$$

$$J = (0.577 + Sum(D_i))0.5$$

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

$$v = (f0 + w.f1)/A$$

with

$$w = \text{acentric factor}$$

$$f0 = a0 + b0T_r + c0T_r2$$

$$f1 = a1 + b1T_r + c1T_r2$$

$$A = 0.176x106 \; T_c0.1667/M0.5P_c0.667$$

a0 = 2.648	a1 = 7.425
b0 = -3.725	b1 = -13.39
c0 = 1.309	c1 = 5.933

In the above relations Pc is in bar and the viscosity is in units of Pa-sec.

8.5.4 Method of Przezdziecki & Sridhar

In this method, the viscosity is related to changes in the specific volume.

$v = V0/E(V - V0)$ centipoise

V = liquid molar volume in cc/mol

$E = -1.12 + Vc/D$

$D = 12.94 + 0.1 M - 0.23 Pc + 0.0424 Tf - 11.58 Tf/Tc$

$V0 = 0.0085 wTc - 2.02 + Vf / \{0.342(Tf / Tc) + 0.894\}$

with

Tc = critical temperature, K

Pc = critical pressure, bar

Vc = critical volume, cc/mol

M = molecular weight

Tf = freezing point, K

w = acentric factor

Vf = specific volume at Tf

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 Tr 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:

$273 < T < 0.6 Tc$: this region is normally covered by published data or by one of the methods 8.5.1, 8.5.2, 8.5.4

$0.76 Tc < T < Tc$: 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

$Tf < T < 273$: 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.

$0.6 Tc < T < 0.76 Tc$: this region may be covered by interpolation between the $273 < T < 0.6 Tc$ region and the $0.76 Tc < T < Tc$ 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:

$vL/vsL = (1 + B.FA)/(1 + w.C.F)$

vL = viscosity at pressure P

vsL = viscosity at saturation pressure Ps

F = (P - Ps)/Pc

w = acentric factor

A = 0.9991 - 0.0004674/(1.0523/Tr0.03877 - 1.0513)

B = 0.3257/(1.0039 - Tr2.573)0.2906 - 0.2086

C = -0.07921 + 2.1616 Tr - 13.404 Tr2 + 44.1706 Tr3 -84.8291 Tr4 + 96.1209 Tr5 - 59.8127 Tr6 + 15.6719 Tr7

8.6 Liquid Thermal Conductivity

8.6.1 Method of Latini at 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/Tr0.167

A = A0TbnMmTcp

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/M0.5)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:

$$k/ks = 0.98 + 0.0079 \, PrTr1.4 + 0.63 \, Tr1.2Pr/(30 + Pr)$$

k = conductivity at P

ks = conductivity at Ps

8.8 Vapor Pressure

The vapor pressure is expressed in its reduced form

$$Pvr = Pv/Pc$$

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

$$\ln(Pvr) = (aX + bX1.5 + cX3 + dX6)/Tr$$

with

$$X = 1 - Tr$$

FKT Equation

$$\ln(Pv) = a + b/T + c\ln(T) + dPv /T2$$

Antoine Equation

$$\ln(Pv) = a + b/(T + c)$$

8.8.2 Gomez-Thodos Vapor Pressure Equation

Gomez-Nieto and Thodos give the following equation:

$$\ln(Pvr) = B(1/Trm - 1) + G(Tr7 - 1)$$

$$G = aH + bB$$

$$a = (1 - 1/Tbr)/(Tbr7 - 1)$$

$$b = (1 - 1/Tbrm)/(Tbr7 - 1)$$

$$H = Tbr\ln(Pc/Pb)/(1 - Tbr)$$

For non-polar compounds,

$$B = -4.267 - 221.79/(H2.5exp(0.038\ H2.5)) + 3.8126/exp(2272.33/H3) + D$$

$$m = 0.78425\ exp(0.089315\ H) - 8.5217/exp(0.74826\ H)$$

$$D = 0$$

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),

$$m = 0.466\ Tc0.1667$$

$$G = 0.08594\ exp(0.0007462\ Tc)$$

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

8.8.2 Gomez-Thodos Vapor Pressure Equations

For polar hydrogen-bonding compounds (water, alcohols),

$$m = 0.0052\ M0.29Tc0.72$$

$$G = (2.464.M)\ exp(0.0000098\ MTc)$$

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

The advantages of this method are,

· fit guaranteed at T = Tb and T = Tc

· good performance with polar compounds

· good performance over Tr = 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:

$$ln(Pvr) = f(0) + wf(1)$$

w = acentric factor

$$f(0) = 5.92714 - 6.09648/Tr - 1.28862\ ln(Tr) + 0.169347\ Tr6$$

$$f(1) = 15.2518 - 15.6875/Tr - 13.4721\ ln(Tr) + 0.43577\ Tr6$$

The characteristics of this equation are,

· guaranteed fit at Tr = 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 at al, Properties of Liquids and Gases, 3rd Ed, McGraw Hill, NY 1977, 4th Ed, McGraw Hill, NY 1987

Schick and Prausnitz, AIChEJ, Vol 14, 673, 1968

Spencer and Danner, J Chem Eng Data, Vol 17, 236, 1972

Spencer, Daubert and Danner, AIChEJ, Vol 19, 522, 1973

Stiel and Thodos, AIChEJ, Vol 8, 229, 1962

Teja A S at al, Chem Eng Sci, Vol 33, 609, 1978, AIChEJ, Vol 26, 337 & 341, 1980, Chem Eng Sci, Vol 36, 7, 1981, Ind Eng Chem Fundam, Vol 20, 77, 1981, Chem Eng Sci, Vol 37, 790, 1982, J Chem Eng Data, Vol 28, 83, 1983, Ind Eng Chem Proc Des Dev, Vol 22, 672, 1983

Thomson, Brobst, Hankinson, AIChEJ, Vol 28, 671, 1982

Tyn M T and Calus W F, Processing, Vol21(4), 16, 1975

Van Velzen D et al, Ind Eng Chem Fundam, Vol 11, 20, 1972

Van Velzen et al, Liquid Viscosity etc, Euratom 4735e, Joint Nuclear Research Centre, ISPRA Establishment, Italy 1972

Vargaftik N B, Tables on Therm Props Liq & Gases, 2nd Ed, Hemisphere, Washington DC, 1975

Wu G Z A and Stiel L I, AIChEJ, Vol 31, 1632, 1985

Yorizane et al, Ind Eng Chem Fundam, Vol 22, 454, 1983

Role of Pressure Junctions - Detailed Discussion

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.

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 Fathom, they do not need to think at the same depth as in the hand calculation. So when AFT Fathom 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 Fathom 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 Fathom and if you try and run this model, AFT Fathom will inform you that it cannot run it because of the lack of a reference pressure. In AFT Fathom 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 Fathom 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 Pump. It is possible the user may not know either the flow or the pressure at the pump. In such cases the user is required to specify the relationship between flow and pressure. This relationship is called a pump curve, and by specifying the relationship between the two, effectively one of the two unknowns can be eliminated. Thus we always end up with one unknown for each junction.

Second, if the user was allowed to specify two flows as in Figure 1, they could specify them with different flow rates. Both ends of the pipe 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 alternate configurations for the model in Figure 1. 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.
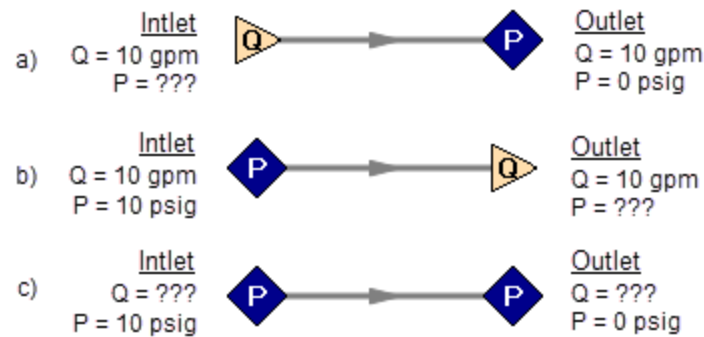
Figure 2a-c: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.

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 at least one pressure a unique solution exists.
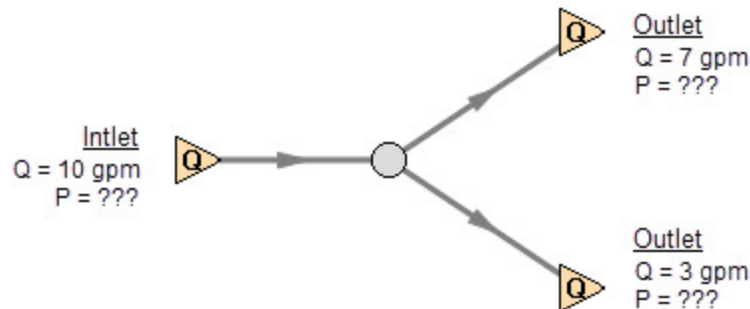

Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.

Now consider the models in Figure 4. The Figure 4a model does have one reference pressure, but in this case there is still a problem. The Flow Control Valve (FCV) is controlling the flow to 10 gpm, and the downstream flow is demanding 10 gpm. The section of the system before the FCV can be solved (because there is a pressure upstream), but the section after the FCV does not have a unique solution. Why? In order to obtain a unique solution, the pressure drop across the FCV must be known. But any pressure drop could exist and satisfy the conditions of this model. It is thus not possible to determine the pressure at the outlet flow demand because it depends on the FCV pressure drop which is not known. The model in 4b does have a pressure downstream of the FCV, and thus there is a unique pressure drop across the FCV and a unique solution exists for the model in Figure 4b.

If you input the remaining data for the Figure 4a model and run it in AFT Fathom, you will get the message shown in Figure 5. AFT Fathom identifies the part(s) of the model where a known pressure is needed.
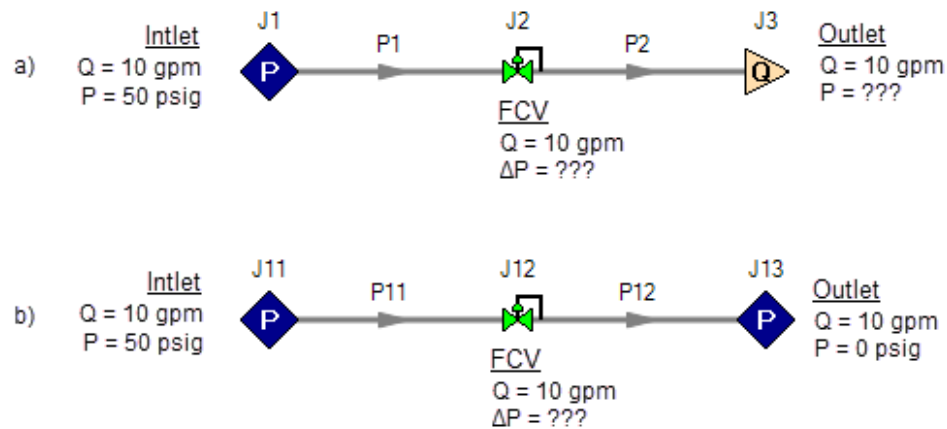
Figure 4a-b: The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists
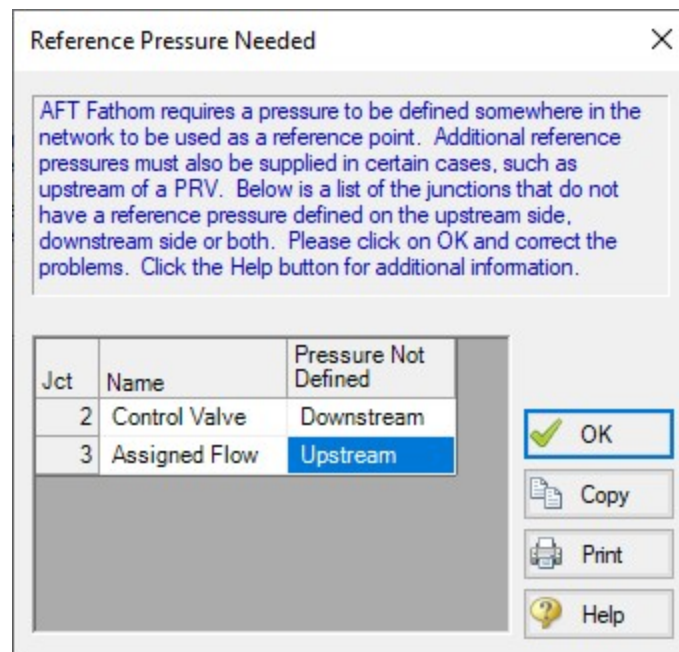
Figure 5: AFT Fathom message when you try to run the model shown in Figure 4a

Sizing Pumps with Flow Control Valves

An analogous situation to Figure 4a is when the user is trying to size a pump using the Pump with a fixed flow. The Pump 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 pump curve for this model, it could be solved.

Figure 6: A pump modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists
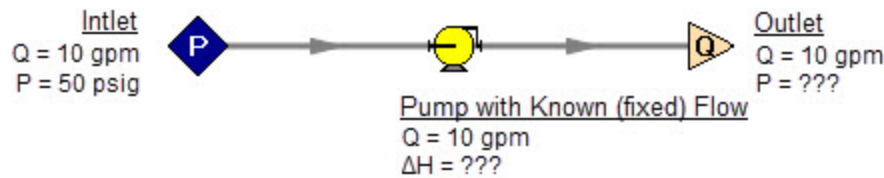
Increasing in model complexity, Figure 7 shows some additional examples of models without a unique solution. In particular, Figure 7b has a pump with a fixed flow that can add as much pressure as it wants, and the downstream FCV can then take out as much as it wants. There are an infinite number of possible 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 pump. How does one size the pump in such a case? There are several ways to do this, one of which is to change the pump from a fixed flow to an actual pump curve. Trying multiple pump curves will guide you to the best pump.

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 pump that results in the pressure drop across the FCV being 0.01 psid. If the system is built and the pump slightly underperforms its published curve, the FCV will not be able to control to its flow control point. Even if the pump 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.
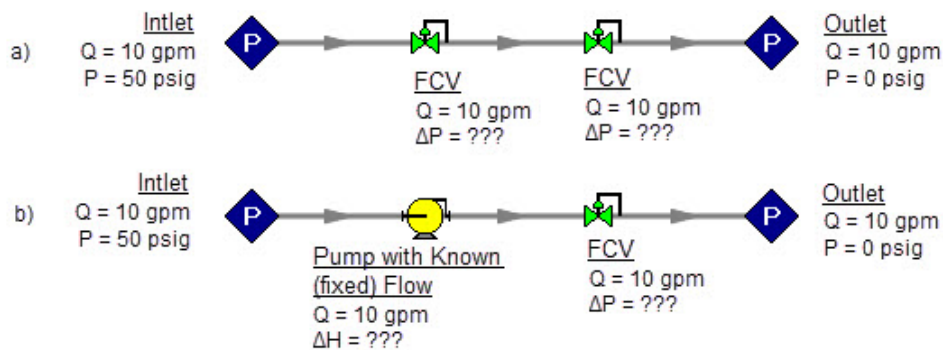

Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a pump modeled as a fixed flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions must 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 pump 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 pump can be sized. Once the pump is sized and an actual pump curve exists, the pump curve can be entered for the pump 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 pump 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 pump such that the FCV pressure drop is minimized. If the FCV has a larger pressure drop than the minimum, the pump must pump harder to overcome this excess pressure drop. In short, it must use more energy and is thus less efficient. Figure 8 depicts this process.
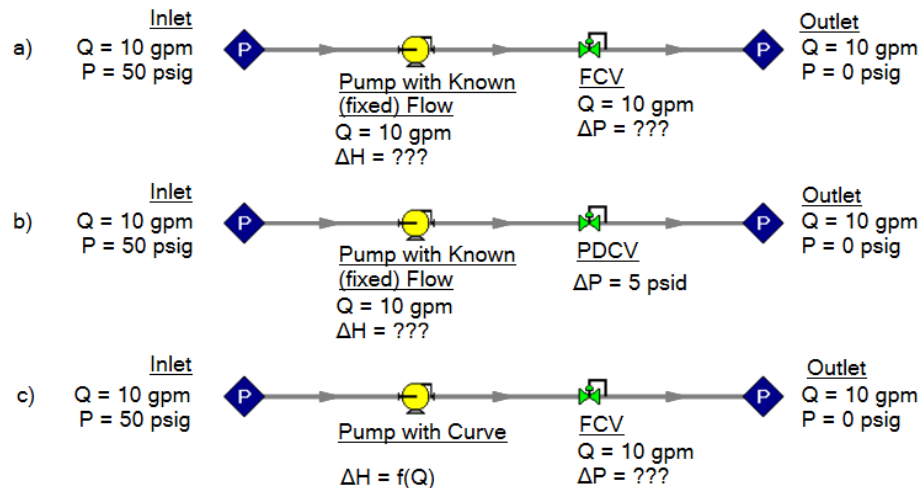
Figure 8a-c: The model at the top is the same as Figure 7b and does not have a unique solution. To size the pump, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the pump is sized, a pump curve exists, and 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 pump, we apply the process described in Figure 8. We choose one of the FCV's, make it a PDCV, size the pump, choose a pump with an actual pump curve and enter it into the model, then return the PDCV to an FCV.

Figure 9: A pump 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 pump, first change the FCV at J7 to a PDCV.

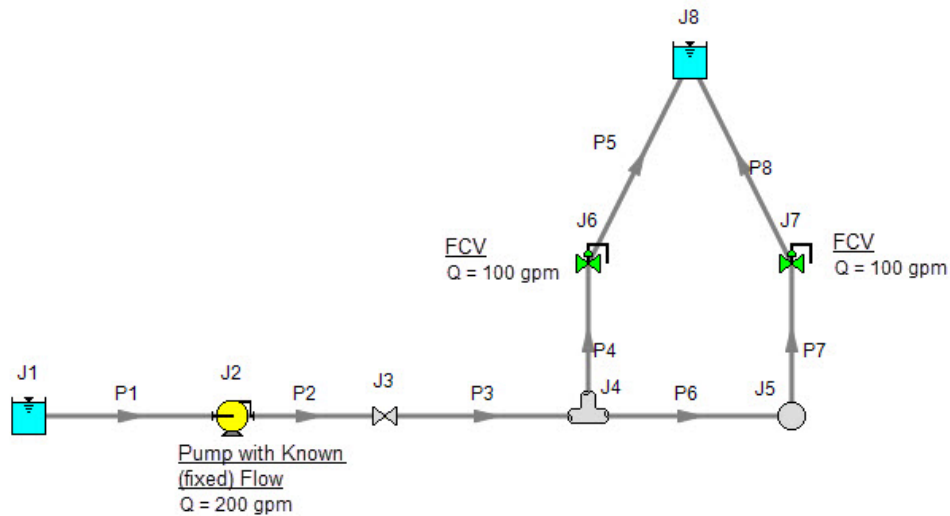This brings up the question, which FCV should one choose to turn into a PDCV? When FCV's are put in parallel, frequently the pipe design has one of the FCV's further away from the pump than the others. Because of the additional piping leading to this FCV, it will be the weakest link in the chain of parallel 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 pump 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 (6.837 psid) is greater than 5 psid because it is closer to the pump. Also notice how the flow through J7 is still 100 gpm even though it is not controlling flow. Once again, the reason for this is that the pump is controlling the flow to 200 gpm, and with the J6 FCV controlling to 100 gpm, the excess must go through J7. The pump is thus sized and its pump pressure rise is shown in Figure 10b (127 ft.).
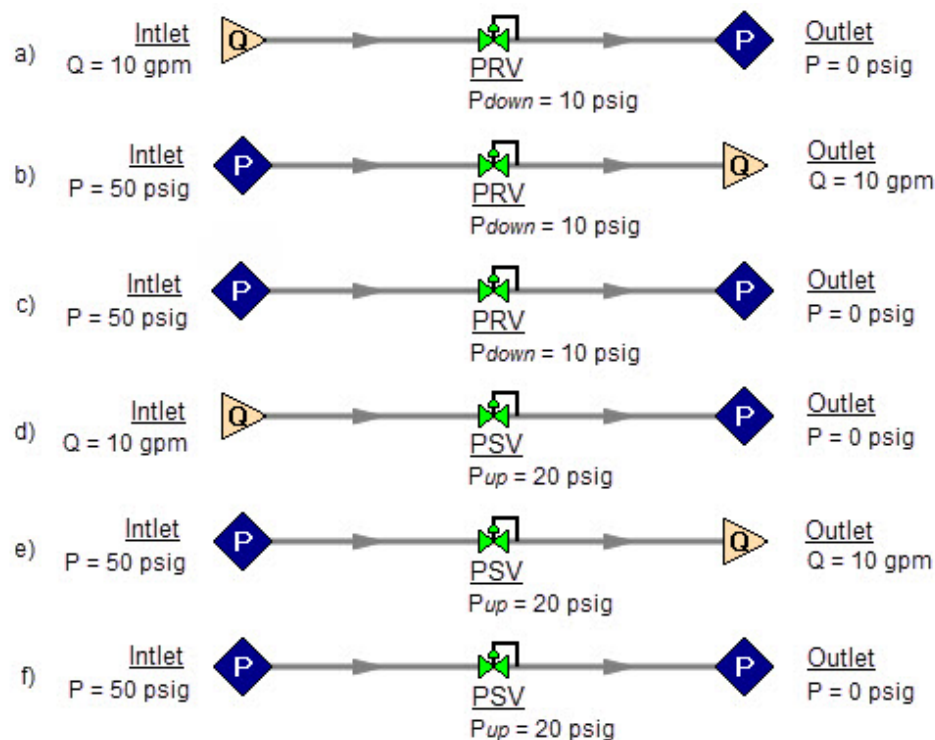
Jct	Name	Valve Type	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP Stag. (psid)	dH (feet)	P Static In (psia)	Cv	K	Valve State
10	Valve	REGULAR	200.0	27.76	1.229	2.841	66.33	180.28	0.5000	Open
4	Control Valve	FCV	100.0	13.88	6.489	14.997	55.34	39.23	10.5569	Open
7	Control Valve	PDCV	100.0	13.88	5.000	11.555	53.85	44.70	8.1340	Open

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)
2	Show ...	Pump	200.0	27.76	54.78	126.6	100.0	N/A	6.390	N/A	N/A	48.04	N/A

Figure 10a-b: Results from changing J7 in Figure 9 to pressure drop control valve

Using Fixed Head Rise Pumps

An alternate method of sizing pumps with FCV's is to use the fixed head rise pump model. Here the pump head can be adjusted while the flow control valves can be modeled as FCV's. This approach is particularly powerful when used with the GSC Module. This would entail specifying the pump head rise as the variable at the min Cv or pressure drop across the control valve as the goal. If there are multiple, parallel control valves, then a group max/min can be used.

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.

Intlet
Q = 10 gpm

PRV
Pdown = 10 psig

Outlet
P = 0 psig

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 pump is being sized in series with a PRV or PSV, modeling the pump as a fixed flow will not permit a unique solution if the pump 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.
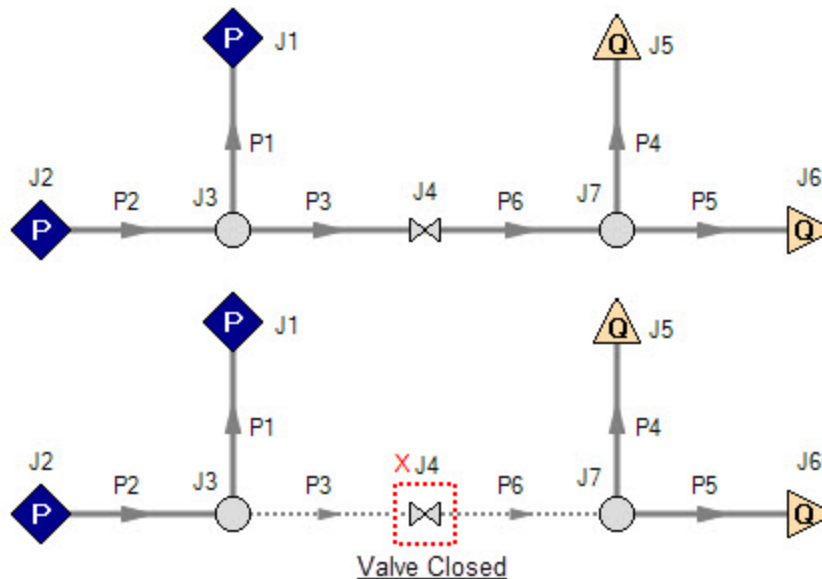
Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.

Summary

AFT Fathom models always require one reference pressure, and may require more if the engineer uses control valves, pumps with fixed flows, or closed pipes or junctions. The actual number of reference pressures needed depends on the model, but AFT Fathom 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

Pipe Material Library References

Prior to AFT Fathom 9, pipe material properties were compiled from a variety of unofficial sources. Many of the original sources are now obsolete. Beginning in AFT Fathom 9, 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 Fathom 7 is still available, but is now called "AFT Customary".

Maintaining several independent pipe libraries facilitates including or excluding specific standards. For example: a pipe engineer in Germany may prefer to use only ISO and DIN standards. The Library 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, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-33
- **Copper Tubing - ASTM -**
  - Source = ASTM B88-09
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-31; www.cranecopper.com.au/sizes/astmb88.asp
- **Ductile Iron - ANSI -**
  - Source = ANSI-AWWA C150-A21.50-08
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Fiberglass - Green Thread -**
  - Source = Fiber Glass Systems product data, http://www.nov.com/Segments/Completion_and_Production_Solutions/Fiber_Glass_Systems/Chemical_Industrial/Green_Thread.aspx

- Roughness References = Fiber Glass Systems product data
- Density References = Average value calculated based on ID, OD and Weight in Fiber Glass Systems product data
- Thermal Conductivity References = Fiber Glass Systems product data
- Modulus of Elasticity References = Fiber Glass Systems product data
- Poisson's Ratio References = Fiber Glass Systems product data
- Additional References = ASTM D2996 - 17
- **HDPE - AFT Customary -**
  - Source = Applied Flow Technology
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **PVC - ASTM -**
  - Source = ASTM D1785-06
  - Additional References = Thermal data from: Materials Science and Engineering, Callister, Table C.4
- **PVDF - AFT Customary -**
  - Source = George Fischer Engineering handbook page 425
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Rectangular Duct - AFT Customary -**
  - Source = Applied Flow Technology
- **Stainless Steel - ANSI -**
  - Source = ASME-B36_19M_2004
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conduct References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Type 304
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ration Referenecs = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for Type 304 @ 20oC
- **Stainless Tubing - AFT Customary -**
  - Source = Applied Flow Technology
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Steel - AFT Customary -**
  - Source = Applied Flow Technology
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3

- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1

- **Steel - ANSI -**
  - Source = ASME B36.10-2004
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - DIN 2391-1 -**
  - Source = DIN 2391-1-1994
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - DIN 2458 -**
  - Source = DIN 2458-1981
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - EN 10216-1 (s1) -**

- Source = BS EN 10216-1-2002
- Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
- Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
- Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - EN 10216-2 (s1) -**
  - Source = BS EN 10216-2-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - EN 10217-1 (s1) -**
  - Source = BS EN 10217-1-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - EN 10217-2 (s1) -**
  - Source = BS EN 10217-2-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3

- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
- Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-4 (s1) -**
  - Source = BS EN 10217-4-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-5 (s1) -**
  - Source = BS EN 10217-5-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
  - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
  - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
  - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
  - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
  - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-6 (s1) -**
  - Source = BS EN 10217-6-2002
  - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for 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

Related Topics

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

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 Fathom

Full Message Text

Several variations are possible depending on which Valve Type is selected in the Control Valve Properties window:

- FCV could not control flow at Junction X - Pressure was added to achieve control flow. See Valve Summary for more information.

- PRV could not control pressure at Junction X - Reverse flow through valve, pressure was added to achieve control pressure. See Valve Summary for more information.

- PSV could not control pressure at Junction X - Pressure was added to achieve control pressure. See Valve Summary for more information.

Explanation

The Control Valve could not control to it's set point and the Always Control (Never Fail) setting is enabled in the Control Valve Properties window. When the Always Control setting is enabled the program has the freedom to add pressure in order to make the solution mathematically possible. Then, the Valve Summary in the Output window gives more information about exactly how much pressure was added via the Pressure Drop Stagnation (dP) or Head Loss (dH) parameters. A negative dP or dH means it was a pressure rise. Normally you would see a positive dP or dH meaning it was a pressure loss. Furthermore, the Valve State output parameter in the Valve Summary will show "Failed - Had to add pressure".

Required Action

If you want the Control Valve to fail open or closed, then you can disable the Always Control setting in the Control Valve Properties window. Whether the valve fails open or closed is dictated by the Use Default Actions setting and related options below it.

If you intended for the Control Valve to actively control, then this critical warning identifies that the pressure is too low at this location and the Control Valve cannot meet the set point. This could indicate a mis-typed input or an operational problem. Double-check the inputs and corresponding units for the model including control valve settings, pumps, pipe sizes, and fluid properties. If there are no input typos, then there are many different conclusions that could be drawn, each one requiring a different solution approach that will depend on your unique system: undersized pump, higher than expected junction losses or K Factors, higher than expected pipe

friction, pipe fouling changing the internal diameter/velocity, or Laminar and ATKF corrections causing higher losses.

Global energy balance disabled by user in Advanced Parameters Panel

A Critical Warning message in AFT Fathom

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

Global mass balance disabled by user in Advanced Parameters Panel

A Critical Warning message in AFT Fathom

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

Junction X had reverse flow - Pump head could not be predicted

A Critical Warning message in AFT Fathom

This message displays for centrifugal pumps when the flowrate goes backwards through the pump. The pump head rise under such conditions is unknown. The dead head value is used.

Pipe sequences of zero friction exist, flow rate cannot be calculated

A Critical Warning message in AFT Fathom

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 hydraulic calculations.

Setting relaxation lower than relative pressure/flow tolerance gives unreliable results

A [Critical Warning message](#) in AFT Fathom

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 Fathom alerts the user to this when inputting data in the [Tolerance panel](#) or [Iterations panel](#) in the [Analysis Setup window](#), and also alerts the user once the solution has been obtained.

There were static pressures below absolute zero in the model

A [Critical Warning message](#) in AFT Fathom

Full Message Text

There were static pressures below absolute zero in the model. While this is mathematically correct, it is not physically realistic. Please check the model.

Explanation

The Solver is mathematically capable of calculating negative absolute pressures which satisfy the [fundamental equations](#), but this of course is not physically realistic. This message notifies the user of this unrealistic situation so that appropriate changes can be made to the model. If the model is based on a real functioning system, then all input data should be double checked. Furthermore, it may be possible that the real system is experiencing cavitation where the static pressure is dropping below the vapor pressure of the fluid at one or more locations. There could be other discrepancies such as worn components, pumps with lower efficiency, or fouling of components that affects the internal pipe areas, velocities, and pressure drops.

Required Action

The user must either correct the model such that this message is not displayed, or understand that all results are not physically possible and the current configuration of the model is not achievable in reality. Something in the model is causing the pressures to be too low which could be from any number of reasons:

- A pipe or junction is providing too much pressure drop.
- The flow demand from an Assigned Flow or Control Valve junction is too high causing too much friction loss.

- The boundary conditions do not provide enough pressure differential.
- Laminar corrections or Non-Newtonian viscosity models may be causing large pressure drops.

One of the best ways to start diagnosing the root cause is to review the Model Data. If all inputs for Analysis Setup, pipes, and junctions has been verified, it is recommended to make various Profile Graphs to visualize how the pressure changes throughout the various flow paths of the model. You may find a specific junction or pipe that is causing the pressure to drop below absolute zero.

These results do not represent a converged solution

A Critical Warning message in AFT Fathom

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.

Related Topics

Weir Liquid Level Convergence Problem

A Critical Warning message in AFT Fathom

This message indicates that the flow required at the weir to maintain flow continuity in the system is higher than the maximum flow the weir can deliver. It is recommended to review the system and weir inputs to resolve the issue.

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

Cavitation May Exist at Junction X

A Warning message in AFT Fathom

The local static pressure at this junction was obtained using the Restricted Area, and it is below the vapor pressure of the fluid.

Chempak temperature and or pressure is out of range for the following

A Warning message in AFT Fathom

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.

Control valve open percentage outside valid range of 0-100% at Junction X

A Warning message in AFT Fathom

This is a cautionary notice that a control valve which includes open percentage data is operating outside the range of 0-100%. This could indicate an error in input, or an improperly sized valve.

Energy may not be balanced at Junction X (total= Y)

A Warning message in AFT Fathom

This warning alerts you to the fact that the converged energy balance results may be incorrect. Once a converged solution is obtained, AFT Fathom 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 tighten 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 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 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.

Related Topics

FCV could not control flow at Junction X

A [Warning message](#) in AFT Fathom

This warning alerts you that the [Flow Control Valve](#) at junction X [was not able to control the flow](#), and backflow may have occurred.

Flow is into system at Reservoir Junction X

A [Warning message](#) in AFT Fathom

Full message is:

"Flow is into system at Reservoir Junction X. Pipe intake elevation above the surface will supply gas, not liquid."

When a pipe exits above a [reservoir](#) junction liquid level it is only meaningful if it is discharging into the reservoir. If the pipe becomes an intake, it will not supply liquid but whatever vapor is above the reservoir liquid. This warning alerts you to the fact that the reservoir had a pipe above liquid level that was taking in fluid. AFT Fathom assumes the intake fluid is the same as the reservoir fluid, but that is likely to not be the case. Results are likely invalid.

Flow may not be balanced at Junction X (total= Y)

A [Warning message](#) in AFT Fathom

This warning alerts you to the fact that the converged results may be incorrect. Once a converged solution is obtained, AFT Fathom 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 tighten 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, 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.

Related Topics

Flow rate at drop off point at Junction X

A [Warning message](#) in AFT Fathom

This warning is for [jet pumps](#), and informs the user that the flow rate is at the edge of the curve. Results may not be valid.

GSC Goal may not be valid. User goal = X. Final goal result = Y.

A [Warning message](#) in AFT Fathom

([GSC Module Only](#))

This is a warning message that appears when the [GSC module](#) search was not able to meet a [goal](#). The solution may still be acceptable if the final goal result is reasonably close to the goal.

HGL, EGL and head loss results may not be meaningful for variable density systems

A [Warning message](#) in AFT Fathom

The concepts of HGL (Hydraulic Gradeline) and EGL (Energy Gradeline) are only valid for constant density systems. This warning is given when HGL, EGL or head loss results are displayed in the output and the system has [variable density](#). It alerts the user that HGL/EGL and head loss output may not be meaningful. Unless otherwise noted, all other output is valid.

This warning can be hidden by setting all head results to not be shown using [Output Control](#).

Junction X %BEP (Y%) is above/below the POR range

A [Warning message](#) in AFT Fathom

The calculated %BEP of Pump X is outside of the POR range as defined in [Pump Configuration](#).

Junction X exceeded runout flow rate

A [Warning message](#) in AFT Fathom

[Centrifugal pump 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 pump is given by the manufacturer. Such data has been defined for this pump in the Pump Configuration window [Configuration Data tab](#), and the system is operating past the runout flow rate. Thus the results are invalid, because the pump likely cannot actually operate at that point.

Junction X flow (Y gal/min) is above/below the AOR range (Z gal/min)

A [Warning message](#) in AFT Fathom

The calculated flow of Pump X is outside of the AOR range as defined by the user in [Pump Configuration](#).

Junction X has insufficient pump NPSH/NPSP…

A [Warning message](#) in AFT Fathom

The available pump NPSH (or NPSP) is less than the required NPSH defined for the [pump](#). The pump will cavitate if changes are not made to increase the NPSH available at the pump.

Negative added pressure by pump at Junction X

A [Warning message](#) in AFT Fathom

Full Message text is:

"Negative added pressure by pump at Junction X."

A pump junction with a fixed flow resulted in a pressure loss at the [pump junction](#). This warning is meant to alert the user that the pump is not adding pressure as they may be expecting.

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes

A [Warning message](#) in AFT Fathom

([SSL Module Only](#))

Full Message Text

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes. Results may be incorrect.

Explanation

This message can only appear when using the [Wasp-Durand Heterogenous Slurry](#) Model.

This message appears when the slope of a pipe is greater than 18 degrees. Pipes with angles between 8 and 18 degrees will display [this caution message instead](#). The Wasp-Durand model does not incorporate [pipe slope corrections](#). [Pipe Slope](#) can impact pressure gradients and deposition velocity. [Vertical conveyance effects](#) will not be captured in the model.

Required Action

If a system contains vertical or significantly-angled pipes, consider using the [detailed Wilson, Addie, Clift](#) or [4-Component](#) slurry models. In some cases, it may be possible to define the system boundaries to exclude inclined pipes, modeling only the horizontal elements of the piping system.

Pipe X temperature is equal to or greater/less than the maximum/minimum temperature limit of...

A Warning message in AFT Fathom

There are two variations of this Warning that are similar:

- 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 Temperature Solver Limits panel of the Steady Solution Control in Analysis Setup (only available when Heat Transfer With Energy Balance is enabled in Fluid Properties). These limits are intended to aid in convergence by restricting the search space while solving, and should generally be left as the AFT default values. This warning indicates that the final solution had one or more pipes outside the limit range, and the temperature results have been artificially limited by the Solver. The results are therefore not valid. To resolve this either remove or change the temperature limit.

Note: These limits should not be used as Design Alerts, or to approximate some physical phenomenon.

PRV could not control pressure at Junction X

A Warning message in AFT Fathom

This warning alerts you that the Pressure Reducing Valve at junction X was not able to control pressure, and backflow may have occurred.

PSV could not control pressure at Junction X

A Warning message in AFT Fathom

This warning alerts you that the Pressure Sustaining Valve at junction X was not able to control pressure, and backflow may have occurred.

Pump X had efficiency greater than 100%

A Warning message in AFT Fathom

Full Message Text

Steady-State:

Pump X had efficiency greater than 100%. This may be due to a density sufficiently greater than the basis of the power input data or an error in the power input data.

Transient with the XTS Module:

Pump X had efficiency greater than 100%. This may be due to a density sufficiently greater than the basis of the power input data or an error in the power input data. First time when efficiency exceeded 100% was (Y seconds)

Explanation

The listed pump had an efficiency over 100% which is mathematically possible, but not physically realistic.

Required Action

Check that the efficiency/power data is entered correctly in the Pump Configuration window, and that the correct selection is made for either efficiency or power in the dropdown header of the column. If power data is being used, a reference density may need to be entered to correct the data, see the Reference Density topic for more information.

If the Variable Speed option is used on the pump, check the control setpoint as it may not be achievable for the given system configuration.

Static pressure lower than vapor pressure at inlet/outlet of pipe/junction X

A Warning message in AFT Fathom

This warning alerts the users that the static pressure is lower than the vapor pressure in the specified pipe or junction and that cavitation may occur.

Tank Junction X began overflowing at Y seconds

A Warning message in AFT Fathom

The total volume of fluid entering the tank junction has exceeded the volume of the tank.

Tank Junction X completely drained at Y seconds

A Warning message in AFT Fathom

The Finite Tank junction has run out of fluid to supply to the system. The simulation will continue under the assumption that fluid is still being supplied at the inlet pipe. The results past the specified time should only be used to troubleshoot the system as in reality the pipe would pull in the vapor is currently residing in the tank.

Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range

A **Warning message** in AFT Fathom

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 an Assigned Pressure 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

Temperature is above the saturation line for the following...

A **Warning message** in AFT Fathom

The solution indicates that the temperature and pressure conditions at the specified pipe are such that one or more gas components has reached saturation conditions. This warning only occurs with Chempak fluids.

Unique solution not possible - Relief valve cycles

A Warning message in AFT Fathom

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.

Velocity in pipe X is less than the deposition velocity

A Warning message in AFT Fathom

(SSL Module Only)

Full Message Text

Steady-State:

Velocity in pipe W is less than the deposition velocity. Results may be inaccurate. (Velocity: Y, Vsm: Z)

Transient with the XTS Module:

Velocity in pipe W is less than the deposition velocity at time X. Results may be inaccurate past this time step. (Velocity: Y, Vsm: Z)

Explanation

Engineers often want to avoid operating settling slurry systems in such a way that the bulk slurry velocity (V_m) is less than the deposition/settling velocity (V_{sm}). This warning indicates that the velocity is in fact less than the deposition velocity which can cause operational problems including stationary beds or plugging of pipes. The methods available in the SSL Module cannot model stationary beds, and thus the results may not be accurate under these conditions. The farther the velocity drops below V_{sm} the more uncertainty in the results.

Required Action

Adjust the model such that the velocity is higher than the deposition velocity. If this is not possible, then it may indicate operational problems in the real system, or you must accept that the results will be incorrect.

Potential adjustments:

- Decrease the pipe size
- Increase the flowrate
- Reduce the slope of the pipe

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 Fathom

This is a cautionary notice that a check valve closed, and the Solver will automatically re-run the model.

Default control valve actions at Junction X used when setpoint not achievable because analysis is transient

A Caution message in AFT Fathom

(XTS Module Only)

The Always Control (Never Fail) option at a Control Valve was enabled by the user but not used in the solution. The Control Valve could not control to its setpoint and so default actions were used.

Flow is negative through Junction X. Loss factors may be in error.

A Caution message in AFT Fathom

This caution alerts you to flow in 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 max-imum/minimum value specified for the resistance curve. Flow rate: Y

A Caution message in AFT Fathom

There are two variations of this Caution that are similar:

- 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.

Head loss, head rise, hydraulic gradeline and energy gradeline output does not have meaning for zero g systems

A **Caution message** in AFT Fathom

If the user sets the gravitational acceleration to zero, presumably for a space application, then the concept of head and gradeline lose meaning. This can be set in Analysis Setup on the Environmental Properties panel. For example see Equation 1 in Network Solution Methodology and see that a zero g will lead to a meaningless HGL.

Users should ignore all head loss, head rise, and gradelines displayed in the output and focus on pressure results.

Head rise across fixed flow pumps is referenced to system g level...

A **Caution message** in AFT Fathom

Full message is:

"Head rise across fixed flow pumps is referenced to system g level. To obtain standard head rise, divide the displayed head rise by the g level."

If the user sets the gravity to something other than standard earth gravity in Analysis Setup on the Environmental Properties panel, then the head rise at a pump is referenced to that g level and not 1 g.

Junction X has a density change across it

A **Caution message** in AFT Fathom

Full Message Text

Junction X has a density change across it. If this is due to a moving interface and not heat input or output then a Volume Balance junction is recommended.

Explanation

This warning is shown when the user has selected Variable Fluid properties on the Heat Transfer/Variable Fluids panel, and the density of the fluid is changing between pipes. The Solver performs a mass balance at junctions, and fluids of different densities on either side of the junction may violate the mass conservation at a junction.

Required Action

If the change in density is due to a change in temperature across the junction, the caution message can likely be ignored. A density change at a junction due to temperature change means the real system likely still has a mass balance at the junction.

If the change in density is due to a change in fluid type or moving interface, the caution message likely should not be ignored. An example of a change in fluid or moving interface is a pipeline carrying fluids of different properties such as oils of different density, or a system is being flushed with water. In this example, the volumetric flowrate is likely conserved across a junction in the real system even if mass is not conserved. The Volume Balance junction can be used at the fluid interface to capture this situation. However, the Volume Balance is limited in that it cannot capture the interface location changing with time, and is only applicable to steady-state.

Losses at one or more pipes or junctions may be significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made

A **Caution message** in AFT Fathom

Full Message Text

Losses at one or more pipes or junctions may be **significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made**. Checking the 'Laminar/Non-Newtonian Correction Strength' output parameter is recommended. If the loss model source was from testing under turbulent conditions, applying a correction is recommended.

Explanation

Flow through one or more pipes or junctions was found to be in the laminar flow regime (Reynolds Number < 2300), but the "Apply Corrections" setting is disabled on the Laminar and Non-Newtonian Corrections panel in Analysis Setup. The "Laminar and Non-Newtonian Correction Strength" output parameter under either the Pipes or Junctions sections of Output Control was calculated to be larger than 1.25 for one or more pipes or junctions. This means that if corrections were applied, the resulting equivalent K factors and pressure losses would be significantly modified (>25%) compared to if no corrections were made.

Required Action

If the loss information used in the model was based on manufacturer data that came from turbulent tests, then the user should review the documentation for the Laminar and Non-Newtonian Corrections panel in Analysis Setup, consider enabling the setting for "Apply Corrections", and review the settings for which the corrections apply. Two scenarios can be compared, one with corrections and one without, to help determine the exact magnitude and location of the impacts. It is also recommended to review the documentation for the ATKF Method.

There is ambiguity in literature about if laminar corrections are required for fittings. A comparison of literature sources is provided, and it is up to the user to decide when/how to apply the corrections.

If the loss information used in the model was based on true laminar data, then the caution message can be disregarded.

Note: No scenarios have been set in the Multi-Scenario Output display

A Caution message in AFT Fathom

This caution is 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 elevation of Y is below Reservoir Z tank bottom elevation

A Caution message in AFT Fathom

(XTS Module Only)

Full message is:

"Pipe X elevation of Y is below Reservoir Z tank bottom elevation of A."

This is a warning that only occurs when running the XTS Module when using a finite tank option at a reservoir junction. If the user defines a pipe elevation which is below the bottom of the tank this warning will display.

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes

A Caution message in AFT Fathom

(SSL Module Only)

Full Message Text

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes. Results may be incorrect.

Explanation

This message can only appear when using the Wasp-Durand Heterogenous Slurry Model.

This message appears when the slope of a pipe is greater than 8 degrees and less than 18 degrees. Pipes with angles above 18 degrees will display this warning message instead. The Wasp-Durand model does not incorporate pipe slope corrections. Pipe Slope can impact pressure gradients and deposition velocity.

Required Action

If the slopes of all pipes in the system are below 18 degrees, Wasp-Durand may still be used, but will reflect the system less accurately. Exercise caution if results for uphill pipes show settling maximum velocities near their mean slurry velocities.

Pipe X length is shorter than the elevation change

A Caution message in AFT Fathom

This caution alerts you to an inconsistency with the input data due to a pipe being defined with a length that is shorter than the elevation change between its connecting junctions. This may be an input error or a result of a junction which itself has an elevation change. If it is due to the junction having an elevation change, you can use different inlet and outlet elevations at the junction to resolve this.

In this situation, pressure loss along the pipe due to friction is calculated using the pipe length the user has defined while static head loss is determined using the elevation change the user has defined.

Pipe X segment length Y is shorter than the elevation change

A Caution message in AFT Fathom

The pipe has intermediate elevations and the length along one of them is less than the elevation change of that segment. In this situation, pressure loss along the pipe due to friction is calculated using the pipe length the user has defined while static head loss is determined using the elevation change the user has defined. This can be corrected by adjusting the intermediate elevations input.

Pipe X steady state Reynold's Number is less than 100. Minor Losses may be significantly lower than expected.

A Caution message in AFT Fathom

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.

Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations

A Caution message in AFT Fathom

This message displays for centrifugal pumps which use viscosity corrections to the performance curves. Typical corrections depend on efficiency data which typically depends on flowrate. No such data was provided by the user. However, the user did provide nominal efficiency and that was used in the viscosity correction calculations. Viscosity correction results are less certain in this case.

Pump X has no efficiency curve or nominal efficiency data...

A Caution message in AFT Fathom

Full message is:

"Pump X has no efficiency curve or nominal efficiency data. An efficiency of 80% was assumed in viscosity correction calculations."

This message displays for centrifugal pumps which use viscosity corrections to the performance curves. Typical corrections depend on efficiency data which typically depends on flowrate. No such data was provided by the user. Further, the user did not provide nominal efficiency either. In order to perform the viscosity correction an efficiency of 80% was assumed. Viscosity correction results are less certain in this case.

Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.

A Caution message in AFT Fathom

This message displays for centrifugal pumps which use viscosity corrections to the performance curves. The methodology used by the Hydraulic Institute has limits which depend on flowrate, head rise, viscosity, and RPM. If the operating case in AFT Fathom is outside the valid range this warning displays.

Spray Discharge Junction X closed because inflow occurred and Special Condition - No Inflow was set

A Caution message in AFT Fathom

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.

System Life specified in Cost Settings is different from the XTS time

A Caution message in AFT Fathom

(XTS Module Only)

The System Life as specified in Cost Settings is of a different length than the transient duration specified in the Simulation Duration panel.

The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace

A Caution message in AFT Fathom

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.

The performance of Pump Junction X may be affected by the fluid viscosity

A Caution message in AFT Fathom

Full message is:

"The performance of Pump Junction X may be affected by the fluid viscosity. Apply the Viscosity Correction on the Pump Property window and rerun the model to assess the impact of the fluid viscosity."

This caution is shown if the fluid used in the model is not water, and the pump viscosity correction factors would be less than one based on ANSI/HI 9.6.7-2015. If the pump and fluid used in the model is in the scope of the ANSI standard it may improve accuracy to use viscosity corrections at the pump.

See the Pump Viscosity Correction topic for more information.

Transient data did not extend to Stop Time at Junction X

A [Caution message](#) in AFT Fathom

([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.

Wasp-Durand relative concentration (X) exceeds 0.8

A [Caution message](#) in AFT Fathom

Full Message Text

Wasp-Durand relative concentration (X) exceeds 0.8 and relative concentration for d95 particles (Y) exceeds 0.5 in pipe Z, indicating the slurry only consists of a pseudo-homogeneous component. However, the Liu Dezhong Model accounts for both pseudo-homogeneous and heterogeneous components.

Explanation

In quasi-homogeneous flow conditions, there are few solids in the heterogeneous slurry component with nearly all contributing to the homogeneous component. The heterogeneous gradient has little contribution to the overall pressure and flow results, but [Wasp-Durand](#) methods may still be appropriate for modeling the pseudo-homogeneous component.

Required Action

As the pseudo-homogeneous component is dictating results almost exclusively, modeling the homogeneous hydraulics accurately is more important than capturing heterogeneous effects in these pipes. Modeling the apparent viscosity appropriately is therefore of particular importance under these conditions.

If the entire system is quasi-homogeneous and the slurry is expected to exhibit [non-Newtonian](#) behavior, it may be better to model the system as a non-Newtonian fluid rather than as a settling slurry.

However, if rheological data is unavailable, use of the Wasp-Durand slurry model with Fei Xiangjun or Thomas viscosity models is likely to provide more accurate results than modeling the fluid as a pure Newtonian fluid.

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

Steady-state conditions are laminar in the following pipes

An [Information message](#) in AFT Fathom

Full Message Text

There are several variations of this Information message:

- Steady-state conditions are laminar (100 <= Re <= 2300) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these pipes. Laminar and Non-Newtonian Corrections **are not applied**.

    Pipes: P1, P2, P3, ...

- Steady-state conditions are laminar (100 <= Re <= 2300) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these pipes. Laminar and Non-Newtonian Corrections **are applied with default settings**.

    Pipes: P1, P2, P3, ...

- Steady-state conditions are laminar (100 <= Re <= 2300) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these pipes. Laminar and Non-Newtonian Corrections **are applied with user-modified settings**.

    Pipes: P1, P2, P3, ...

Note: If a [Zero-Length Connector (ZLC)](#) had laminar flow, there will be an additional note appended to the message stating that ZLCs have been excluded from the list of pipes. ZLCs cannot have Fittings & Losses, so the message does not apply to them.

Explanation

This Information message is ultimately intended to communicate that pressure losses in junctions and fittings may be different under laminar flow conditions compared to turbulent conditions. If flow through a junction or fitting is laminar, then extra

consideration may be warranted for how this will impact the pressure drop, and if corrections should be made. Fathom offers methods to correct losses for laminar flow, but there is also debate in literature about if corrections should be made. Specifics will be discussed below, otherwise consult the Required Action section below.

It should be noted that this message is not intended to discuss pressure drop from pipe friction. Some notes on laminar pipe friction will be provided at the end of this explanation.

Turbulent vs Laminar Loss Coefficients

It is generally believed that the pressure drop through junctions and fittings derive from the physics of bulk fluid motion, wall separation, turbulence, etc. Therefore, the losses during laminar flow may be different compared to turbulent conditions. These pressure losses are typically modeled using data from manufacturers (Resistance Curves and Valve Cv or Kv values) or loss coefficients from academic references (K Factors). The problem is that data from these sources are typically based on laboratory tests which usually occurred under turbulent flow conditions with water.

Laminar flow is often associated with high viscosity fluids, and usually the pressure drops through junctions are higher under laminar conditions. For example, a manufacturer may provide a pressure loss data point at a specified flowrate for a strainer that was tested with turbulent water in their laboratory. If this strainer is installed in a system that pumps high viscosity oil, it is very likely that the pressure drop through the strainer at the same specified flowrate will be much larger. Therefore, it is important to consider these effects when designing systems to ensure they can meet design requirements.

Correction Options

The Laminar and Non-Newtonian Corrections panel in Analysis Setup offers some capabilities that can help estimate adjusted loss coefficient values for laminar flow. AFT generally recommends to compare scenarios with and without the Adjusted Turbulent K Factor Method (ATKF) enabled to determine the magnitude of effect. Results can be easily compared using Multi-Scenario Output.

By default the laminar flow regime is defined by a Reynolds Number less than 2300, but this value can be changed on the Environmental Properties panel of Analysis Setup. The lower bound of Re = 100 exists because there is a separate Caution message if the flow is in the "deep laminar" flow regime which is defined by Reynolds Numbers less than 100. The Reynolds Number for a pipe can be checked by adding the "Reynolds Number" output parameter from the Pipes section of the Output Control. The "Laminar and Non-Newtonian Correction Strength" output parameter can be added from both the Pipes section and Junctions section of the Output Control. The correction strength parameter for pipes will refer to the K Factors for any lumped Fittings & Losses, and not the pipe friction itself. A correction strength larger than 1.0 acts as a multiplier that increases the equivalent K Factor and pressure loss for the pipe fittings or junction.

Comment on Area Restrictions Inside Junctions

When flow in a pipe is laminar, there is also an implication that flow through connected junctions would also be laminar. This also applies to components and fittings that have been lumped into the pipe Fittings & Losses. One caveat to mention is that in reality some junctions could have an area restriction such as the path of flow through a valve, orifice, etc. In an area restriction, the

velocity will increase, potentially to the point where the Reynolds Number is no longer laminar, and laminar corrections might not be needed. This might be a consideration for systems that operate in the transitional flow regime. Even when accounting for area restrictions within junctions, there is a high probability that if the connected pipe experiences laminar flow that the junction will also experience laminar flow, or at least operate under a condition with different pressure loss characteristics compared to reference K Factors.

Pipe Friction Under Laminar & Non-Newtonian Conditions

For more information on how to account for laminar or Non-Newtonian effects on pipe friction, consult the Pressure Drop in Pipes Detailed Discussion and Viscosity Model panel topics. Below is a brief summary:

- If the pipe uses the default Friction Model (Standard), then the Solver will use the Darcy-Weisbach loss model which has an explicit correlation for the laminar friction factor (f=64/Re). In this case laminar flow is automatically accounted for in pressure drop calculations. Other Friction Models may or may not account for laminar flow - consult the equations for each friction model on the Pressure Drop in Pipes Detailed Discussion topic.
- If a Non-Newtonian viscosity model is used, then the friction factor will be calculated according to the corresponding methods - consult the equations for each viscosity model.

Required Action

First, the user should consult the help topic on the debate of if laminar corrections are required for junctions and fittings, and use their engineering judgment to decide if they believe corrections are required. If the user does not want to apply corrections, then keep the laminar corrections disabled on the Laminar and Non-Newtonian Corrections panel (this is the default selection).

If corrections are desired, and if junction loss coefficients were from turbulent conditions, then it is recommended to enable laminar corrections. The user should consider enabling the Adjusted Turbulent K Factor method and read the corresponding help topic to learn more.

> **Note:** All of the K Factors from Fathom's internal Fittings & Losses Library are from sources that tested under turbulent flow conditions.

If loss factors were from laminar conditions and this is confirmed with the manufacturer or reference source, then keep the laminar corrections disabled so as to not double count the effect of laminar conditions. Corrections can be enabled or disabled for an entire junction type. This may be useful if you are accounting for laminar flow in bends and valves using the 3-K Darby Method, but not for General Component junctions, and therefore you only want to apply corrections to General Component K Factors.

Previous Variations of the Messages

From the initial release of Fathom 13 to an intermediate maintenance release there were alternate variations of these messages (release 2023.09.14 up to and including 2024.02.05). Changes

were made because AFT changed our philosophy on how and when to alert users. Previously an Information message would communicate when significant corrections were made (>25%), and a Caution message would communicate if no corrections were made but would have been significant if applied. Whether the Information or Caution message was displayed depended on if the user enabled corrections from the Laminar and Non-Newtonian Corrections panel. Models created in Fathom 12 or previous versions had laminar corrections enabled by default, and importing them into Fathom 13 will keep the corrections enabled. New models created in Fathom 13 have laminar corrections disabled by default, but the Informational message documented on this topic is shown to notify the user. The new philosophy is to simply alert users to laminar flow, which may or may not be expected, and allow the user to decide if applying corrections is required.

Previous Information message:

Losses at one or more pipes or junctions were **significantly modified (>25%) with a Laminar or Non-Newtonian correction**. The strength of the correction can be seen by checking the Laminar/Non-Newtonian Correction Strength output parameter. If the loss model source was from testing under laminar conditions, applying a correction is NOT recommended.

Previous Caution message:

Losses at one or more pipes or junctions may be **significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made**. Checking the 'Laminar/Non-Newtonian Correction Strength' output parameter is recommended. If the loss model source was from testing under turbulent conditions, applying a correction is recommended.

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

The Model Diagnostics window is a feature introduced in AFT Fathom 11. 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 all input will be checked, and the results will be displayed in a tabular format. Items of note will be highlighted based on the impact each identified item may have on the results. It should be noted that **not all items identified by this window will have a negative impact on results**. Rather, the identified items are settings that have frequently been incorrectly applied by other users. This is not an exhaustive list of troubleshooting items, though the most common issues are all included.

Data in the Model Diagnostics is organized by the Data Source, which is the window where that setting/input is defined. Any item that may negatively impact the model will be highlighted based on its potential to affect the model, as shown in the Potential Impact column. For pipe and junction diagnostic items, the pipe(s)/junction(s) that were identified will be listed below the diagnostic item. For an example, see Figure 1 below. The Diagnostic window indicates that pipe P1 has unusual pipe dimensions, and should be reviewed.

Figure 1: Example of the Model Diagnostics window

Related Topics

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.
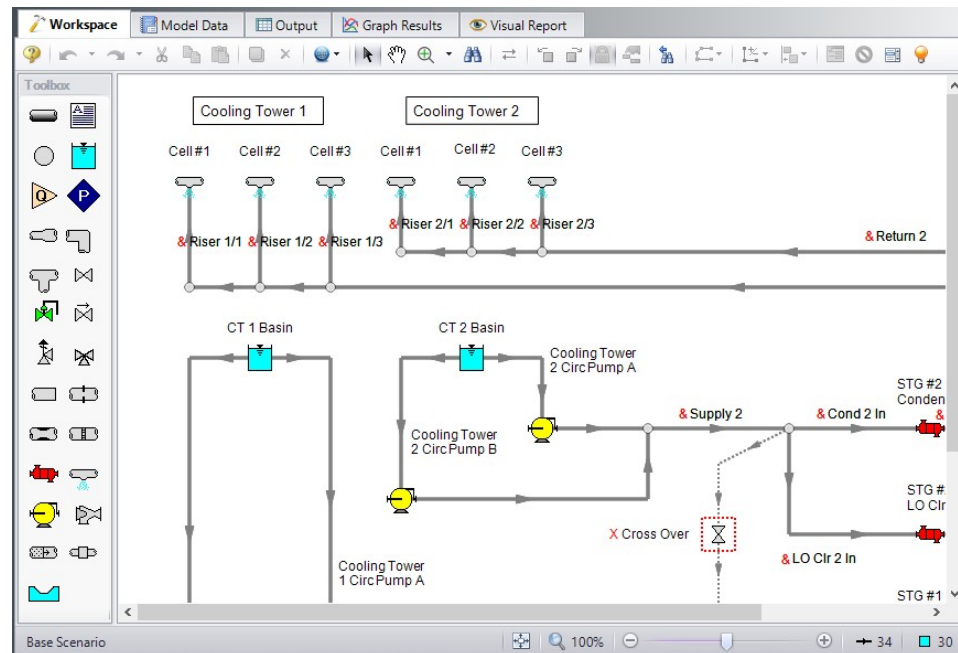
Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown)

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 (#107) has length data entered in miles rather than feet. This is not automatically an input error, as AFT Fathom accepts pipe input with mixed engineering units. However, more frequently than not the user intends to use consistent units, so this probably is an error.

Related Topics

Transfer Results to Initial Guesses

AFT Fathom's iterative Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) 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. This causes much 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 located on the Output window Edit menu and Toolbar.

Related Topics

Poor Pump Curve Fits

The Pump Properties window allows curve fits of pump data up to fourth order. Depending on the number of data points entered, pump curves fits that look like the orange curve in Figure 1 are possible.

AFT Fathom 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 pump curves cross the flow rate axis as shown in the blue curve in Figure 1.

AFT Fathom will attempt to warn the user when a model containing a pump 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 pump is not operating to the right of the inflection point.
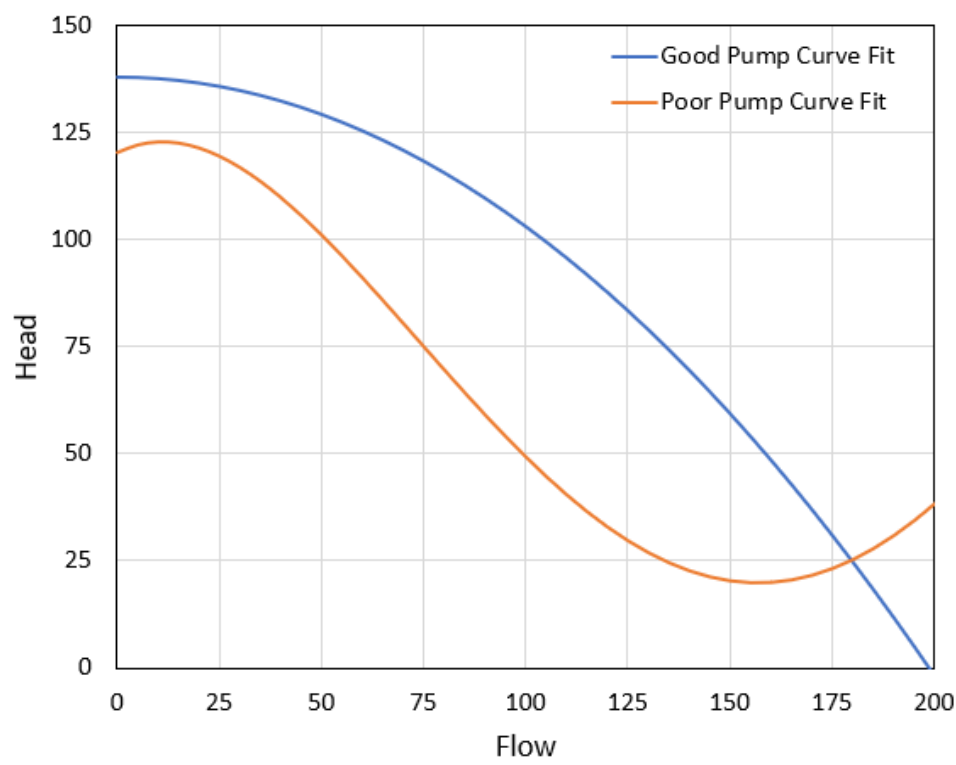
Figure 1: A good pump performance curve fit in blue and a poor pump performance curve fit in orange. All pump curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.

Related Topics

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.
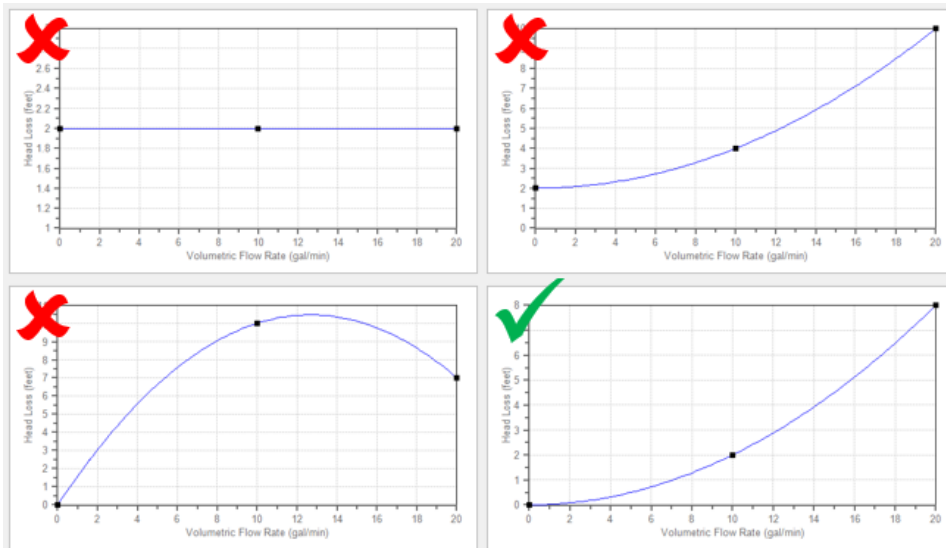
Figure 1: Example resistance curves showing correct and incorrect curve types

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

Use the Output Window Sort Feature

If a model runs but the results look incorrect, use the **Sort feature** found in the Arrange menu or toolbar in the Output Window to look for extremes in velocity, pressure drop or other parameters. An input error (like incorrect diameter) may be easier to find by looking at its effect on the results.

Related Topics

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, which 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 correlations 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 loss model itself. Rather, it is due to multiple tees coupling together in such a way that the Solver cannot converge on all of the flow splits. The tee loss coefficients are dependent on the flow splits, but the flow splits are dependent on the other losses in the system. Therefore, if a pipe network contains many tees using the Detailed loss model, when one tee's loss coefficient is updated from an intermediate Solver iteration, the entire network needs to be re-calculated to determine how the flow splits changed. Sometimes this process is not able to terminate within the Tolerances specified by the model.

A common issue with detailed tees is having the pipes connected in the Properties window differently than how connections are set up in the Workspace. Make sure to double check those connections before running the models.

You can quickly and easily check if a convergence problem is being caused by using the Detailed loss model. Create a child scenario, then using the Global Junction Edit window, you can change all tees to the Simple loss model and rerun the model. If it now converges then the problem was likely due to the use of Detailed tee modeling.

Experience has shown that this problem can frequently be resolved by using absolute tolerance criteria on the Tolerance panel of Analysis Setup rather than relative tolerance. Use this option at your own discretion and avoid setting the tolerances too loose such that you get false convergence. Often times, using the "Both Absolute and Relative" option gives the best of both worlds.

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

Lower the Flow Rate Relaxation

The Iterations panel in Analysis Setup 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 Fathom'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

Try Absolute Tolerance

Most models converge more reliably with relative tolerances, which is the default on the Tolerance panel in Analysis Setup. However, some models converge much better with absolute tolerance. This most often applies to models which have very small values for flow or pressure, as it is much harder to meet a relative tolerance using small numbers.

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

Make Initial Flow Rate Guesses for Pipes

If you do not provide any initial guesses, AFT Fathom will make its own guesses. Occasionally the AFT Fathom 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. However, note that initial guesses that are too far from the final solution can also interfere further with convergence.

Related Topics

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 towards these elements.

Related Topics

Change Boundary Conditions

Changing the boundary pressures or flow rates can sometimes give you insight into what is giving the model trouble. For example, obtaining convergence only within a certain range of flowrates could indicate an issue with curve fit data being unrealistic outside of that range of flowrates.

Related Topics

Turn Off Parts of the Model

Using the Special Conditions feature, you can turn off elements in your model and in effect shutdown entire sections. If the model runs when certain sections are closed, this could indicate a problem with one of the components in the closed section.
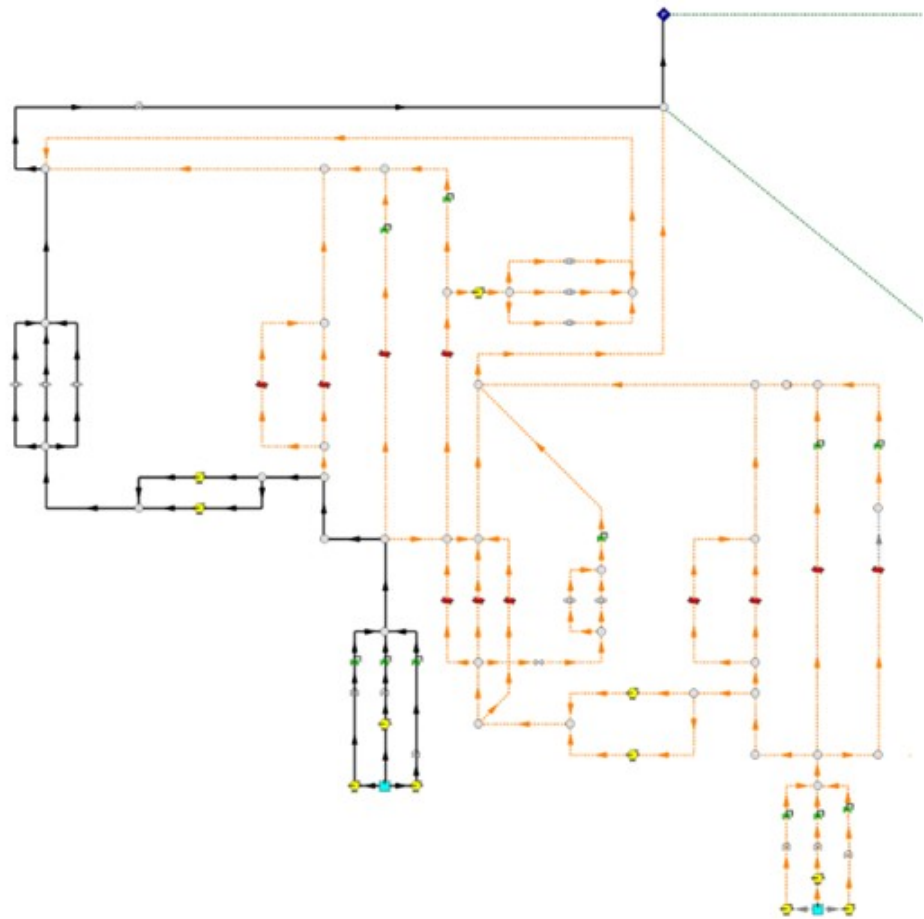
Figure 1: The pipes in black are open to flow, while the pipes in yellow have been closed using special conditions, meaning that Fathom will automatically set their flow solution to 0

Related Topics

Break a Large Model Into Submodels

AFT Fathom provides convenient features for deleting parts of your model and merging parts back in. A section of your model can be quickly deleted by selecting it on the Workspace and choosing Delete from the Edit menu. If the section is instead Cut, it can be Pasted into a new model of its own for testing or holding. At a later time the removed sections can be added back in to the current model by using the Merge feature on the File menu.

The advantage of breaking your model into sub-models is that you can investigate the convergence characteristics of each part of your model independently. If you can get each sub-model to converge, you should be able to get the original model to converge also. If one of your sub-models will not converge, it is doubtful your original model will converge either.

Related Topics

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 Fathom. 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.
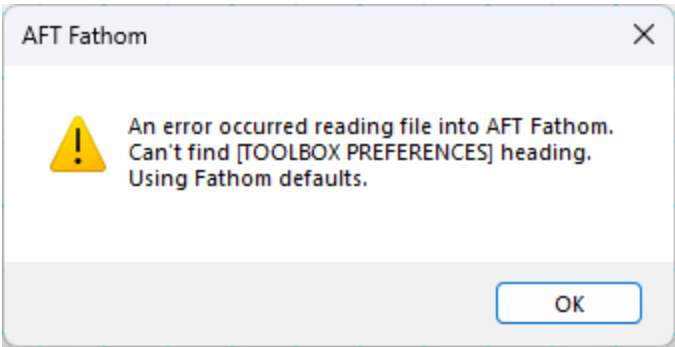
Figure 1: Error message indicating that some portion of the model file cannot be found

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).
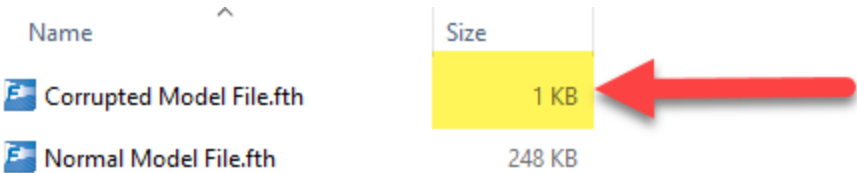
Figure 2: Very low file sizes can be an indication of model file corruption

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.

Figure 3: The newest backup file, identifiable by file extension numbering and date modified

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.
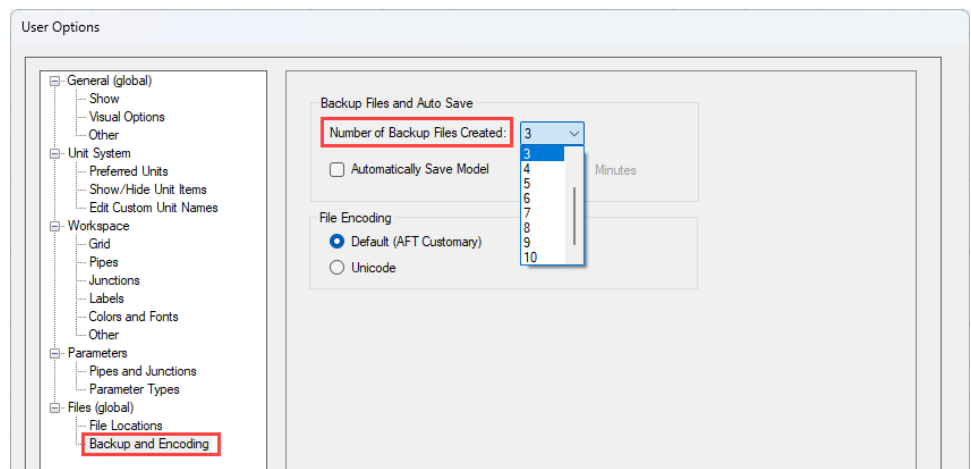
Figure 5: The number of backup files created and options to save automatically can be customized in User Options

Related Topics

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 Fathom.

Contents

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.pmtechgroup.com/, or call at (719) 687-9145 for more information.

Related Topics

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

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

Chempak Database Installation

AFT Fathom 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 Fathom to access the Chempak Fluid Library in the software.

Chempak File Location

The Chempak Data is installed in **C:\AFT Products\Chempak Data** by default. If both AFT Fathom and the Chempak Database are installed to their default locations, no further configuration is necessary. However, if AFT Fathom or the Chempak Database are not installed in a default location, AFT Fathom will need to know where to look for the Chempak Database. This is specified in the control.aft file, located in the same folder as Fathom.exe. If the files cannot be found the following message is displayed:
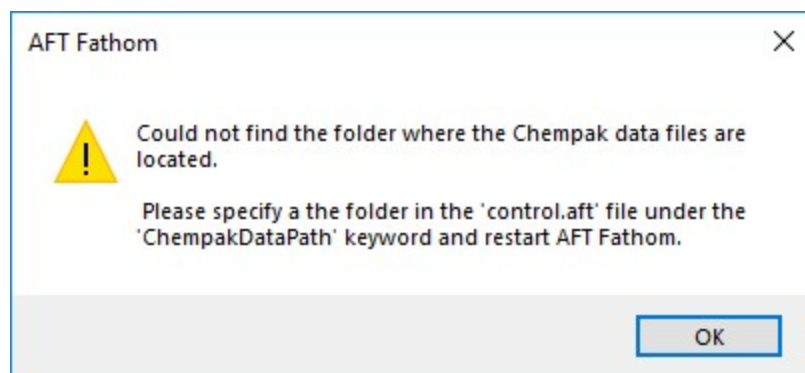
Figure 1: AFT Fathom cannot find the Chempak Database

Control.aft can be edited with a text editor, and by default looks like:
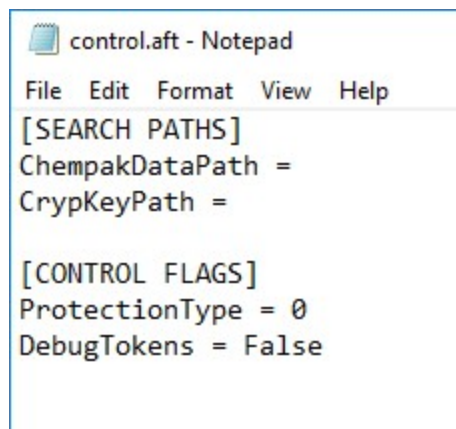
Figure 2: Default control.aft file

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 Fathom, you may receive a message that the database
is outdated.
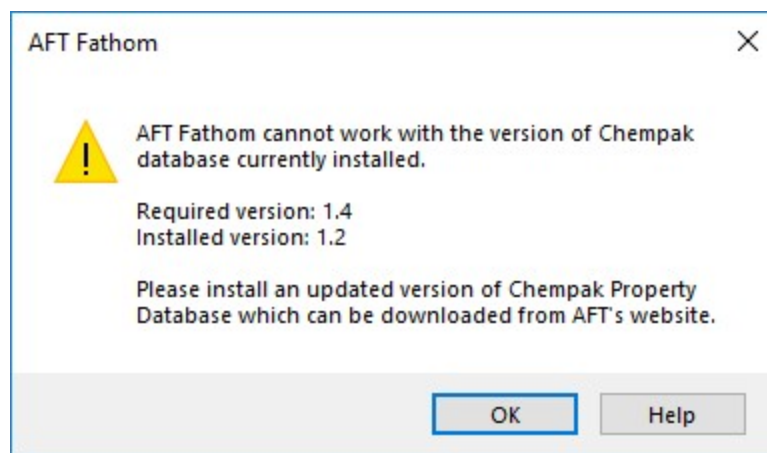
Figure 3: Outdated Chempak database

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

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.

Updated Fluids Library

If the data in a shared fluid library is changed outside of the model you are opening, the following informational message will appear when opening the file:

The following fluid(s) used in the scenario have different property data than what is currently in the library:
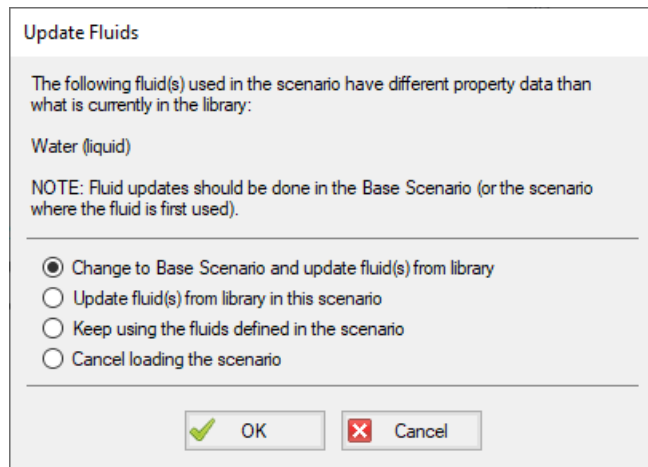
Figure 1: Update Fluids dialog window which appears if the fluid properties for the model have changed since it was last opened

There are a few reasons that the fluid data may have been changed:

1. The model was defined in a previous version of Fathom using a fluid from the AFT Standard fluid library. The AFT Standard fluid library was updated in AFT Fathom 12 in order to standardize the reference data. If the model uses the same fluid properties for all scenarios, it is recommended to use the "Change to Base Scenario and update..." option to update the fluid properties throughout the model. If different scenarios use different fluid properties then it is recommended to use the "Update fluid(s) from library in the scenario" option to update each scenario individually.

2. The model is defined with a custom fluid in the Fluid Library and the custom fluid was modified outside of this model. If the fluid was changed in error, then choose the "Keep using the fluids defined in the scenario" option or cancel loading the model and change the fluid. If the changes were intended then update the fluid data used in the model using the "Change to Base Scenario and update..." option.

Special File Control Features

What is the Control.aft File?

Control.aft is read as AFT Fathom begins to load and instructs AFT Fathom 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 Fathom 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 Fathom will not find the important data.

Specifying Where Chempak is Located

Before AFT Fathom can use the Chempak Fluid Database 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 Fathom to look in the C:\AFT Products\Chempak Data directory for the data.

Also see Chempak Data Installation.

Related Topics

Converting Models from Earlier Versions

Models from earlier versions of AFT Fathom 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.

Related Topics

Index

H

I