AFT Impulse™

Product Help Guide

AFT Impulse Version 9

Waterhammer Modeling in Piping Systems

Published: May 02, 2023

Dynamic solutions for a fluid world ™

Contents

Overview of AFT Impulse

Welcome to AFT Impulse™ - a visual platform for modeling fluid transients in pipe networks! See what's new in version 9!

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

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

What Can AFT Impulse Do?

AFT Impulse 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
- Systems with valve transients
- Systems with pump transients
- Systems with pressure or flow control valve transients
- Systems with transient cavitation and liquid column separation
- Systems with surge suppression devices such as accumulators, surge tanks and air valves
- Systems with variable density and viscosity
- Multiple design cases in a single model file (Scenario Manager)
- Non-Newtonian fluid behavior

AFT Impulse's steady-state 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. Once a steady-state solution is obtained, AFT Impulse simulates the transient using the Method of Characteristics, the most widely used method for solving fluid transient problems.

Where to Start

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

Add-on Module Capabilities

AFT Impulse has two add-on modules that extend modeling capabilities.

- Pulsation Frequency Analysis (PFA) allows the engineer to study the pressure pulsation effects a positive displacement pump has on a system.
  - Predict resonant frequencies
  - Graphically show the frequency response of a system
  - Examine the pressure response of a system when a positive displacement pump is running at a resonant frequency
  - Ensure the pressure pulsation magnitudes comply with the API-674 standard
- Settling Slurry (SSL) allows the engineer to perform waterhammer analysis while accounting for the complex property and system interactions associated with slurry flows.
  - Model homogeneous and pseudo-homogeneous slurry mixtures
  - Slurry pump performance de-rating

Who Can Use AFT Impulse

AFT Impulse assumes that the user possesses a good general knowledge of engineering pipe system hydraulics and has at least a basic understanding of the fluid transients and surge phenomenon. Even the most advanced and easy-to-use software package cannot make up for a lack of fundamental knowledge on the part of the user.

Related Topics

New in Version 9

Analysis Setup

The Analysis Setup window consolidates windows for System Properties, Solution Control, Section Pipes, Transient Control, and Activate Modules all into one location. This allows users to activate and enable modules, specify fluids, change solution tolerances or iterations, and so on, all from a single window. For a list of windows that have merged into the Analysis Setup
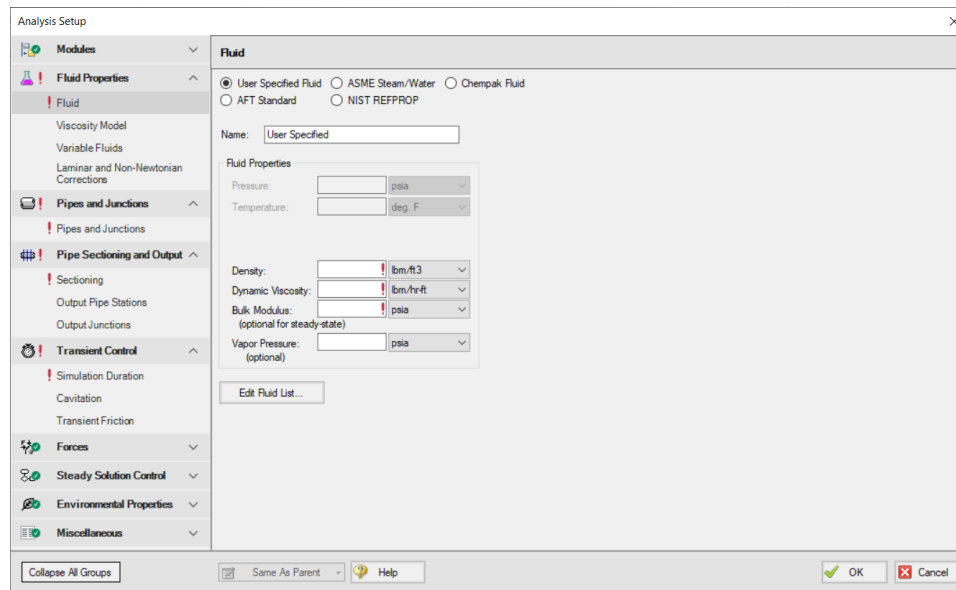
Libraries

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.
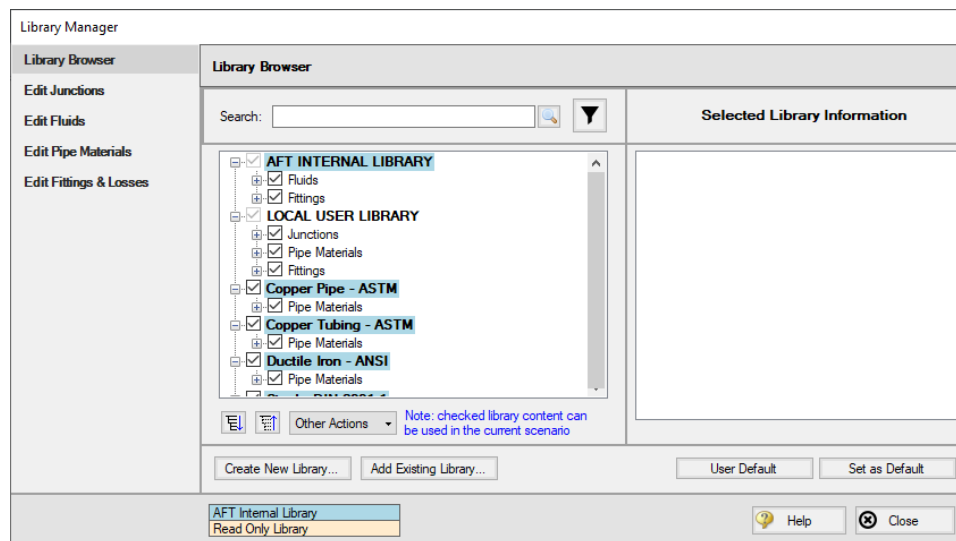
Gas Accumulator Updates

The Gas Accumulator Properties window has been redesigned, and several new features have been added as follows:

- The tank geometry for the gas accumulator can now be defined, allowing the liquid level in the gas accumulator to be tracked, and limiting the gas volume accordingly
- Additional options for defining the steady state gas accumulator volume have been added, allowing the thermodynamics for the steady state to be defined separately from the transient

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
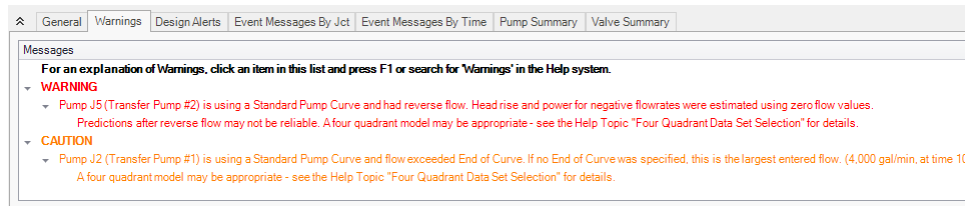
More Improvements

- **Pipes/Junctions**
    - Pipes can be defined as zero-length connectors to connect two junctions which are directly flanged together
    - Several new junctions have been added including the Bend, Orifice, Screen, and Venturi junctions
    - Submerged pump can now be defined using surface pressure and pump depth
    - Valve transients can be defined as open percent vs time
- **Workspace and Interface**
    - Pipes and Junctions can be set the Same as Parent directly from the Workspace
    - Custom Unit names can be defined in the Unit System section of the User Options window
    - Pipe/Junction notes can be searched in Select Special
    - Special Conditions can be directly changed from the toolbar for junctions that have multiple special condition options
- **Output and Reporting**
    - Show the number of warnings and design alerts in the batch run window
    - Design alerts are grouped together in the General Output section
    - New output parameters are available for the Relief Valve junction
- **Miscellaneous**
    - Herschel-Bulkley non-Newtonian viscosity model added for representing Power Law fluids which have an apparent yield stress
    - New run time messages are shown in the Solution Progress window to help users adjust their model to decrease run time for the simulation
    - 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

Interface Improvements for Impulse 9

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

Analysis Setup Sources in Impulse 8

- **Steady Solution Control**: The Steady 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

- **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.

- **Section Pipes**: The Section Pipes window has been moved to the **Sectioning panel** in the Pipe Sectioning and Output group.

- **Transient Control**: The Transient Control parameters were distributed into multiple groups:

    - The settings from the Transient Solver Control tab were moved to the **Transient Control group**

    - The Pipe Station Output and Junction Output tabs which specified what output to save to file were moved into the **Pipe Sectioning and Output group**

    - The Force Output tab was moved to the **Forces group**

    - The Advanced transient control settings were moved to the **Advanced Parameters panel** in 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.

- **Add-on Modules Activation**: The activation window for add-on modules has moved to the **Modules panel**

- **Slurry Data** (SSL Module Only): Settings have been moved to the **Fluid Properties group**

- **Pulsation Data** (PFA module only): Settings have moved to the **Pulsation Setup group** and the **Sectioning panel**

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
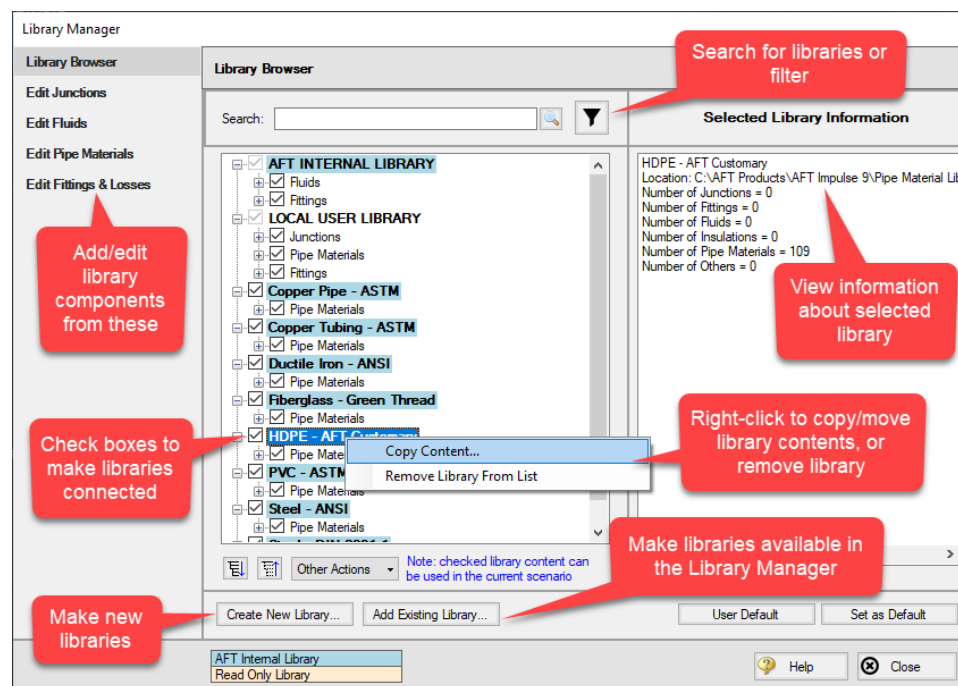
New in Version 8

Scenario Comparison Tool

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

Multi-Scenario Graphing

- Users can now create graphs of data from multiple scenarios in the Graph Results window
- Transient Junction graphs can be created for any scenarios in a model, while Transient Pipe, Profile, and Force graphs can be created for scenarios with the same pipe sectioning
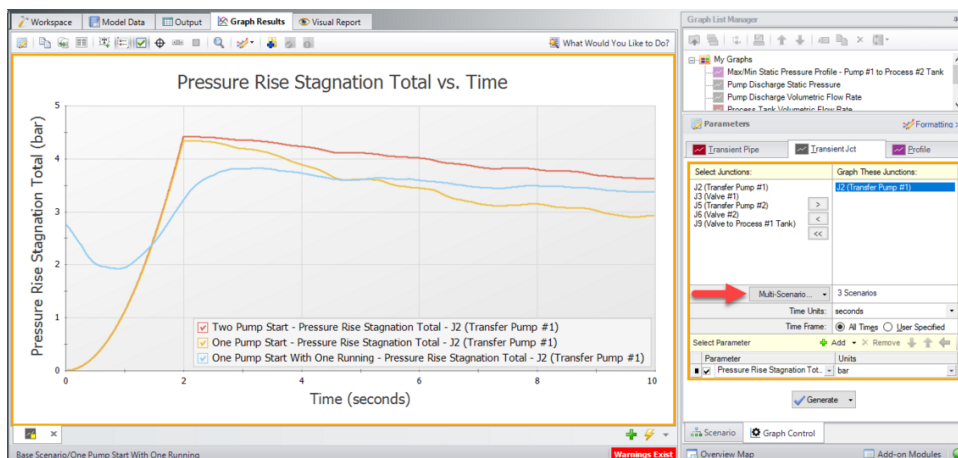
New Time-Varying Friction Model

- An additional model for unsteady friction has been added to the Transient Control window
- The Brunone model adds an unsteady shear stress term which varies based on the velocity at the current time step. This addition allows Impulse to better account for the frictional losses as fluid velocity changes during a transient.

Improved Pump Window and Four-Quadrant Data

- Reconfigured selections for pump performance curve options to further clarify how different selections will behave
- 'Automatically Selected' option for Four-Quadrant data sets to automatically choose a data set for a given pump based on Estimated Specific Speed
- Four-Quadrant data sets clarified to reflect the quality of the data

Junction Specific Parameters in Visual Report

- Output such as percent of BEP for a pump or the Cv of a control valve can now be displayed in the Visual Report
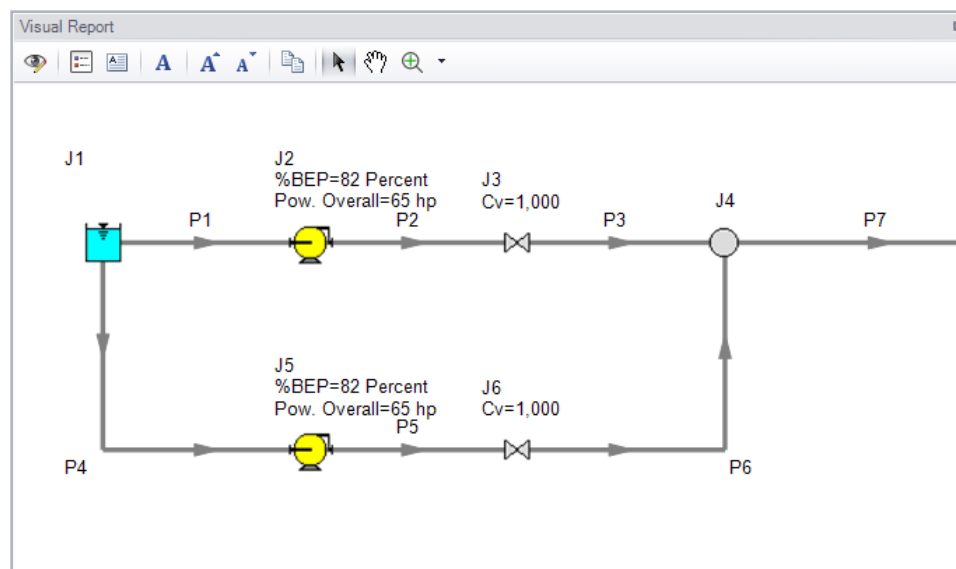
Improvements to Force Set Capabilities

- Export forces to AutoPIPE
- Vectors for force sets can be defined using X-Y-Z unit vector information. This force set information can then be included with exported force data into a relevant pipe stress analysis software.
- Force Sets can be defined directly from the workspace by selecting pipes and using the right-click menu.

Notes in Output

- Notes can be added to 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.
- **Magnifier Tool** - The magnifier tool is available in both the Workspace and the Visual Report, and provides an enlarged view of sections of the model to make it easier to manage and navigate large models.
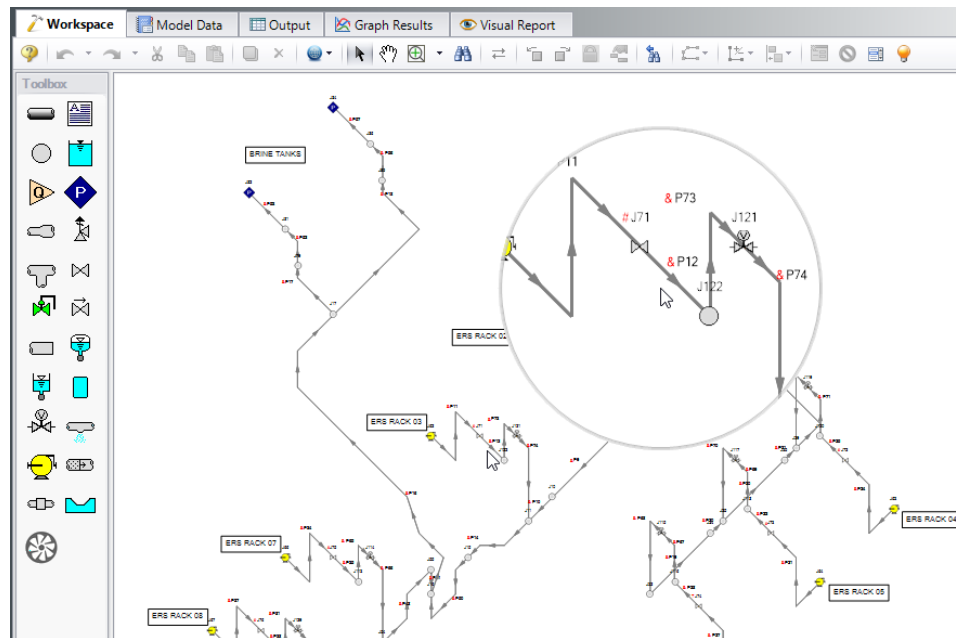
- **Automatic Junction Rotation** - Impulse will automatically rotate junctions to visually align with neighboring pipes to speed-up model formatting
- **Split Pipes by Intermediate Elevations** - Pipes with intermediate elevations can be split with branches at the intermediate elevation points. See Combining 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 (launched from clicking "Define Pipes and Junctions" in the Checklist) remains open when pipes/junctions are being edited
- Double-clicking a junction allows more than one of that junction to be placed in the Workspace at a time
- Snapping can be turned off for pipes when the isometric drawing mode is enabled
- **Output**
  - Output parameters can now be filtered/searched for simpler selection
- **Visual Report**
  - Visual Report Control has been redesigned to better filter parameters
- **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
  - 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

Related Blogs

Five Fantastic New Features for Fast Fluid Transients: Impulse 8

New Feature Highlight - Scenario Comparison Tool

New in Version 7

New & Improved Microsoft Excel Integration

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

	A	B	C	D	E	F	G	H
1	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	FCV setting set	Yes	Control Valve	71	Control Setting	Set equal to value	300	High Demand
3	FCV setting reduced by 20 gpm	No	Control Valve	72	Control Setting	Change by value (+/-)	-20	Low Demand
4	FCV setting increase 10%	No	Control Valve	73	Control Setting	Change by percent (+/-)	10	Average
5	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	
6	Pipe Diameter increase	Yes	Pipe	58	Nominal Size Increment	Change by value (+/-)	1	

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

New Isometric Drawing Mode

- New Isometric Pipe Drawing Mode
    - Show and Snap to an isometric grid on the Workspace
    - View models in an intuitive "3D" layout, and easily build from existing isometric drawings
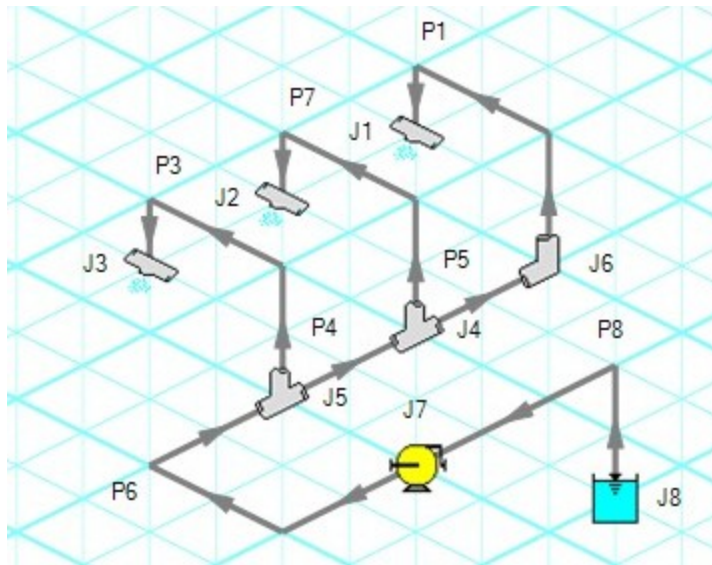
NIST REFPROP Fluid Database Now Included

- Over 150 fluids from the National Institute of Standards and Technology (NIST) Reference Fluid Thermodynamic and Transport Properties Database (REFPROP) Version 10

- Select a NIST REFPROP fluid directly in System Properties
- Use accurate and complete fluid information in your models without the need to locate and enter fluid properties manually

New Pump Features and Improved Pump Window

- New Positive Displacement Pump Model
  - Clarifies the specification of Positive Displacement Pumps
- New Pump as Turbine Model
- Added options for Reverse Flow to Centrifugal Pump Model
  - Clarifies required data and model intention
  - For a detailed description see Pump Properties Window
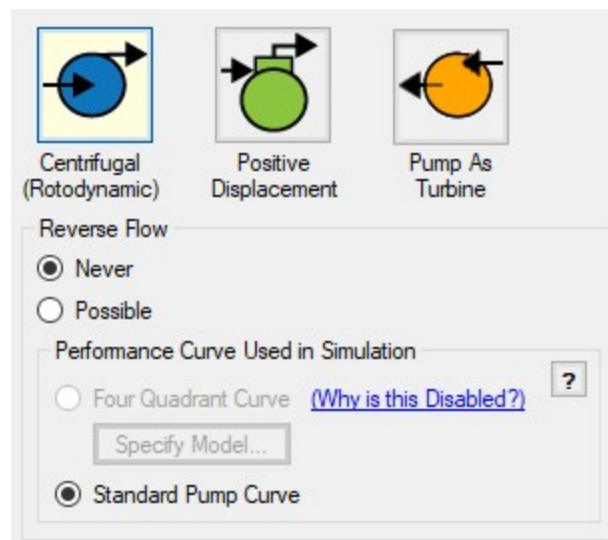
- Specify a centrifugal pump's Preferred Operating Region (POR) and Allowable Operating Region (AOR)
  - Quickly determine if a pump's behavior is acceptable
  - Automatically calculate a POR from *ANSI/HI 9.6.3-2017 - Rotodynamic Pumps - Guideline For Operating Regions*
  - For a detailed description see Pump Configuration > Configuration Data
- Significant enhancements and clarity added to the Four Quadrant modeling capabilities of the pump
- Added ability to model a pump trip for a VFD pump
- See a full list of changes to the Pump Properties window here

New Multi-Condition Events

- Create complex triggers for transient events
    - 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
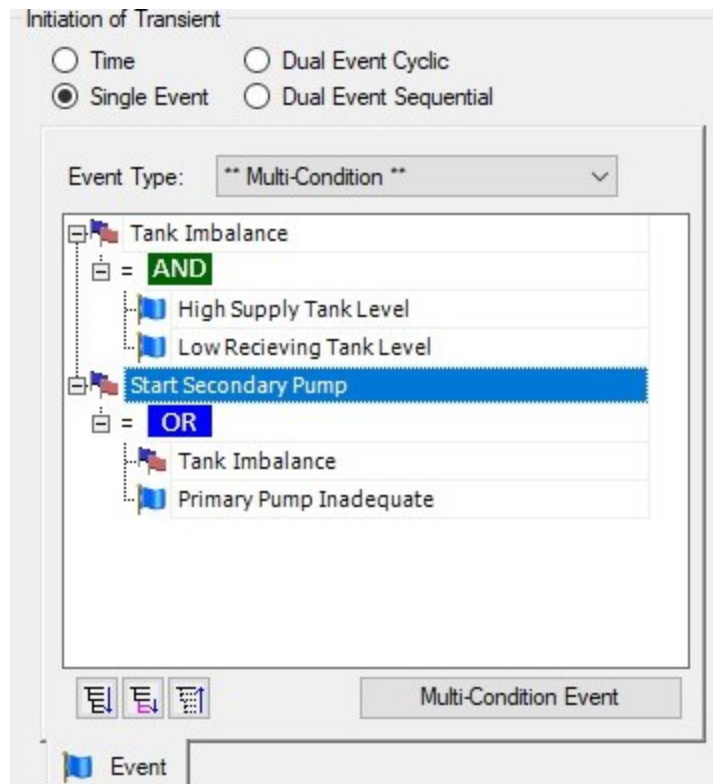
Visual Report Animation

- Revived visual report animation feature from AFT Impulse 1
    - Use color animation to show absolute values or values relative to the steady state
    - Static color map option to show maximum/minimum values from the transient data
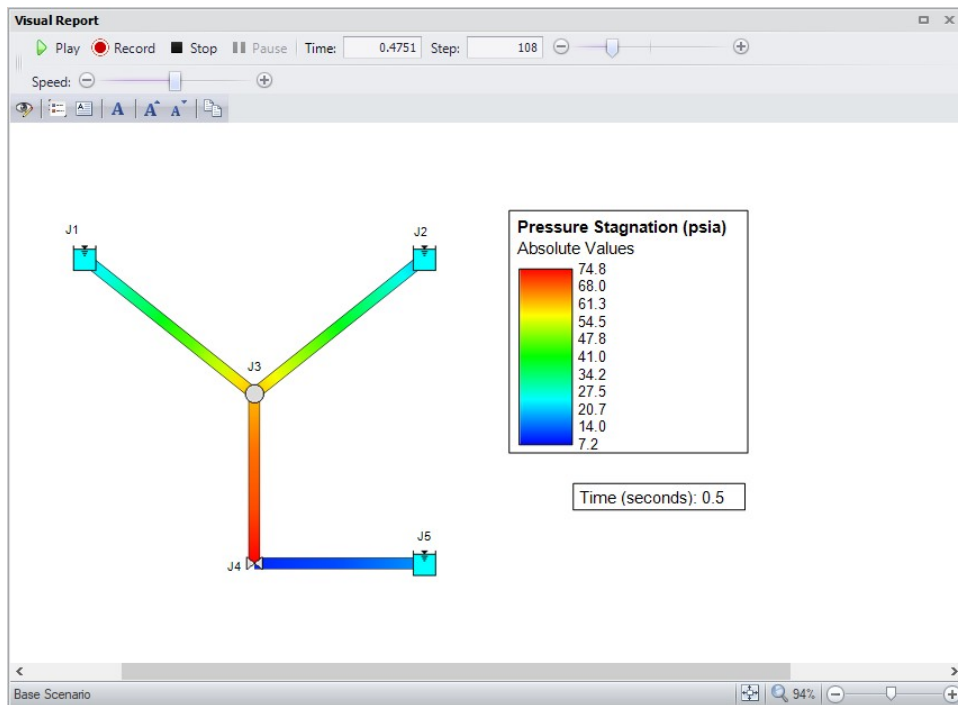
New Features for Valves

- **General**
  - Pre-defined curves available for valve Cv vs open percent
- **Check Valves**
  - Check valve closing velocity can be estimated using dimensional data for deceleration vs. maximum velocity to close chart
  - Ability to enter manufacturer data for deceleration vs. maximum velocity to close
  - New inertial model for translating nozzle/plug check valves
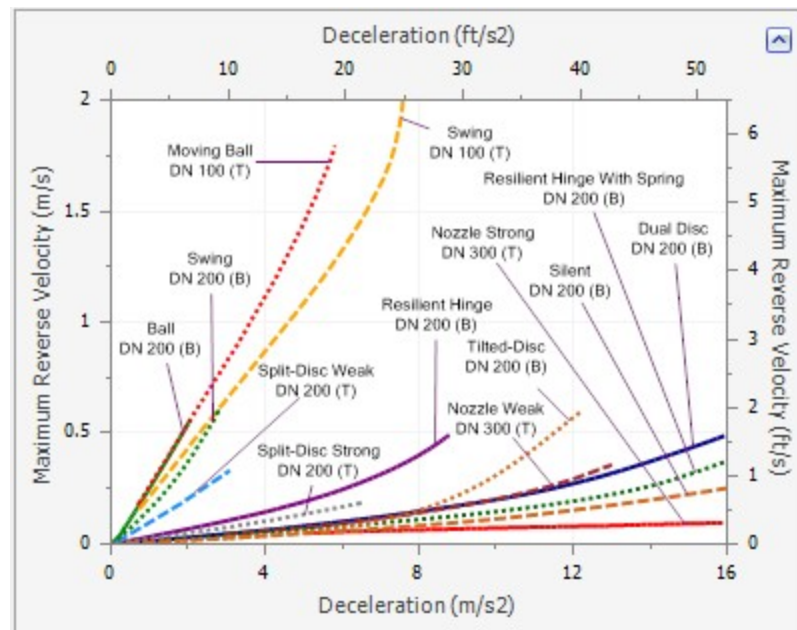
- **Control Valves**
    - Control Valve setting for Always Control (Never Fail) can be set to a default to off
    - Added Control Valve transient option to fail open or closed from current Cv value
- **Valve**
    - Standard valve junction has the option to create Cv vs time transient from open % vs Cv data

Improved Startup Window

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

New Integration with Common Piping Layout formats

- Save modeling time by importing existing Piping Layouts from various neutral file types
    - Import CAESAR II 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

  - Increased the loading speed for large models

  - Applying Design Alerts now shows Design Alert Violations in transient Output tables, and all time steps that violate the Design Alert are highlighted

  - Time delay option added for air valves

  - Added Buried Pipe model for calculating pipe wavespeed

  - Reservoir junction now has a finite tank option

  - New Force Set types for exit differential forces

  - Forces are calculated during the run for improved output display and graphing speed

  - Option available to add more rows to entry tables

- **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.
    - Data can be sorted by clicking on column heading
    - Output reports in portuguese
- **Graph Results**
    - More parameters available on the Transient Junction tab, including parameters such as delta pressure/head
- **Model Data**
    - Quickly locate a specific object on the Workspace by right-clicking the item in Model Data
- **Visual Report**
    - Reworked Display Option controls for Pipes and Junctions in Visual Report Control

Related Topics

Related Blogs

Iso love the new Pipe Drawing Mode!

Paint a Clear Picture Faster Than Ever with AFT's New Excel Export Manager

New in Version 6

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
- Main menus are translated into available output languages

Graph Results

- Multiple Graphs can be added to Graph Folders for easy generation and display
- Multiple graph tabs on primary window Graph tab

- Multiple parameters can be stacked on separate graphs using the same X-axis
- Secondary Y-parameter for most graph types
- Multiple graphs tiled on same Graph tab
- Multiple graphs can be animated at the same time
- New parameters such as Kv, Cv rate of change, valve position, liquid accumulator volume
- 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
- Animations can be saved in MP4 and QuickTime formats

Workspace

- New paste preview mode allows the pipes and junctions to be placed desired and will indicate interference with existing objects on the Workspace
- GIS shape files can be imported to create a model
- Text alignment options in annotations
- Annotations can be layered on the Workspace

Junctions

- New Turbine junction type
- Two new Check Valve models, including an inertia-based Swing Check Valve model
- Surge Anticipator valve capability added to the Relief Valve
- Remote pressure sensing capability added to the Relief Valve
- Improved clarity for data entry in the Surge Tank Property window
- Automatic generation of Cv vs. Time using equal percentage or linear profiles
- New events based of Cv or Kv
- Maximum pressure setting for Gas Accumulators simulates a relief valve
- When the speed is set for a pump, both the original and modified pump curves are shown on the Pump Property window
- Design Alerts can be applied to junctions

Pipes

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

Forces

- New Force Summary report available in the output showing all applied force sets
- New Design Alerts can be applied to Force Sets and shown in the Force Summary
- New Force report in Model Data showing the input details
- Force definitions can be imported from an Excel file

Scenario Manager

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

Output

- Results Diagrams which graphically show data for pumps

Visual Report

- Automatic Color Map generation

Related Topics

Impulse 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 provides a short introduction to the interface of AFT Impulse, and works through a simple example model.

- [Online Quick Start Tutorial Video (English)](#)
- [Online Quick Start Tutorial Video (Metric)](#)

Beginner: Valve Closure Problem

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

- Beginner: Valve Closure 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 Impulse.

- Walk Though Examples

Module Walk Though Examples

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

- Module Walk Through Examples

Related Topics

Engineering Assumptions

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

- Liquid-phase flow only
    - Entire system is wet (100% liquid full)
    - AFT Impulse is not meant for analysis of two-phase flow
- One dimensional flow
- Fluid properties remain constant during transient
    - Non-condensable gas release and dissolution is negligible
    - Wavespeed remains constant
    - No chemical reactions
    - Fluid density is not affected by pressure waves in the system
- Vapor pockets formed during transient cavitation in pipes are considered point-volumes at section boundaries, and do not move
    - AFT Impulse is limited in its ability to model sustained cavitation and two-phase flow. The accuracy of results will be reduced when sustained two-phase flow conditions exist. Small cavitation volumes over short durations are handled with good accuracy.
- Additional assumptions exist for particular modeling components and features, discussed where relevant

Related Topics

Startup Window

The Startup Window is displayed when AFT Impulse 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 Impulse 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 the Fluid panel before running the model.

- **Select A Recently Used Fluid** - Also requires Temperature.

- **Modeling Preferences** - Displays various settings for the application, such as specifying the units to use or which grid to use. These options can be modified later. Click the **Modify** button to reveal the following settings, as shown in Figure 2. Click the **Remember My Preferences and Hide** button to update the modeling preferences for the model and to update the default settings for the application.

  - **Units** - US, Metric, or both sets of units with one as default can be displayed. Additionally, the displayed units can further be reduced to only show common units and those used by a certain industries.

  - **Language** - Select a language to display Output in.

  - **2D or Isometric Workspace** - Set up the grid.

  - **Default Pipe Material** - Set the pipe material that is initially selected for new pipes.

- **Resources**

  - **User Setup** - Open the AFT website to view the latest AFT Impulse release, or sign up for notifications of new releases via email.

  - **Quick Start Guides** - Open the included PDF of the Quick Start Guide, or a Quick Start Video. These materials are designed to help get a new user up and running with AFT Impulse as quickly as possible.

  - **Help Files** - Open our detailed help file (you are here!) or the collection of worked Example problems.

  - **Video Tutorials** - Open the AFT website to our library of instructional videos.
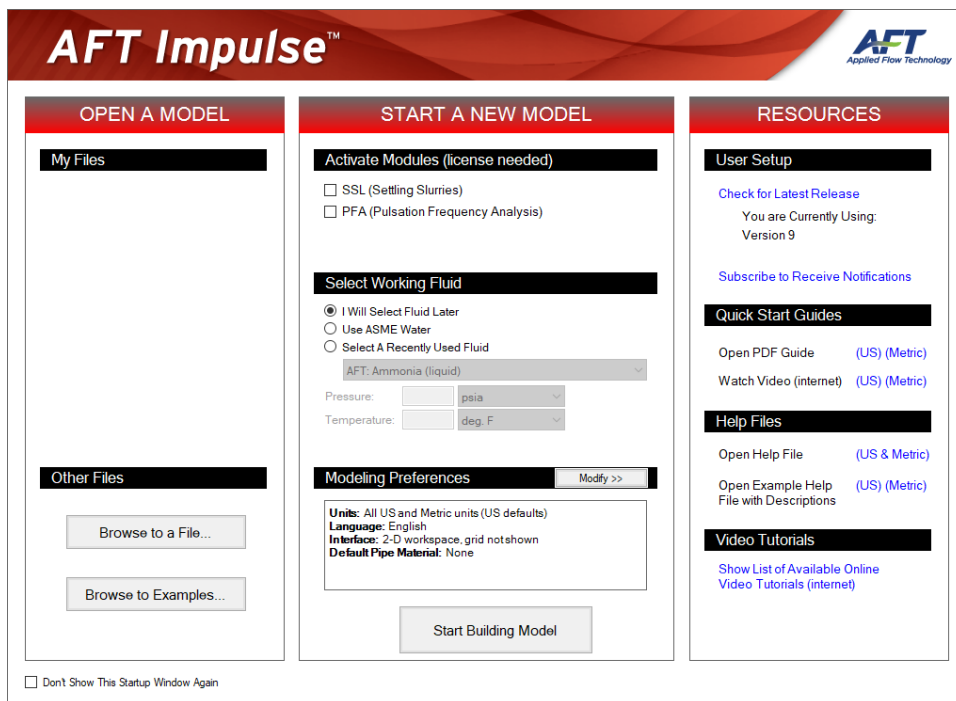
Figure 1: Startup Window in AFT Impulse

Figure 2: Startup Window showing Modeling Preferences

Related Topics

Frequently Asked Questions (FAQ)

Can AFT Impulse model transient cavitation?

Yes, AFT Impulse uses the Discrete Vapor Cavity Model or Discrete Gas Cavity Model.

Can AFT Impulse model gas transients?

No, AFT Impulse only models transients in liquid systems.

Can AFT Impulse model pump transients?

Yes. Pumps can be modeled as a prescribed speed transient or the speed can be calculated based on pump inertial characteristics.

Can AFT Impulse model transients cause by positive displacement pumps?

Yes, positive displacement pumps can be modeled as steady operation with periodic flow or during startups and shutdowns.

Can AFT Impulse model air (i.e., vacuum breaker) valves?

Yes, see the Air Valve junction topic.

Can AFT Impulse model pressure and flow control valves?

Yes, see the Control Valve junction topic.

Can pipes be modeled with varying elevation profiles?

Yes, users can assign an elevation profiles to individual pipes.

Can I specify maximum and minimum operating pressures?

Yes, AFT Impulse allows you to include these as "design alerts". Design alerts can be cross-plotted against pressure profiles.

What fluid properties are available in AFT Impulse?

AFT Impulse offers a standard set of properties for common fluids in the AFT Standard Fluids library. It also offers the NIST REFPROP library included with all AFT applications. Users can also utilize the optional Chempak data add-on which is a chemical property library of up to 700 fluids with mixing capability. Use the library as a stand-alone application with the Chempak Viewer or access the library from within Microsoft Excel with the Chempak Add-in.

Can AFT Impulse model non-newtonian fluid behavior or pulp and paper systems?

Yes, AFT Impulse offers Power Law and Bingham Plastic non-newtonian fluid models, and offers the Duffy method and Brecht & Heller for pulp and paper system modeling.

How does the steady-state solution capabilities in AFT Impulse differ from AFT Fathom?

The steady-state solver in AFT Impulse uses the same method as AFT Fathom. However, its purpose is different. AFT Fathom is a general purpose steady-state modeling tool. On the other hand, AFT Impulse uses steady-state only as it relates to waterhammer. Thus some capabilities found in AFT Fathom do not exist or are more limited in AFT Impulse. The most important differences are as follows: AFT Impulse does not model heat transfer. It does not have junctions for static elements like elbows, orifices, heat exchangers, etc., although these can still be modeled in AFT Impulse but through other methods such as Additional Losses in pipes or as General Components. AFT Impulse does not offer polynomial resistance curves except for General Components. AFT Impulse does not offer graphing of pump and system curves, which only have meaning for steady-state.

Does Impulse have the ability to model partially filled pipes?

Impulse can model partially filled pipes at the end of a flow path or run. This capability is primarily intended to handle the situation where a supply riser to a cooling tower or condenser, for example, partially drains when the system is not in operation.

Overview of Fluid Transients

A system is steady-state if its [properties](#) - pressure, velocity, etc - do not change with time. Any disturbance to a steady-state system will cause these properties to change - this change is a **fluid transient**.

Note: Fluid transients and their effects are commonly referred to with the term *waterhammer*. While the term is most appropriate for water systems, waterhammer analysis applies to any liquid.

All fluid transients cause disturbances which propagate throughout the fluid as coupled pressure and velocity [waves](#). This behavior is perhaps best explained with a [conceptual example](#). Like any acoustical wave, these waves reflect, combine, and dissipate. This behavior can be extremely complex, even in relatively simple systems.

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

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

Related Blogs

[What Is Transient Analysis?](#)

Conceptual Example

The wave mechanics of a [fluid transient](#) are not always clear at first glance. It is helpful to consider a simplified example to understand the dynamic events following a single transient event.

In the example below, a rigid, frictionless pipe is connected to a fixed pressure upstream (a large reservoir, for example) and a valve downstream. The fluid is flowing at a steady velocity. The downstream valve is instantaneously closed, which [instantaneously halts flow](#).

Phase 1: Initial Wave

When the valve is closed, the fluid immediately upstream no longer has anywhere to flow to and therefore must come to a complete stop. This stopped fluid acts like another closed surface for the fluid further upstream - stopping that fluid in turn. This is a wave which propagates upstream at a constant [wavespeed](#). The wavespeed remains constant through all phases.

When the fluid is brought to a complete stop, the kinetic energy of the moving fluid is transferred into potential energy, raising the pressure of the fluid.

The fluid properties at any given point do not change until the wave reaches that point. This means that the steady flow into the pipe remains unchanged, increasing the total amount of fluid in the pipe, until the wave reaches the reservoir.
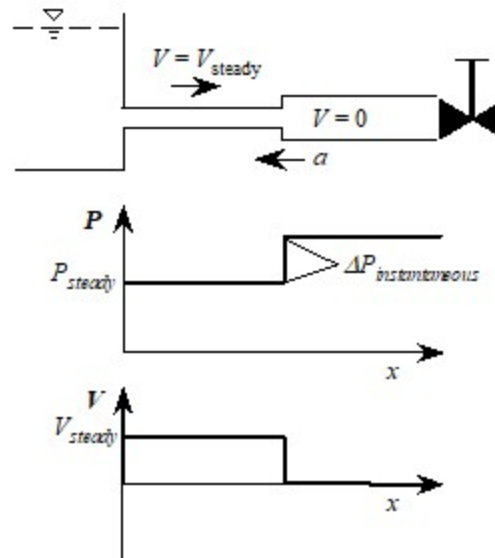
Phase 1 - Initial Wave

Phase 2 - Reflected Wave

The upstream reservoir is considered a fixed pressure. Because the pressure at this point is unable to change, the high pressure transient wave is unable to be absorbed into the reservoir, and the wave is reflected completely.

The wave is now traveling downstream, reducing the high pressure back to the reservoir pressure.

Lowering the pressure requires converting the stored potential energy back into kinetic energy. However, flow cannot proceed downstream. Instead, the fluid behind the wave travels upstream, discharging back into the reservoir. This magnitude of this velocity is the same as the original steady-state velocity.
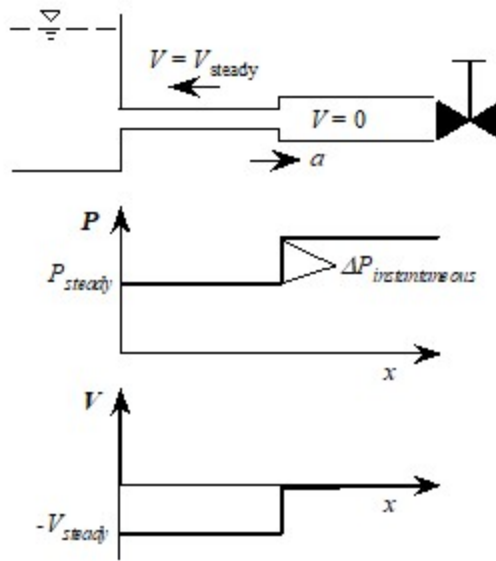
Phase 2 - Reflected Wave

Phase 3 - Negative Wave

When the wave from phase 2 reaches the closed valve, the entire pipe is at the original steady-state pressure. However, the fluid in the entire pipe is now flowing in the reverse direction of steady-state.

When the reverse flow reaches the closed valve, there is no longer any fluid available to move and relieve the high pressure originally built up in phase 1. The flow is again completely halted at the valve. Like the upstream reservoir, the closed valve does not absorb the pressure wave and it is again completely reflected. However, the behavior is different due to the opposing flow direction.

This time, the flow velocity is brought from negative velocity to rest, instead of positive velocity to rest. This causes a negative pressure compensation. In effect, the fluid leaving the system begins drawing a vacuum at the closed valve.

The negative wave that results travels upstream at a pressure lower than the steady-state pressure. This pressure drop is the same magnitude as the pressure rise in phase 1.

Note that fluid has been flowing out of the pipe and into the reservoir during the entirety of phases 2 and 3.
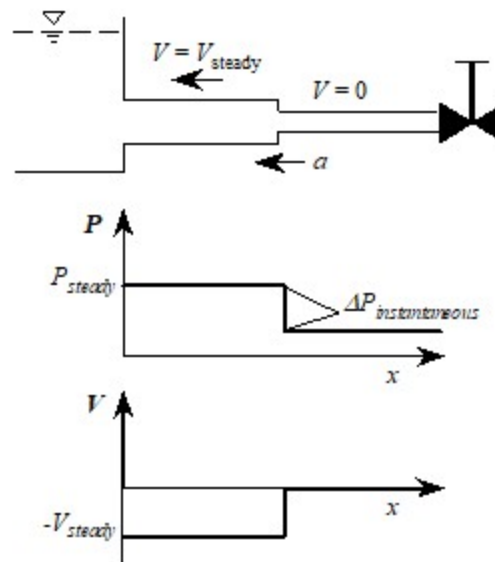
Phase 3 - Negative Wave

Phase 4 - Reflected Negative Wave

When the negative wave reaches the upstream reservoir, it is again completely reflected, bringing the pressure back to the steady-state pressure.

To increase the pressure, fluid must flow back into the pipe. The fluid velocity and pressure behind the wave are back to the original steady-state magnitudes. However, the wave is traveling in the opposite direction to phase 1.

Upon meeting the valve, the wave is again perfectly reflected. As before, the fluid is entirely halted, and the forward fluid momentum converted into a rise in pressure.

The behavior after this final reflection is identical to phase 1. In a frictionless pipe, and with perfect reflections, there is no mechanism to reduce the energy of the system - the wave reflects indefinitely, cycling through the four phases.
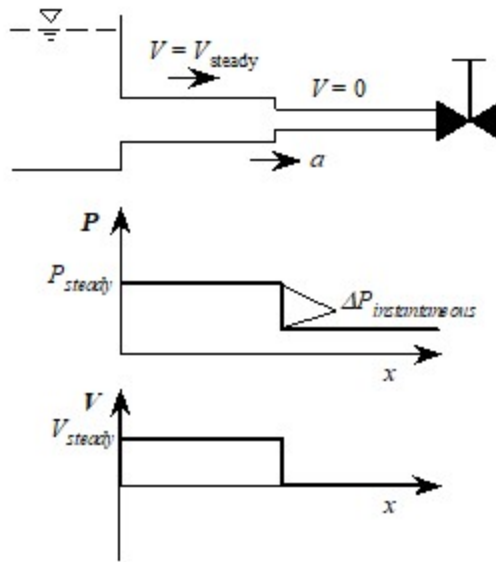
Phase 4 - Reflected Negative Wave

Related Topics

Related Blogs

What Is Transient Analysis?

Wavespeed and Communication Time

Wavespeed

The speed at which a wave propagates throughout a medium is termed the wavespeed. The wavespeed depends on the acoustical velocity of the fluid itself as well as the physical properties of the pipe and the method of support.

Acoustical velocity through the fluid is related to the bulk modulus of elasticity (*K*) and density (*ρ*) of the fluid.

$$a_{acoustic} = \sqrt{K/\rho}$$

This wavespeed is the propagation speed in a fluid where there is no mechanism to impede the wave motion - for example, in an unbounded body of fluid or in a completely rigid pipe.

In piping systems, some of the energy in a pressure wave distorts the pipe itself, reducing the wavespeed. This depends on the pipe modulus of elasticity (E), the inner diameter (D), the wall thickness (e) and a constant ($c1$) which depends on pipe supports.

$$a = \sqrt{\frac{K/\rho}{1 + c_1 \dfrac{K}{E} \dfrac{D}{e}}}$$

For more information, see Wavespeed - Detailed Discussion.

Also note that the wavespeed equation presented here is only valid for fluids with no solids present. If the SSL module is being used to model settling slurries, the wavespeed is adjusted as is discussed in the Wavespeed Calculation For Slurries topic.

Communication Time

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

The time it takes for a wave to travel from one end of a system to the other is simply the distance (L) divided by the wavespeed.

$$\Delta t_{end-to-end} = L/a$$

When a transient event occurs at one end of a pipe, the wave travels at the wavespeed to the other end of the pipe. However, this is not sufficient time to analyze the problem. While the disturbance has propagated through the entire system, the point of transient initiation has not yet seen the result of the reflected wave. For the wave to both travel to the end of the pipe and back requires double the above time. This time is known as the Pipe Period or Communication Time.

$$Communication\ Time = \frac{2\,L}{a}$$

Note: The Communication Time only accounts for half of the wave cycle (described in the Conceptual Example). This is how much time it takes for a disturbance to *return* to its starting point, but analyzing a system for only this amount of time neglects part of the transient cycle. Transient simulations should run for at least double the Pipe Period, or 4L/a.

Related Topics

Related Blogs

A Story About Fluid Communication

Minor Wave Reflections

Every point of resistance to flow in a piping system is a source of wave reflections. Without any resistance in a system there would be no reflections or pressure loss.

It is beneficial to include all of the resistances in a system during a transient analysis, to capture the pressure losses that occur in both steady-state and transient conditions.

However, the magnitude of a reflection from a minor loss such as a locked open valve or an elbow is generally small enough to be ignored. Including every minor loss as a component requires numerous Pipe Sections, which has the potential to exponentially increase run time.

Rather than ignoring the component entirely, and thus ignoring the pressure loss, it is possible to effectively ignore *only the minor reflection* by lumping the loss of the component into a pipe. By increasing the overall resistance of a pipe, the additional pressure loss is accounted for and run times are significantly improved, with usually only minor changes in results.

> **Note:** It can be difficult to tell what components can be lumped into pipe resistance, and what reflections are important. Generally, components that do not change position or behavior during a transient, and have low steady-state pressure loss can be lumped. If it is not clear that a certain component has no significant reflection it is best to include this component in the analysis. Comparing analyses with a component explicitly modeled versus lumped gives an indication of the impact this simplification has.

AFT Impulse provides an easy way to lump these resistances with Fittings & Losses, by allowing the user to define common components directly "within" a pipe.

Automatically Combine Pipes

The Combine Pipes window is opened from the Section Pipes window and will automatically combine adjacent pipes of the same diameter separated by a static junction. When pipes are combined, all length, elevation, pipe K factors, intermediate junction loss, and visual segment data is preserved into the resulting pipe.

Related Topics

Pipe Sectioning - Introduction to Method of Characteristics

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

What is Sectioning?

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

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

The time Δt it takes to travel from one point to another is determined by the linear distance Δx and wavespeed a between those points. We have defined this time to be the time step for the transient simulation.

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

Each section must be continuous and unchanging fluid path. In other words, the physical properties of the pipe cannot change within a single section. This means that every Pipe in the model must have a whole number of sections - otherwise the section would be interrupted by a Junction. In mathematical terms, a pipe of any length L must have N sections of length Δx:

$$L = N * \Delta x \qquad N = 1, 2, 3, \dots$$

In any system there will be a single **controlling pipe**. This is the pipe with the fewest sections, generally only one. As this pipe cannot have any fewer sections, the sectioning of the entire system is effectively controlled by this pipe.

What happens if the system cannot be evenly divided? For example, consider a system of two pipes - each with identical wavespeed. One of the pipes is 20% longer than the other.
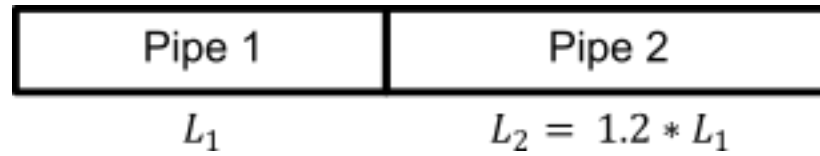
Figure 1: Mismatch in pipe lengths

If we state that Pipe 1 - the controlling pipe - should have 1 section, Pipe 2 must have 1.2 sections to meet the time step restriction. However, this fails the whole number of sections requirement. One solution here is to find a common denominator that evenly divides both pipes. If we state that Pipe 1 has **five** sections, then Pipe 2 can be divided into **six** sections of the same length. But what if the common denominator is a very large number? Consider the below system.
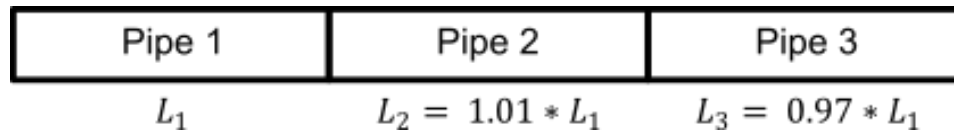
Figure 2: Smaller mismatch in pipe lengths

If all three pipes were the exact same length, we only need **one section** in each pipe. Due to the length variation Pipe 3 - the controlling pipe - would require **97 sections**. This also means that the time step must be 97 times smaller! We have increased the number of calculation points by nearly a factor of 100 and the number of time steps required by the same factor. The small variation in length will cause the simulation to take nearly 100*100 = 10,000 times longer! Clearly, this is unacceptable for any practical purposes.

The solution is to allow some **variance (ψ)** into the system. The time step cannot be adjusted independently across sections. Instead, we can adjust the *wavespeed* in each pipe.

$$a_{modified} = a_{actual} \left(1 \pm \psi \right)$$

Again consider the three pipe example - if we decrease the wavespeed in Pipe 3 to 97% of its original value, and increase it in Pipe 2 to 101% of the original value, we can arrive back at our ideal situation - 1 section in each pipe. A 3% change in wavespeed can be considered negligible for any reasonable transient simulation - there are greater uncertainties than this in almost all studies.

Note that you can decrease the maximum variance further by adjusting the wavespeed in all three pipes. Above, we kept Pipe 1's wavespeed fixed - introducing the 3% variance in Pipe 3. If we vary the speed in Pipe 1 as well, we introduce some variance in Pipe 1, but we also make the Pipe 3 value smaller.

The standard allowance for variance in AFT Impulse is 10%. It is suggested to increase the number of sections in the controlling pipe until this value is under 10%. Some references such as Wylie accept up to 15% variance.

Sectioning Effect on Run Time

Extremely long run times are often a result of a large number of sections. As discussed above, the **run time is approximately proportional to the square of the number of sections**.

Large numbers of sections are rarely required for accurate results. There are a few common methods for reducing the number of sections:

1. Do not include minor sources of wave reflections as junctions.
2. Limit the difference between the longest and shortest pipes in the model. A very short pipe must have at least one section. If there is a very short pipe (1 meter) and a very long pipe (10,000 meters) then the model must have a large number of sections (10,001 sections). If the short pipe cannot be lumped into its adjoining pipe, it is often an acceptable approximation to artificially lengthen the short pipe. This may increase the error in the short pipe, but experience shows that this error is usually negligible in a fluid transient context. **Increasing the very short pipe's length from 1 to 10 meters decreases the number of sections required by a factor of 10 - and therefore the runtime by a factor of 100.**
3. Do not include extremely long pipes unless the transient behavior in those pipes is being directly analyzed. Instead, consider using the Infinite Pipe feature.

Sectioning Effect on Accuracy

Because each pipe section is explicitly solved along the characteristic lines from the Method of Characteristics, breaking pipes into continuously smaller sections offers no improvement in accuracy.

That is, using twice the number of sections in a pipe will not yield a more accurate prediction. This is in contrast to other kinds of finite difference methods where more increments offers improved accuracy.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

Cavitation

What is Cavitation?

Cavitation is a commonly observed phenomenon in liquid systems which occurs any time local pressure reaches the liquid's vapor pressure. At vapor pressure, the liquid boils, and forms vapor pockets within the liquid. These vapor pockets grow while the system pressure remains at the vapor pressure, and will collapse when the local pressure increases above the vapor pressure.

Cavitation is seen in many situations. An everyday example is boiling a pot of water. Raising the temperature of the water raises the vapor pressure of the water. Eventually, the water at the bottom of the pot is heated enough that the vapor pressure is equal to atmospheric pressure, and the water begins to boil. Vapor pockets are formed, and as they rise to cooler regions of water where the vapor pressure is lower, they collapse.

Another common example is cavitation at a pump impeller. Pressure in the liquid decreases as it approaches the impeller, and vapor pockets will form if the pressure at the impeller eye reaches the liquid's vapor pressure. The vapor pockets move out away from the impeller eye, and collapse as the pressure rises above the vapor pressure.

A final example is cavitation downstream of a valve. Pressure downstream of this valve is initially above vapor pressure during normal operation. The valve is closed rapidly which generates a low-pressure wave and causes the downstream pressure to reach vapor pressure. A vapor void forms while the pressure is at vapor pressure. The low-pressure wave is reflected as a high-pressure wave, returns to the valve, and collapses the vapor pocket.

Transient Cavitation

The three examples of cavitation above all represent situations where the liquid pressure reaches vapor pressure and cavitation will occur. However, Impulse only attempts to model situations similar to the final example, where pressure waves are generated by transient events in a system forming vapor voids. This type of cavitation is called *transient cavitation*.

Transient cavitation can be particularly harmful to a system because the vapor void formed must eventually collapse when high pressures return. The liquid on each side of the vapor void will slam together causing large pressure surges, large transient forces, and potentially significant damage to a system. The pressure surge following a transient cavitation event can often be larger than the pressure rise predicted by the [Joukowsky Equation](#) for the system, meaning engineers should always evaluate these events carefully.

Any transient event can generate a pressure wave which results in transient cavitation. Some common events include:

- Pump trips
- Valve closures
- Pump startups

- Sudden changes in control valve position
- Constructive wave interference

Cavitation Modeling in Impulse

AFT Impulse attempts to model transient cavitation in pipes using one of two cavitation models: the Discrete Vapor Cavity Model (DVCM) and Discrete Gas Cavity Model (DGCM). These models are described further on the Cavitation Models page. Both models are meant to capture limited, transient cavitation in pipes, and are not meant to capture cavitation inside a junction, or cavitation which exists for a sustained period of time. Impulse also does not allow simulations to proceed when cavitation is present in the steady-state solution.

A fundamental assumption behind Impulse's solution methodology is that all pipes are 100% liquid-full. Cavitation inherently violates this assumption because vapor is introduced to the system. Users should always treat results with any amount of cavitation as uncertain results and carefully review all results.

AFT Impulse reports cavitation in a system with two parameters: Vapor Volume and Percent Vapor Volume. The Vapor Volume reports the actual volume predicted by Impulse's cavitation models. The Percent Vapor Volume reports the vapor volume as a percent of the pipe section volume.

AFT Impulse also uses Percent Vapor Volume to trigger two different warning messages in the output. A Warning message is generated when the Percent Vapor Volume is greater than 10%, while a Critical Warning message is generated when the Percent Vapor Volume is greater than 100%. Each of these warnings is meant to convey increasing uncertainty in the results to the user.

Related Topics

Related Blogs

How does AFT Impulse model cavitation?

Instantaneous Waterhammer Equation

With some exceptions, it is safe to calculate the maximum possible transient fluid pressure surge by using the instantaneous waterhammer equation.

Note: The instantaneous waterhammer equation is often credited to Joukowsky and referred to as the Joukowsky Equation.

Derived from the mass and momentum equations, this equation calculates the pressure change (ΔP) when instantly changing the flow velocity (ΔV) for a given density (ρ) and wavespeed (a):

$$\Delta P = -\rho \, \alpha \, \Delta V$$

There are cases where this value can be exceeded - these are discussed below in the Exceptions section.

Example

Assume flow of water is instantaneously stopped by a valve closure.

ρ = 62 lbm/ft3

a = 4000 feet/sec

ΔV = 10 feet/sec

P_{steady} = 50 psig

$$\Delta P = -\left(62 \, \frac{lbm}{ft^3}\right) \left(4000 \, \frac{ft}{s}\right) \left(-10 \, \frac{ft}{s}\right)$$

$$\Delta P = \left(62 * 4000 * 10 \, \frac{lbm}{ft \cdot s^2}\right) \left(\frac{1 \, lbf \cdot s^2}{32.2 \, lbm \cdot ft}\right) \left(\frac{1 \, ft^2}{144 \, in^2}\right)$$

$$\Delta P = 535 \, psi$$

$$P_{max} = \Delta P + P_{steady}$$
$$P_{max} = 535 \, psi + 50 \, psig$$
$$P_{max} = 585 \, psig$$

Exceptions

- **Cavitation**

  When a positive pressure surge occurs, it is always followed by negative pressure surge that drops the pressure. If the drop in pressure causes cavitation, then the pressure spike that occurs when the cavity collapses can cause the local pressure to exceed that predicted by the instantaneous waterhammer equation.

  The potential to cavitate can be assessed by subtracting ΔP from the steady-state pressure. If this value is lower than the fluid vapor pressure, then cavitation is possible.

- **Pipe Size Changes**

  When using this equation, the case considered is usually a valve closure. It is frequently applied with the *ΔV* obtained based on the pipe diameter at the valve. However, if smaller diameter pipes exist upstream that carry the same flowrate, their *ΔV* will be larger because they have less area. This will result in a larger pressure surge in the smaller pipe.

  To be safe, the equation needs to be applied to the pipe with the largest *ΔV* potential in the entire system.

- **Multiple Pressure Surge Sources**

  If there is potential for multiple elements to generate pressure surge at the same time, the surge pressures can interact and add together, thus exceeding the individual surge pressures.

- **Line Pack**

  In long pipelines or shorter systems with high friction the drop in Hydraulic Grade Line may be higher than the initial pressure surge. In these cases, the flow does not come to a complete stop with the initial compression wave - some flow continues towards the closed valve at a lower velocity, compressing the fluid and therefore increasing the pressure. See Wylie (1993) for more information.

Derivation

Consider a 1 dimensional fluid flow where the local velocity has been disturbed a small amount (*ΔV*). This small disturbance causes small disturbances in pressure (*ΔP*) and density (*Δρ*), along with a wave that travels with speed *a*. This is represented in the below control volume.
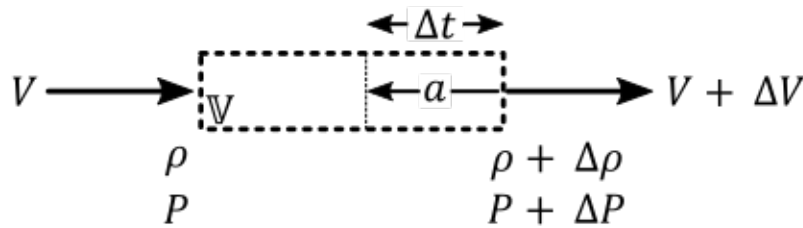
Figure 1: A downstream change in velocity causes a change in density and pressure, which propagates upstream as a wave.

First, we apply conservation of mass to the control volume above.

$$\oint_{CS} \rho \left(\vec{V} \cdot \hat{n} \right) dA = -\frac{d}{dt} \int_{C\!\!\!V} \rho \, d\!\!\!V$$

The left side of the equation, relating flow into and out of the control volume, reduces to:

$$\oint_{CS} \rho\,(\vec{V} \cdot \hat{n})\,dA = (\rho + \Delta\rho)(V + \Delta V)\,A - \rho\,V\,A$$

The right side of the equation, relating the change in mass within the control volume, reduces to:

$$-\frac{d}{dt}\int_{C\rlap{\,\diagdown}V}\rho\,d\rlap{\,\diagdown}V = -\frac{\Delta\rho\,\rlap{\,\diagdown}V}{\Delta t}$$

The density is only changing in the area swept by the wave front (moving at speed *a*, against the fluid velocity) over the time step. Therefore, we only need to include the change in density over this volume, as the other terms would cancel out. The volume swept by the wave is:

$$\rlap{\,\diagdown}V = (a - V)\,A\,\Delta t$$

reducing the above to:

$$-\frac{d}{dt}\int_{C\rlap{\,\diagdown}V}\rho\,d\rlap{\,\diagdown}V = -\Delta\rho\,(a - V)\,A$$

equating the sides and canceling the common *A* term:

$$\boxed{(\rho + \Delta\rho)(V + \Delta V) - \rho\,V = -\Delta\rho\,(a - V)}$$

Next, applying conservation of momentum:

$$\sum F = \frac{d}{dt}\,(m\,V)$$

The forces involved are simply the pressures over the areas. Because the pipe is considered *frictionless* for this method, the upstream and downstream pressures are equal.

$$\sum F = P\,A - (P + \Delta P)\,A$$

The right hand side of the equation can be expanded into a time derivative and flux term:

$$\frac{d}{dt}(mV) = \frac{d}{dt}\int_{C\kern-0.5em\vee} \rho V\, dV + \oint_{CS} \rho V\, (\vec{V} \cdot \hat{n})\, dA$$

Where the time derivative term follows similar logic to the mass conservation, with the volume affected defined by the swept area of the wave:

$$\frac{d}{dt}\int_{C\kern-0.5em\vee} \rho V\, dV = \frac{[(\rho + \Delta\rho)(V + \Delta V) - \rho V](a - V)\, A\, \Delta t}{\Delta t}$$

The flux term is relatively straightforward:

$$\oint_{CS} \rho V\, (\vec{V} \cdot \hat{n})\, dA = (\rho + \Delta\rho)(V + \Delta V)^2\, A - \rho V^2\, A$$

Combining and simplifying:

$$\boxed{-\Delta P = [(\rho + \Delta\rho)(V + \Delta V) - \rho V](a - V) + (\rho + \Delta\rho)(V + \Delta V)^2 - \rho V^2}$$

The two boxed formulas, representing conservation of mass and momentum, can be combined and simplified to the instantaneous waterhammer equation:

$$\boxed{\Delta P = -\rho\, a\, \Delta V}$$

Related Topics

Related Blogs

Line Packing: Recovering the Irrecoverable

Typical Approaches to Reducing Transient Pressures

General Considerations

Excessively high surge pressures can burst pipes and damage equipment. Low pressure surge can result in pipe collapse if the difference between internal and atmospheric pressure is high, cavitation and liquid column separation, and sub-atmospheric pressures unacceptable for drinking water pipelines.

If we consider the simplified case of the instantaneous waterhammer equation, pressure change is directly proportional to the fluid's density, wavespeed, and change in velocity. Therefore, a reduction in pressure response requires a reduction in density, wavespeed or velocity change.

While lowering the fluid density does reduce the pressure response, it is not usually a practical solution.

Note: The below suggestions are some common ways to mitigate pressure surge. This page is not intended to cover all possible means of reducing pressure change.

Lowering the Wavespeed

If the wavespeed of the fluid can be lowered, the pressure response will also be lowered. There are a couple of ways that the effective wavespeed can be lowered:

- Change the construction of the pipe to deform a greater amount under stress.
    - The simplest way to accomplish this is to use a more flexible pipe material.
    - It is also possible to do this by changing the pipe geometry. Under internal pressure, any pipe will want to become circular. An already circular pipe cannot deform to become "more" circular, however a non-circular pipe (such as a rectangular duct) can deform more easily. This deformation action increases the effective elasticity (decreases E), decreasing the wavespeed.
- Change how much the liquid deforms under stress.
    - Entrain gas in the liquid. In effect, this makes the bulk modulus of the liquid lower.

Lowering the Rate of Velocity Change

The other practical way to reduce transient pressures is to reduce the rate of velocity changes in the system. There are a few ways that this can be accomplished:

- Slow down controlled equipment as much as possible.
    - Slow down the closure of valves. Making a valve closure as slow as possible will reduce the transient pressure response as much as possible.

- It is important to note here that the closure time of a valve does not proportionally relate to the change in velocity at the valve except in special cases. An oversized valve may close almost all the way before any significant pressure rise is seen. This is because there may be very little pressure loss across the valve initially, leading to very small velocity change. The *effective* closure time may be significantly less than the entire actuation period. In cases like this the pressure response can often be dramatically improved by modifying the pressure loss vs. time characteristics of the closure, rather than the total closure time.

  - A speed controlled pump will cause fluid transients on speed change - slowing this speed change down reduces surge.

  - Any other similar change - variable flows, pressures, control setpoints, etc have a similar effect.

- Reduce the speed at which components adjust to the system.

  - Control Valves that quickly change position in an attempt to maintain control can worsen a transient event. Limiting their opening and closing rates can lessen this.

  - Relief Valves that are poorly designed can open and close very quickly, again propagating pressure waves they should mitigate.

  - Check Valve slam can be problematic - and preventing it may require changing the system or other equipment.

  - Tripping or starting pumps causes significant pressure response. This rate can be reduced by increasing the inertia of the pump and driver assembly - making the pump take longer to stop or start.

- Add equipment to mitigate high or low pressures.

  - If using this type of device, it is generally desirable to locate the device as close as possible to the cause of the transient. The practicality of the different options depend highly on the particular case of interest.

  - Relief Valves - Avoid excessive pressures by providing a means for fluid release. Inward relief valves can be used for low pressure situations.

  - Air Valves - Allowing ambient pressure gas into the pipe during low pressure surge raises the system pressure. Usually located at high elevation points along a pipeline.

  - Surge Tanks - Surge tanks can be an effective way to reduce surge pressures if the steady-state system pressures are relatively low. High steady-state pressures result in high hydraulic gradelines. Since the liquid level in a surge tank rises to the local hydraulic gradeline, they would have to be very tall if located in a high pressure region.

  - Gas Accumulators - Gas accumulators can be effective in spreading out the wave front of a transient and thus reducing peak pressures. Gas accumulators change the frequency response of the system and can in fact amplify pressure surge if not sized and located properly.

  - Check Valves - Appropriately located check valves can reduce transient pressures in the system by preventing reverse flow.

- Parallel Valves - Rather than use one large valve to stop the flow, perhaps two or more valves can be used in parallel. The timing of the valve closures is staggered, thus allowing a portion of the flow to be stopped with each valve.
- Modify the piping
  - Increasing the diameter of a pipe decreases velocity for the same flow. With a lower initial velocity, the maximum potential velocity change is reduced.
  - Appropriately placed bypass piping can reduce pressure responses.
  - As with any wave phenomenon, the system may have resonant effects at certain frequencies. Changing the length or configuration of certain pipes can reduce (or increase) the pressure response. Adding dead end pipes tuned to a certain frequency is one method of accomplishing this.

The above and additional methods for surge reduction or discussed in detail in Swaffield et al. (1993, Chapter 6), Wylie et al. (1993, Chapter 10), and Chaudhry (1987, Chapter 10).

Related Topics

Related Blogs

Raise the bar using Vacuum Breaker Valves

A Clever Way to Mitigate Waterhammer During a Valve Closure

Transient Forces

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

Contents

Related Blogs

Q&A: Calculating Transient Forces for Pipe Stress Analysis

You Should Combine Your Pipes

Defining Forces

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

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

Defining Force Sets

In order to process force data, the locations where the forces are to be calculated must be defined. Force Sets can be defined either from the Workspace or from the Force Definitions panel of the Forces group in Analysis Setup.

Defining from Analysis Setup

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

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

Each individual force set has a number of required parameters:

- **Force Type** - There are several force set types available. They are described in detail in Force Set Types.
- **Force Set Name** - A unique name for the force set.
- **Start Pipe** - Where the first (only for Point type) node is located.
- **Length To Start Node** - The length from the beginning of the pipe to where the force set starts.
- **Start Station (Actual Length)** - Because calculations must be performed at calculation stations (nodes), the exact length entered in the previous field may not be a valid

calculation point. In this case, the nearest valid node will automatically be determined and displayed here. **Keep in mind that the actual force set is between the two "Actual Length" fields, and can be different than the entered values**.

- **End Pipe** - The second (for difference types) node. Note that the selection must be valid for the specified force type. See Force Set Types.

- **Length To End Node** - The length from the start of the pipe to the end node.

- **End Station (Actual Length)** - As described above, the nearest calculation station to the entered length.

- **Pipe Group** - For force sets across 3+ pipes, a group is required and is selected here.

- **Exit Area** - For Exit type forces, an exit area is required.

- **Force Unit Vector Coordinates** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. Impulse is still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the Export Force Data tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

Defining from the Workspace

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

Figure 2: Create Force Set window in the Workspace

Each force has several of required and optional parameters:

- **Pipe Name** - If the pipe has a name entered other than 'Pipe', the user can choose whether to include that pipe name or not in the Force Set name.

- **Force Set Name** - Users can add a name for the Force Set to better organize and manage Force Sets in the model.

- **Force Unit Vectors or Length** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. Impulse still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the Export Force Data tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

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

Related Topics

Related Blogs

Q&A: Calculating Transient Forces for Pipe Stress Analysis

Five Fantastic New Features for Fast Fluid Transients: Impulse 8

You Should Combine Your Pipes

Force Set Types

There are several types of forces that can be defined for analysis with AFT Impulse:

- **Point** - Point forces are simply the difference in pressure between the fluid and ambient multiplied by the pipe area. This can be useful for places where the fluid is leaving the system, such as a relief valve or exit valve.

- **Difference** - Difference forces are defined as the unbalanced force between two nodes, or computation stations along two sequential pipes. An example of this is the hydraulic loading between two elbows along the length of a pipe. See Figure 1 below.

- **Difference Exit** - Similar to the difference force set in that two nodes are required. However, the user defined exit area will be used for calculating the force across the section. This is useful for modeling forces on something such as a nozzle.

- **Difference (3+ Pipes)** - The same as difference forces, but spans across more than two pipes. A pipe group must be defined that contains the pipes between the two locations. The Difference (3+ Pipes) force set type must still include only sequential pipes.

- **Difference Exit (3+ Pipes)** - The same as difference exit, but spans across more than two pipes, requiring a group.
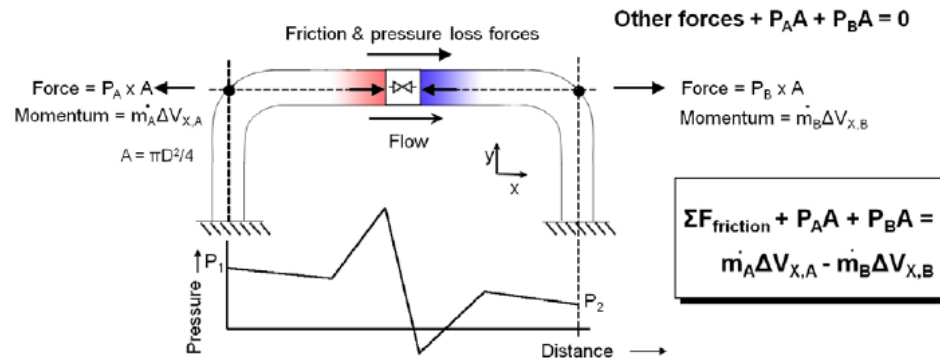
Figure 1: Difference force type schematic. For Difference Exit, the area used at End Node is user defined.

Related Topics

Related Blogs

Q&A: Calculating Transient Forces for Pipe Stress Analysis

Exporting Force Data

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

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

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

Note: Force Set unit vectors can only be exported to CAESAR II (v12 and higher) and AutoPIPE. To export any Force Sets to AutoPIPE, users must have defined force unit vectors for each Force Set.
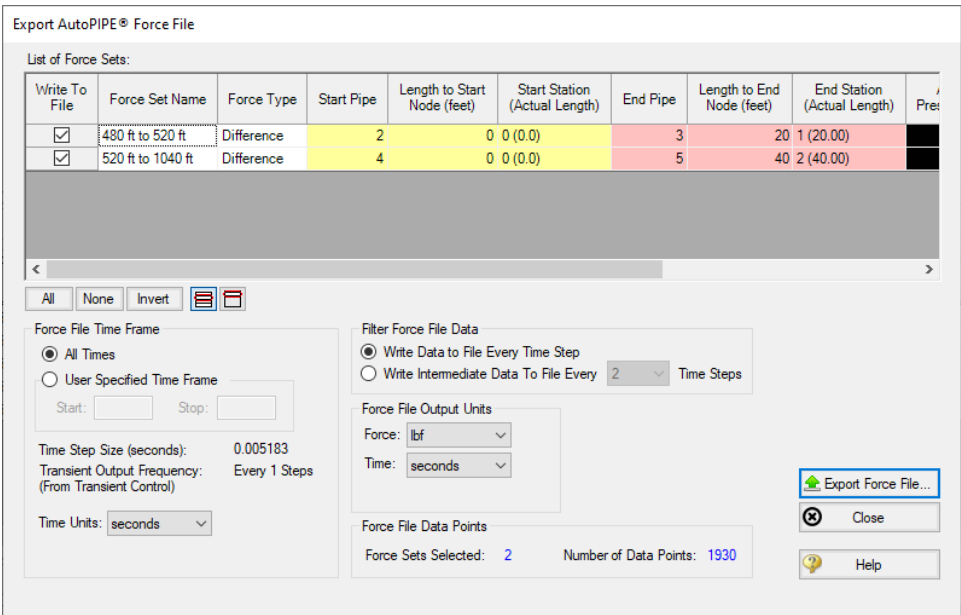
Figure 1: Exporting force results to an AutoPIPE Force File

Related Topics

Force Theory

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

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

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

- All piping and components are assumed to be completely rigid. No deflections or deformations are considered.

- The force due to gravity is neglected.

- The volume of any junctions (bends, valves, etc.) is considered negligible. This is consistent with how junctions are modeled throughout AFT Impulse.

- All Difference Force sets comprise a pipe or set of pipes that are axially aligned between the beginning or ending nodes.

- Friction effects are modeled using the steady-state Darcy-Weisbach equation throughout the entire simulation.

- The total force acting on the piping is assumed to be equal and opposite to the forces acting on the fluid.

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

Description of Force Calculations

An example case for a difference force set can be considered for a stretch of pipe at constant elevation with a valve in the middle. The nodes on either end of this Force Set are a pair of elbows, as depicted in Figure 1 below.
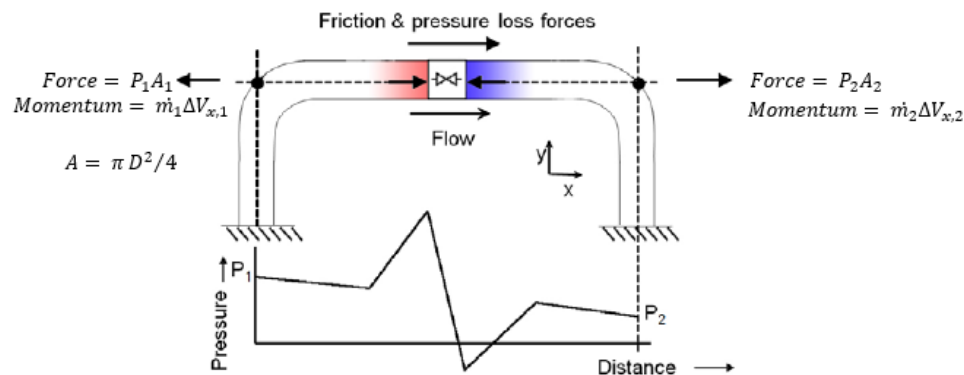
Figure 1: Simple force diagram for a fluid flow in a pipe

With flow through such a pipe, there are several forces acting on the fluid, including friction, pressure, gravity (which AFT Impulse neglects for force set calculations), normal forces from the pipe to the fluid, and forces due to redirecting the flow through the elbow's directional changes (i.e., momentum changes). Only the forces which are axially aligned with the pipe will be considered. The forces upstream (1) and downstream (2) are strictly related to the static pressure and flow area at those points.

Using Newton's second law, the momentum changes and forces can be related, and we can solve for the overall reaction force, Fx. Equation 1 below shows the complete force balance used by AFT Impulse.

$$F_x = P_1 \, A_1 + \dot{m}_1 \, V_1 - F_{friction,Pipe \ A}$$
$$+ \left[P_{dn} \, A_{dn} - P_{up} \, A_{up} - \dot{m}_{up} \, V_{up} \right] \quad (1)$$
$$- F_{friction,Pipe \ B} - P_2 \, A_2 - \dot{m}_2 \, V_2$$

Note that Figure 1 does not show an area change at the valve, as shown below in Figure 2. If upstream and downstream pipe areas are identical, the momentum terms for the junction will always cancel out – in steady and transient conditions. AFT Impulse will account for this momentum change when it exists.

Steady State

For steady flow the velocity through a pipe is constant. Thus, the momentum terms will cancel out, leaving us with Equation 2 below.

$$F_x = P_1 \, A_1 - F_{friction,Pipe \ A}$$
$$+ \left[P_{dn} \, A_{dn} - P_{up} \, A_{up} \right] \quad (2)$$
$$- F_{friction,Pipe \ B} - P_2 \, A_2$$

The force due to friction is equal to the pressure drop due to friction multiplied by the flow area.

$$F_{friction} = \Delta P_{friction} \, A \quad (3)$$

In steady state, the only source of pressure drop (at constant elevation) is friction. Therefore, the friction and pressure terms in Equation 2 cancel out, resulting in a net force of zero at steady state.

$$F_x = \left[(P_1 \, A_1 - P_{up} \, A_{up}) - F_{friction,Pipe \ A} \right]$$
$$+ \left[(P_{dn} \, A_{dn} - P_2 \, A_2) \right. \quad (4)$$
$$\left. - F_{friction,Pipe \ B} \right] = 0$$

Transient Conditions

What happens when we consider the above system in a transient case? We may have a large pressure wave propagate through this section of pipe, perhaps due to the instantaneous closure of a valve somewhere downstream:
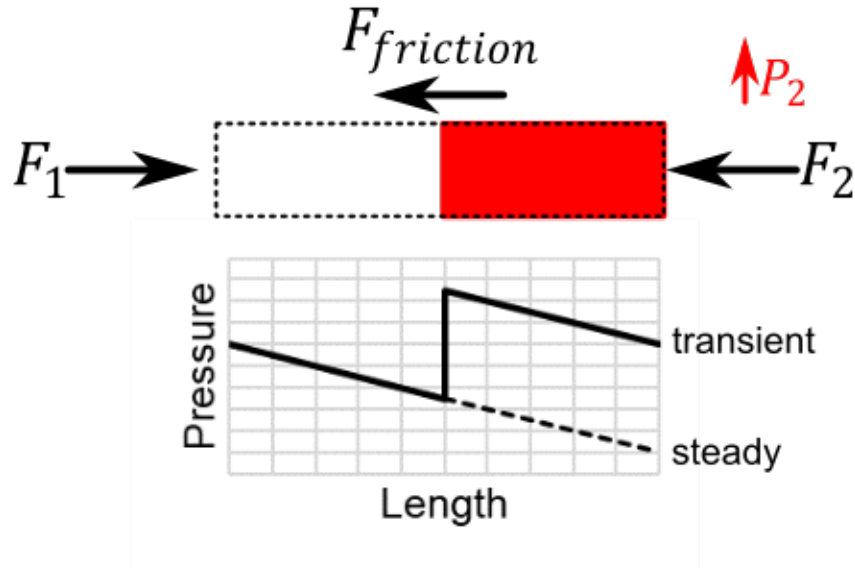
Figure 2: Simple case during a fluid transient, with a pressure wave travelling upstream

Pressure and flow in the pipe are no longer at steady state, meaning we must calculate every component of Equation 1. Additionally, the force due to friction varies over the force set, since flow is no longer constant along the length of the pipe. To accurately account for the friction force, we must sum the force from each section.

$$\sum F_{friction,pipe} = \sum_0^{N\ sections} \sum F_{friction,section} \qquad (5)$$

The entire balance is calculated for a single time step, and must be repeated for every time step to determine the dynamic forces on the force set.

Force Components

In some fields, it is common practice to not include all terms of the force balance in calculations.

The simplest method is to look only at the forces due to pressure and momentum changes across junctions. The sum of forces in this force balance will not be zero even at steady state, as the friction and total momentum balance terms are not included in the calculation.

With AFT Impulse 7 and earlier, the option to ignore the friction and overall momentum terms was presented to users by default. Starting with AFT Impulse 8, those options were moved into the Advanced Parameters panel. Since those terms are often a significant part of a complete force balance, and are always real forces, it is strongly recommended to always include the momentum and friction force balance components, as they will give more accurate results.

Related Topics

Related Blogs

Q&A: Calculating Transient Forces for Pipe Stress Analysis

Warnings For Pipe Force Calculations

Transient Cavitation Modeling

Transient pipe forces calculated in Impulse attempt to capture the imbalanced hydraulic loading on a series of pipes caused by a pressure wave moving through those pipes. Transient cavitation can cause significant pressure waves in a system while also adding inherent uncertainty to the results. These factors combine such that transient forces calculated while cavitation is present in an Impulse model are also uncertain and should be treated with caution. For more information, see the discussion on interpreting cavitation results.

Variable Density

When the density is variable, AFT Impulse cannot accurately account the hydropressure differences that affect force calculations. The magnitude of the error introduced by variable densities across the length of a Force Set is proportional to the density differences.

Related Topics

Related Blogs

Q&A: Calculating Transient Forces for Pipe Stress Analysis

Interpreting Transient Results

This section discusses how to begin understanding complex transient results. Topics included discuss how to use the output animation features in the Graph Results and Visual Report windows, and how to interpret and troubleshoot the model when transient cavitation or artificial transients occur.

Contents

Animating Graph Results

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

Creating an Animated Graph

Since animations display information along a flow path, only Profile type graphs may be animated.
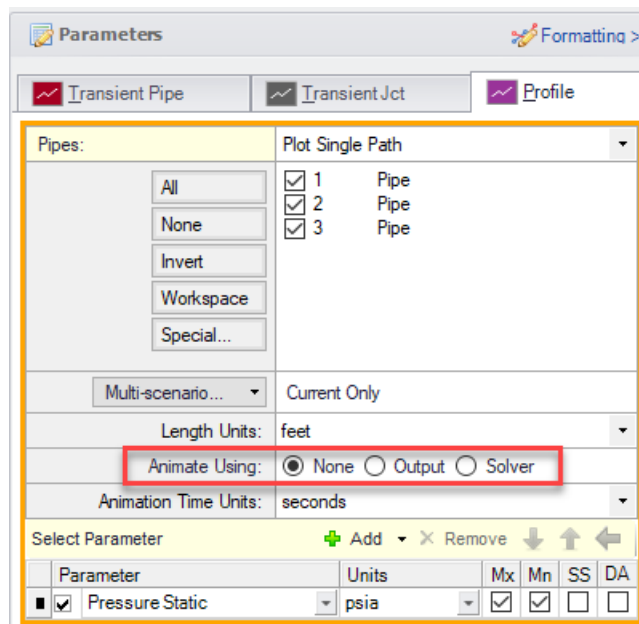
Figure 1: Options for animation under Profile type graphs in the Graph Results window

There are two ways to animate:

- **Animate Using Output** - This option uses data saved to the transient output file. This option requires that the data for all stations in the path of interest was saved to the output file. This method is generally preferable as it is faster and allows the user to advance (or reverse) the animation to any given point. The only disadvantage of using Output for animation is that the required output files may be large.

- **Animate Using Solver** - Instead of using saved data, this option re-runs the transient solver to generate the required data on the fly. This has the advantage of reducing output file size, but the animation must be displayed step-by-step and the transient calculation will be slower than reading from a file. For smaller models with graphs you do not intend to revisit, this may be acceptable. Generally, it is easier to save the data and use Animate Using Output. This option is not available for Multi-Scenario graphs.

Using an Animated Graph

Once the animated graph is created, it will look much like a normal Profile graph, but a few components have been added.
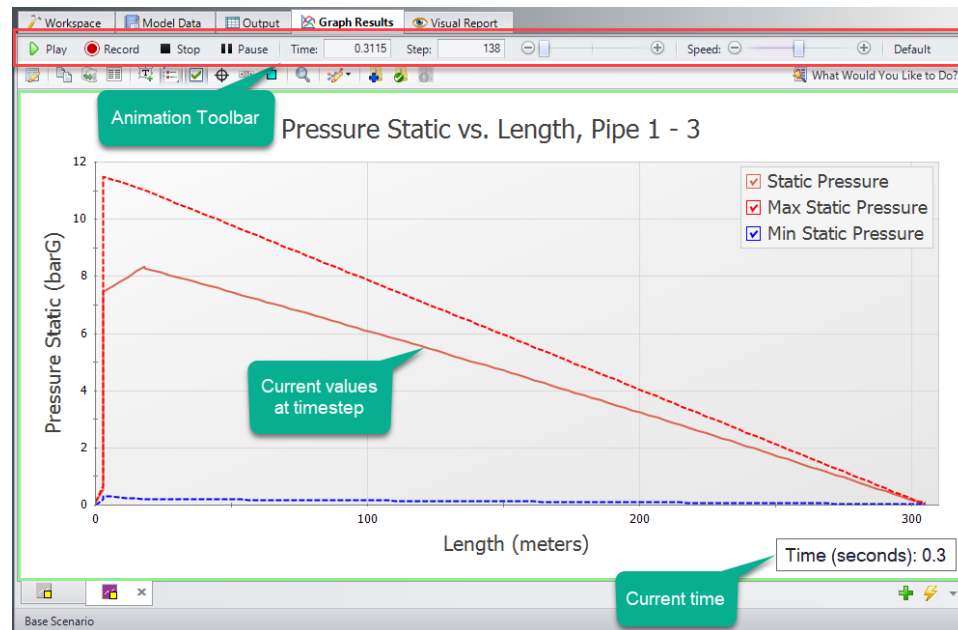
Figure 2: Components added to the Graph Results window when animating a Profile graph

The animation toolbar has several controls:

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

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

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

Using Animations to Interpret Behavior

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

Like normal Profile graphs, multiple parameters can be displayed and animated as a single Graph List Item. The images below show the relationship between pressure and flow (velocity) for this case.
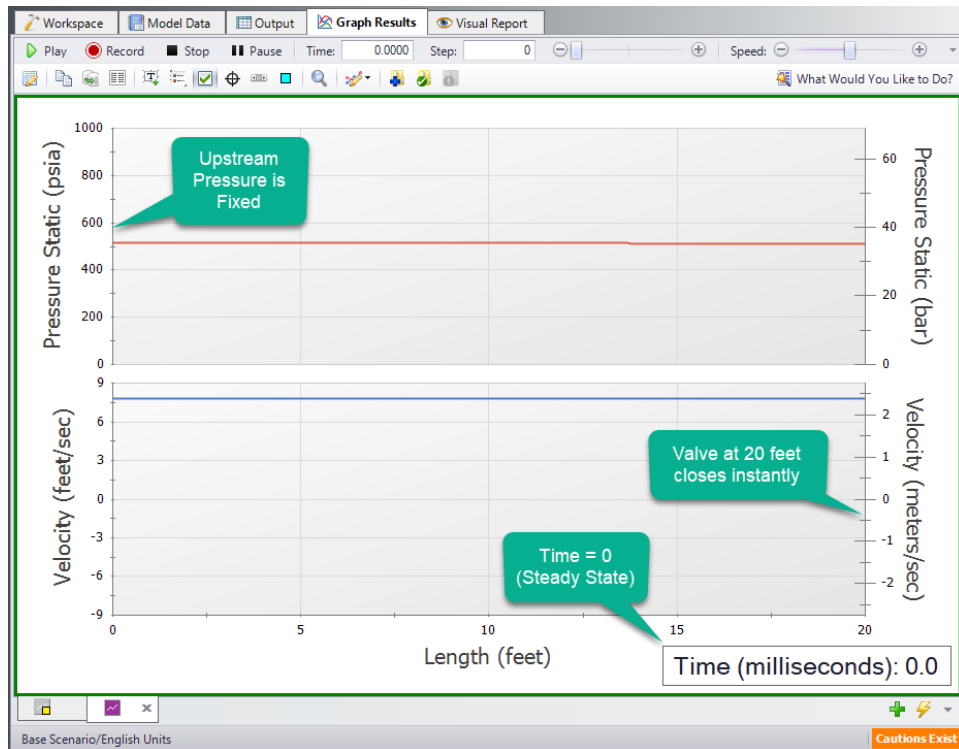
Figure 3: Animation showing steady-state (time step zero) results
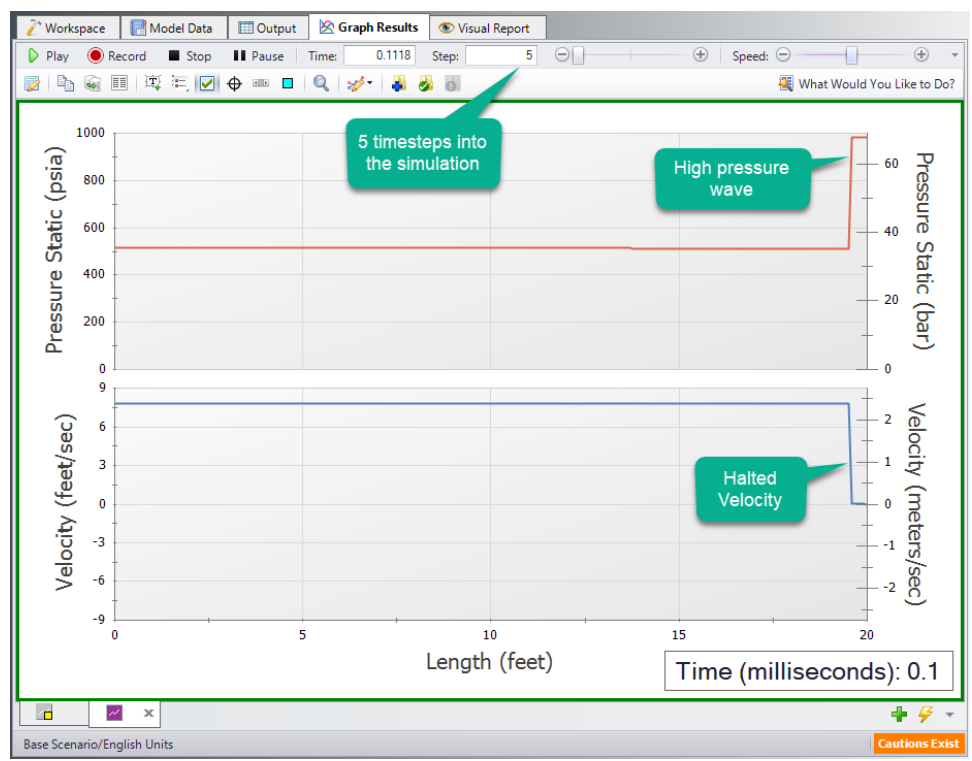
Figure 4: The instant closure of the valve halts the flow, causing a large rise in pressure
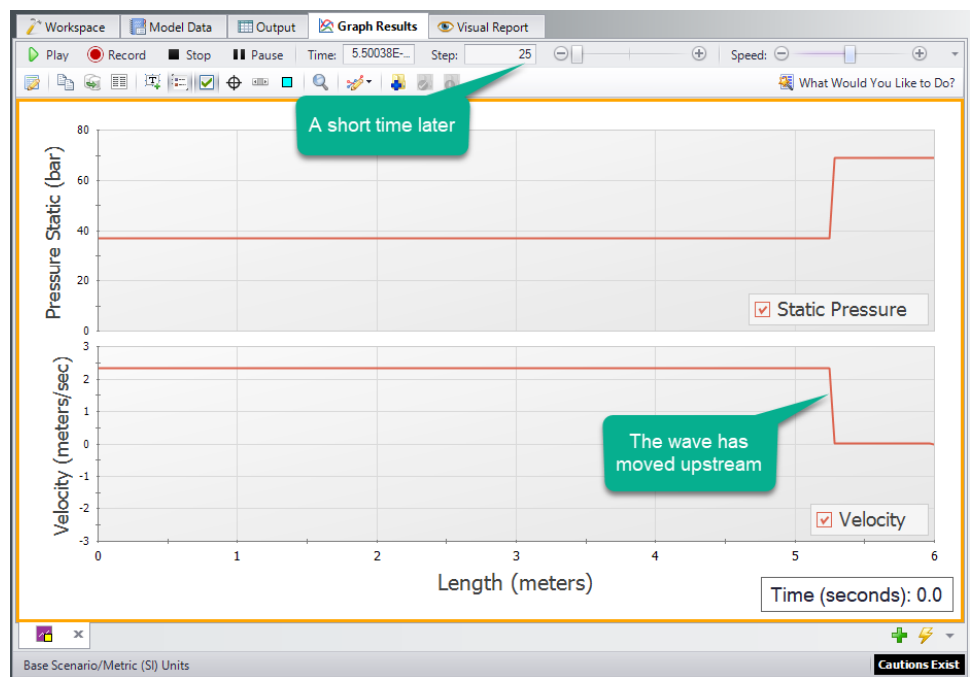
Figure 5: The movement of the wave within the pipe can be directly visualized

Related Topics

Animating Visual Report Results

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

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

Saving Output for Animation

The Visual Report Animation feature works by referencing the output for each computational pipe section in the animated pipes. In order to accomplish this, data for each pipe station being animated must be saved to the output file. By default, Impulse will only save the data for the inlet and outlet pipe stations. To change this the settings will need to be adjusted to save All Stations for the pipes being animated by, which can be done in the Output Pipe Stations panel in the Pipe Sectioning and Output group.

> **Note**: Saving data for all stations will slow down the model run time, and may require a very large output file. To save time, it is recommended to save output only for the pipes which will be animated.

Creating an Animation

To enable animation in the Visual Report, The Display Mode will need to be set to "Animate" in the Visual Report Control window.

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

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

It should be noted that once the animation settings have been changed for a scenario, those settings will be retained even if the scenario is re-run. Each scenario can retain different Visual Report Settings if desired; however, there is no way to save multiple Visual Report configurations within the same scenario.
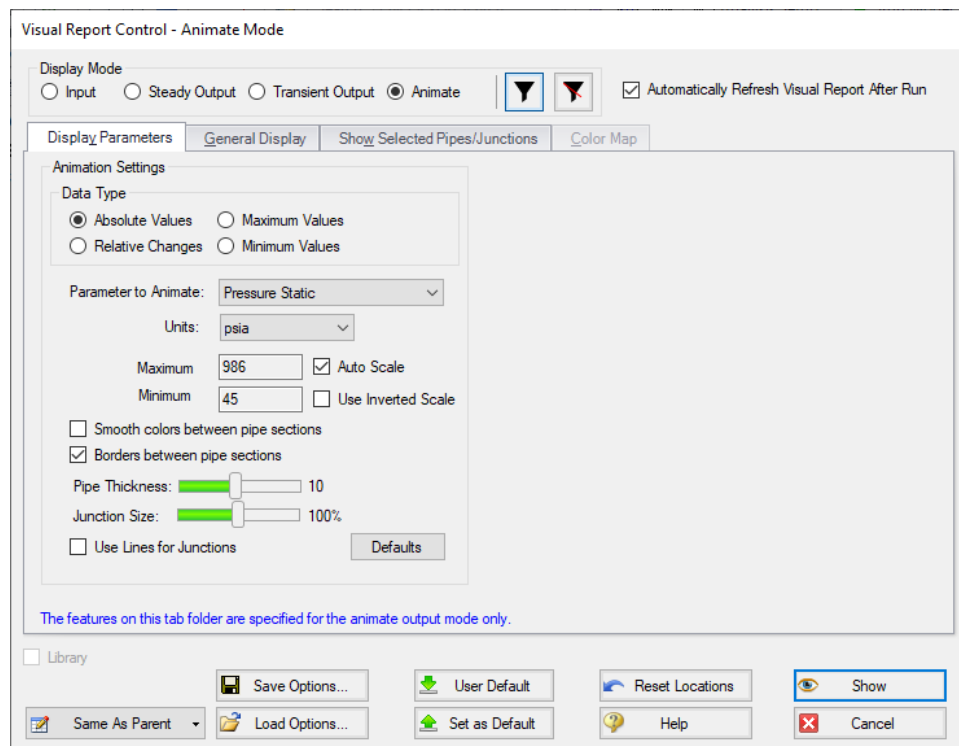
Figure 1: Options for the Animate display mode in Visual Report Control

Viewing Animation

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

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

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

The animation toolbar has several controls:

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

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

Using Animations to Interpret Behavior

There are many ways that animations are helpful in interpreting behavior. Below, we consider the simple case of a valve closure.
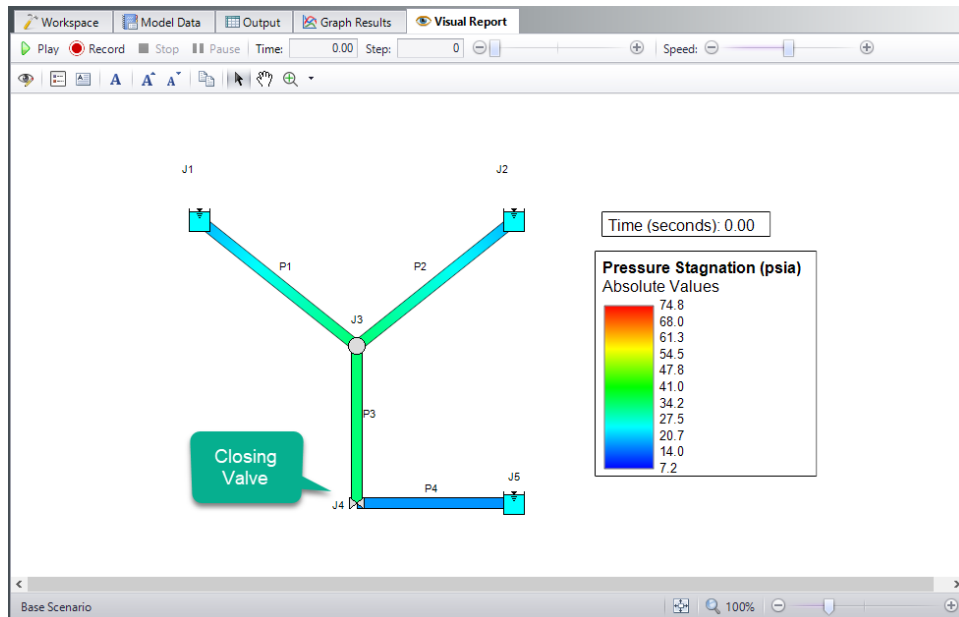
Figure 3: Visual Report animation showing steady-state (time step zero) results
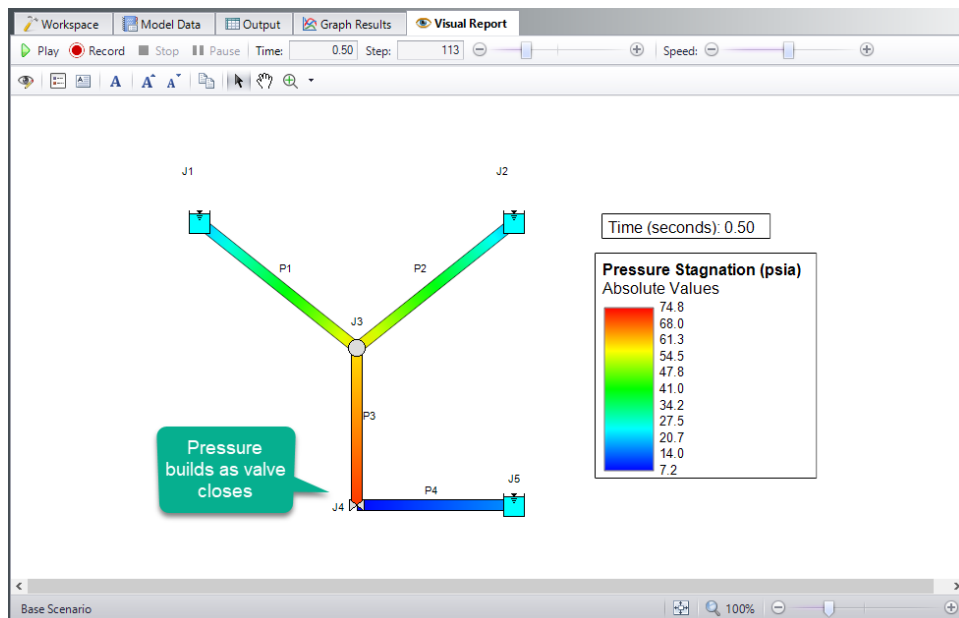
Figure 4: Pressure builds as the valve rapidly closes
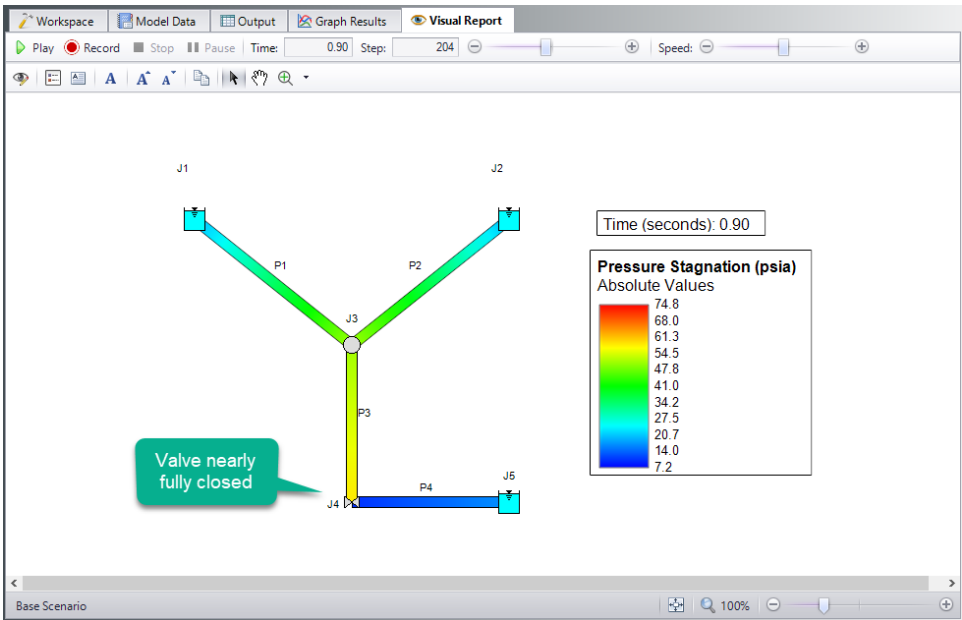
Figure 5: The movement of the wave within the pipe can be directly visualized

Related Topics

Interpreting Cavitation Results

Transient Cavitation is one of the most complex physical phenomena which AFT Impulse tries to model. Impulse's solution methodology assumes that all pipes are 100% liquid-full, yet cavitation inherently violates that assumption. As such, potentially significant uncertainty is inherent to results from any system which sees cavitation, and the user must carefully analyze their results to determine how reliable those results are.

Determining reliability is neither straightforward nor easy. The below discussion attempts to give guidance on how to best approach the problem, but finding an exact answer is almost impossible. As with any complex engineering task, the engineer must use good judgement to draw meaningful conclusions.

Percent Vapor Volume

A first indicator of how reliable results showing cavitation are, is how much vapor volume is predicted during cavitation relative to the size of the system. Impulse's cavitation models are meant to capture limited amounts of transient cavitation, and as the vapor volume in the system increases, confidence in the results decreases.

Impulse reports on Percent Vapor Volume in the Transient Max/Min tables in the Output window. Percent Vapor Volume divides the vapor volume at each station by the pipe volume for each computation station, and is reported as a percentage.

A Warning message is shown when Percent Vapor Volume exceeds 10% at any station, and a Critical Warning is shown when Percent Vapor Volume exceeds 100% at any station.

These messages are meant to convey increasing uncertainty as Percent Vapor Volume increases. When vapor volume is small relative to the pipe volume, Impulse's 100% liquid-full assumption is violated, but to a small degree, and more confidence can be placed in the results. As vapor volume increases, it grows past what either DVCM or DGCM is meant to capture, and confidence in the results decreases. When calculated vapor volume exceeds the physical pipe volume, reported results are unreliable, and the engineer must take great care when using them.

Numerical Noise

A second indicator of how reliable results showing cavitation are, is if mathematical noise is present in the cavitation results. Both cavitation models – DVCM and DGCM – perform complex, sensitive calculations which are sensitive to small changes in calculations.

There are many situations in which numerical noise can be present in the results. DVCM in particular can encounter the situation where multiple adjacent pipe stations are at or near vapor pressure, and pressure spikes generated by the formation and collapse of vapor cavities oscillate between the stations. These oscillations can grow in amplitude and generate significant noise in the system. An example of this sort of noise is shown in Figure 1 below.
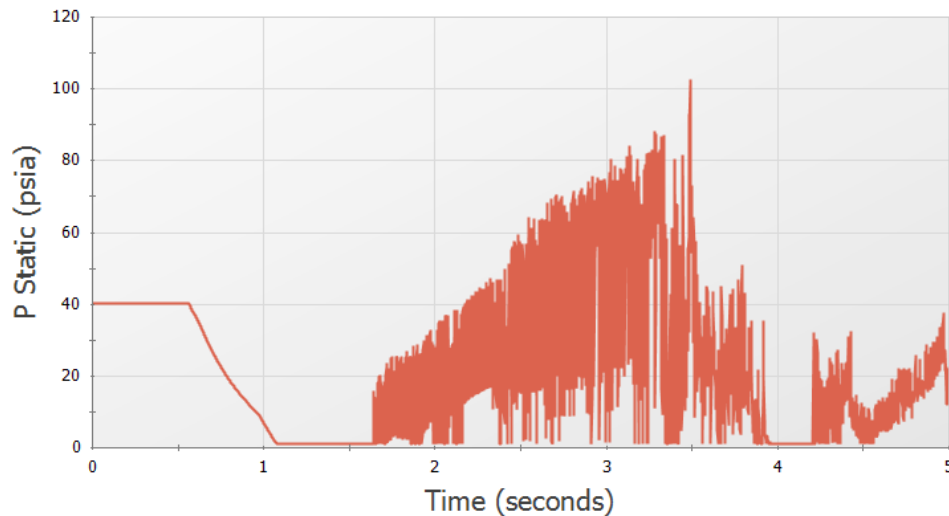
Figure 1: Numerical noise in the static pressure results for a pipe following the collapse of a vapor void.

When cavitation results with numerical noise are present, the engineer must attempt to see past the noise and find the major system trends. A key approach is by comparing results from each cavitation models. In many cases, DGCM will give results with less noise than DVCM. If both models show similar system-level trends, greater confidence can be assigned to the results without noise.

DGCM generally produces less noise than DVCM, but in some situations, DGCM can also see results with numerical noise, particularly when complex junctions such as relief valves or pumps with four quadrant curves are used. Results from each cavitation model should still be compared to evaluate what the true system response is.

Other Confidence Indicators

Numerous other indicators to help the engineer assign confidence to cavitation results exist. Discussing all of them is not practical, so four further indicators are described here.

1. **Using DVCM vs. DGCM**
   Comparing results using DVCM and DGCM gives two independent sets of results to compare. If both cavitation models give similar results, higher confidence can be assigned to those results. If each model gives different results, they should be treated with lower confidence.

2. **Sectioning**
   The sectioning scheme used for a model impacts the assumption used for both DVCM and DGCM that the vapor volume is negligible compared to the liquid volume in each section. As section count increases, the same vapor volume represents a larger percentage of the pipe volume. If results are stable as section count is changed, higher confidence can be assigned to those results. If results are not stable as section count is changed, it

indicates the assumptions behind Impulse and its cavitation models are violated, and the results should be treated with lower confidence.

3. **Vapor Void Formation and Collapse**

   Vapor void formation and collapse in a real system must behave realistically. The vapor voids must form and collapse in a continuous motion based on the presence of low-pressure waves. Multiple bubbles are likely to form, but if those bubbles are captured by a discrete model (like both DGCM and DVCM are), the model should show a single vapor void forming and collapsing rather than numerous voids with very short time spans.

   If results show behavior consistent with expected physical behavior, higher confidence can be assigned to those results. If results show excessive noise or chaotic behavior, it indicates the cavitation model may not be giving a good representation of real results. Figures 2a and 2b below compare results which show behavior that is likely to be physically realistic compared to behavior that is likely to be numerical noise.
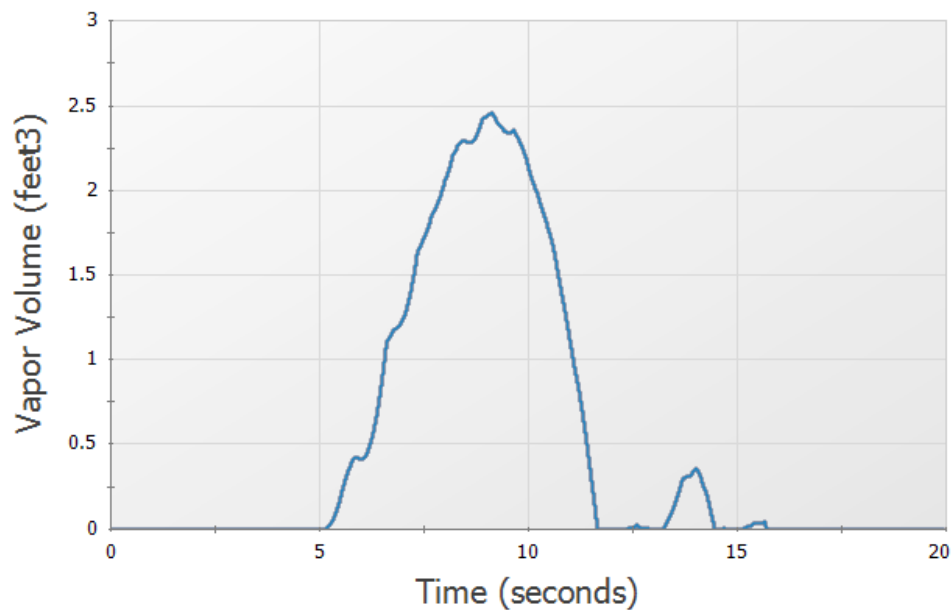
Figure 2a: Cavitation results showing smooth vapor void formation and collapse, adding confidence to the results.
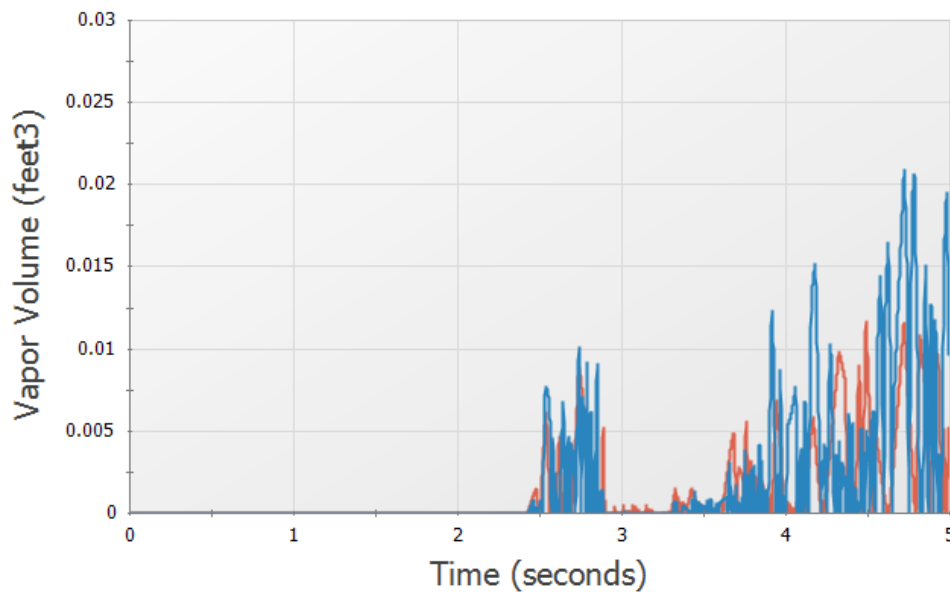
Figure 2b: Cavitation results showing noisy vapor void formation and collapse, limiting con-fidence in the results.

4. **Disabling Cavitation Modeling**

   The uncertainty inherent to cavitation modeling can make it difficult to determine the shape and magnitude of a low-pressure wave which causes cavitation. Disabling cavit-ation modeling in Impulse can allow the engineer to better see the low-pressure wave in their system to determine the extent to which it is likely to cause cavitation and to determ-ine if a subsequent high-pressure wave will form and violently collapse the vapor void.

   Note that this step is only meant for troubleshooting purposes. AFT always recommends that cavitation modeling is enabled for transient simulations as all liquids have a vapor pressure and can experience cavitation.

Transient Forces

The transient forces calculated in AFT Impulse are intended to capture the imbalanced hydraulic loading on a series of pipes caused by a pressure wave moving through those pipes. In order to have confidence in the transient force results, we must also have confidence in the simulation's pressure results.

Transient cavitation in a system inherently introduces uncertainty to the results. Specifically, it introduces uncertainty regarding the shape and timing of the pressure waves in the system. The magnitude of the transient forces calculated by Impulse is highly dependent on the shape and timing of those pressure waves, meaning that transient force results can have significant uncer-tainty when cavitation is present in a simulation. A Warning message is presented in the output to alert users of this uncertainty.

Engineers attempting to use transient force results for further analysis (e.g.: pipe stress analysis) when cavitation is present must carefully examine the cavitation results in their system to

determine the level of confidence they can place in those results. The engineer should then apply a conservative safety factor to the transient force results before performing further analysis.

Detailed Decision Tree

Interpreting transient cavitation results is complex and rarely straightforward given that each system is unique and reacts to low pressures differently. The above discussion reflects that complexity by numerous approaches the engineer can take.

Two papers published at the 2018 ASME Pressure Vessels and Piping Conference attempt to give a more rigorous decision tree to help guide the engineer as they work to evaluate cavitation results in their system and apply safety factors within the context of ASME B31.3 code. The full papers are too lengthy to include here, but a summary is presented below. Full versions of each paper are available on our website.

Cavitation Classifications

Simulation results with cavitation can be broken into four categories based on the extent of cavitation:

1. **None** - No cavitation present in the system
2. **Limited** - Maximum vapor volume percent less than 10%
3. **Major** - Maximum vapor volume percent between 10% and 100%
4. **Extreme** - Maximum vapor volume percent greater than 100%

The Limited and Major categories are broken into further categories based on where cavitation occurs and the confidence assigned to the results.

Safety Factors for Pressure Results

Four levels of safety factors can be applied to maximum pressure results based on the extent of cavitation. These safety factors are used when the engineer is working to ensure the system pressures never exceed the rated pipe pressures.

1. **Small** - 1.25:1 - Used when no cavitation is present
2. **Moderate** - 1.5:1 - Used when limited cavitation is present and the engineer has higher confidence in the cavitation results
3. **Large** - 2:1 - Used when major cavitation is present or when limited cavitation is present, but the engineer has low confidence in the cavitation results
4. **Very Large** - 3:1 - Used when extreme cavitation is present, and the system cannot be miitgated to avoid cavitation

Safety Factors for Force Results

Three levels of safety factors can be applied to transient force results based on the extent of cavitation. These safety factors are used when the engineer is reporting transient force results

for further pipe stress analysis. The recommended safety factors for force results are generally higher than those recommended for pressure results given the greater uncertainty.

1. **Small** - 1.25:1 - Used when no cavitation is present
2. **Large** - 2:1 - Used when limited cavitation is present and the engineer has higher confidence in the cavitation results
3. **Very Large** - 3:1 - Used when major cavitation is present or when limited cavitation is present, but the engineer has low confidence in the cavitation results

Related Topics

Related Blogs

How does AFT Impulse model cavitation?

Troubleshooting Artificial Transients

It is relatively clear how an Artificial Transient is detected, but diagnosing its source and fixing the problem can be much more complicated. **Often, the best course of action will be to send the model to AFT Support at support@aft.com**. The tools provided by AFT Impulse can help point you in the right direction.

First, take a look at the Output. Even though the transient simulation is halted when an Artificial Transient is detected, the Output will still provide valuable information.
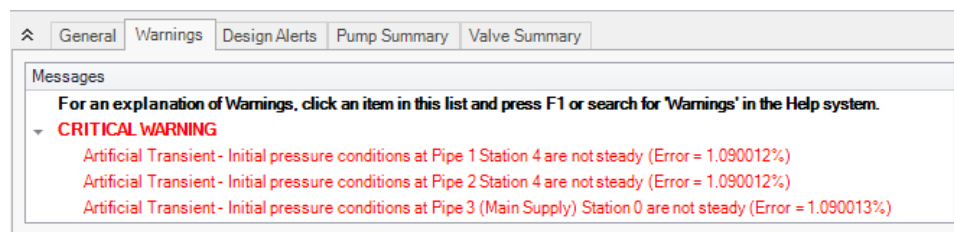
Figure 1: Artificial Transient Warnings

The Warnings tab will give information on the Artificial Transient(s). In Figure 1, we can see that the pressure is inconsistent at the entrance to pipe P1, P2, and P3. These 3 pipes connect to a branch junction. A pressure change of 1% will not cause excessively large non-real pressure transients to move through the system, but it may be enough to negatively impact the real results in the system. With this information alone, it may be possible to determine and solve the issue.

Second, if the location alone does not provide enough information, observe the behavior of the Artificial Transient. This will require allowing the transient simulation to proceed, by disabling the Stop Run option on the Artificial Transient Detection panel. To properly understand the Artificial Transient, it is **critical** that all other user-defined transients in the model are disabled. This can be most easily accomplished with a Transient Special Condition.

Note: An Artificial Transient is an affect strictly related to the beginning of the simulation. When troubleshooting, it is not necessary or advised to run a long simulation. Shorten the Simulation Duration as much as possible to make running the model quick. In many cases only a handful of time steps are needed.

The most powerful Output tool for analysis of an Artificial Transient is the Animation of a Profile Graph. By animating the behavior it can be made clear where the event originates, as well as what kind of impact it has on the system. Again, **user transients should be disabled which means the simulation *should* remain at a constant steady-state throughout the transient simulation**.
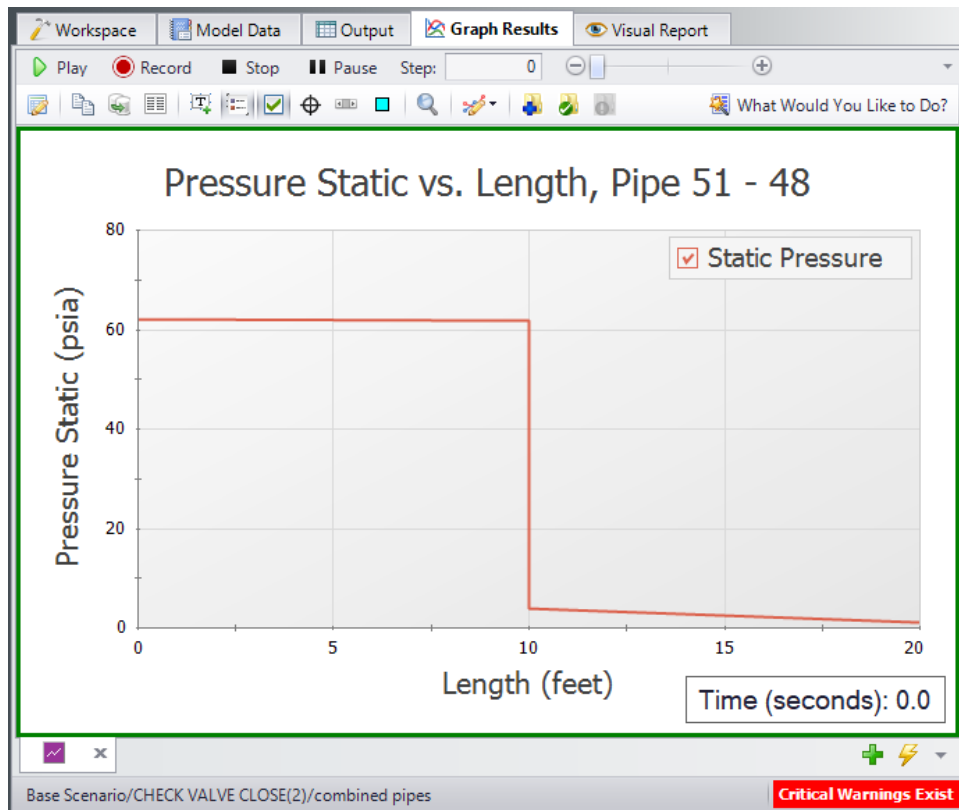
Figure 2: A profile graph showing time step zero. This represents the initial steady-state solution.

Advancing the animation **just one time step** shows a small change in the pressure profile:
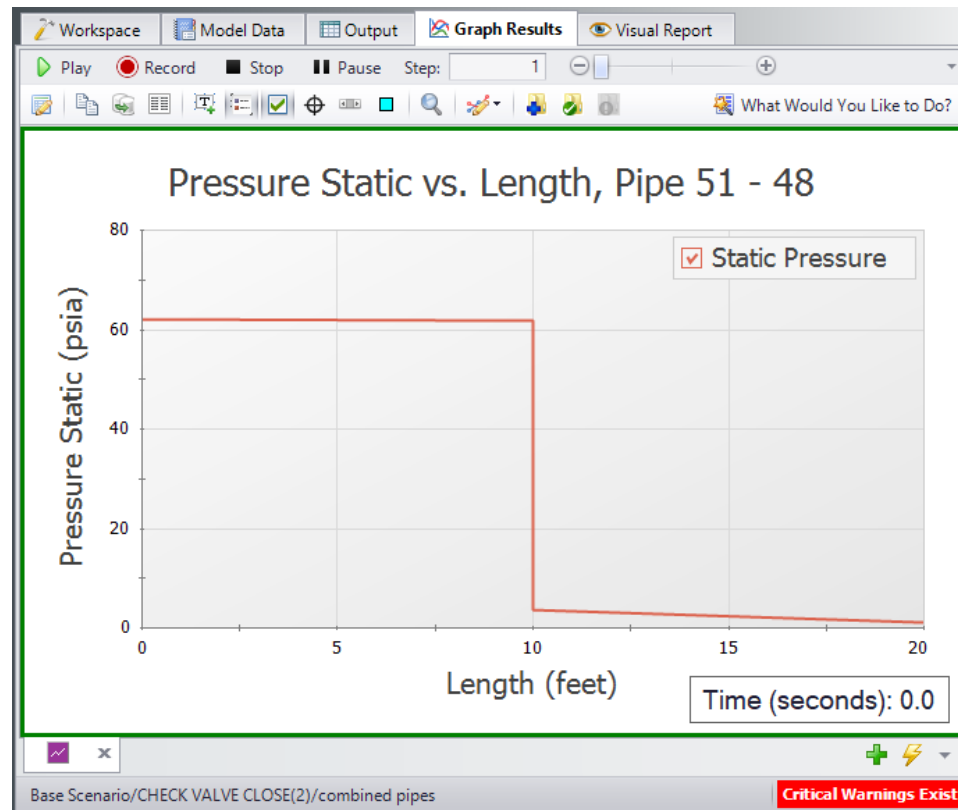
Figure 3: The same profile graph showing the first time step. There is a dramatic change in pressure at J2, as reflected by the Warnings.

By allowing the animation to proceed, an assessment of the impact the Artificial Transient has on the system can be made. In this case, the effect is likely relatively small. However, in many cases the problem may not be small enough to be considered negligible. This depends heavily on the nature of the problem and the system, making general guidelines difficult to develop.

In this case, it's difficult to say definitively if this artificial transient needs to be addressed or not. However, it's always a good idea to attempt to resolve the issue and ensure that your transient analysis is not showing non-real results.

Third, the root of the Artificial Transient must be determined if it is not acceptable. This is often the most difficult step as there is no one potential cause.

The best approach is to consider the results above as well as the assumptions made by AFT Impulse and the various junctions. In the case above, a pump junction is using a four-quadrant data set that is giving invalid data during the transient. Note that there is an additional Critical Warning in Figure 1 to this effect. While this warning is being generated 2 seconds into the simulation rather at the same time as the artificial transient, it can still give a hint to the problem.

In this case, a four-quadrant data set is being used that is not well-suited to the pump in the model. Adjusting the model to use a better-suited data set resolved both the Artificial Transient and the error later in the simulation.

Note: Do not forget to re-enable the detection of Artificial Transients!

Related Topics

Related Blogs

[Are you for real? Understanding and Troubleshooting Artificial Transients in AFT Impulse](#)

Fundamental Modeling Concepts

This section focuses on the critical concepts and theories that comprise the model building process in AFT Impulse. 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 Impulse employs two fundamental pipe system constructs: pipes and junctions.

Pipes are conduits for one-dimensional liquid flow. Pressure drop due to friction occurs in pipes.

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

In addition to balancing flow, junctions also influence the flow or pressure behavior of the system. For example, 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 transient behavior, irrecoverable pressure losses or fluid behavior.

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

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

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

It is important to specify the proper flow direction for pipes which connect to junctions such as pumps or control valves. AFT Impulse uses the flow direction of the pipes to determine in which direction to increase or reduce pressure.

Related Topics

Features for Modeling Irrecoverable Losses

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

Related Topics

Convention for Positive Flow Direction

Reference Positive Flow Direction

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

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

Convention for Flow Entering and Exiting

Many of the junction types permit fluid flow to come into or pass out of the system in a prescribed manner. The most obvious of these is the reservoir junction. Reservoirs maintain a constant pressure at a node, and the flow adjusts according to the governing equations. While in principle the gas is stored or removed from the reservoir, AFT Impulse considers the gas 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 Impulse object is that flow in is positive (that is, the object gains the flow) and flow out is negative (the object loses the flow).

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 Impulse 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 Impulse, you should follow these guidelines:

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

- Learn and use the tools in AFT Impulse 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 Impulse 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 Impulse 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 Impulse to review your results for input errors. Pay attention to the information in the output report. Don't neglect warnings in the output. Perform hand analysis checks of results.

- Bound your problem. Instead of trying to calculate the exact resistance in a network, or the right loss factor for an orifice, use your judgment, guided by data, on how much variation might occur. By running your model at the upper and lower limits, you can obtain answers, for example, on the maximum and minimum possible pressure drops in the system. If the results at the maximum and minimum are acceptable, your analysis is probably finished. If they are not acceptable, you can further refine the input model.

- Recognize whether your problem is one of maximization or minimization. That is, recognize whether the design goal is to minimize the flow rate, for example, or maximize the flow rate. Then make your assumptions conservative enough to provide high confidence that your design requirements are being satisfied.

Related Topics

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 Impulse 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 Impulse and if you try and run this model, AFT Impulse will inform you that it cannot run it because of the lack of a reference pressure (Figure 2). In AFT Impulse there are several junctions that can act as a reference pressure including the reservoir, assigned pressure, valve (with the exit valve option), spray discharge, 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 assigned flow mode), the Need Reference Pressure window (shown below) is displayed. This window informs you which junctions bound the sections where a reference pressure is needed to obtain a unique solution.

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

See Role of Pressure Junctions - Detailed Discussion (Long) for more information.
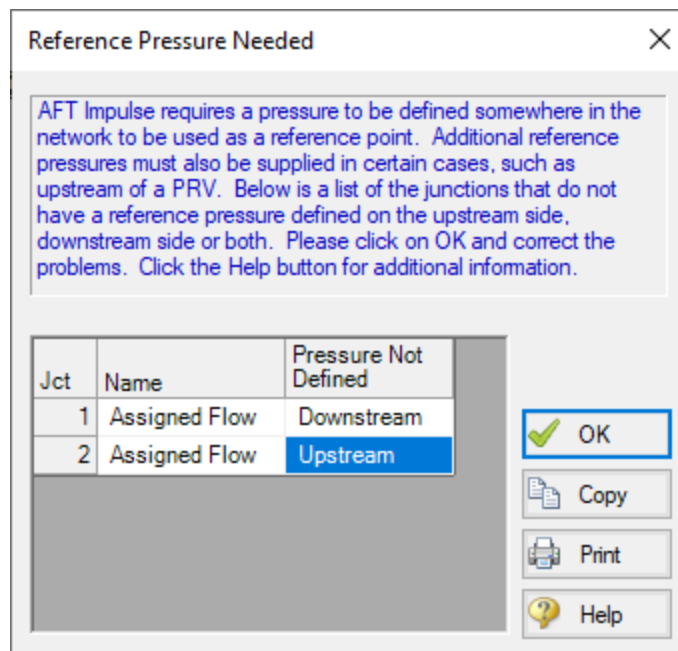
Figure 2: AFT Impulse message when you try to run a model without sufficient reference pressures

How Pressure Junctions Work

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

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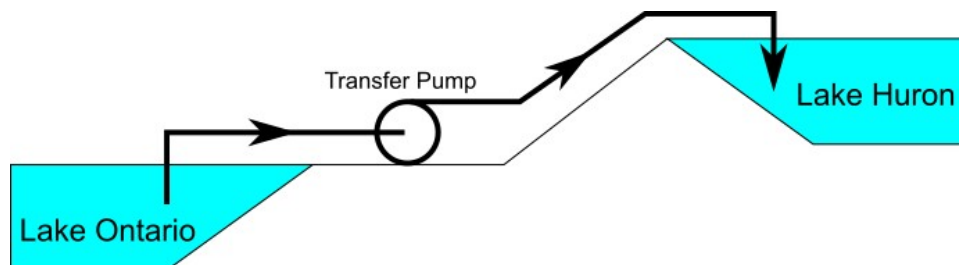
Figure 3: Physical pumping system to transfer water from shoreline supply tank (J1) to discharge tank (J2) at top of hill

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 Impulse 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.
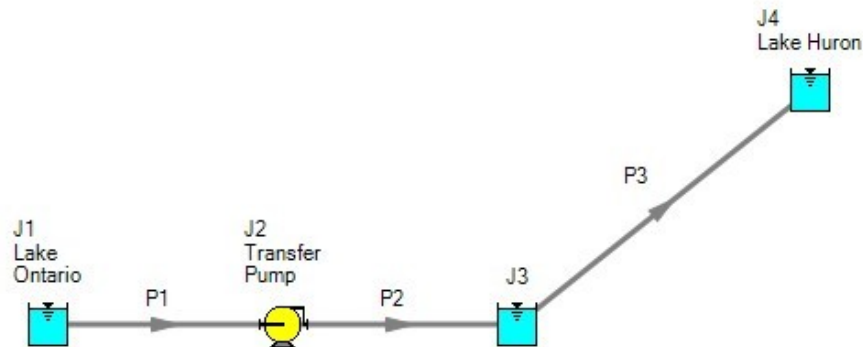
Figure 4: Model to size pump attempting to represent system in Figure 3. It does not accurately represent Figure 3.
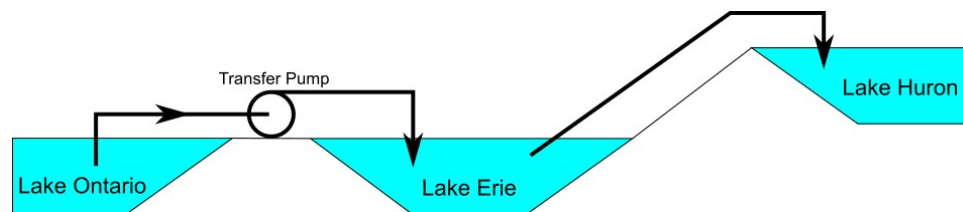
Figure 5: Physical system represented by model in Figure 4.

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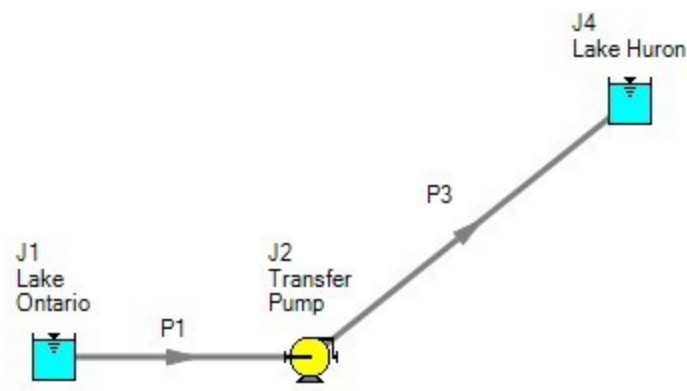
Figure 6: Proper model of physical system shown in Figure 3.

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

Related Blogs

Reference Pressure Needed: Where It Comes from and How to Address it

Open vs. Closed Systems

Model a Closed Loop System

A closed circulating system can easily be modeled with AFT Impulse 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 out-flow rate for the system.

Balancing Mass

In order to model a closed system, only one pressure junction is used in the model. Typically this would be 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 Impulse views a closed system model. AFT Impulse 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 Impulse's Steady-State 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.
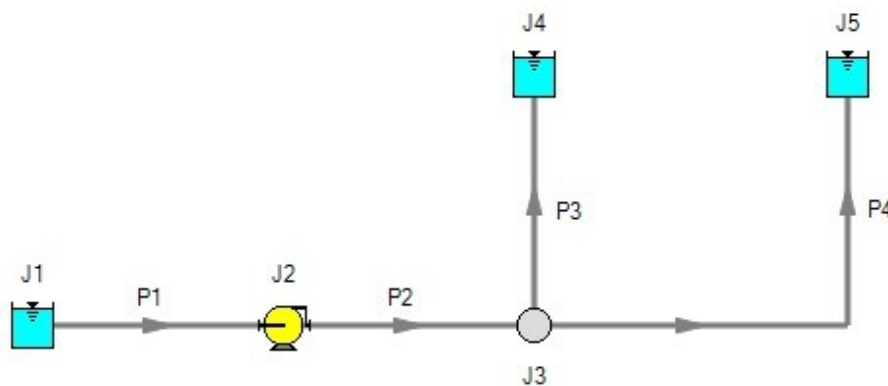
Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5

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

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

Impulse is not applying a mass balance to J1 directly. It is merely the result of an overall system balance.
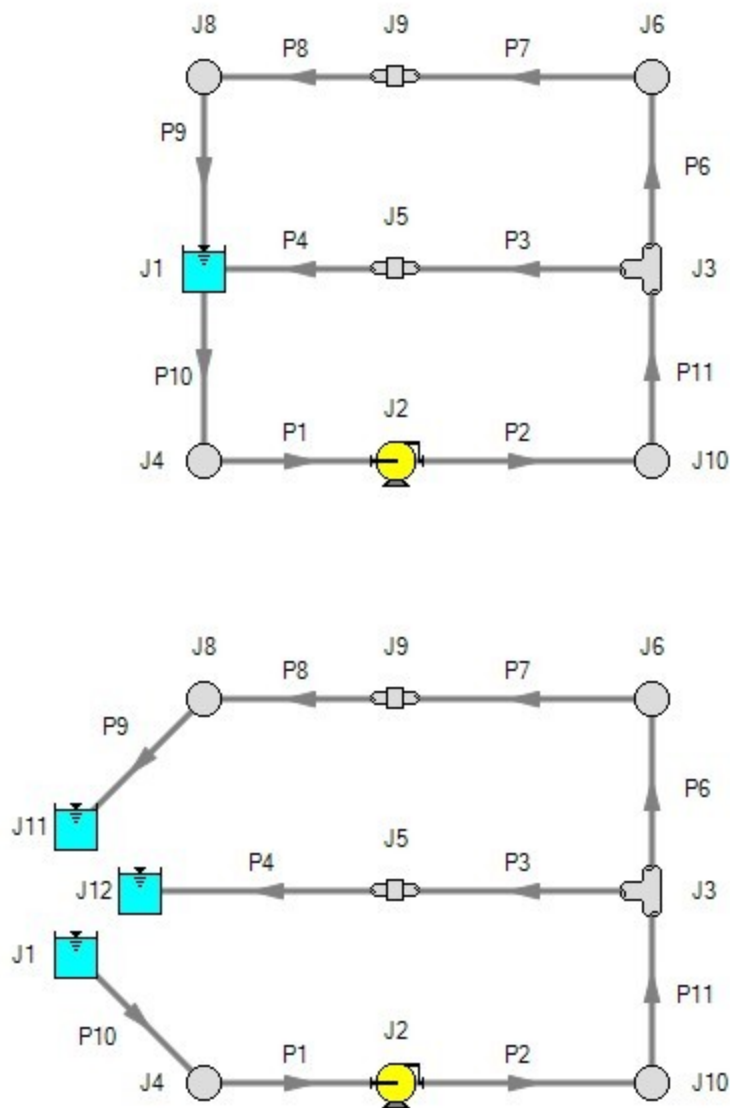
Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two system will appear identical to AFT Impulse.

Related Topics

Curve Fitting

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

- Pump Curves
- Junction Resistance Curves
- Custom Fluids
- Pipe Physical Properties

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

Figure 1: Custom Fluid Definition

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

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

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

- **Polynomial** - The data is best fit to a polynomial of order 0,1,2,3, or 4. Note that a 0th order polynomial is a constant value, and a 1st order is a linear fit.

- **Interpolated X-Y Data** - The data has a linear interpolation drawn between every data point.

One of the critical items to note with any curve fit is that you may not be operating close to one of your defined points if the data does not conform well to a polynomial relationship. For example, the 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:
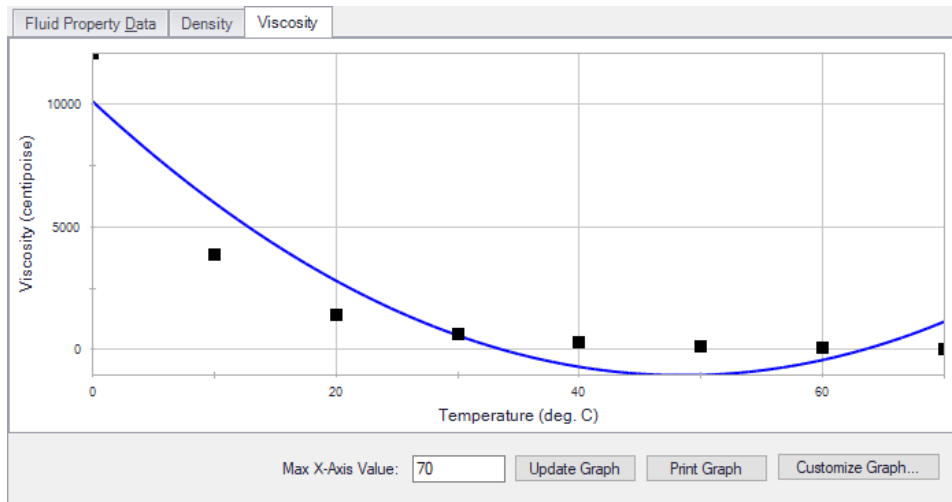
Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data

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. For example, in Figure 1 above there are a, b, and c constants shown for the Density (Rho) curve fit. These values use a base unit, which is not necessarily the unit the user has defined for that parameter.

Related Topics

Stagnation vs. Static Pressure Boundaries

With two exceptions (to be discussed), all pressure-type boundary conditions in AFT Impulse 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 Impulse 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 Impulse 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.
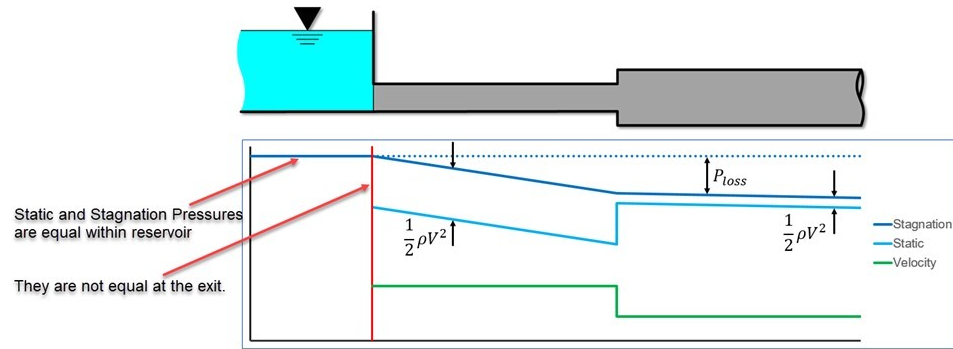
Figure 1: Pressure profile along horizontal pipe shown

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

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

When to use Static Pressure

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

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

From this equation it is clear that application of the static pressure is not sufficient to specify location 1. The velocity and elevation is also required.

In AFT Impulse, 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 Impulse 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, pipe sectioning, transient controls, force definitions, environmental properties, and other related options. Launch Analysis Setup from either the tool-bar or the Analysis menu.

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

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

The groups in Analysis Setup are

- Modules group
- Fluid Properties group
- Pipes and Junctions group
- Pipe Sectioning and Output group
- Transient Control group
- Pulsation Setup group - (PFA Module only)
- Forces group
- Steady Solution Control group
- Environmental Properties group
- Miscellaneous group
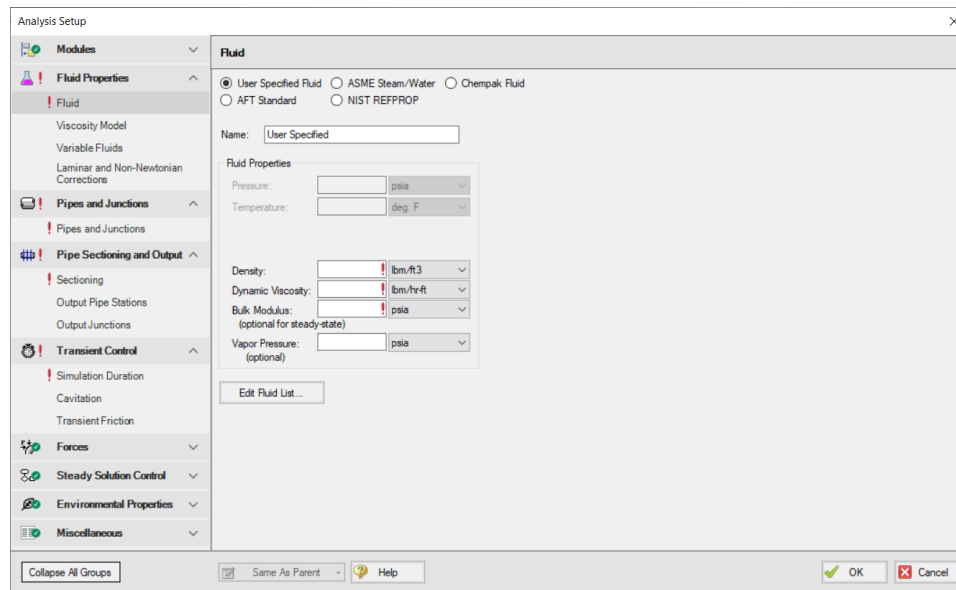
Figure 1: Default state of Analysis Setup in a blank model

Modules Group

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

- PFA (Pulsation Frequency Analysis) - Helps identify and avoid resonant frequencies in systems, especially in those caused by positive displacement pumps.
- SSL (Settling Slurries) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.

Related Topics

Modules Panel

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

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

Enabling a module turns on all functionality for the module. By default activating a module will also automatically enable the module for use. Disabling a module keeps the license seat

checked out but the module calculations are not applied to the model. Disabling a module may be desired for various reasons, as is discussed further for each module below.

Note: A module specific license is required to activate the PFA or SSL add-on module.
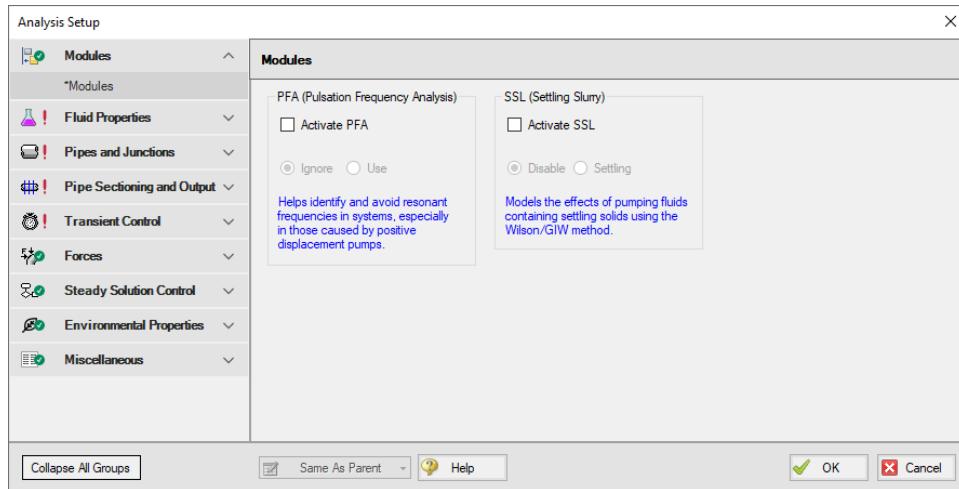
Figure 1: The default state of the Modules panel

PFA (Pulsation Frequency Analysis)

Activate the Pulsation Frequency Analysis (PFA)module by checking the box next to Activate PFA. Once activated, enable it by selecting Use or disable it by selecting Ignore. When the PFA module is activated, the Pulsation Setup group appears in the Analysis Setup. The Pulsation Setup group has two items: Pulse Setup and PD Pump Settings.

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

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

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

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

Feature	PFA Not Activated	PFA Activated	
		PFA Disabled	PFA Enabled
Pulsation Setup group in Analysis Setup	Not Visible	Visible	Visible
Pulsation Frequency Analysis in Analysis menu	Not Visible	Visible	Visible
Pulsation Setup in Analysis menu	Not Visible	Visible	Visible
Steady-State Pulsation Tolerance on Tolerance panel	Not Visible	Not Visible	Visible
Pulsation Settings on Sectioning panel	Not Visible	Not Visible	Visible
Frequency tab in Graph Control, Graph Results window	Not Visible	Not Visible	Visible
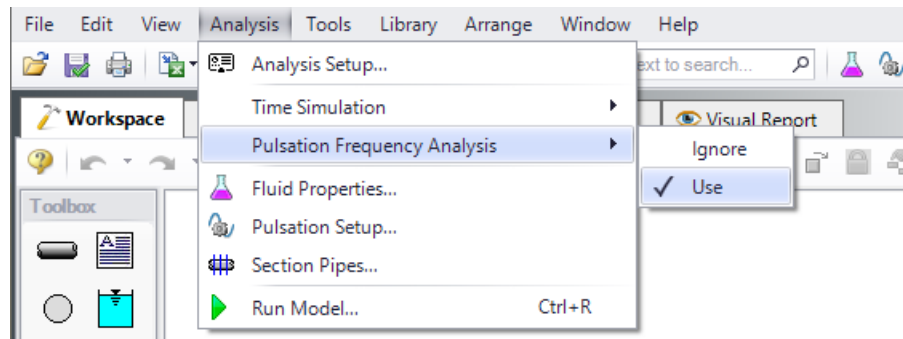
Figure 2: Select Use from the Pulsation Frequency Analysis in the Analysis menu to perform pulsation analysis

SSL (Settling SLurry)

Activate the Settling Slurry (SSL) 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 SSL module fluid properties items as shown in Table 2 below. These items are Carrier Fluid, Solids Definition, and Slurry Definition. Additionally, the Library menu will have a Solids Library option. These items are present while SSL is activated, regardless of whether SSL is enabled or disabled.

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

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 Impulse 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 Impulse. If a model with SSL activated is opened in standard AFT Impulse without SSL activated, then the SSL data will be lost once the model is saved. Table 2 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 per-

formed 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 Impulse feature accessibility based on the SSL module activation and enablement

Features	SSL Not Activated	SSL Activated	
		SSL Dis-abled	SSL Enabled
SSL items in Fluid Prop-erties group	Not Visible	Visible	Visible
Slurry in Ana-lysis menu	Not Visible	Visible	Visible
Solids Library in Library menu	Not Visible	Visible	Visible
Flowrate spe-cified in terms of slurry, solids or clear liquid in sev-eral 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
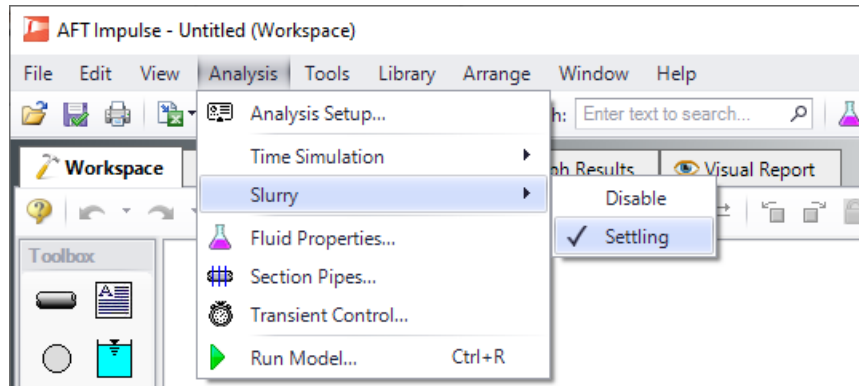
Figure 3: Select Settling from the Slurry option in the Analysis menu to perform settling slurry calculations when running the model

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.

- Variable Fluids panel - Optional panel. Allows user to select constant or variable fluid properties.

- 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 Blogs

Get More Flexible With Your Fluids

Fluid Panel

The Fluid Panel controls the properties of the fluid in the model. A default fluid is not selected - one must be defined to run any simulation.

The first selection that must be made is what Fluids Library to use, if any. The default selection of User Specified requires the manual definition of Density, Dynamic Viscosity, Bulk Modulus (optional for steady state runs), and Vapor Pressure (optional if cavitation modeling is not required).

The available Fluid Libraries are:

- **AFT Standard** - An included library that offers 71 common fluids to the user.
- **ASME Steam/Water** - Data from the 1997 ASME Steam Tables.
- **NIST REFPROP** - Data compiled by the National Institute of Standards and Technology (NIST) using thermophysical property models for a wide variety of fluids. Fluid mixtures are supported with this library. For more info see the following sources: REFPROP FAQ, NIST Chemistry WebBook, and NIST Chemical Name Search.
- **Chempak Fluid** - A large library of fluids with detailed property information, including data not found in the AFT Standard fluids such as saturation lines. Fluid mixtures are supported with this library.

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: Fluids in the supercritical region will be treated as incompressible in Impulse. For increased accuracy fluids above the critical point should be modeled in a transient compressible flow solver such as AFT xStream.

AFT Standard Fluid Library

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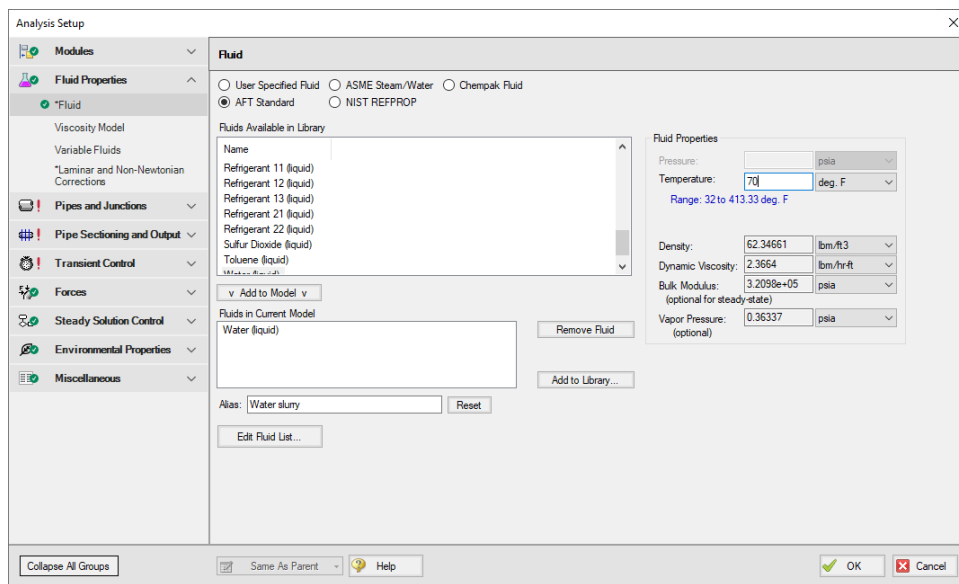
Figure 1: Fluid Panel in Analysis Setup with AFT Standard fluid library selected

ASME Steam/Water Library

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

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

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

For more information on AFT SteamCalc Viewer see the SteamCalc Help file.
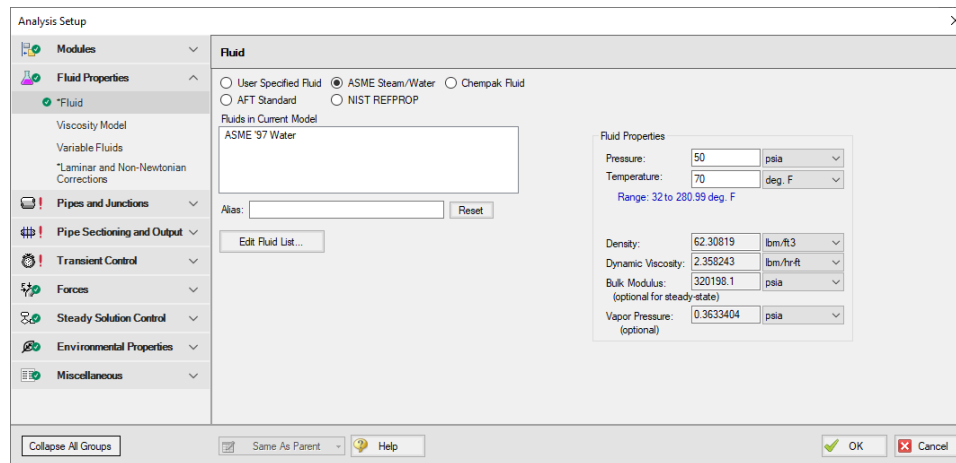
Figure 2: Fluid Panel in Analysis Setup with ASME Steam/Water library selected

NIST REFPROP Library

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

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model. Enter 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.
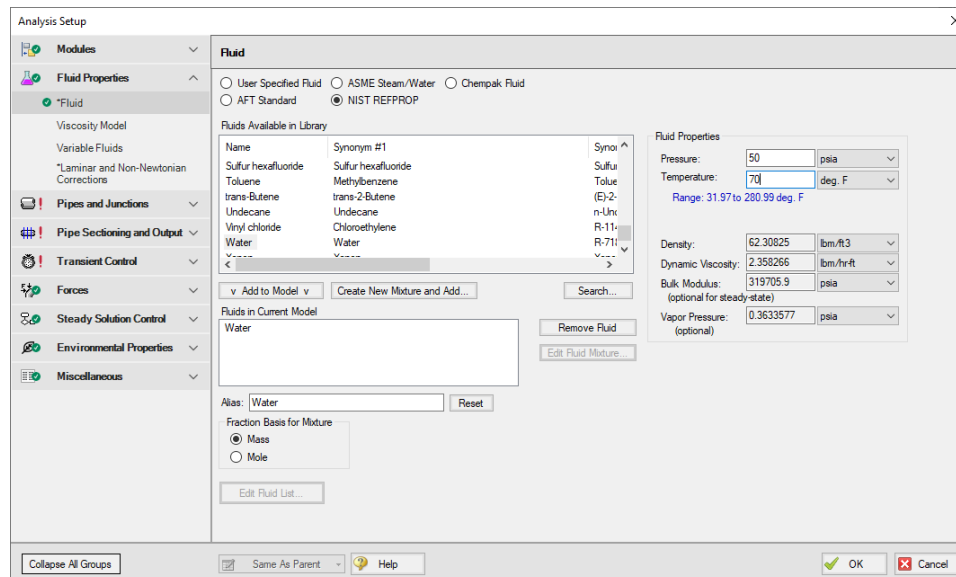
Figure 3: Fluid Panel in Analysis Setup with NIST REFPROP library selected

Chempak Fluid Library

The Chempak Fluid Library offers approximately 700 fluids to the user. In addition, it offers non-reacting mixture calculations. A Chempak Data license is required to use fluids from this library.

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

Fluids from the Chempak Fluid Library support fluid mixtures.

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

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

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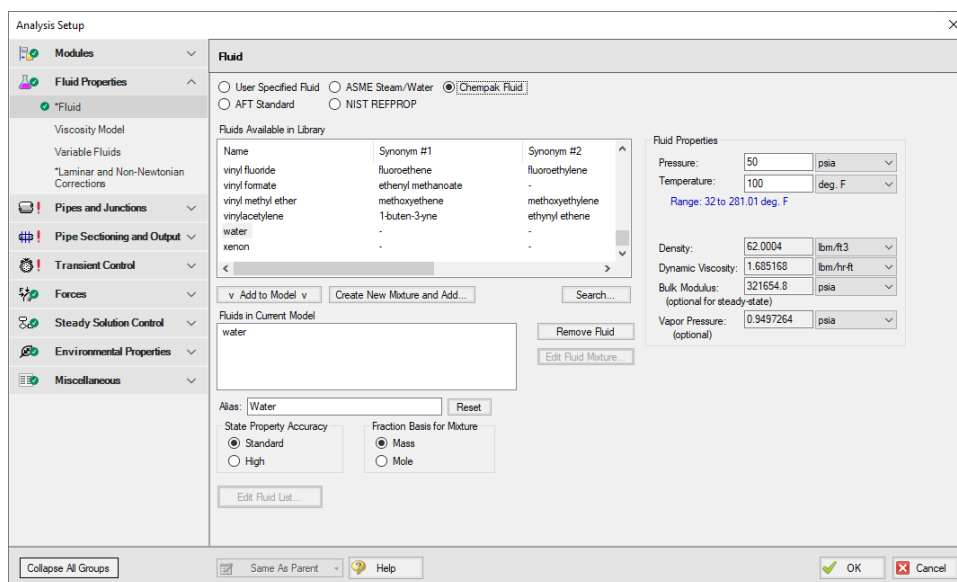
Figure 4: Fluid Panel in Analysis Setup with Chempak Fluid library selected

State Property Accuracy Options

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

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

The State Property Accuracy options are located on the Fluid Panel below the Fluids in Current Model area.

This option is applicable to only to Chempak fluids.

Related Topics

Related Blogs

Get the Most Out of AFT's Fluid Property Viewers

Get More Flexible With Your Fluids

Creating Liquid Mixtures

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

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

Note: The fraction basis of all mixtures is either mass or mole, and it is specified on the Fluid panel itself. If you would like to change the fraction basis for the mixture (from mass to mole or from mole to mass), click on the desired fraction basis located at the bottom of the Fluid panel.
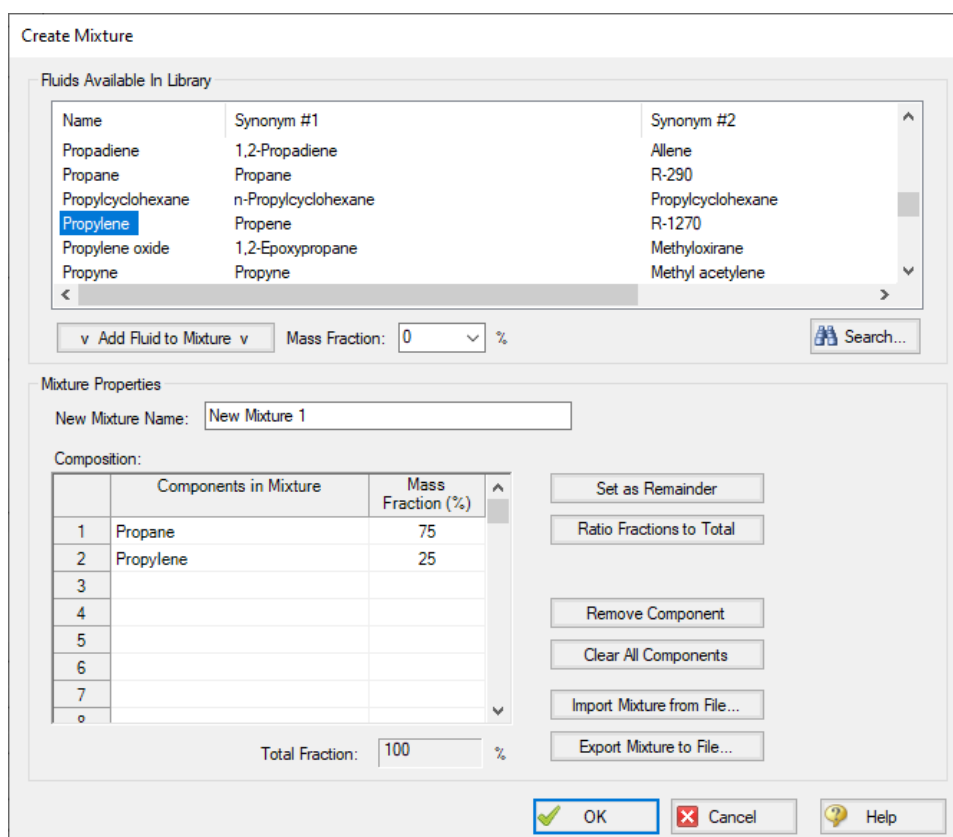
Figure 1: The Create Mixture window (opened from the Fluid panel) allows you to create predefined mixtures for the model.

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

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

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

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

Related Topics

Related Blogs

[Get More Flexible With Your Fluids](#)

Viscosity Model Panel

AFT Impulse 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.

AFT Impulse offers seven viscosity models:

- Newtonian
- [Duffy Method (Pulp and Paper Stock)](#)
- [Brecht & Heller Method (Pulp and Paper Stock)](#)
- [Power Law Model](#)
- [Bingham Plastic Model](#)
- [Herschel-Bulkley](#)
- [Homogeneous Scale-up Method](#)

Modeling non-Newtonian fluids is a complex task. A myriad of test equipment and theoretical models exist, and each have various names unique to an industry or application. This discussion along with other discussion related to non-Newtonian flow in Impulse seeks to help the reader understand 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 on the [Laminar and Non-Newtonian Corrections panel](#) topic.

Applicability in Transient Flow

AFT Impulse uses the same non-Newtonian models and correlations for both steady-state and transient flow. The applicability of these viscosity models to transient flow is uncertain. Studies suggest that the viscosity models available in Impulse remain valid in transient systems, but no

definitive answer exists. Users should treat all transient non-Newtonian hydraulic results with caution.

Impulse models these non-Newtonian fluids in a transient system by recalculating friction factor or pressure loss information each time step as dictated by the settings on the Transient Friction panel. It is strongly recommended that transient systems with non-Newtonian fluids are modeled with the Time Varying – Simple model (selected by default for all models created in Impulse 8 or later). The Constant model will not capture the non-Newtonian behavior of the fluid as velocity changes. The Time Varying – Advanced (Unsteady Brunone) model has not been studied for use with non-Newtonian fluids.

Related Topics

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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.

Impulse contains data on 20 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.
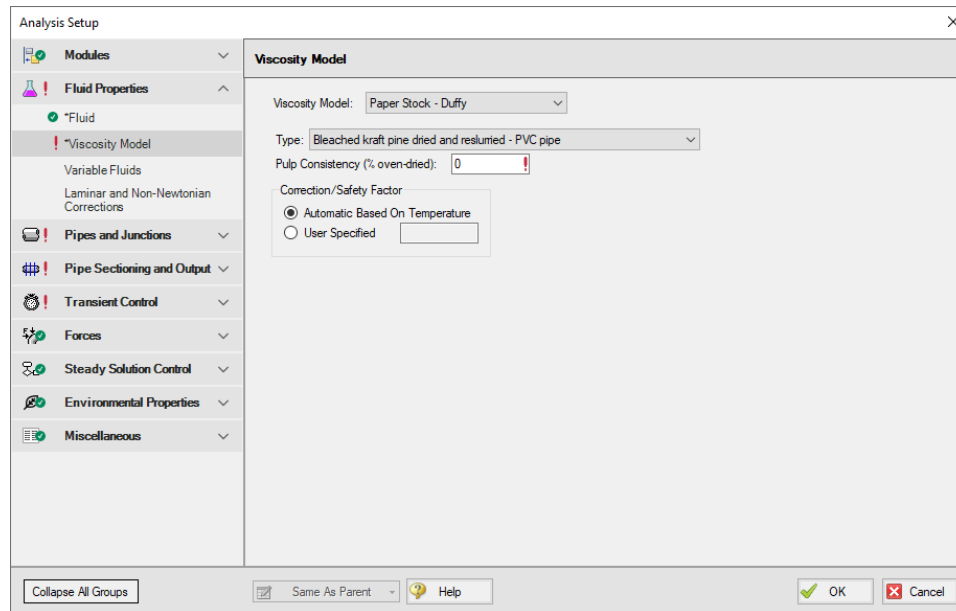
Figure 1: Viscosity Model interface for the Duffy method.

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 Impulse are based on a pulp temperature of 35 deg. C. The Automatic Based on Temperature will use the temperature specified in 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 Impulse 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 0 <Vmax:

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

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

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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.

Impulse contains data on 8 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.
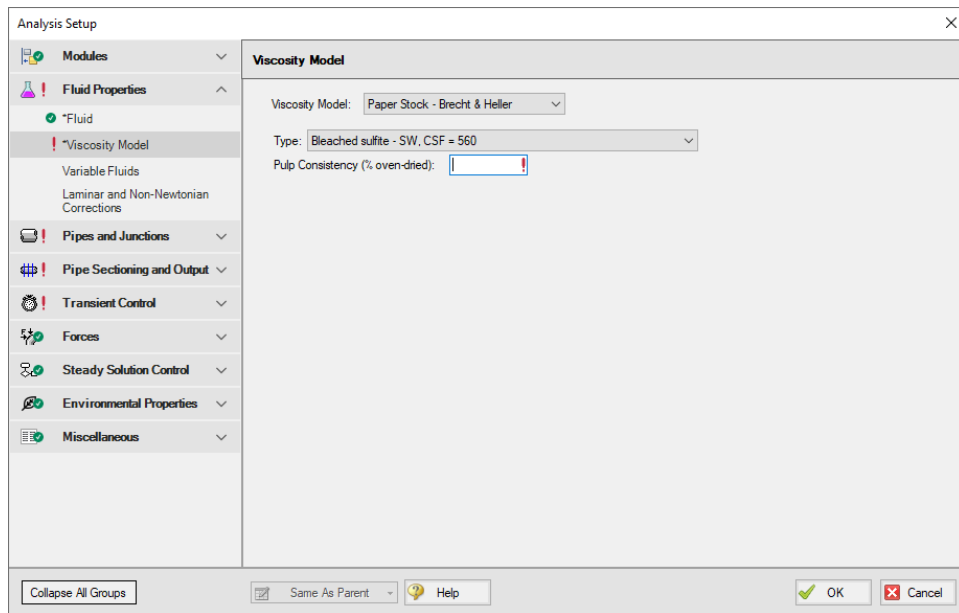
Figure 1: Viscosity Model interface for the Brecht and Heller Method.

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 Impulse 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:

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

The pipe friction factor is:

$$f = \frac{3.97}{R\acute{e}^{1.636}}$$

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 (ft/s)
- ρ = stock fluid density (lbm/ft3)
- C = pulp consistency (% dryness)
- K = friction factor constant, specific to pulp type (dimensionless)

Related Topics

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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.
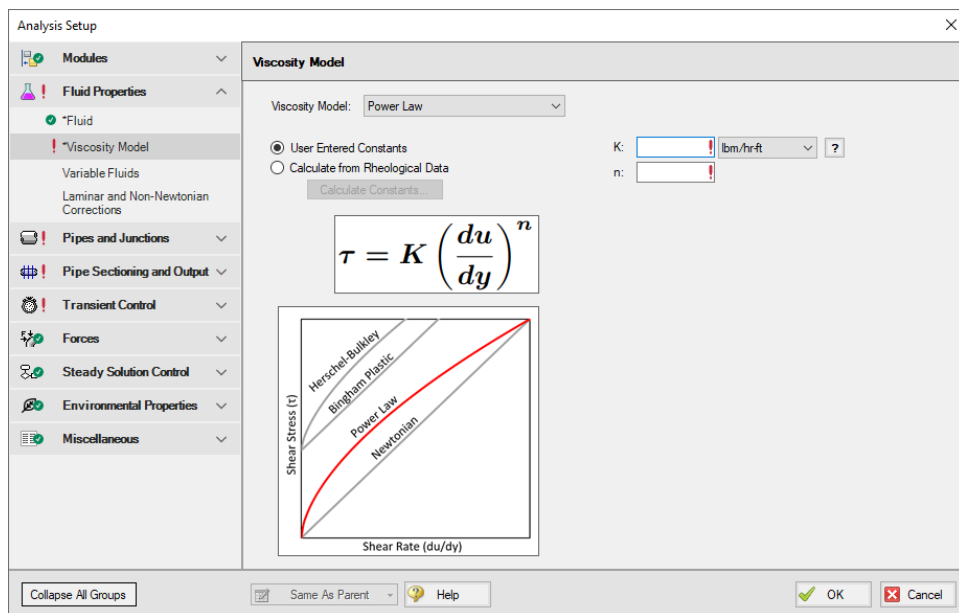
Figure 1: Viscosity Model interface for the Power Law model.

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 Impulse 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.

$$f = (1 - \alpha)\, f_L + \frac{\alpha}{(f_T^{-8} + f_{Tr}^{-8})^{1/8}}$$

Where:

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

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

$$f_{Tr} = 7.16 * 10^{-4}\, e^{-5.24\, n}\, Re_{pl}^{0.414 + .757\, n}$$

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

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

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

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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:

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

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.
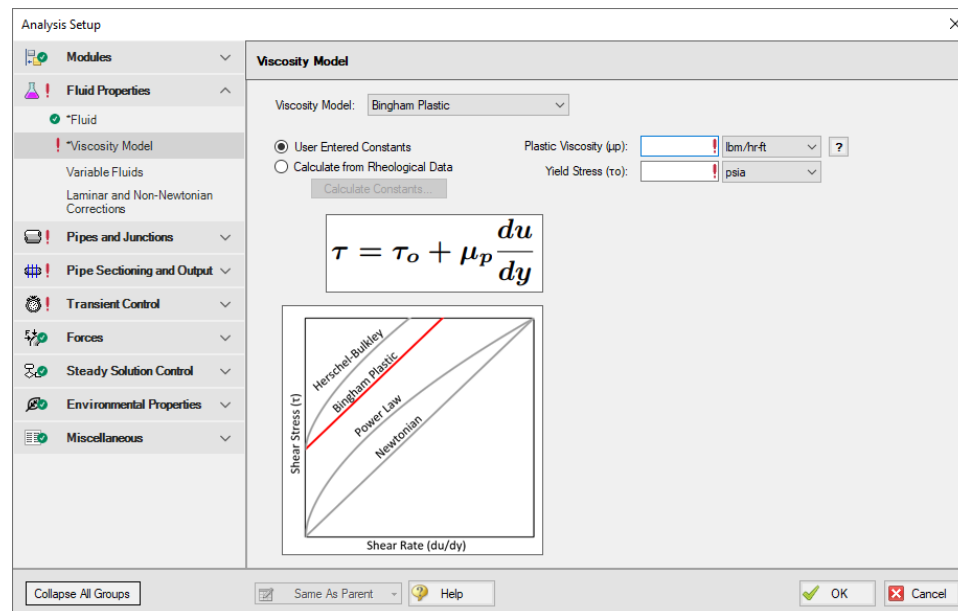
Figure 1: Viscosity Model interface for the Bingham Plastic model.

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 Impulse 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.

$$f = (f_L^m + f_T^m)^{1/m}$$

Where:

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

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

$$f_T + 4\left(\frac{10^\alpha}{Re^{0.193}}\right)$$

$$\alpha = -1.47(1 + 0.14e^{-2.9*10^{-5}\,He})$$

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

Related Blogs

[Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network](#)

Herschel-Bulkley Model

The Herschel-Bulkley 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:

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

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.
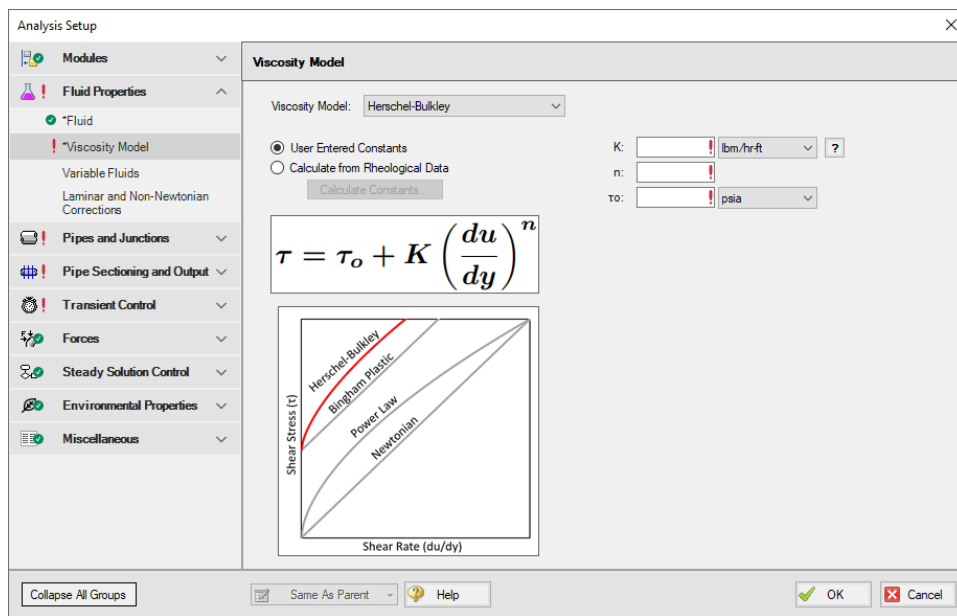
Figure 1: Viscosity Model panel for the Herschel-Bulkley model

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:

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

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.

Related Topics

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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.
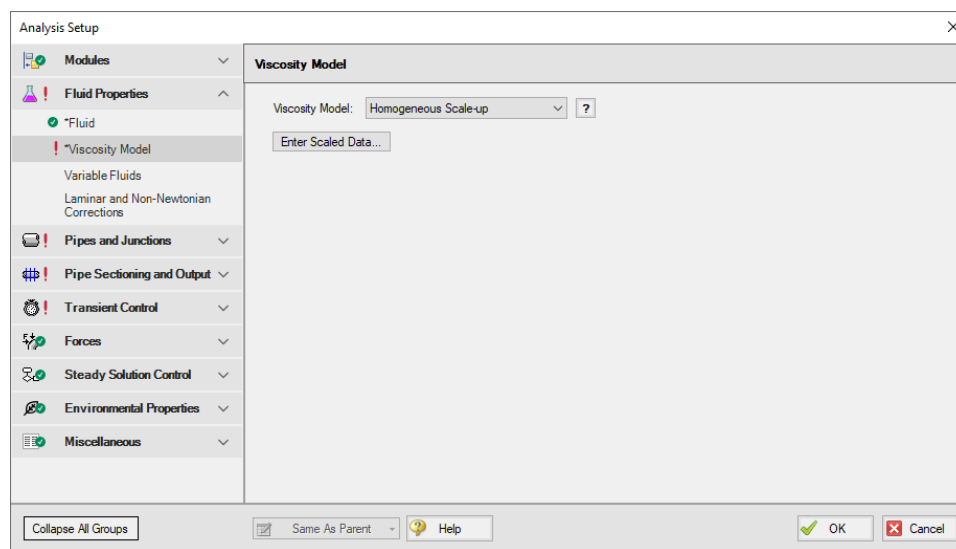
Figure 1: Viscosity Model interface for the Homogeneous Scale-up method.

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 Impulse will use those models in the system. The cross-plot is intended only to give a visual comparison of the data.
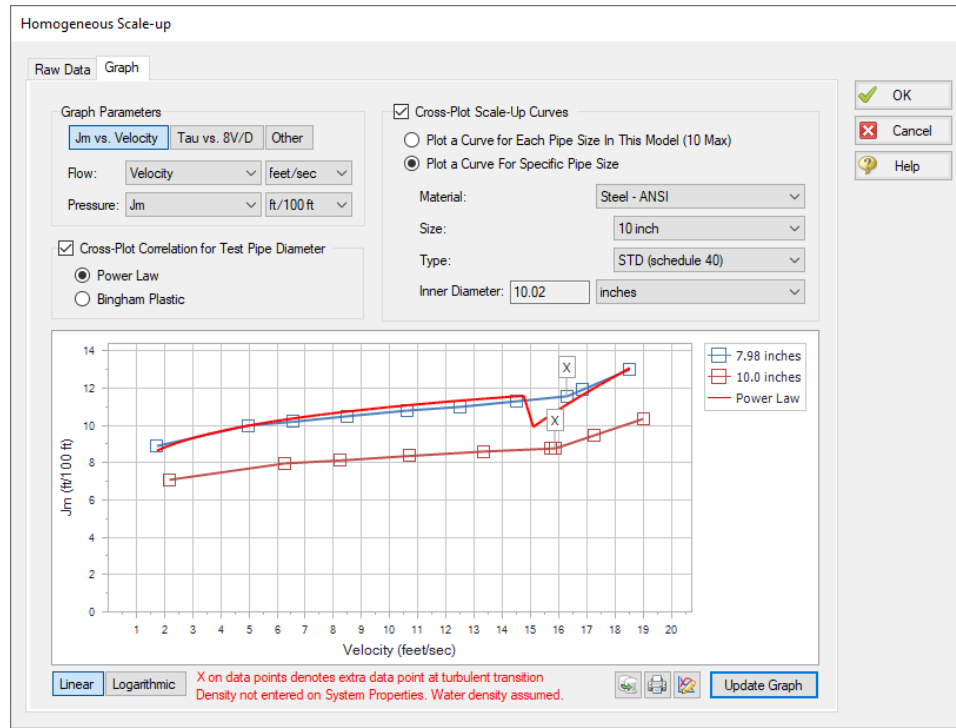
Figure 3: Graphed data for the Homogeneous Scale-up method.

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:

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

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

Turbulent data scales as follows:

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

$$(V_m)_2 = (V_m)_1 + 1.5\,U \ln\left(\frac{D_2}{D_1} \right)$$

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

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

Related Blogs

Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network

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, Impulse can calculate the viscosity model constants from that data.

The interface used to enter this data and the processes Impulse 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.
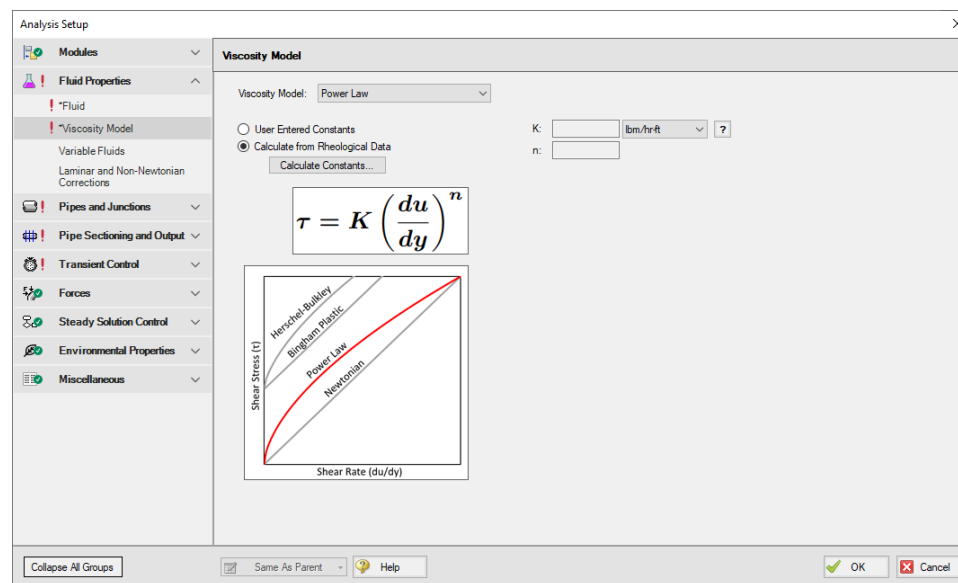
Figure 1: Power Law viscosity model using the Calculate from Rheological Data option.

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 ($8V/D$) 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. Impulse version 8 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 Impulse 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.
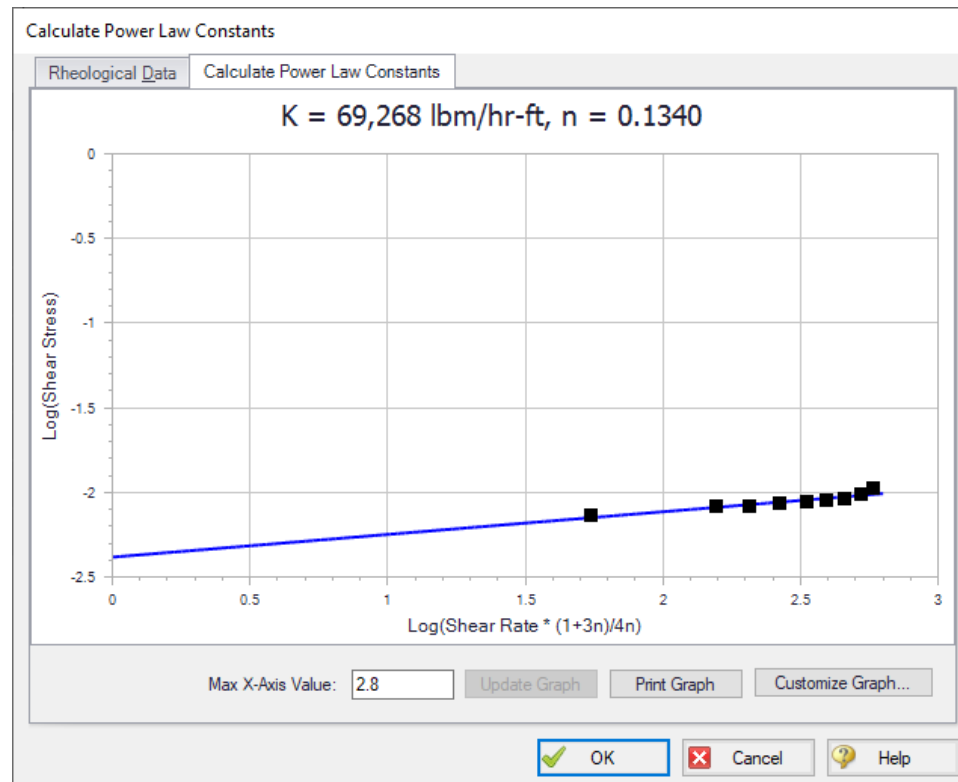
Figure 3: Curve fit rheological data and constants.

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 Impulse 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. Impulse 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:

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

Impulse 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

Related Blogs

[Non-Newtonian Fluids - How to Model the Soap, Silly-Putty, or Shampoo in Your Pipe Network](#)

Variable Fluids Panel

AFT Impulse allows users to model constant or varying fluid properties.

- **Constant Fluid Properties** – all fluid properties are the same for every pipe and are as specified in 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 in the Fluid panel are the default fluid properties. You can then 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.

Laminar and Non-Newtonian Corrections Panel

AFT Impulse 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 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 those losses may not be similar for both Newtonian and non-Newtonian fluids. AFT Impulse offers three options to model the losses due to pipe fittings or junctions during laminar flow. They are:

1. None
2. The [Adjusted Turbulent K Factor (ATKF) Method](#) which is derived from the Crane hydraulic handbook.
3. Using the ratio of non-Newtonian to Newtonian pipe friction

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.
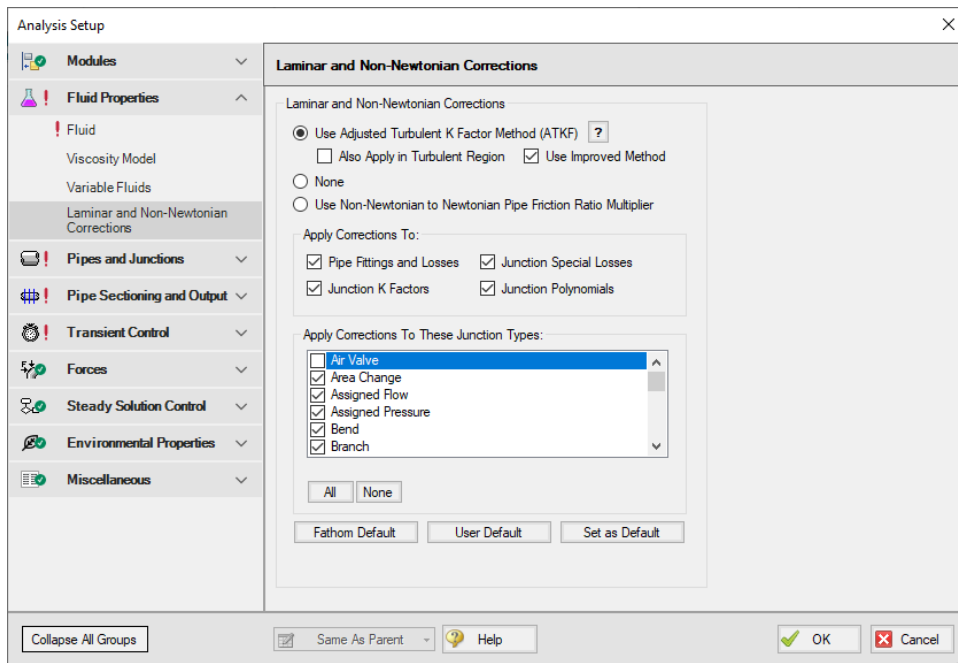
Figure 1: Options to modify fitting and junction losses in the Laminar and Non-Newtonian corrections.

Users must select which of the three options they wish to use. They must also select which non-pipe losses to apply the corrections to.

Non-Newtonian to Newtonian Pipe Friction Ratio

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

Related Blogs

Applying the Improved ATKF Method for Laminar and Non-Newtonian Flow

Adjusted Turbulent K Factor (ATKF) Method

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 *fL/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 Impulse K-factor data by looking at large Reynolds number and hence fully turbulent conditions

$$L_{eq,turb} = K_{turb} \; \frac{D}{f_{turb}} \qquad (5)$$

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

$$K = K_{turb} \; \frac{f}{f_{turb}} \qquad (6)$$

To apply the method, take the AFT Impulse 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 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, which is discussed in detail in the technical paper by Wood et. al, 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 Impulse 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 Impulse. 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 4.6 method allows reliable adjustment of turbulent pressure loss data for fittings and junctions to laminar applications.
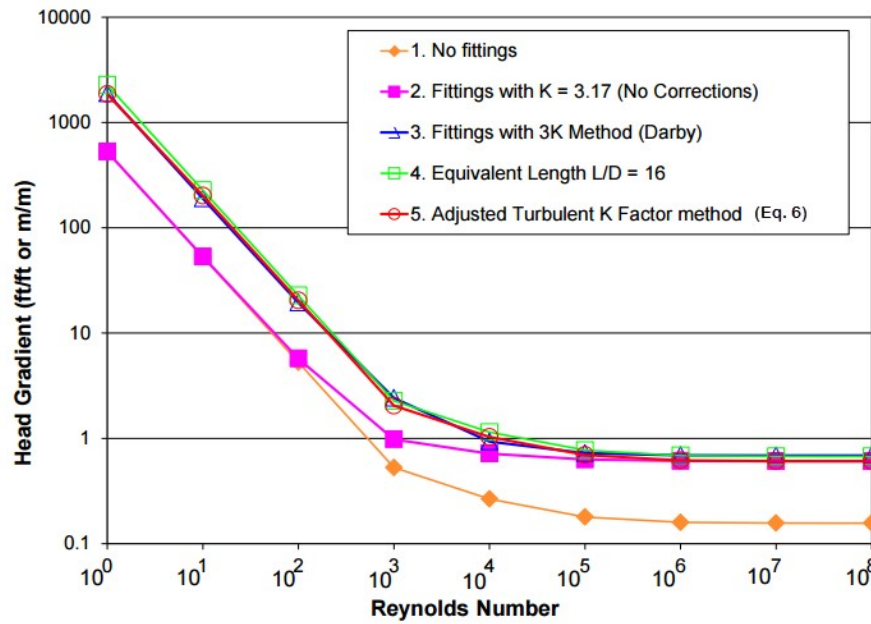
Figure 1: Log-log plot of dimensionless head gradient vs. Reynolds number for various fitting loss models

Related Topics

Related Blogs

Applying the Improved ATKF Method for Laminar and Non-Newtonian Flow

Improved ATKF Method

For commercial steel pipe and pipes with a similar roughness, it has been shown that the equivalent length, 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 on the left Figure 1. The 3-K method shows no dependence on pipe roughness in the laminar regime, as is shown on the right in Figure 1. 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.
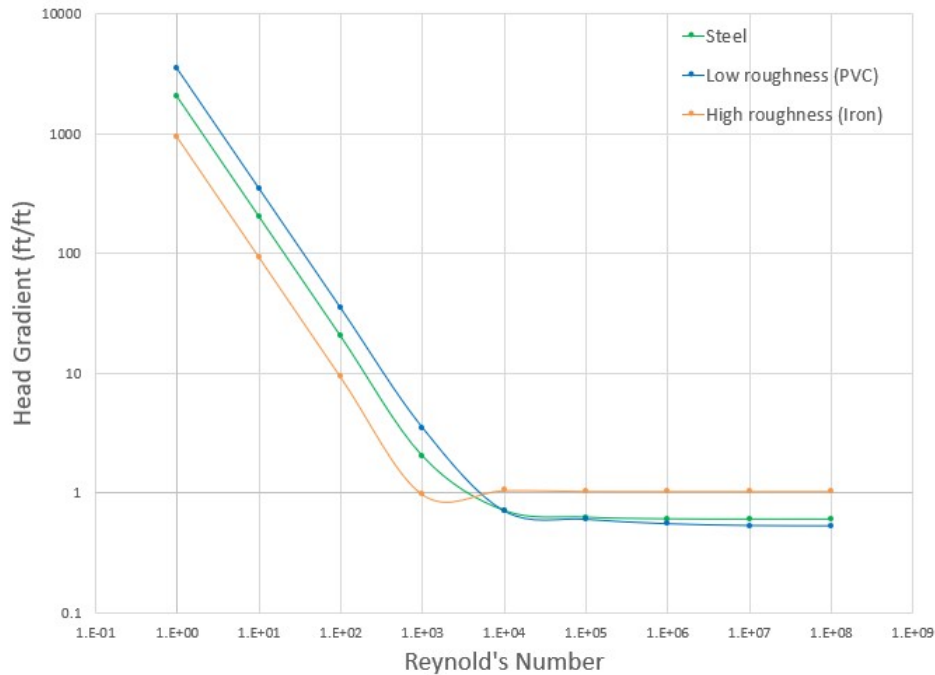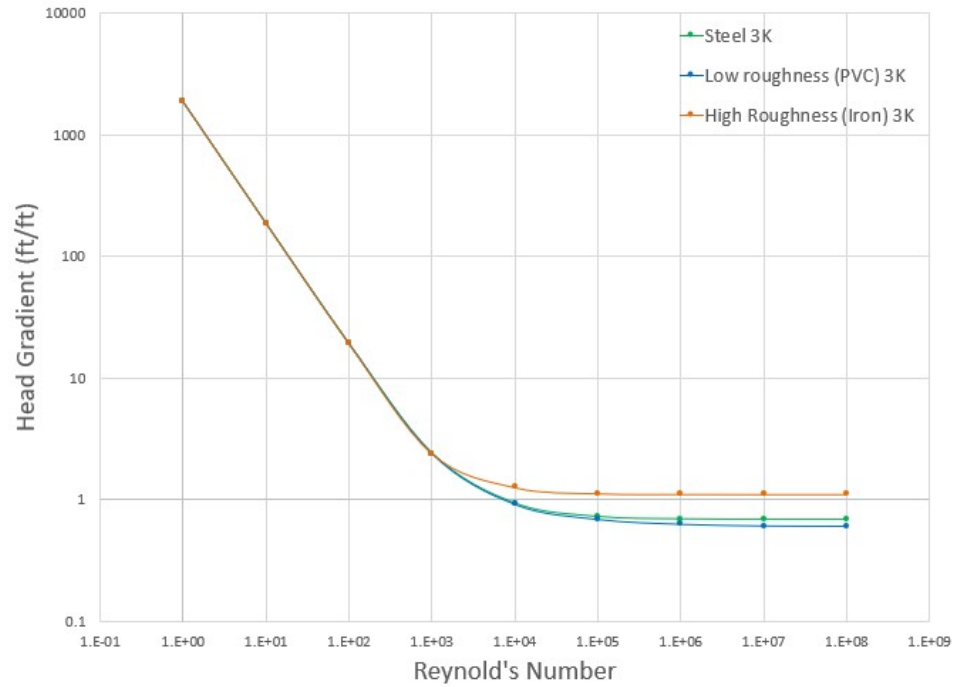
Figure 1: Head Gradient calculated with the original ATKF method (left) and 3-K method (right) at different roughness values.

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 2).
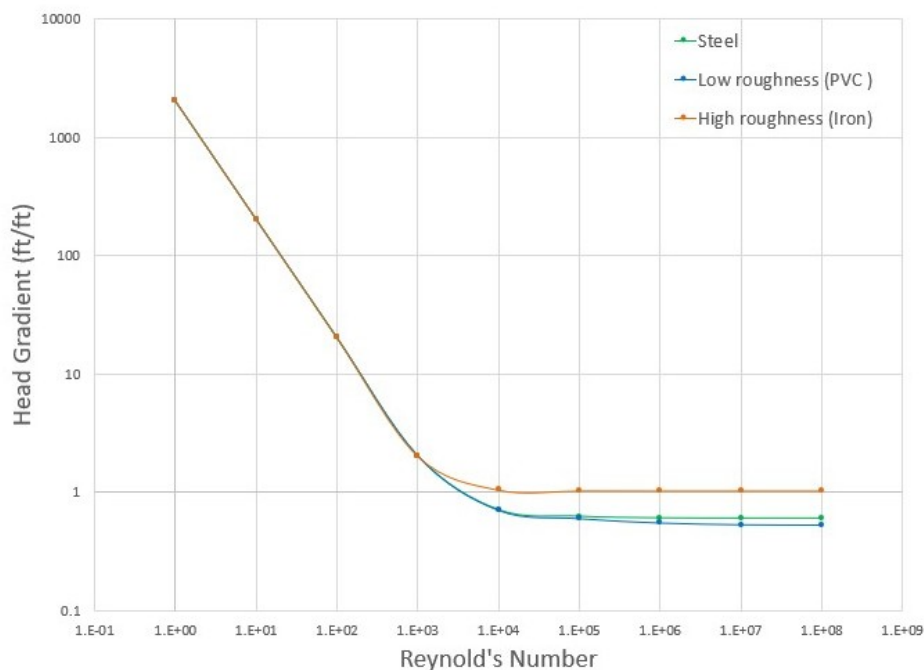
Figure 2: Head gradient calculated with the Improved ATKF method at different roughness values

Related Topics

Related Blogs

Applying the Improved ATKF Method for Laminar and Non-Newtonian Flow

Note about Laminar Corrections for Fittings

There is ambiguity in the literature about how to calculate fitting pressure loss for fittings. 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 Impulse 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 Turbulent Flow Through One-Half-Inch Valves and Fittings", Trans. ASME, v. 79, (Nov. 1957), p. 1759.

Related Topics

Related Blogs

[Applying the Improved ATKF Method for Laminar and Non-Newtonian Flow](#)

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 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

On the Carrier Fluid Panel, users select and define the carrier fluid for the slurry. The two carrier fluid options are:

- Basic Water Slurry Input
- Advanced Slurry Input

Basic Water Slurry Input

The Basic Water Slurry Input option (see Figure 1) uses water as the carrier fluid and treats all pipes as having constant slurry properties.

Specify the water temperature for the slurry property calculation.
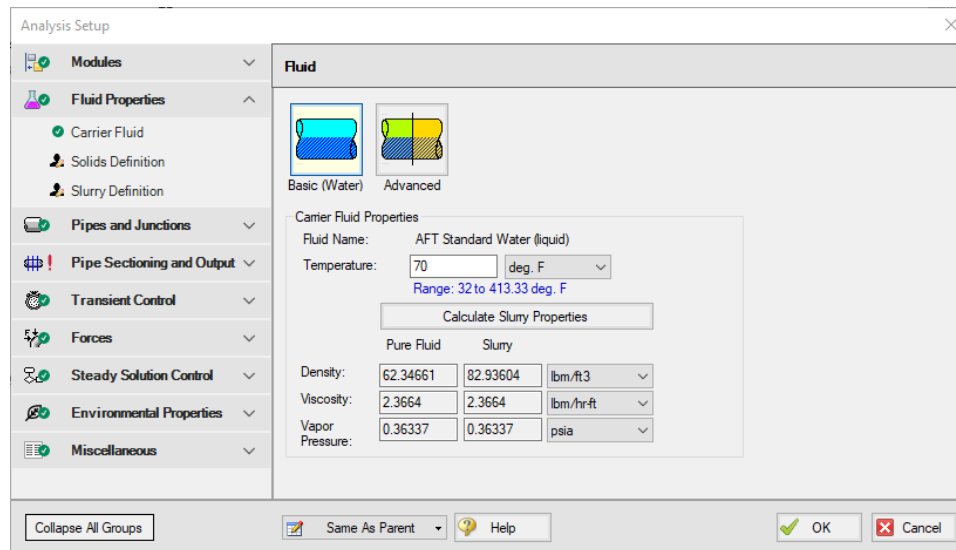
Figure 1: Basic Water Slurry Input option on the Carrier Fluid Panel models water as the carrier fluid and constant slurry properties in the system

Advanced Slurry Input

The Advanced Slurry Input option (see Figure 2) uses any fluid as the carrier fluid and allows slurry properties to vary among pipes.

The Carrier Fluid Panel looks very similar to the Fluid Panel when 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 fluid data from the AFT Standard library, the ASME Steam/Water library, the NIST REFPROP library, and the optional Chempak library.
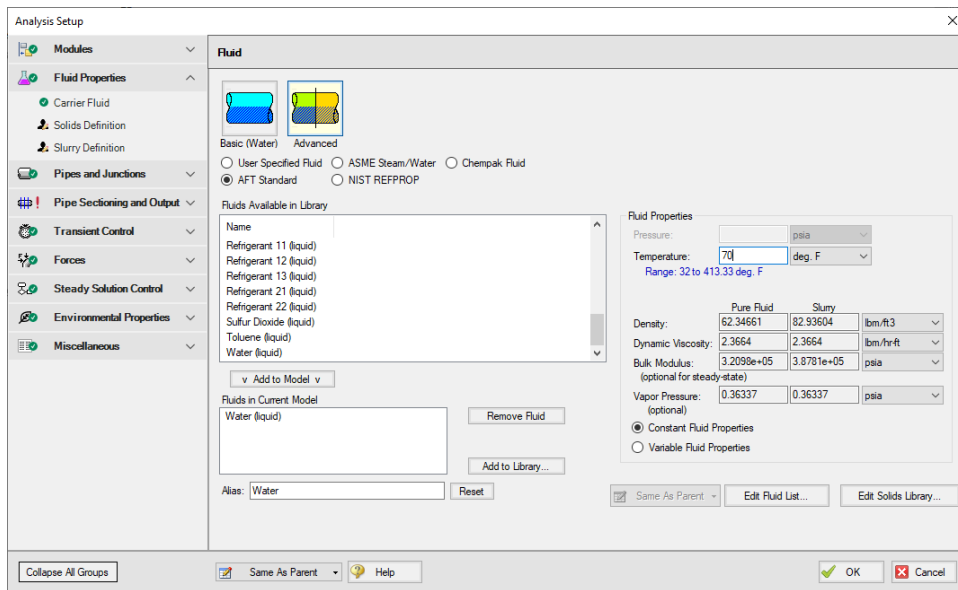
Figure 2: Advanced Slurry Input option on the Carrier Fluid Panel models any fluid as the carrier fluid and allows variable slurry properties in the system

Related Topics

Solids Definition Panel

On the Solids Definition Panel, users define the Slurry Calculation Method and Solids Specifications. These inputs are the same as for both the Basic Water Slurry Input and the Advanced Slurry Input.

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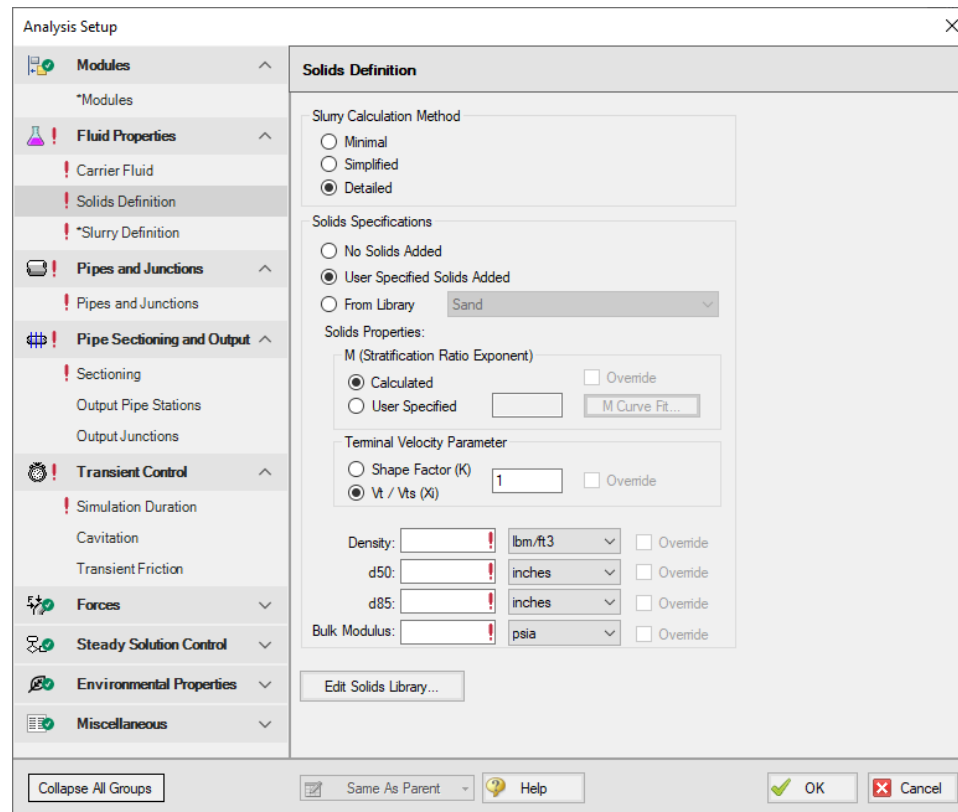
Figure 1: Solids Definition Panel when Detailed Slurry Calculation Method and Calculated M are selected

Slurry Calculation Method

There are three calculation methods available to model slurries. Each method requires different amounts of input data and uses different calculations. These calculations are described in the Slurry Calculation Methods section.

- Minimal Method - Used in the preliminarily design phase where only rough data is available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data used, the calculations should only be used for early stage approximations. 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 withing suspended particles. The Simplified Method is a more accurate model than the Minimal method because it accounts for more slurry definition.

- Detailed Method - The default method, and the method which provides the most rigourous solution methodology. The solids density, d50 particle size, terminal velocity parameter,

and the solids concentration are required. Additionally, if the *M* exponent is calculated, the d85 particle size is required. As with the Simplified Method, this method also assumes heterogenous 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
- 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 first be defined 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 on the Slurry Model 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 Impulse 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 (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).

Related Topics

Slurry Definition Panel

The Slurry Definition Panel allows input of solids concentration relative to the carrier fluid and estimation of bulk slurry properties, as shown in Figure 1.

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: 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. Each method requires different amounts of input data and uses different calculations. These calculations are described in the Slurry Calculation Methods section.

- Minimal Method - Used in the preliminarly design phase where only rough data is available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data used, the calculations should only be used for early stage approximations. 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 withing suspended particles. The Simplified Method is a more accurate model than the Minimal method because it accounts for more slurry definition.

- Detailed Method - The default method, and the method which provides the most rigourous solution methodology. The solids density, d50 particle size, terminal velocity parameter, and the solids concentration are required. Additionally, if the *M* exponent is calculated, the d85 particle size is required. As with the Simplified Method, this method also assumes heterogenous 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 two **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

- Settling Slurry - Wilson, Addie, Clift - Default option, applies slurry calculations from Wilson

The **Sliding Friction Coefficient (μs)** is the coefficient of mechanical sliding friction between the granular mass and the boundary - where the boundary is the 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. This field is only available when the Minimal Slurry Calculation Method is selected at the top of the panel.

The **Bed Concentration (C_vb)** is an optional parameter 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 as V_{sm} and is the usual parameter of interest in design-ing a system to avoid operating with a stationary bed. This field is only available when the Sim-plified 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** - 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 $V_{sm,max}$ cal-culated 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** - This option will cause the V_{sm} to be calculated solely using Equation 5.11.
- **User Specified** - This option allows the user to directly specify V_{sm}.

As Wilson describes there are times where operating at a velocity lower than V_{sm} is eco-nomically 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.

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 junctions are fully defined. When a red exclamation point is next to its name, there is at least one pipe or junction that is undefined. The model cannot run until all pipes and junctions are defined.

Related Topics

Pipes and Junctions Panel

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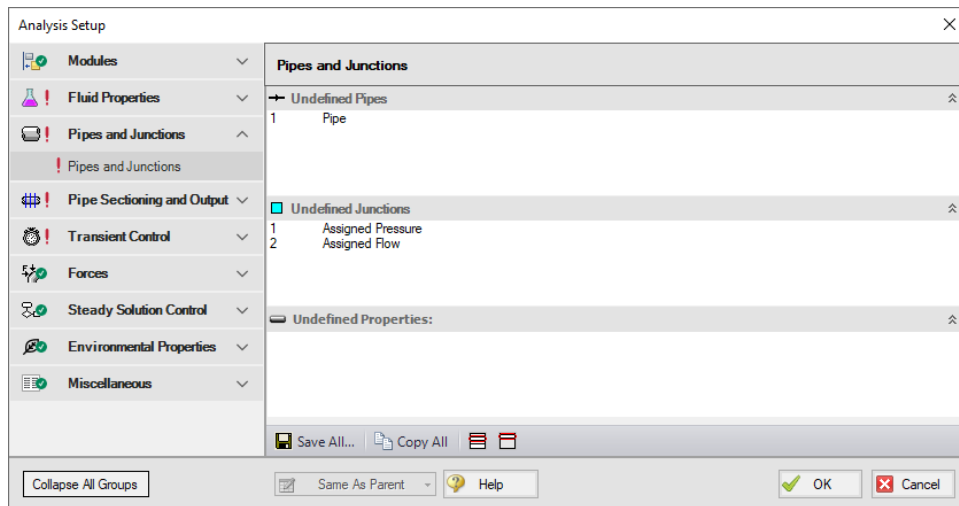
Figure 1: Default state of the Pipes and Junctions panel

This panel is made up of three sections:

- Undefined Pipes
- Undefined Junctions
- Undefined Properties

The Undefined Pipes and Undefined Junctions sections display 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.
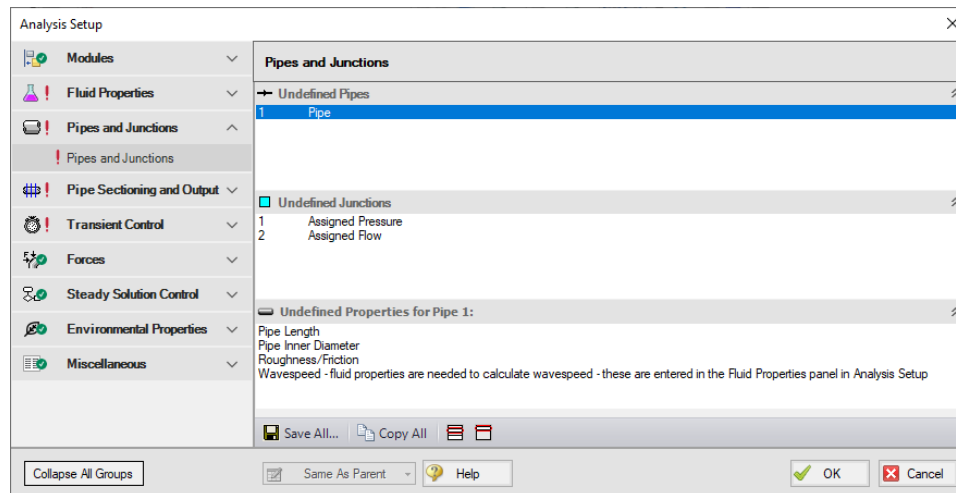
Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed

Related Topics

Pipe Sectioning and Output Group

The Pipe Sectioning and Output Group in the Analysis Setup window has three items: the Sectioning panel, Output Pipe Stations panel, and Output Junctions panel. These panels control the sectioning used for the transient solution, as well as what output is saved to the file for the pipes and junctions.

- Sectioning panel - Required panel. The pipe sectioning used for the transient Method of Characteristics solution is defined here. Some settings on this panel will change when the PFA Module is activated and enabled.
- Output Pipe Stations panel - Optional panel. This panel controls which pipe stations are saved to file.
- Output Junctions panel - Optional panel. This panel controls what transient junction output is saved to file, such as pump speed, valve Cv, etc.

Related Topics

Sectioning Panel

Before running a transient simulation, the pipes must be sectioned as described in Pipe Sectioning - Introduction to Method of Characteristics. This forces a common time step, and forces the section length to be equal to the product of the pipe wavespeed and time step. The Sectioning panel controls how the pipes are sectioned.

Sectioning is a required item in the Analysis Setup window when transient simulation is enabled.
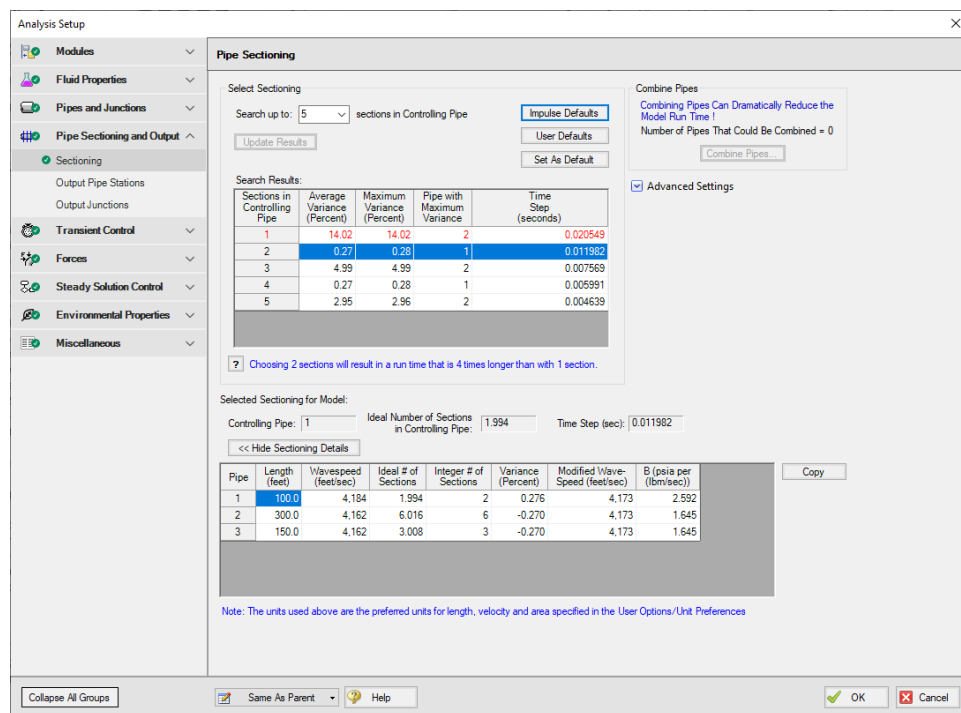
Figure 1: The Sectioning panel

This window automates the selection of an appropriate sectioning scheme for a particular model.

Clicking Update Results analyzes the system based on the length and wavespeed of every pipe. The sectioning window automatically selects the lowest number of sections in the controlling pipe that will produce a Maximum Variance below the accepted amount. This accepted amount defaults to 10%, but can be changed as discussed below in Sectioning Options. The meaning of Variance is discussed in Pipe Sectioning - Introduction to Method of Characteristics.

The Selected Sectioning for Model area shows the effects of the currently selected sectioning on the controlling pipe, as well as the overall time step required.

> **Note:** Be careful when selecting the number of sections. The run time required increases with the number of sections squared, so the number of sections should be minimized where possible. **There is not a direct correlation between number of sections and accuracy of results.** It is not good practice to increase the number of sections to obtain more accurate results. For large models it may be wise to accept additional variance for the decrease in run time, at least for preliminary runs.

Sectioning Options

- **Search up to X sections in Controlling Pipe** - the number of sectioning options to search up to can be defined here. This can help prevent unnecessary searching in large models, or allow additional sections in models where it is required.

- **Combine Pipes** - The [Combine Pipes](#) utility is accessible here. Combining Pipes can greatly reduce model run time in many cases. See [Minor Wave Reflections](#).

- **Advanced Settings**

    - **Show Detailed Searching Options** - This allows for searching a certain range. This is useful if you do not want to spend time searching for low numbers of sections when it is already known that these are unacceptable.

    - **Maximum Allowable Variance** - This has no effect on the model, it only affects which rows are deemed unacceptable (colored red) and which entry is automatically selected.

    - **Search Options**

        - **Bandwidth** - How far away from the specified number of sections should be searched.

        - **Increment** - How many individual points should be tested. The searching algorithm will check all of the pipes for all combinations in a 2*Bandwidth range at Increment intervals. Small Increment and large Bandwidth could potentially find a better result at the cost of computational time.

- **Show Sectioning Details** - This shows the resulting number of sections, variance, and other information about every pipe in the model for the selected sectioning.

Numbering Convention for Pipe Sectioning

Once a pipe is broken into sections, computation takes place where the sections join together. These are called pipe *stations*. The total number of stations in a pipe is equal to the number of sections plus 1. The station at the beginning of the pipe is numbered zero.
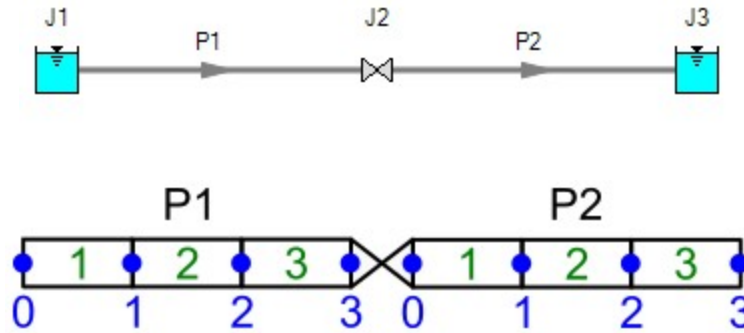
Figure 1: Computing Stations (blue) exist at the boundary of every section (green)

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

Related Topics

Sectioning Panel with PFA Module

([PFA module only](#)) In AFT Impulse (with or without the PFA module), the pipes are sectioned in order to comply with the Method of Characteristics. You can read about pipe sectioning in the [Method of Characteristics Pipe Sectioning topic](#). Essentially, the pipe with the smallest L/a value is the controlling pipe, and the maximum time step will be equal to Equation 1.

$$\Delta t_{max} = \left. \frac{L}{a} \right|_{min}$$

(1)

In the traditional AFT Impulse, the Sectioning panel will display from one section in the controlling pipe (that is, the pipe with the least number of sections) to the specified number of sections in the controlling pipe.

However, in the PFA module, the pipe sectioning must be further refined to prevent any number of pipe sections in the controlling pipe that results in a time step larger than the Maximum Time Step Based on Cutoff Frequency, which is calculated using Equation 2.

$$dt = \frac{1}{6 f_c}$$

(2)

Figure 1 shows the modified Sectioning panel that is used in the PFA module.
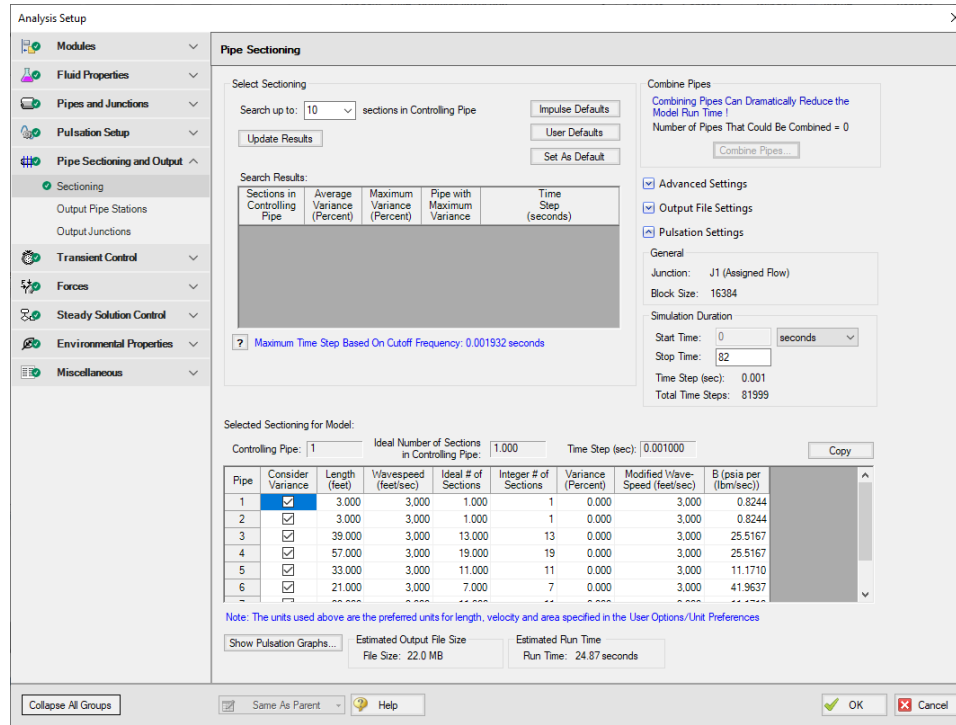
Figure 1: Sectioning panel in the PFA Module

In Figure 1, this example shows that the when the controlling pipe has one section, the time step is still large enough for the Cutoff Frequency. However, if the Maximum Time Step Based on Cutoff Frequency was larger than the 0.001 seconds shown here, the option for one section in the controlling pipe would not be shown. Note that the number of sections in the controlling pipe can drastically affect the model run times, so it is recommended to select the smallest amount of sections in the controlling pipe as possible.

A feature unique to the PFA module in the Sectioning panel is the ability to exclude certain pipes in the sectioning variance or modification calculations. This allows the user to use pipes to model dampeners or devices that are not strictly a pipe. To exclude certain pipes from the sectioning calculations, ensure that the box next to the pipe in the Sectioning panel is unselected. These pipes will still be sectioned based on time step.

The Pulsation Settings section of the Sectioning panel automatically calculates the Block Size based on Equation 3.

$$ BS = 2^m \qquad where: m = 1 + INT\left(\frac{\ln(5) - \ln(dt_b)}{\ln(2)}\right) \qquad (3) $$

The Sectioning panel allows you to specify the Simulation Duration. The minimum required Simulation Duration is automatically calculated.

Note: The Simulation Duration and Output File Setting inputs are located on the Simulation Mode/Duration panel when the PFA module is not activated.

Once the sectioning has been completed, click on the Show Pulsation Graphs icon on the bottom of the Sectioning panel to see graphs of the Pulse, the FFT of the Pulse, the LPF of the Pulse, and the FFT of the LPF. Close out of the graphs, and then click OK to accept the pipe sectioning.

When you exit the Sectioning panel after fully defining the sectioning, a message appears informing you that the PFA module will now place the forcing function onto the junction you selected as transient data. Click OK. You will notice that the forcing function has now been input in the specified junction Properties window on the Transient data tab.

Note: If the forcing function is being added to a pump junction the pump must be defined as a positive displacement pump with a steady flow rate. This requirement is in place to avoid artificial transients in the system.

Related Topics

Output Pipe Stations Panel

The Output Pipe Stations panel allows specific calculation stations to be saved to the output file. Similar to only saving a subset of time steps, saving only certain stations reduces the amount of space required. This does not affect the results of the simulation.

For most purposes, saving only the Inlet and Outlet stations of each pipe is sufficient. There are several reasons to save more data than this:

- If you are interested in the transient behavior at a specific point other than the inlet or outlet, that point must be included. For example, an analysis of a long pipeline with the default options will not permit graphing the pressure over time at the center of the pipeline. Adding the appropriate station to the output file will allow this.

    - Related, a profile graph of a pipe with many sections will become more representative of the simulation results if more points are available to plot.

- Animating a profile plot from the output file requires that all of the stations in the pipes composing the path are saved.

To change the saved stations for multiple pipes at once, check the box next to the pipes to be changed, then change the update the dropdown box next to "Change Selected Pipes To" to the desired option, and click the "Change Selected Pipes To" button to make the change.
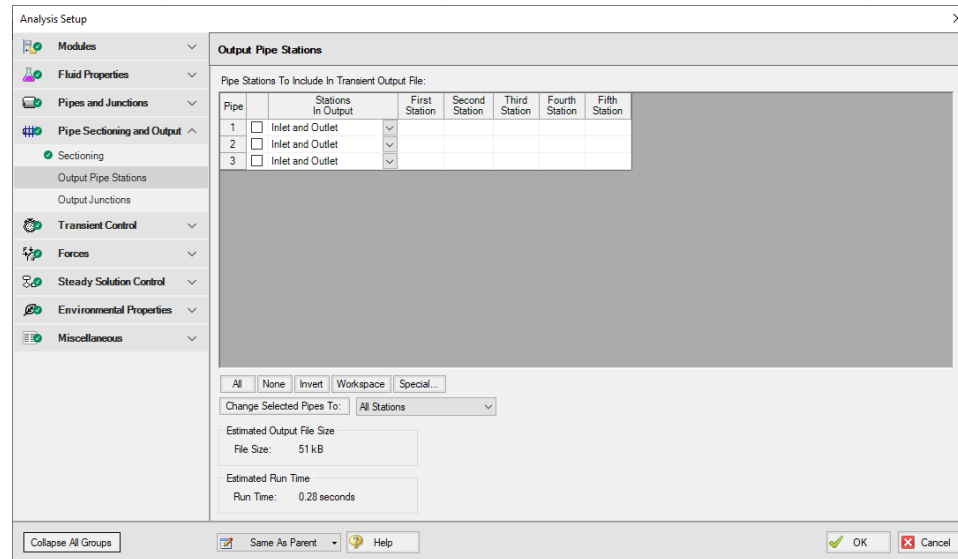
Figure 1: Output Pipe Stations panel

Related Topics

Output Junctions Panel

The Output Junctions panel allows additional parameters to be saved to the output file for some junctions. In general, most relevant data can be obtained from the connected pipes. However, certain data is not available directly from the pipes - for example, Spray Discharge outflow or Pump speed.

The output saved from the Output Junctions panel allows Transient Junction graphs to be created for the listed parameters. By default all junction output will be saved except branch flow rate data. However, junction data can be excluded to reduce model run time and file size if the outputs for that junction are not required for analysis.

The Output Junctions panel for a model with Spray Discharge and Pump junction output being saved is shown below in Figure 1.
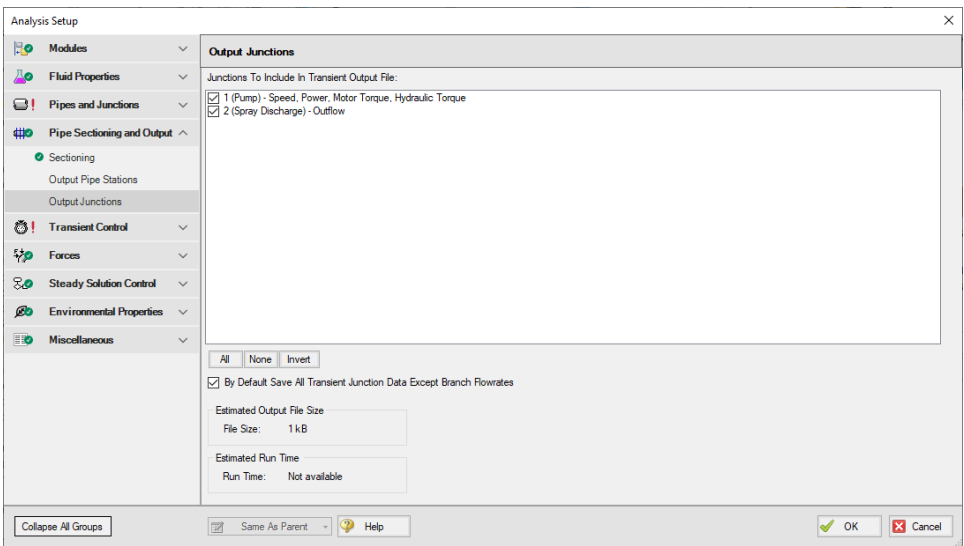
Figure 1: Output Junctions panel

Related Topics

Transient Control Group

The Transient Control group in Analysis Setup allows the user to specify transient parameters such as the simulation duration, cavitation model, and transient friction calculation method. Only the Simulation Mode/Duration panel is required to run the model.

- Simulation Mode/Duration panel - Required panel. Contains the controls for simulation type, simulation duration, and options to save less output to file.

- Cavitation panel - Optional panel. Allows specification of the cavitation model used and relevant model parameters.

- Transient Friction panel - Optional panel. Allows selection of the transient friction calculation method.

Related Topics

Simulation Mode/Duration Panel

This panel contains settings that are **required** for a transient simulation, most of which affect the simulation itself.

Time Simulation

The Time Simulation section defines the type of analysis that will be performed, Steady Only or Transient.

Transient selected by default. A transient simulation always performs a steady-state analysis first, and then proceeds to the transient solution. It can be useful for both design and troubleshooting purposes to run the steady simulation on its own.

These options can also be toggled from the Analysis menu Time Simulation item.

Simulation Duration

As the name implies, this setting controls the duration of the simulation. The total number of time steps required for the specified interval is shown. Note that the Start Time can be a value other than zero. This does not mean that the solver is able to start mid-transient. This is only a reference value, and will affect events defined according to the simulation time. If no such events exist, only the specified duration will impact the overall output.

Save Output to File

- **Every Time Step** - Default selection. Every Time Step is saved to the output file. For long simulations, a very large number of time steps may be required. This will correspondingly require a very large amount of data if this is selected.

- **Every *X* Time Steps** - To reduce file size, only a subset of the data can be saved to file. This will **not affect** the solution, but **will** affect the presentation of results in Output and Graph Results. Generally, transient behavior is sufficiently linear that this does not affect the interpretation of the results. However, in some instances - especially when transient cavitation is present - this may "skip over" short duration spikes in pressure. Note that the maximum and minimum values reported in the Output tab are tracked throughout the simulation and remain valid regardless of this selection. Maximum and minimum lines on a profile graph may be affected, causing an apparent discrepancy.

Transient Simulation Estimations

- **Estimated Output File Size** - The size of the resulting output file can be accurately estimated because it is known beforehand what data is being saved, for how many stations,

and for how many time steps.

- **Estimated Run Time** - The estimated time to run the model is much more difficult to estimate. This should be considered an extremely rough estimate - a better estimate can be obtained by beginning the transient simulation and observing how long it takes to reach 2% complete. The transient simulation process is generally quite linear with time, so a complete solution will take approximately 50 times longer than this.
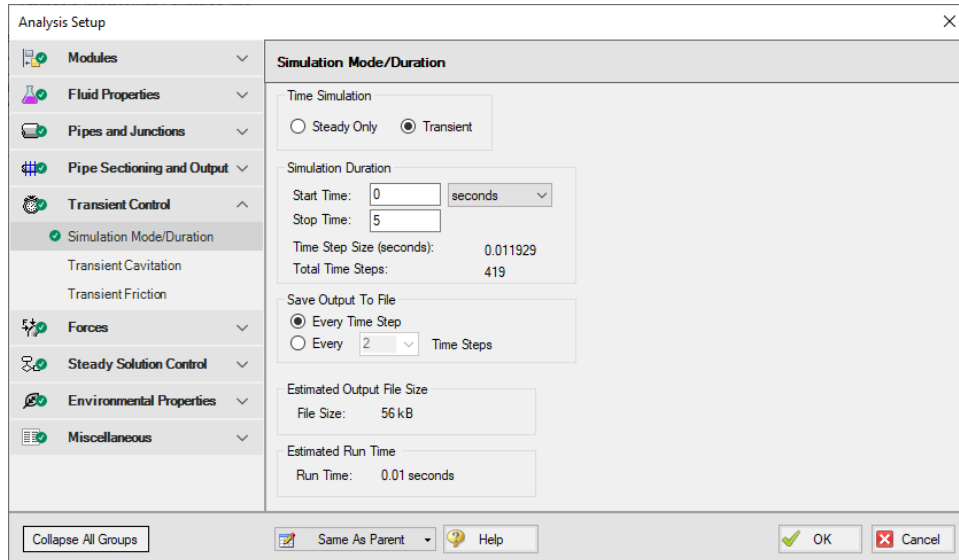
Figure 1: Simulation Mode/Duration panel

Related Topics

Transient Cavitation Panel

The Transient Cavitation panel contains settings for the transient cavitation models.

Transient Cavitation modeling can be enabled or disabled using the check box next to Model Transient Cavitation. Transient cavitation modeling is enabled by default, and it is strongly recommended to keep it enabled. Some situations exist where it can be beneficial to disable cavitation modeling for troubleshooting purposes. When cavitation modeling is enabled, Impulse will use the fluid's vapor pressure and the selected cavitation model to capture transient cavitation. When cavitation modeling is disabled, the Method of Characteristics solution will be allowed to calculate pressure below the fluid's vapor pressure and even below absolute zero.

Two cavitation models are available: the Discrete Vapor Cavity Model (DVCM) and the Discrete Gas Cavity Model (DGCM). DVCM is the default model due to the lower computational complexity and simulation run time.

DGCM is the more advanced model and contains several additional parameters which are all defined by default. The user can modify the DGCM parameters if desired, but the default values are strongly recommended.

Further detail regarding the transient cavitation models and inputs can be found on the [Cavitation Models theory page](#).

Related Topics

Related Blogs

[How does AFT Impulse model cavitation?](#)

Transient Friction Panel

Impulse provides several options for modeling the pipe friction during the transient simulation:

- **Constant** - The pipe frictional resistance (shear stress) obtained in the steady-state part of the simulation will be used as a constant in the transient part of the simulation

- **Time Varying** - The pipe friction will differ for each pipe section because the flowrate (and hence Reynolds number) is varying along the pipe due to the fluid transient. Further, the resistance will vary with time as the flowrate changes.

  - **Simple**: A steady state approximation of the pipe frictional resistance (shear stress) will be calculated as the fluid velocity for each pipe station changes by more than 1%. In AFT Impulse 7 and earlier, this model was included as the "Use Variable Pipe Resistance" option.

  - **Advanced - Unsteady Brunone**: This model uses the same calculation method as the Simple model described above, with the addition of an unsteady shear stress term which varies based on the velocity at the current time step. The unsteady term is calculated using an unsteady friction model commonly referred to as the "Brunone" model. This calculation is a more theoretical model than the Simple Time Varying model, and should be used with caution. See [Adamkowski and Lewandowski (2006)](#) for more information.

Which Friction Model to Use

The three friction models available in AFT Impulse are each intended for different use cases.

The Constant friction model is best used when flow in the simulation stays in the turbulent or highly turbulent regimes. Here, pipe friction is either weakly dependent on flowrate (turbulent flow) or independent of flowrate (highly turbulent flow in the Fully Rough zone of the Moody chart). As a result, the friction factors determined in the steady state system likely remain valid throughout the transient simulation.

The Time Varying - Simple friction model is best used when zero or low flow is seen in some pipes either in the steady state system or in the transient simulation. This situation could arise

when events like pump startups or valve closures are being modeled. Using the steady state friction factor throughout the transient simulation could lead to incorrect estimations of pressure drops in the system, especially when there is zero flow in a pipe during steady state.

The Time Varying - Advanced friction model is best used when it is important to account for the effects that a change in velocity have on pipe friction. The Time Varying - Simple model assumes that each velocity pipe friction factors are calculated for is a 'steady state' velocity, and that a change in fluid velocity does not lead to any additional friction losses other than the friction factor changing. The Time Varying - Advanced model adds in an unsteady shear stress term that helps to account for velocity changing. In models that see flow in a pipe change direction, this term can have a large impact on the transient results.

Note: Both Time Varying friction models require additional calculations and run time when compared to the Constant friction model. Experience suggests that the Time Varying - Simple model increases run time by 20% while the Time Varying - Advanced model increases run time by roughly a factor of three.

Related Topics

Related Blogs

[Five Fantastic New Features for Fast Fluid Transients: Impulse 8](#)

[When Should You Use Variable Pipe Resistance?](#)

Pulsation Setup Group

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

There are two panels which are required for frequency analysis with a PD pump.

- [Pulse Setup panel](#) - Required panel. The Pulse Setup panel allows the user to define properties of the pulse including the source, cutoff frequency, low-pass filtering, and magnitude.

- [PD Pump Settings panel](#) - Optional panel. This panel is only required if the pulsation source is specifed as a PD pump.

Pulse Setup Panel

([PFA module only](#)) The Pulse Setup panel allows the user to define properties of the pulse including the source, cutoff frequency, low-pass filtering, and magnitude.

Pulse Properties

The Pulse Properties section allows you to specify the junction that will act as the source of the pulse, as well as the start time and magnitude of the pulse. By default, the magnitude of the pulse is twice the steady-state flow, but you have the option to define this magnitude as "User Specified". This pulse will be assumed to last for two time steps and then return to steady-state flow. The pulse can only be applied to specific junctions, including a fixed flow pump or an assigned flow junction (these can be specified on a fixed steady mass flow or volumetric flow basis).

Cutoff Frequency

The PFA module will filter out content above the cut-off frequency (the expected maximum frequency) while passing content below this frequency through. If the pulsation source has been modeled as a PD pump, the cutoff frequency will be calculated using Equation 1.

$$f_c = \frac{n\,N}{10} + \left(Pump\ Speed\ Buffer * \frac{n\,N}{10} \right) \tag{1}$$

Where:

- *n is* the number of pump heads
- *N is* the pump speed

Equation 1 is a typical rule-of-thumb that is somewhat arbitrary. As the analyst, you may have good reason to select a value different from this, so the option to define a User Specified Cutoff Frequency is also available.

Low-Pass Filtering

Frequencies above the cutoff frequency are not of interest for pulsation analysis, so a low pass filter is applied to simplify the analysis.

The filter type is the filter used to calculate which signals to pass. The Chebyshev filter is the only filter available in the PFA module. The default low-pass filter settings should be sufficient for most analyses. These settings should not be changed without an understanding of digital signal processing, such as can be found in Vazquez et. al, 2012.

Pulsation Source

The pulsation source section allows you to specify what will be applying the pulsation. Most commonly, users model the pulsation caused by a PD pump. Specifying the Pulsation Source as a PD Pump and inputting the number of heads, the pump speed, and speed buffer will allow the PFA module to automatically calculate the cutoff frequency. The speed buffer default is 15%, and this accounts for modeling uncertainties. This buffer is customizable.

Modeling the pulsation source as Other is used if the pulsation results from something other than a pump, such as a valve that is chattering. If this option is used, the natural frequencies can be graphed, but other PD Pump specific features will not be available.

Related Topics

PD Pump Settings Panel

If a PD pump is the source of pulsation, the PD Pump Settings panel is required to calculate the pump flow forcing function which is used in the Pump Speed child scenarios.

The required pump properties are as follows:

- **Minimum Pump Speed** - affects the lower range of pump speeds at user-selected frequencies after the model has been run.

- **Bore** - diameter of PD pump bore

- **Stroke** - pump stroke length

- **Rod Length** - connecting rod length of pump slider crank

- **Clearance** - clearance ratio of cylinder expressed in volume units or percent of swept volume (dead volume/swept volume), also often referred to as Dead Volume in literature

- **Suction Pressure** - average suction pressure

- **Discharge Pressure** - average discharge pressure

Related Topics

Forces Group

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

Force Definitions Panel

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

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

Defining Force Sets

In order to process force data, the locations where the forces are to be calculated must be defined. Force Sets can be defined either from the Workspace or from the Force Definitions panel of the Forces group in Analysis Setup.

Defining from Analysis Setup

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

- **New** - Add a new force set to the table
- **Duplicate** - Copy an existing force set

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

Each individual force set has a number of required parameters:

- **Force Type** - There are several force set types available. They are described in detail in Force Set Types.
- **Force Set Name** - A unique name for the force set.
- **Start Pipe** - Where the first (only for Point type) node is located.
- **Length To Start Node** - The length from the beginning of the pipe to where the force set starts.
- **Start Station (Actual Length)** - Because calculations must be performed at calculation stations (nodes), the exact length entered in the previous field may not be a valid calculation point. In this case, the nearest valid node will automatically be determined and displayed here. **Keep in mind that the actual force set is between the two "Actual Length" fields, and can be different than the entered values**.
- **End Pipe** - The second (for difference types) node. Note that the selection must be valid for the specified force type. See Force Set Types.
- **Length To End Node** - The length from the start of the pipe to the end node.
- **End Station (Actual Length)** - As described above, the nearest calculation station to the entered length.
- **Pipe Group** - For force sets across 3+ pipes, a group is required and is selected here.
- **Exit Area** - For Exit type forces, an exit area is required.
- **Force Unit Vector Coordinates** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. Impulse is still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the Export Force Data tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

Defining from the Workspace

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

Figure 2: Create Force Set window in the Workspace

Each force has several of required and optional parameters:

- **Pipe Name** - If the pipe has a name entered other than 'Pipe', the user can choose whether to include that pipe name or not in the Force Set name.

- **Force Set Name** - Users can add a name for the Force Set to better organize and manage Force Sets in the model.

- **Force Unit Vectors or Length** - Define unit vectors for your linear pipe to give the resulting forces a three-dimensional direction. Impulse still only a one-dimensional fluid modeling software, and specifying a unit vector will not change the force calculations. However, by defining the XYZ vector for a given force set, that data can be brought into other software for further analysis using the Export Force Data tool. These unit vectors should be defined based on the coordinate system in the force stress analyzing software that the force sets will be exported to.

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

Related Topics

Steady Solution Control Group

The user has control over a number of parameters that influence numerical convergence of the steady state solution.

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

- Tolerance panel - The iterative tolerances that define convergence for the steady state solution can be defined on this panel.
- Iterations panel - Additional iteration controls that can affect whether or not a model converges for the steady state, or how quickly it will converge, can be defined here. Most settings on this panel will not affect the final result.

Related Blogs

Having model convergence problems? Start here!

Tolerance Panel

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

A solution is to use *iterative tolerance*. This tolerance is related to the change between iterations. When this change is smaller than the iterative tolerance, the solution is said to be *converged*. These settings are specified in the Tolerance panel.

Tolerance and Solution Accuracy

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

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

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

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

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

Avoiding False Convergence

It is important to keep the tolerance at least one order of magnitude smaller (two orders or more is recommended) than the relaxation. A false convergence can occur when the change from the old value to the new value is small enough to be within the tolerance because it is restricted by the relaxation. False convergence in the steady state can lead to artificial transients in the Transient Solver.

Absolute Tolerance

The **absolute change method** is generally more reliable for iterative solvers like the steady state solver for AFT Impulse because it is less sensitive to the magnitude of the solution, whether close to zero or very large. However, in principle, specifying absolute tolerance requires some knowledge of the final solution. If you specify mass flow convergence as sufficient when the flow no longer changes by 1 lbm/sec each iteration, then your solution will be compromised if the magnitude is also near 1 lbm/sec. In this case it would be better to set the tolerance 3 or 4 orders of magnitude below the solution.

Relative Tolerance

Relative tolerances, on the other hand, only check the relative change of the number. That is, each successive change is divided by the number itself. This removes the requirement to be knowledgeable about the steady state solution, but can give trouble for problems where one of the solutions is too close to zero (because the change is divided by a number close to zero, making a very large number).

There are options for combining absolute and relative tolerance. You can tell AFT Impulse to assess convergence based on whether either an absolute or relative criteria is satisfied, or whether both absolute and relative criteria are satisfied.

Table 1 shows an example of how the four different criteria would be applied to a flow rate iterations.

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

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

Related Topics

Related Blogs

Having model convergence problems? Start here!

Iterations Panel

Relaxation

Relaxation slows iterative progress toward a solution reducing the destabilizing effect of non-linear systems. By default, the Solver implements an adaptive relaxation scheme that reduces the relaxation dynamically in response to highly non-linear features of the problem. Relaxation can be directly specified in the Iterations panel.

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

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

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

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

- **Flow Rate Relaxation -** For highly non-linear systems, a flow rate relaxation of 0.1 is advised. Experience shows that using flow rate relaxation factors between 0.5 and 1 has limited usefulness. If a flow rate relaxation of 1 does not work, you should generally try flow rate relaxation parameters between 0.1 and 0.5. In more extreme cases, a flow rate relaxation factor smaller than 0.1 may be required.

- **Pressure Relaxation -** Because of the nature of the matrix solver, experience indicates that if pressure relaxation is used, it should **always** be set to 0.5 **or** 1 to maintain stability.

Iteration Limit

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

- **Iteration Span for AutoDetect** - If the 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 very robust and usually provides the fastest convergence.

Iteration History

Iteration History shows a history of which pipe or junction is most out of tolerance, which is helpful for troubleshooting.

Related Topics

Related Blogs

Having model convergence problems? Start here!

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.
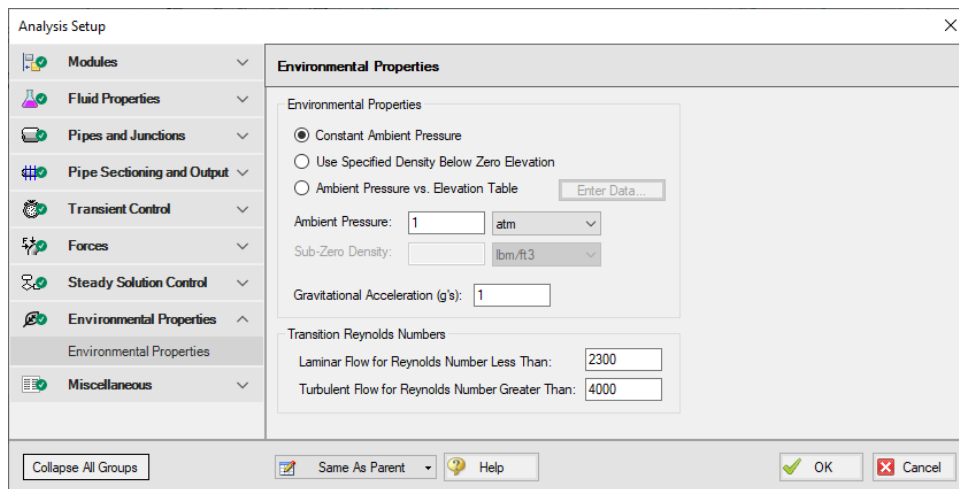
Figure 1: Default state of the Environmental Properties Panel

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 Impulse 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. Impulse can adjust the atmospheric pressure

automatically using the Ambient Pressure vs. Elevation Table option. Users can input a pressure profile to adjust ambient pressure depending on a junction's elevation.

Gravitational Acceleration

The default for gravitational acceleration is 1 standard earth gravitational acceleration, or 32.174 ft/s^2. 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.

If the Variable g feature in the [Waterhammer](#) section of User Options is enabled, users are also able to add a transient to the gravitational acceleration. This could be used for systems in rockets or on a centrifuge.

Transition Reynolds Number

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

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 has three panels:

- [Artificial Transient Detection panel](#) - Contains controls to determine when an artificial transient has occurred and to help troubleshoot artificial transients
- [Threading panel](#) - Allows the user to enable or disable threading for the transient solver
- [Advanced Parameters panel](#) - Contains advanced parameters that can be used for troubleshooting purposes

Related Blogs
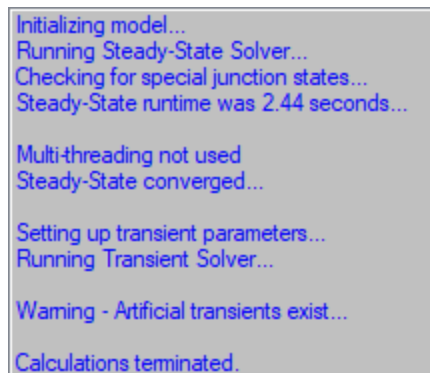
[Are you for real? Understanding and Troubleshooting Artificial Transients in AFT Impulse](#)

Artificial Transient Detection Panel

Artificial Transients are non-real fluid transients that occur when the steady-state results do not match the initial step of the transient calculation. This mismatch will propagate an artificial wave through the system. If this Artificial Transient has significant magnitude and is not detected, it will interact with the real, physical transients being modeled and affect the results.

After the steady-state simulation has been completed, the results will be used as the initial conditions for the Method of Characteristics solution. To check for an Artificial Transient, the Method of Characteristics solution is allowed to proceed a single time step without any user transients applied. Because the system should remain steady-state without any imposed transients, the results after this "zero time step computation" should not change.

The relative change in pressure and flow from steady-state to this "zero time step" is determined for each calculation station. If any of these changes exceed the **Artificial Transient Criteria**, the simulation will be halted.

Figure 1: Halting of transient simulation when Artificial Transients are detected

Due to the nature of the changing calculation methods, it is possible that negligible differences between the two solutions will exist. While technically any difference no matter how small is an Artificial Transient, small differences are unavoidable in practice due to the differing solution methods. For this reason, the values are allowed to change within some tolerance. The Artificial Transient Detection panel provides several options to help troubleshoot and resolve artificial transients:

Note: See the Troubleshooting Artificial Transients topic for more information on how to change these settings appropriately
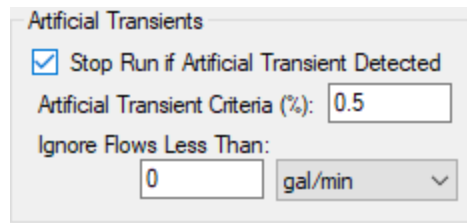
Figure 2: Default detection settings

- **Stop Run if Artificial Transients Detected** - The one and only reason for this option to be disabled is for troubleshooting the cause of an Artificial Transient (see below). It should never be left disabled for non-troubleshooting simulations.

- **Artificial Transient Criteria (%)** - The relative change that is considered small enough to ignore. For some models, it may be reasonable to increase this value. However, this depends in part on the nature of the Artificial Transient. In particular, it should be clear that allowing the Artificial Transient does not significantly impact the results.

- **Ignore Flows Less Than** - Extremely small changes in flow can still be relatively high on a relative basis. For this reason, there may be Artificial Transients that meet the Artificial Transient Criteria but still have very little impact on the model. If this occurs in the model, small flows can be ignored in the detection, allowing the calculation to proceed. Again, when modifying this value it should be clear that the results are not significantly affected.

Related Topics

Related Blogs

[Are you for real? Understanding and Troubleshooting Artificial Transients in AFT Impulse](#)

Threading Panel

Threading is enabled by default to decrease the run time of Impulse models, but can be turned off here if desired. Threading does not affect results.

Advanced Parameters Panel

Advanced settings can be changed here. Users are strongly encouraged not to change these settings from their defaults unless directed by AFT Support or for troubleshooting purposes.

Working with Pipe and Junction Objects

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

Contents

Selecting and Finding Objects

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

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

Related Topics

Selecting Objects

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

Selecting multiple objects can be accomplished by:

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

Select Flow Path

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

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

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

Select Special

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

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

- Selected on Workspace
- Closed in Model
- Special Conditions Set
- Number Range
- Junction Type
- Pipe Input Properties
- Zero-length Connectors
- Notes
- Undefined Objects
- User-Colored Pipes
- Transient Data Exists
- Groups
- Pipe Output Properties (Only available when output is available)
- Junction Output Properties (Only available when output is available)

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

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

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

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

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

ted, 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 models that expand over multiple Workspace pages.

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

Workspace Search

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

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

Find From Model Data or Output

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

Jump Feature

By clicking the Jump button in a Properties window, you can move directly to another Properties Window of another a pipe or junction that you specify.

The Jump button is located in the upper right corner of the Properties Window.

Bookmarks

Use the Bookmark tools on the View menu to set Bookmarks and to display different book-marked 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 the pipes and junctions you want in the group
2. Choose Groups -> Create from the Edit menu.
3. You will be prompted for a name for the group. Enter a meaningful name and click OK.
4. The Group Manager then opens. The selection of pipes and junctions in the Group will match those currently selected on the Workspace. If you want to modify the selections once more, you can do it here.

You can include Subgroups inside of your Groups. Subgroups are existing Groups. If a Group contains one or more Subgroups and the Group is selected on the Workspace, then all pipes and junctions in the Group and Subgroup(s) will be selected. Additionally, if you include a Subgroup in your Group and then later change the contents of the Subgroup, the Group that contains the Subgroup will also be changed.

After you have created a Group, you can then quickly reselect that Group of pipes or junctions on the Workspace by choosing Group -> Select from the Edit menu and choosing the name of the Group.

The Group Manager can be opened by choosing Group -> Manager from the Edit menu. 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 Impulse has 3 different pipe drawing modes. These pipe drawing modes are

- 2D Freeform
- 2D Orthogonal
- Isometric

This section will discuss the features of each drawing mode as well as how to adjust the pipe presentation in each including Snap to Grid.

Related Topics

Related Blogs

Iso love the new Pipe Drawing Mode!

Snap to Grid

Toggled from Arrange > Snap to Grid.

All junctions, pipe endpoints, and pipe verticies will be snapped to the grid intersection points when Snap to Grid is on. Turning Snap to Grid on in an existing model will optionally attempt to move all existing junctions to the grid points.

Related Topics

2D Freeform Grid

The standard grid in AFT Impulse, enabled by default in the Arrange -> Pipe Drawing Mode menu.

This Pipe Drawing Mode allows placing pipes or junctions anywhere on the Workspace, with pipes at any angle.
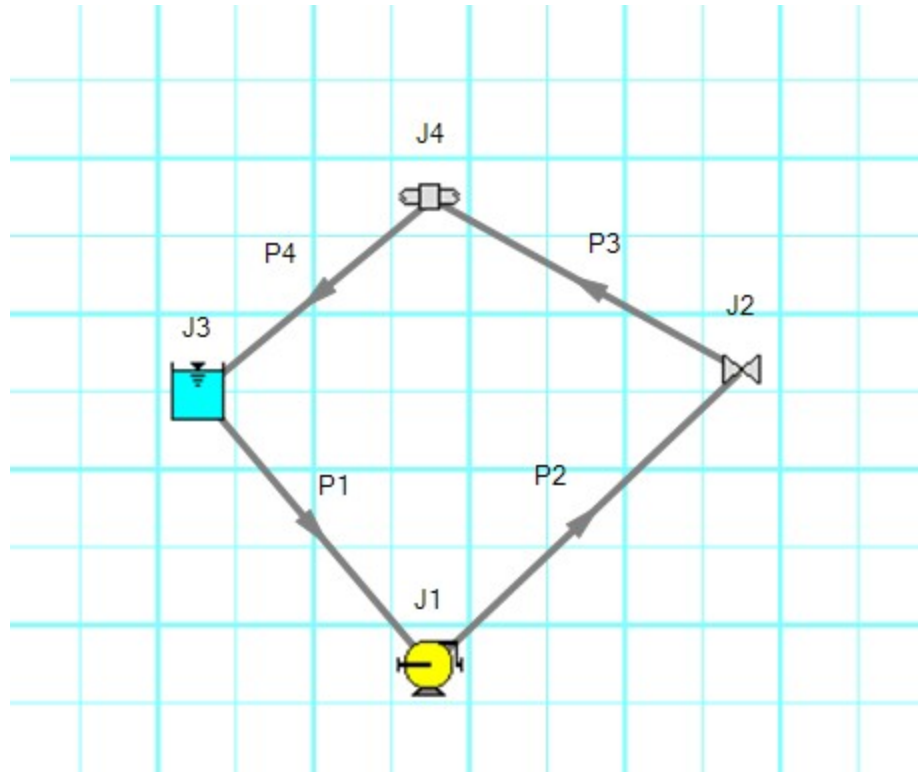
Figure 1: 2D Freeform with Snap to Grid off
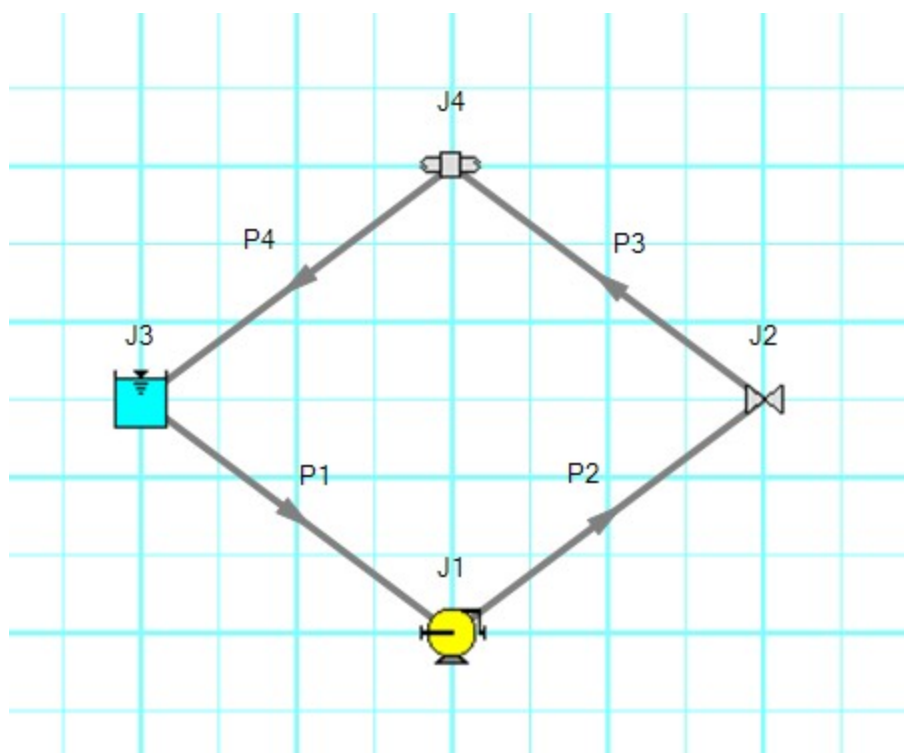
Figure 2: 2D Freeform with Snap to Grid on

Related Topics

2D Orthogonal

The 2D Orthogonal Pipe Drawing Mode forces all pipes drawn in the model to be either horizontal or vertical. This is enabled in the Arrange -> Pipe Drawing Mode menu.

In 2D Orthogonal mode, segments are automatically inserted into pipes to enforce drawing only along the horizontal and vertical.
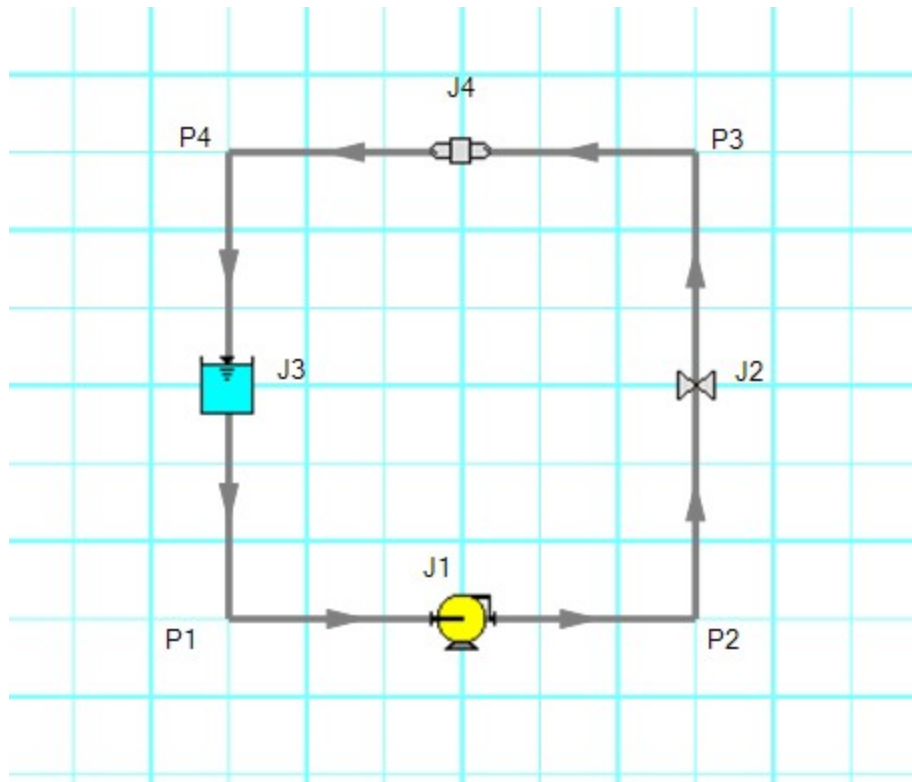
Figure 1: Model in 2D Orthogonal mode with Snap to Grid on

Tips for Using the Orthogonal Drawing Feature

Orthogonal drawing can create some confusion for the user because segments from different pipes may overlap each other and hence become indistinguishable.

- Use the Add/Remove Segments button to add new vertices
- Use a fine grid and Snap to Grid features
- If you have a junction with several connecting pipes that interfere, enlarge the junction to 150% or 200% of normal size

Related Topics

Isometric

AFT Impulse allows the user to place pipe or junction objects anywhere in the Workspace.

In order to make building clean looking models easier, objects are snapped to the 2D Freeform grid by default. However, it may be convenient or desired to demonstrate the three dimensional nature of a system. A typical 2D grid has two sets of gridlines representing the x and y axes. An isometric grid has three sets of gridlines that are offset by 60°, representing the x, y, and z axes.

Enable the isometric grid by selecting Pipe Drawing Mode -> Isometric from the Arrange menu. There will be a prompt about converting non-isometric pipes - this is a required step, but may result in some unintended object placement. It is best practice to set the grid type before building a model and retain that type of grid if possible. It is also very helpful to show the Grid (Arrange -> Show Grid) when building an isometric model.
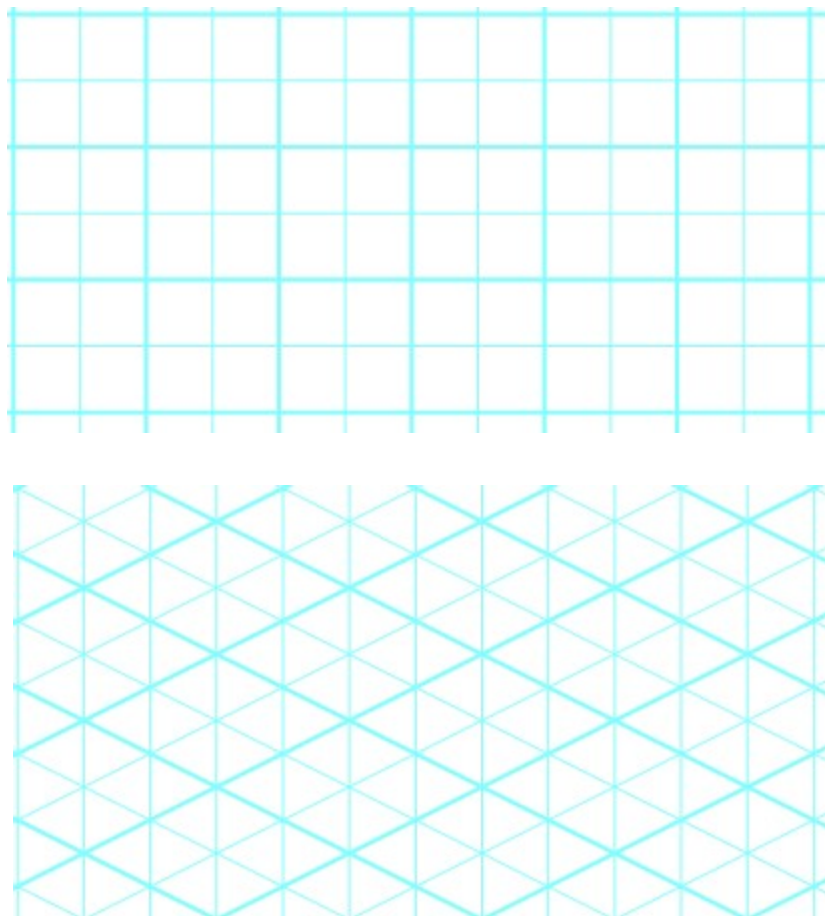
Figure 1: 2D (left) and isometric (right) grids

When drawing on the isometric grid, pipes are automatically segmented to lie on the gridlines as shown below. Dragging one of the handles on the pipe shows a preview line of where the pipe will be placed if adjustment needs to be made.

The placement of the preview line can be cycled through all possibilities by pressing the arrow keys or by scrolling the mouse scroll wheel.

Holding the alt key while dragging a pipe endpoint will retain the existing vertex and auto-matically add a new segment to the pipe - the placement of the line can be controlled with the arrow keys or scroll wheel as before.
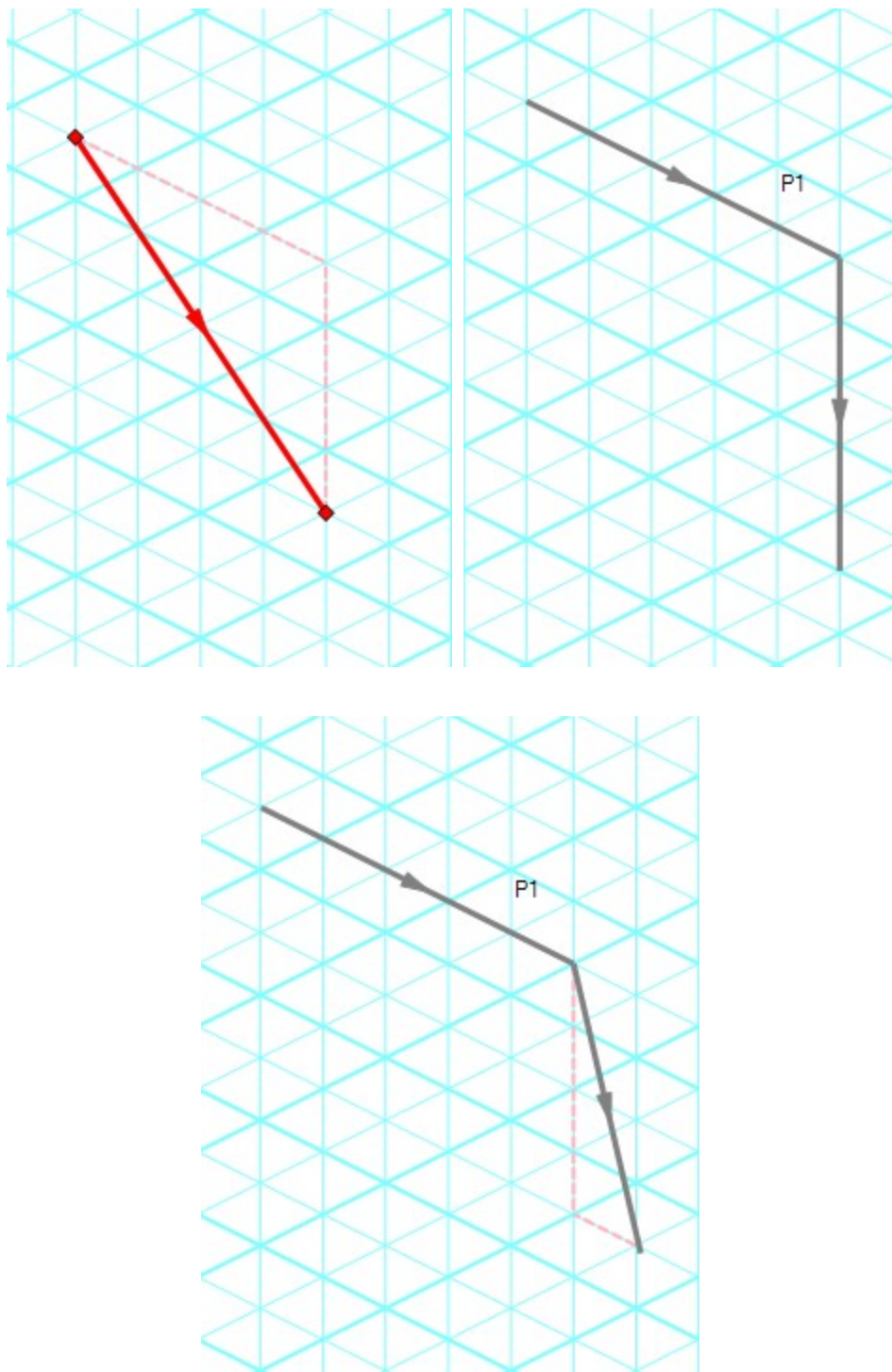
Figure 2: Automatic layout of isometric pipes

The default action is for pipes in the workspace to snap to the isometric gridlines. However, if desired, a given pipe can be set to not snap to the grid by clicking Edit -> No Snapping for Pipe or by right-clicking a selected pipe and clicking the Disable Pipes Snapping to Isometric Grid option.

Placing Junctions

Placing junctions onto the Workspace follows the usual rules - however, the visual appearance of the icon is more complex than in an orthogonal grid. When icons are added after pipes are placed, AFT Impulse will rotate the icon automatically following the workspace arrangement. When pipes are connected to icons already placed in the workspace, right-click and select Auto-rotate Icon to adjust the icon. For manual control of icon appearance, right-click and select Customize Icon to choose the desired appearance.

Note: It may be easiest to make use of the Rotate button in the Customize Icon window to find the appropriate icon. With many possibilities to choose from, finding the proper rotation can be difficult.
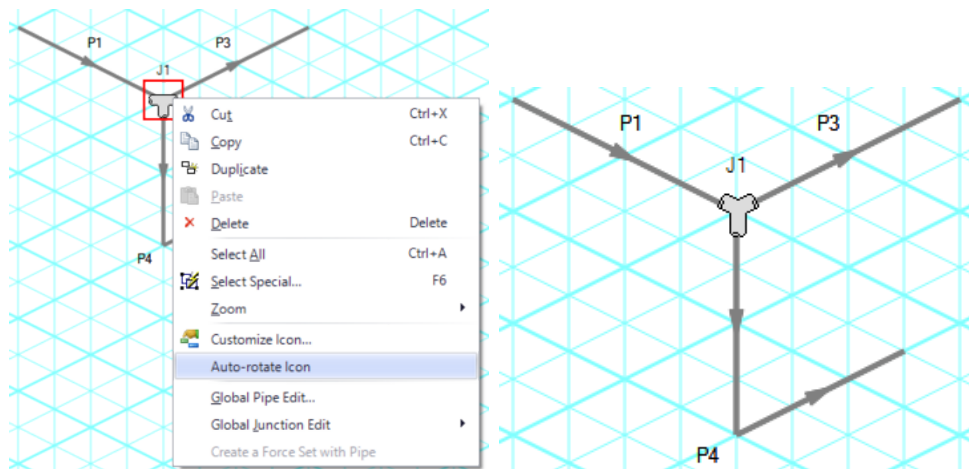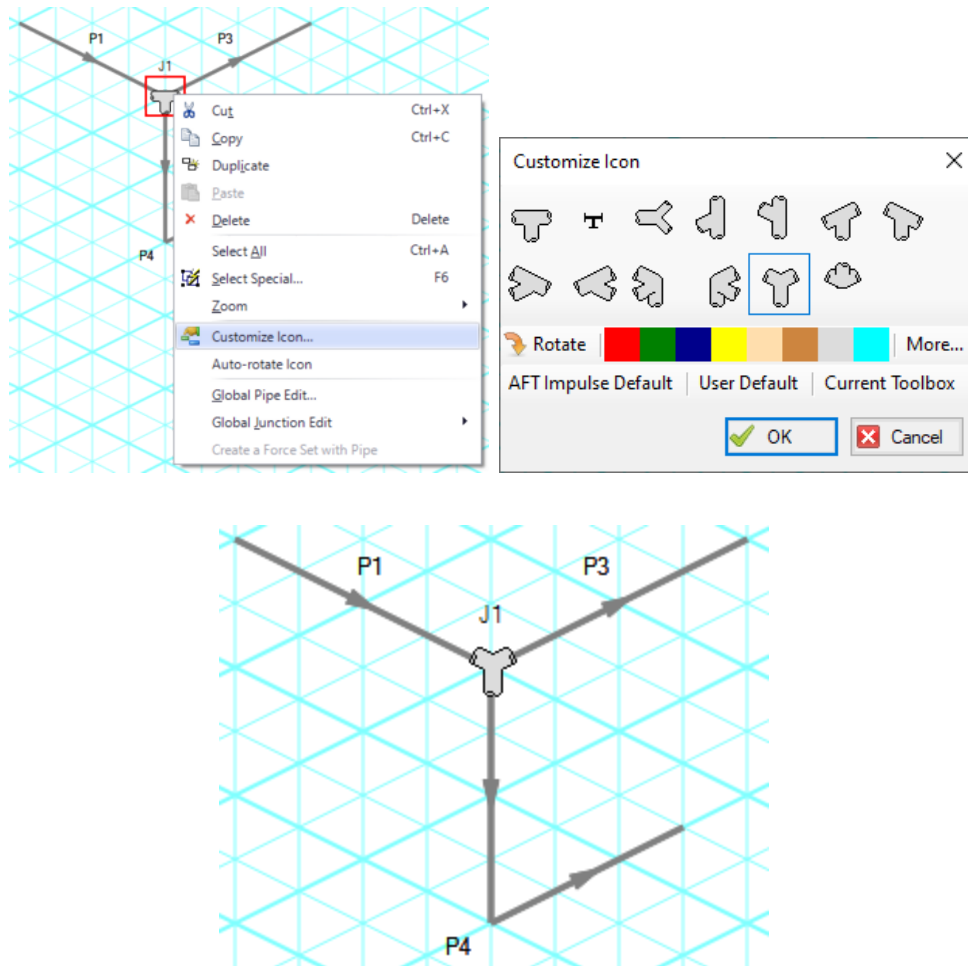
Figure 3: Icon rotations in isometric space

Related Topics

Related Blogs

Iso love the new Pipe Drawing Mode!

Global Edit

AFT Impulse has the powerful capability to apply input parameters across a wide range of objects. This can be done for pipes by using the Global Pipe Edit or across junctions of the same type by using the Global Junction Edit.

Related Topics

Related Blogs

Mass Modify your Model with Global Edit

Global Pipe Editing

The Global Pipe Edit window allows you to edit data for multiple pipes at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process.

Here is how you use it:

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

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

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

Related Topics

Related Blogs

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

Global Junction Editing

The Global Junction Edit window allows you to edit data for multiple junctions at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process, or the instructions are provided below in this topic.

Junction data falls into the following two broad categories:

- Common Data - data that applies to all junctions regardless of junction type; examples include elevation and initial pressure data.
- Junction Specific Data - data that applies only to certain junctions; examples include pump curves and number of heat exchanger tubes.

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

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

Editing Common Junction Data

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

Here is how to globally edit junction common data:

1. Open the Global Junction Edit window.
2. Select the Common option for Junction Template in the upper left.
3. Select the junctions you want to edit from the Junction list on the left. Note that all junctions in the model are shown, no matter what the junction type.

4. Click the Select Common Junction Data button. This will open the Common Junction Data Edit window in which you can enter the data you want to change on each of the selected junctions.

5. Enter the junction data you want to change and click the OK button in the Junction Properties Window.

6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.

7. Click the Apply Selections button.

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

Editing Specific Junction Data

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

Here is how to globally edit specific types of junctions:

1. Open the Global Junction Edit window.

2. Select the Junction Template option in the upper left. Alternatively, if you want to copy the data from an existing junction, you can choose "Existing Junction", then select the desired junction number and skip to step 6.

3. Select the type of junction you want to edit in the dropdown list in the upper left. Here Spray Discharge is selected.

4. Click the Select Spray Discharge Data button (the button name will change based on the type of junction selected). This will open the appropriate junction Properties Window, in this case the Spray Discharge Properties Window.

5. Enter the junction data you want to change and click the OK button in the Spray Discharge Properties Window (or whatever window type is displayed).

6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.

7. Select the junctions you want to edit from the Junction list on the left. Note that only junctions of type selected in the upper left are shown (Spray Discharge junctions in this case).

8. Click the Apply Selections button.

Related Topics

Related Blogs

Mass Modify your Model with Global Edit

Toolbox

The bundle of tools at the left of the Workspace window that allows you to create pipes and junctions. The toolbox can be docked to the left, right, or floated within the workspace by clicking on the arrows on the top of the toolbox.
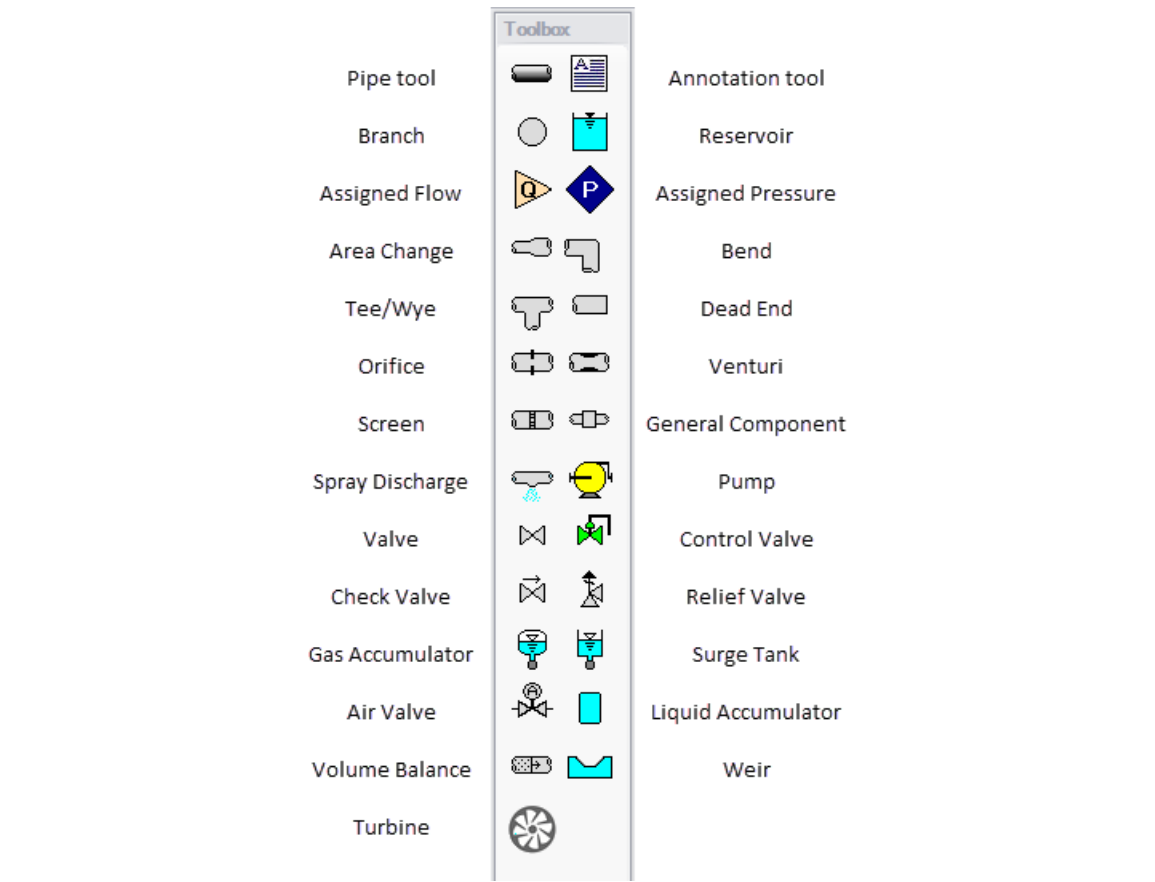
Figure 1: Toolbox items

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 Impulse eliminates the line immediately after you release the mouse.

When a single pipe is selected, two squares (or more if the pipe is segmented), called *handles*, appear. When the cursor is placed over a pipe handle it changes to a crosshair and the pipe handle can be moved or stretched by dragging with the left mouse button.

Another way to create new pipes is to duplicate existing pipes with the Duplicate or the Copy and Paste features found on the Edit menu. When a new pipe is derived from a previous pipe, the new pipe retains the input associated with the original pipe.

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

Pipe ID Numbers

When a new pipe is created, it is assigned a default ID number. When the pipe is displayed on the Workspace, the ID number is preceded by the letter "P" to indicate that it is for a pipe. From the Pipe Properties Window, you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the pipe and cannot be replicated in any other pipes in the model. The pipe number remains with the pipe in descendant scenarios in the Scenario Manager. An arrow near the ID number indicates the reference flow direction for the pipe. The reference positive flow direction can be reversed by choosing Reverse Direction from the Edit menu or by selecting the reverse direction button on the Toolbar.

Tip: The pipe or junction ID number text can be dragged to a new location to improve visibility. The new location is saved with the model. If you want the text of one or more pipes and/or junctions returned to the default location, select those pipes/junctions on the Workspace and choose Reset Label Locations from the Arrange menu. From this box, click the Reset Labels For Currently Selected Pipes and Junctions. If you want all pipe and junction text returned to the default locations, choose Reset Label Locations from the Reset Label Locations box on the Arrange menu.

Pipe Segments

A pipe can be segmented or "bent" into numerous sections. This only affects the visual representation and does not affect the calculated results. It does not add an "elbow" to the model or

the resulting pressure drop effects of an elbow. This feature is useful in graphically bending the pipe around another part of the model or graphically showing that there are embedded elbows as fittings & losses.

You can add a segment to a pipe in several ways. The easiest way to add a pipe segment is to first select the pipe, then click on the +/- image on the Add/Remove Segment icon from the Toolbar. A segment will automatically be added. One or more segments can be added by clicking on the down-facing arrow on the Add/Remove Segment icon from the Toolbar, and then choosing the number of segments to be added. Alternatively, you can first select the pipe, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to choose Add a Segment or Add Multiple Segments.

The easiest way to remove a segment is to first select the pipe, then click on the down-facing arrow on the Add/Remove Segment icon from the Toolbar. A drop down list will appear that will allow you to choose Remove Segment. If you wish to remove all segments, click Remove All Segments. Alternatively, you can first select the pipe from which you want to delete one or more segments, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to remove one or all segments by clicking on either Remove Segment if you want to remove one segment, or by clicking on Remove All Segments if you want to delete all segments.

Keyboard Shortcuts

See the Keyboard Shortcuts for Workspace topic for more information on additional Pipe Drawing Tool shortcuts.

Related Topics

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, friction model, and wavespeed to be combined. In addition, the pipes must be pointing in the same direction.

Joining pipes may cause some junction data to be lost such as special conditions, loss factors for reverse flow, and ignored transients.

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

Note: For maintenance releases prior to November 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.

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 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 Impulse 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 Impulse, 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

Related Blogs

You Should Combine Your Pipes

Creating Junctions

Below the pipe drawing and the annotation tools on the Toolbox are junctions, which allow you to model a large variety of pipe system components. To add a junction to a model, drag the desired icon from the Toolbox and drop it anywhere on the Workspace.

As you move the mouse pointer over the Toolbox, a tooltip identifies the type of junction under the mouse pointer.

When you duplicate a junction using the Duplicate, Copy, or Paste features on the Edit menu, the new junction retains the input associated with the original junction.

Junction ID Numbers

Junctions are shown on the Workspace with default ID numbers, which you can change in the Junction Properties Window. The letter "J" precedes the ID number to signify that it is for a junction. From the junction's Properties window, you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the junction and cannot be replicated in any other junctions in the model. 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 for analysis, you must connect the objects properly.

Remember that *connectivity only exists between* junctions and pipes. There are no junctions that connect to junctions, and no pipes that connect to pipes.

AFT Impulse will attempt to automatically connect pipes and junctions when they are moved. If Impulse 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 Impulse. 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 Assign 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 Impulse'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 Impulse 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 it, drag it within the Workspace, and drop it in the desired location. When an object is dragged beyond the existing Workspace area, the Workspace is expanded accordingly.

A pipe object can be stretched by grabbing the handles (small red squares) at the pipe endpoints and moving an endpoint to a new location. To make these handles visible, select the pipe you want to move.

There are numerous ways to select a group of objects for movement or other operations.

To prevent accidental movement on the Workspace, lock the objects. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction. The Lock feature is accessed from the Arrange Menu or the lock button on the Toolbar. To lock an object on the Workspace, select it and then activate the Lock feature either from the Arrange menu or from the lock button on the Toolbar. If you would like to lock multiple objects at the same time, ensure that all of the objects are selected prior to activating the Lock feature.

Keyboard Modifiers

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

Related Topics

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 editing operatings on objects in the workspace. Alternately you can use the Undo button on the Toolbar.

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

Related Topics

Morphing Junctions

Occasionally you may want to change a junction's type in your model. You can do this by morphing the old junction into a new one.

If you hold down CTRL key and select a junction from the Toolbox and drop it onto an existing Workspace junction, the old junction will morph into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.

Related Topics

Defining Objects

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

1. Double-click the object on the Workspace
2. Select the object (by clicking on it once, or using the Selection Box) and press ENTER
3. Select the object and click the Open Pipe/Jct Window icon on the Toolbar
4. Double-click the left column of the appropriate table in the Model Data window (where the object ID number is shown)
5. Jump from another Properties window
6. Double-click the connected pipe or junction number in a Properties window

Parameters that functionally describe the pipe or junction are entered in the Properties window. Some parameters in the Properties windows are required and others are optional. The optional parameters offer you added control over your model and the ability to obtain specialized information.

The highlight feature shows the required input in pipes and junctions.

Undefined Objects Window

You can open the Undefined Objects window to view all undefined objects and the undefined properties. This is available on the View menu.

Use Status Feature

Each pipe and junction Properties window has a Status tab. The Status tab shows what required information for the pipe or junction is missing.

Satisfying Connectivity Requirements

In addition to entering the required information for each object in the model, you must satisfy these connectivity requirements:

- You must accept the connectivity information shown in each object's Properties window
- Each junction must have the appropriate number of pipes connected to it. For example, an area change junction must have exactly two connecting pipes, whereas a 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 check valve on the Workspace. A check valve requires inlet and outlet piping. AFT Impulse reflects this by requiring each check valve to be assigned an upstream pipe and a downstream pipe. Entering only the Valve model details and setting an elevation is *necessary but not sufficient information to define the check valve* according to AFT Impulse's requirements. Once you have entered the check valve data and properly connected it, the check valve is defined.

Show Object Status

The Show Object Status feature helps you identify which objects on the Workspace are defined and which are not. This feature can be toggled on and off by selecting it from the View menu or by clicking the floodlight button on the Toolbar.

Turning Show Object Status on displays the undefined junction and pipe ID numbers in the Undefined Color, which is red by default. The color can be changed through the User Options window on the Tools menu. Show Object Status helps you rapidly assess where your model requires additional information.

Leaving Show Object Status always toggled on is not recommended for large models, as it will slow the Workspace graphics response. When this feature is on, every Workspace graphical

operation you perform must be accompanied by a check of all pipe and junction connections. This check takes time, and could degrade your graphics performance.

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

Related Topics

Inspecting Objects

The Inspection feature is a time saving feature for reviewing the data associated with a Workspace object. Inspection displays the data for an object in read-only format (see Figure 1).

To inspect an object, position the mouse pointer on the object and hold down the right mouse button. Inspecting is quicker than opening the Properties Window if you want to look at an object's input data but not edit it.

The Inspection feature is also available in each Properties Window. This is useful if you want to check the input data of objects connected to the one you are working on.

For example, when working in a Valve Properties window, you may wish to quickly check the upstream and downstream pipe diameters previously defined. Holding down the right mouse button on each connected pipe ID number will show you that pipe's primary input parameters.

The Inspection feature works similarly in Pipe Properties windows when you inspect connected junctions. Judicious inspecting can save you time when assembling or debugging a model.

If output data exists it will also be displayed.
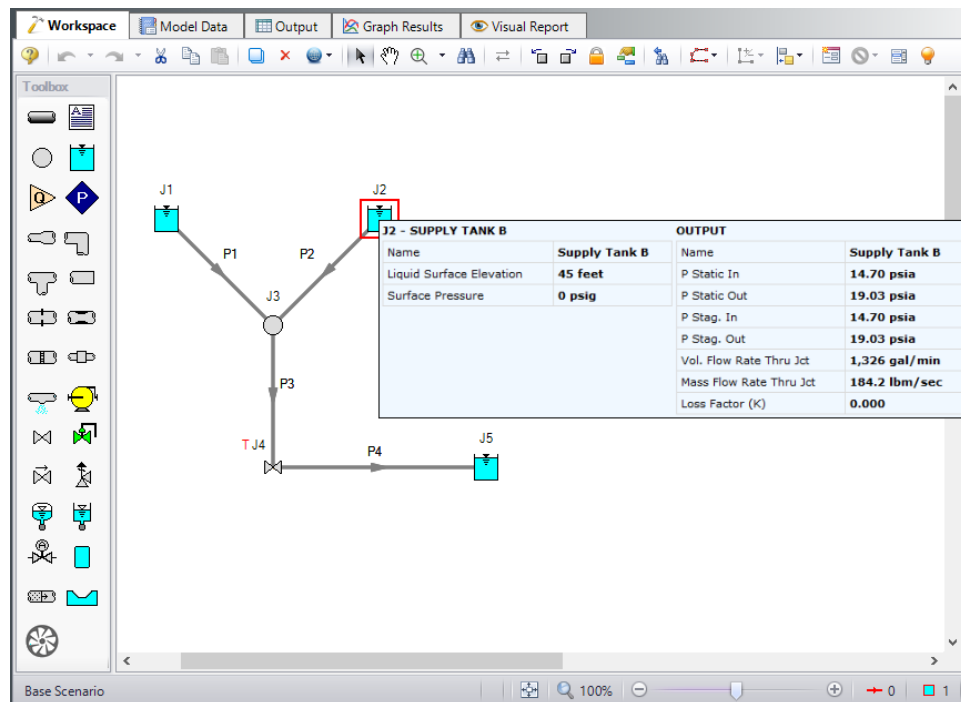
Figure 1: Inspecting from the Workspace with right mouse button

Related Topics

Related Blogs

AFT Responds to 2020 Customer Survey Feedback

Notes

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

Related Topics

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

Related Blogs

AFT Responds to 2020 Customer Survey Feedback

Moving Pipe and Junction Labels

Each pipe and junction on the Workspace also displays text (or label) referring to the number and/or name of the object. This label can be dragged to improve model aesthetics. AFT Impulse keeps track of the relative location of the label to the object so that if the object is moved, the label moves with it.

Returning Label to Default Location

To return the label to its default location, you can:

- Right-click the label for an individual object and choose Reset Label Location
- Use Arrange -> Reset Label Locations and respond to the dialog

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.

The special conditions are set in one of three ways:

- By selecting the junction(s) on the Workspace and choosing Special Conditions from the Edit Menu

- By selecting the junction(s) on the Workspace and clicking the Special Condition icon on the Toolbar
- By opening the junction's Properties window and clicking the appropriate condition on the Optional folder tab.

The junctions which have special conditions set are shown using a special symbol before the ID number (an X by default). This symbol can be customized in the Tools/User Options window.

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

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

Special Condition: Ignore

The Air Valve, Gas Accumulator, Liquid Accumulator, Surge Tank, Relief Valve and Weir junctions have a unique kind of Special Condition called Ignore. This makes the steady-state and transient solvers ignore the presence of the junction. Thus not only is the data ignored, the junction itself is ignored. This is convenient when locating one of these devices by trial and error. Rather than having to delete the junction, it can be ignored.

As an example, assume that one is trying to find the best location for a gas accumulator. If one does this with a single gas accumulator junction, it will be required to run a case, view the results, then relocated the junction by changing the pipe lengths and rerunning. If one uses the Special Condition Ignore, one can place multiple gas accumulator junctions in the model and then set all of them except one to Ignore so that only one of the junctions is used at a time.

Related Topics

Pipes

This section focuses on specifying and modifying the Pipe object in the AFT Impulse 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 Impulse, 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, and two connecting junctions. If the pipe is not purely cylindrical, additional geometry information is also required.

The Pipe Properties Window allows the user to define all data for a pipe on the Workspace. An example on a fully defined Pipe Properties Window is shown below in Figure 1. The various Pipe inputs are discussed in the respective pipe pages listed in the Related Topics section below.
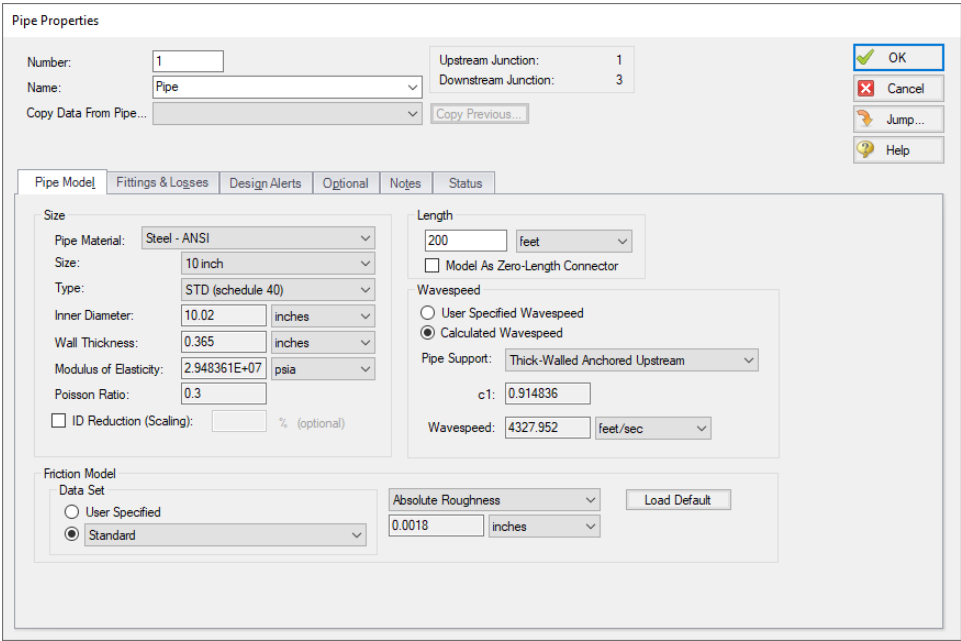
Figure 1: Fully Defined Pipe Properties Window

Related Topics

Related Blogs

Intermediate Elevations and Splitting Pipes

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 auto-matically allocated.

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

Name

A name can be assigned to each pipe for reference purposes. The default name is simply "Pipe". Names do not need to be unique. The name can be shown on the Workspace, in the Visual Report, and in the Output window.

Copy Data From Pipe...

The Copy Data From Pipe dropdown list allows you to copy all or some of an existing pipe's prop-erties to the current pipe. Selecting an existing pipe will prompt the user on what parameters should be changed. The Copy Previous button functions similarly, copying parameters from the previously edited pipe.

In child 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.
- **Size -** Select from a list of defined nominal sizes for the selected Pipe Material. Only available for library materials.
- **Type -** Select from a list of defined types (classes/schedules) for the selected Pipe Material. Only available for library materials.
- **ID Reduction (Scaling) -** Reduction in hydraulic diameter due to scaling can be accounted for by entering a percent reduction in this field. Zero represents no reduction.

Length

The Length is defined as the distance between the two connecting junctions. This length has no relationship to the visual length of the pipe drawn on the Workspace.

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, and they can only be used in up to two sequential pipes.

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.

Due to the nature of the solution method for ZLCs, there are several configurations that are not allowed to be used. See below for a list of restricted options:

- **Junctions that can only be connected to one zero-length connector**
    - Reservoir with Infinite Reservoir tank model and no loss factors defined on the Pipe Depth & Loss Coefficients tab
    - Assigned Pressure

- Tee/Wye with simple loss model
- Spray Discharge
- Branch with Flow Source/Sink defined on Optional tab and no loss factors on the Pipe Depth & Loss Coefficients tab
- **Junctions not allowed to be connected to zero-length connectors**
  - Reservoir with Finite Open tank model
  - Check Valve with Force Balance valve model
  - Gas Accumulator
  - Surge Tank
  - Air Valve
  - Liquid Accumulator
  - Weir
  - Turbine
  - Tee/Wye with detailed loss model
  - Branch with loss factors on the Pipe Depth & Loss Coefficients tab
  - Reservoir with loss factors on the Pipe Depth & Loss Coefficients tab
  - Pumps with Four Quadrant Curve
  - Relief Valve with Internal or Inline Exit valve model
  - Volume Balance

Wavespeed

The wavespeed is the speed at which transient events propagate through the pipe. Typical values for the wavespeed are 2000-5000 ft/sec (700-1600 m/sec). This parameter depends on the liquid acoustic velocity and the pipe material and support.

There are two options to determine a wavespeed for the pipe:

- **User Specified Wavespeed** - If this option is selected the wall thickness, modulus of elasticity, poisson ratio, and pipe support fields will no longer be used and will be grayed out so that the user can enter their own wavespeed.
- **Calculated Wavespeed** - This option will use information from the Fluid panel, the Pipe Material Library, and the chosen Pipe Support model to estimate the wavespeed for the user.

The Pipe Support selection is made from the provided drop down list to obtain the constant c1. This constant is required for pipes in which the wavespeed is calculated. AFT Impulse provides nine common pipe support types from Wylie, et. al, 1993.

- **Thin-Walled** - Piping in which the wall is relatively thin with respect to diameter so that D/e > 25, where D is diameter and e is thickness. Three types are available:

- **Thin-Walled Anchored Upstream** - Pipe anchored at its upstream end only.
  - **Thin-Walled Anchored Throughout** - Pipe anchored throughout against axial movement.
  - **Thin-Walled With Expansion Joints** - Pipe anchored with expansion joints throughout.
- **Thick-Walled** - Piping in which the wall is relatively thick with respect to diameter so that D/e < 25, where D is diameter and e is thickness. Three types are available:
  - **Thick-Walled Anchored Upstream** - Pipe anchored at its upstream end only.
  - **Thick-Walled Anchored Throughout** - Pipe anchored throughout against axial movement.
  - **Thick-Walled With Expansion Joints** - Pipe anchored with expansion joints throughout.
- **Circular Tunnel (unlined)** - Models a conduit through solid rock or concrete.
- **Circular Tunnel (lined)** - Accounts for increase in wavespeed due to a steel liner in contact with a solid rock or concrete tunnel. An additional input for the Rock Modulus of Rigidity is required for this model.
- **Buried Pipe** - Accounts for wavespeed in a buried pipeline. An additional input for the Soil Modulus of Rigidity is required for this 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 the Viscosity Model panel, 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.
- **Hazen-Williams -** A method using an empirical factor to relate flow and pressure drop, common in water distribution.
- **Explicit Friction Factor -** friction factor for the pipe, you can enter the value explicitly.
- **Resistance -** You can specify a pipe resistance in terms of head loss and volumetric flow rate. The pipe head loss will then be as follows: $dH = RQ^2$.

- **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.

- **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 library losses, click Specify Fittings & Losses to open the Pipe Fittings & Losses window. An additional K Factor loss can also be user specified.

Pipe Fittings & Losses

The Fittings & Losses library contains many standard loss values. Because many of these values are dependent on pipe diameter, a pipe diameter must be specified to open the Pipe Fittings & Losses window.

Losses are divided into several categories represented by tabs. To add fittings to the pipe, select the type, and adjust the quantity column next to the desired fitting.

After accepting the changes by clicking OK, a summary of the pipe's Fittings & Losses will be displayed in the pipe's properties window.

Note: Area Change fittings do not change the size of the pipe, they only represent the loss factor based on the pipe diameter and the area ratio of the loss. When including area changes, notice the illustration of how the loss is defined, where Apipe is the area of the pipe and As is the area of the loss. These area changes are useful to model fittings between a pipe and another junction like a pump.
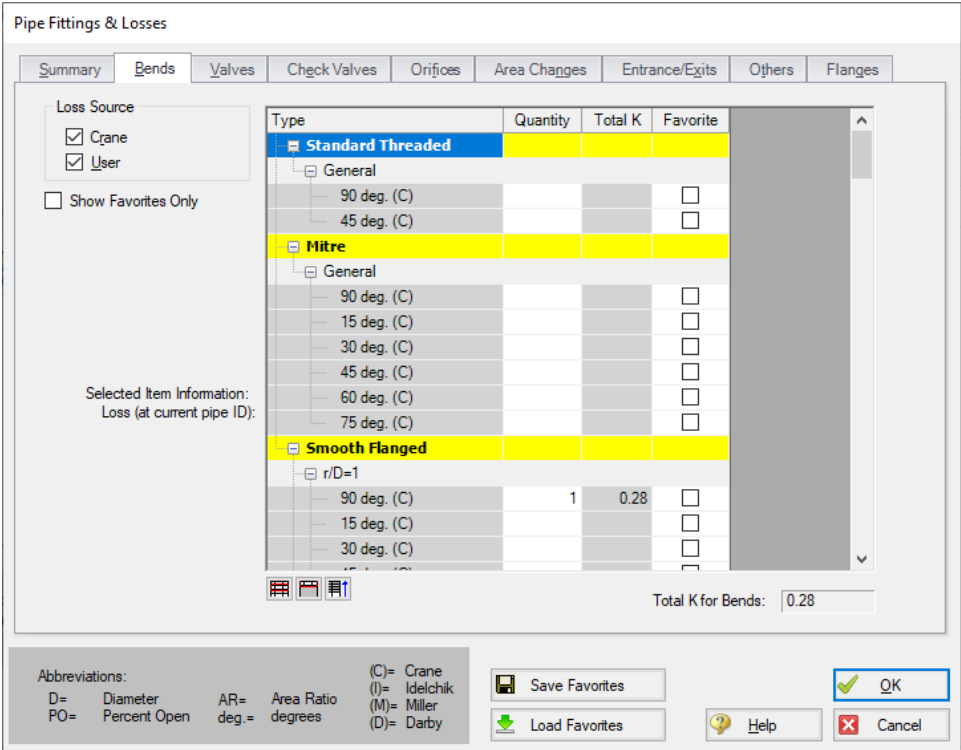
Figure 1: Pipe Fittings & Losses

Fittings & Losses Workspace Symbol

When a pipe has any loss value assigned through the Fittings & Losses tab, a symbol (default of &) is added before the pipe number.

Favorite Fittings & Losses

In the Pipe Fittings & 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

Fluid Properties for Pipes

This folder tab is only visible when variable properties are being modeled. Variable property modeling is enabled in the Analysis Setup window.

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 varied from pipe to pipe. Thus, some pipes in the model can be modeled as Newtonian, with others as non-Newtonian.

Related Topics

Optional Input

Initial Flow Rate Guess

The Initial Flow Guess for a pipe can be specified. An appropriate initial guess can decrease the number of iterations required to arrive at a converged solution. This initial guess will only affect the steady-state solution process.

The Solver generates initial guesses on its own, which is usually sufficient to get the Solver going in the right direction so a converged solution results. If this is not the case, user defined guesses can help obtain a solution.

Stagnant Region Steady-State Pressure

You can specify a reference pressure in a pipe that will be closed during steady-state which will be used for the transient simulation. Only specify one pressure per stagnant region or you will

receive an error stating there is more than one stagnant pressure defined. A single stagnant pressure will allow Impulse to calculate the pressures in the region with hydrostatic pressure differences due to elevation change being taken into account.

Pipe Line Size and Color

Each pipe may have its line size and color changed from the defaults configured in Workspace Options.

Workspace Display

By default, only the pipe number is displayed on the Workspace. In addition to this, the pipe name, nominal size, and type/schedule can be displayed.

Design Factors

A design factor can be used to add safety margins into the hydraulic analysis. For pipes, two design factors exist:

- **Pipe Friction:** friction factor is multiplied by the design factor assigned to the pipe.
- **Pipe Fittings & Losses:** total loss factor (K factor or Equivalent Lengths) is multiplied by the design factor assigned to the pipe.

Support for Partially Full Pipe

There is checkbox which indicates the pipe supports partially full pipe calculations. This means partially full along the axial direction (length), as opposed to partially full along the radial direction (radius). This function only works for pipes that have a slope, and when the pipe is partially full it drains from the end with the higher elevation.

If this feature is not selected, the pipe will always be assumed to be liquid full. Selecting this feature implies that one end of the pipe is connected to a junction at its higher elevation endpoint which allows gas to flow in, thus allowing the pipe to drain.

If the option is selected, and the entry is 100%, it means the pipe is initially 100% full, but can drain if backflow occurs at the high elevation endpoint. The percent full can also be set to something less than 100%. In this case, the pipe is assumed to be partially full initially. This is only valid when the steady-state condition for the pipe is such that there is no flow, since if there were flow in that pipe during steady-state, the pipe would not be partially full.

This feature only works with pipes which connect to a air valve, spray discharge, assigned pressure or exit valve junction at the high elevation end. When selected, the predicted level of the liquid surface and other related parameters can be graphed in the Graph Results window. When pipes can be partially full, more accurate predictions result when the pipe is modeled with more sections. At least two sections should be used, and more if possible. This is because Impulse will

model each section in the pipe as either completely liquid full or empty. Therefore, more sections in the partially full pipe will allow Impulse to more precisely determine the liquid level.

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.

Intermediate Pipe Elevations

The inlet and exit elevations of a pipe are equal to the elevations of the upstream and down-stream 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:
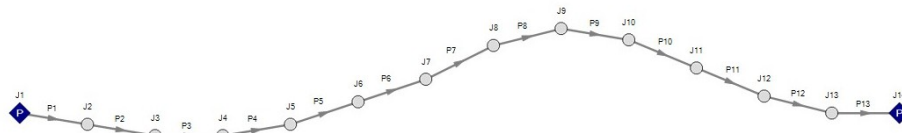
Figure 1: Accounting for elevation changes with many pipes and junctions

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 Impulse 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. The maximum number of rows available in this table can be changed in User Options.

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.
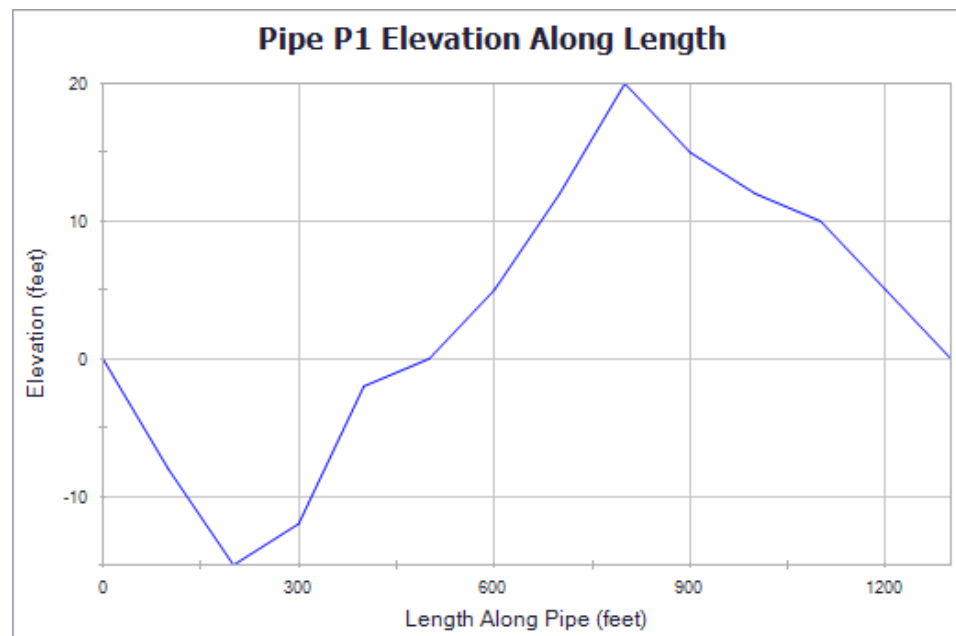
Figure 3: Intermediate Elevation Graph

Related Topics

Related Blogs

Intermediate Elevations and Splitting Pipes

Add/Remove Pipe Segment

Visual segments can be added to pipes in the Workspace.

To add or remove segments, use the Arrange -> Pipe Segments submenu. These options are also available on the Workspace Toolbar.

Note: These segments are useful for creating a clear layout of the system, but have no impact on the solution - a visual "bend" is not the same as adding a Branch junction.

Related Topics

Related Blogs

Intermediate Elevations and Splitting Pipes

Reverse Pipe Direction

Every pipe has a reference positive flow direction as indicated by its arrow. This direction can be easily reversed by selecting Reverse Direction from the Arrange menu or clicking the Reverse Direction button on the Workspace Toolbar.

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 Impulse utilizes the Darcy-Weisbach loss model to relate the Darcy Friction Factor, the pipe geometry, fluid density, and fluid velocity to pressure drop in the pipe. The Darcy Friction Factor differs from the Fanning Friction Factor by a factor of 4.

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

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

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

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

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

- 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:

$$\Delta H = R\,Q^2$$

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

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

Non-Newtonian Pressure Loss

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

Hazen-Williams Method

AFT Impulse also offers the Hazen-Williams method of specifying irrecoverable loss information. The Solver converts the Hazen-Williams factor to a Darcy-Weisbach friction factor (Walski 1984, 37). This allows a consistent solution approach to be used for all pipe system models, while retaining the flexibility of the two approaches to account for losses.

$$C_{HW} = \frac{17.25}{f^{0.54}\,(V\,D)^{0.081}}$$

where *V* is in ft/s and *D* is in ft.

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*):

$$\frac{dP}{L} = \frac{(0.241 \, f \, s \, Q^2)}{d^5}$$

$$r = \frac{0.0119 \, Q}{dv} \quad MIT \; Reynold's \; Number$$

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

where:

- dP = pressure drop (psi)
- L = length (miles)
- f = friction factor, MIT
- Q = volumetric flow rate (barrels/hour)
- 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 Impulse 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 Impulse 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 Impulse 10 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):

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

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 lbm/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

SSL Module

Settling Slurry Module

An optional add-on module to AFT Impulse which allows users to simulate settling slurries.

Junctions

This section focuses on the Junctions in the AFT Impulse workspace as well as the theories that govern the junctions. Topics included discuss the types of junctions that AFT Impulse 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 types of junctions offers specialized features to allow you to prepare a good conceptual model of the physical system of interest. Table 1 compares the features of the junction types.

There is a junction Properties window for each junction type. Each junction Properties window falls into one of two basic window formats, depending on the number of connecting pipes allowed for that junction type.

> **Note:** AFT Impulse does not prevent negative flow through junctions. Should this occur, AFT Impulse will use the same loss factor or pressure drop data referenced to the defined upstream pipe, just as in forward flow. Because it is possible that the loss factor or pressure drop data would not be the same for both flow directions, the flow solution could be misleading. For this reason it is important that you either properly define the pipe flow directions for junctions or pay close attention to the flow directions that result. AFT Impulse will not always give a warning for this, though it will for certain junctions.

Table 1: Comparison of junction capabilities

Junction Type	Connecting Pipes	Special Conditions	Custom Library	Window Format
Air Valve	1-2	Yes	Yes	1
Area Change	2		Yes	1
Assigned Flow	1	Yes	Yes	1
Assigned Pressure*	1/1-25		Yes	1
Bend/Elbow	2		Yes	1
Branch	2-25	Yes	Yes	2
Check Valve	2	Yes	Yes	1
Control Valve	2	Yes	Yes	1
Dead End	1			1
Gas Accumulator	1-25	Yes	Yes	2
General Component	2	Yes	Yes	1
Liquid Accumulator	1-25	Yes	Yes	2
Orifice	2		Yes	1
Pump	2	Yes	Yes	1
Relief Valve	1-2	Yes	Yes	1
Reservoir	1-25		Yes	2
Screen	2		Yes	1
Spray Discharge	1-4	Yes	Yes	2
Surge Tank	1-25	Yes	Yes	2
Tee/Wye**	3			2
Turbine	2	Yes	Yes	1
Valve	1-2	Yes	Yes	1

Venturi	2		Yes	1
Volume Balance	2			1
Weir	2	Yes	Yes	1

*Assigned Pressure junction connects to only one pipe for static pressure, and up to 25 for stagnation
**Tee/Wye 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, those with one or two pipes adopt a direction from the connecting pipes. For these junction types, upstream and downstream pipes are recognized by AFT Impulse 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 important to have the pipes' 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.

To inspect in this window format, position the mouse pointer over the connected pipe ID and hold down the right mouse button.

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

Format #2: Junctions With More Than Two Connecting Pipes

The second basic window format for junction Properties windows is for junctions that allow more than two connecting pipes (see Figure 2). These junctions typically allow up to 25 connecting pipes; the exception is the Tee/Wye junction, which allows only three.

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 Impulse'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 pipe for reference purposes. Names do not need to be unique. The name can be shown on the Workspace, in the Visual Report and in the Output window.

Library List

A list of all available custom components. It displays components from any connected library.

Selecting a component from this list automatically retrieves the junction settings and populates the properties window. Changing the input manually will deselect the component in the list.

Copy Data From Jct

The Copy Data From Junction dropdown list allows you to copy all or some of an existing junction's properties to the current junction. Selecting an existing junction will prompt the user on what parameters should be changed.

In child scenarios,Copy Data From Junction lists "Parent Junction Data," allowing the user to copy all or some parameters from the parent junction.

Pipe Connectivity

The connected pipes are displayed. This is helpful to determine potential connection issues and easily identify which pipes are upstream and downstream of the junction.

Elevation

Every junction must have an elevation reference. This value is the centerline elevation of the connected pipe(s).

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 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.

Jumping to Another Junction

Quickly move to the next junction. See Jump Feature in [Finding Objects](#).

Related Topics

Optional Input

Initial Guesses

A user defined initial guess can be entered for pressure (or head) 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 [Steady-Solver](#) iterations required.

Workspace Display

By default, only the junction number is displayed on the [Workspace](#). In addition to this, the junction name can be displayed.

Special Conditions

The [Special Condition](#) can be changed for some junctions. For example, a valve can be set to closed, preventing all flow. A symbol (default of "X") will be placed before the junction label indicating that a special condition is applied.

Design Factor

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

- **Pump Junctions:** head rise at the pump is multiplied by the design factor assigned to the pump.
- **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

You may want to change the icon to better represent a specific junction type. This is accomplished by clicking the Change Icon button on the Optional folder tab. The Change Icon window appears, allowing you to choose a new icon from a list of available icons.

You may want to change the size of the junction to distinguish it from other junctions. This is accomplished by clearing the use Workspace Default for Size checkbox and selecting the desired size in the drop down list.

Related Topics

Specifying Losses

Losses can be specified in two ways. The most flexible way of including a loss factor in an AFT Impulse 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.

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. When modeling for waterhammer, it is preferred that losses for all static components be included in pipes as Fittings & Losses. This reduces the number of pipes and model run time.

Related Topics

Conventions for Specifying Base Area

Loss factors are area dependent. When specifying a loss factor at a junction, the base area is of critical importance. By default AFT Impulse adheres to the following convention: whenever it makes sense physically, the base area is always the upstream pipe flow area. However, this can be modified by the user. In most Junction Properties windows (on the Optional tab), the base area can be specified as the upstream pipe, downstream pipe, user specified area, or user specified diameter.

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 Tee/Wye, Branch, Reservoir, Assigned Pressure, and Assigned Flow. Other exceptions are Valves and Relief Valves specified as exit valves.

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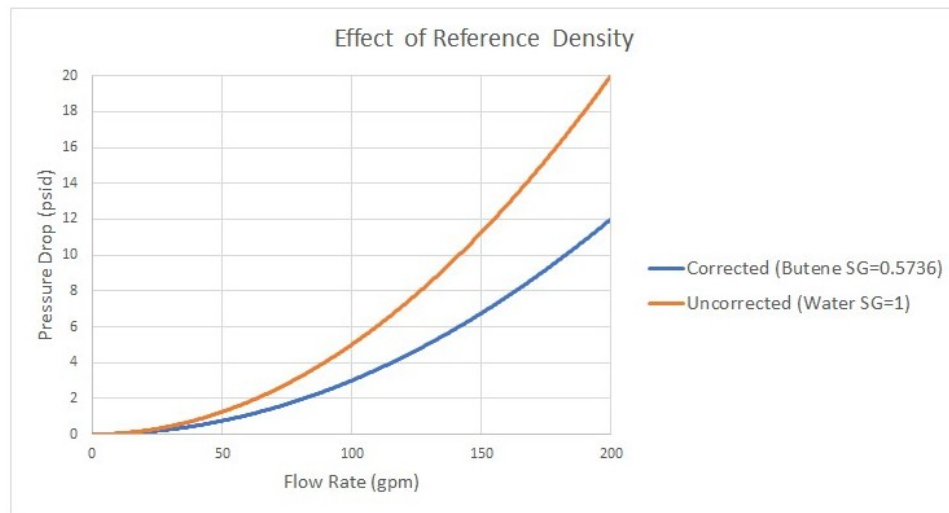
Figure 1: Visual comparison of Pressure Loss Curves for differing densities. Head curves would be identical for this component.

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 any Resistance Curve allow for the specification of a reference density if one of the above parameters is selected. The selection appears below the curve fit area as shown in Figure 2.

Figure 2: Reference Density Selection

There are three options:

- No Correction (default) - This selection assumes that the component was tested with the fluid defined in the Fluid panel.

- Water @STP - This selection assumes density of the test fluid - is that of water at standard temperature and pressure. If the System Property fluid differs, the entered curve will be corrected.

- User Specified - This directly specifies the reference density. If the System Property fluid differs, 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

Lumped Inertia Pipe with Orifice Waterhammer Theory

Some junctions in AFT Impulse use a pipe as a connector for some elements, although it is not a complete element by itself. Such an element may also include an orifice as shown in Figure 1. These connector pipes are modeled as pipes with lumped inertia (Wylie, 1993), such that their losses are included with the junction calculations.
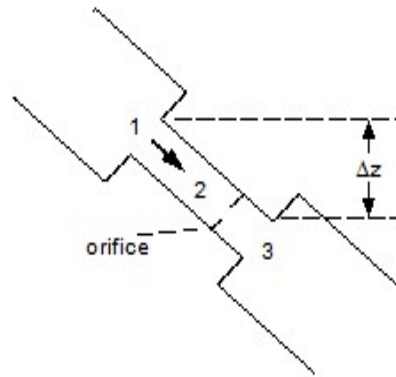
Figure 1: Lumped inertia with friction schematic

Summing forces acting on the lumped section, 2,

$$F_1 - F_3 - F_{friction} - F_{orifice} + F_g = \rho \, A_2 \, L_2 \, \frac{dV_2}{dt}$$

or,

$$F_1 - F_3 - F_{friction} - F_{orifice} + F_g = L_2 \, \frac{d\dot{m}_2}{dt}$$

Now,

$$F_1 = P_1 \, A_2$$

$$F_3 = P_3 \, A_2$$

$$F_{friction} = \frac{f \, L_2}{D_2} \frac{\dot{m}_2{}^2}{2 \, \rho \, A_2{}^2} A_2$$

$$F_{orifice} = \frac{\dot{m}_2{}^2}{2 \, \rho \, C_D{}^2 \, A_0{}^2} A_2$$

$$F_g = \rho \, A_2 \, g \, \Delta z$$

Substituting terms and dividing by area A2

$$P_1 - P_3 - \frac{f \, L_2}{D_2} \frac{1}{2 \, \rho \, A_2{}^2} \dot{m}_2{}^2 - \frac{1}{2 \, \rho \, C_D{}^2 \, A_0{}^2} \dot{m}_2{}^2 - \rho \, g \, z = \frac{L_2}{A_2} \frac{d\dot{m}_2}{dt}$$

Taking average values over time, and denoting current time by a prime,

$$\frac{P_1' + P_1}{2} - \frac{P_3' + P_3}{2} - \frac{f \, L_2}{D_2} \frac{1}{2 \, \rho \, A_2{}^2} \dot{m}_2' \, |\dot{m}_2| - \frac{1}{2 \, \rho \, C_D{}^2 \, A_0{}^2} \dot{m}_2' \, |\dot{m}_2|$$

$$- \rho \, g \, z = \frac{L_2}{A_2} \frac{(\dot{m}_2' - \dot{m}_2)}{\Delta t}$$

Collecting terms and simplifying

$$P_1' - P_3' = C_1 + C_2 \, \dot{m}_2'$$

where

$$C_1 = P_3 - P_1 + 2\,\rho\,g\,z - \frac{2\,L_2}{A_2\,\Delta t}\,\dot{m}_2 \tag{1}$$

$$C_2 = \frac{f\,L_2}{D_2}\frac{1}{\rho\,A_2{}^2}\,|\dot{m}_2| + \frac{1}{\rho\,C_D{}^2\,A_0{}^2}\,\dot{m}_2 + \frac{2\,L_2}{A_2\,\Delta t} \tag{2}$$

If a connector pipe exists but no orifice then

$$C_1 = P_3 - P_1 + 2\,\rho g\,z - \frac{2\,L_2}{A_2\,\Delta t}\,\dot{m}_2$$

$$C_2 = \frac{f\,L_2}{D_2}\frac{1}{\rho\,A_2{}^2}\,|\dot{m}_2| + \frac{2\,L_2}{A_2\,\Delta t}$$

If an orifice exists but no connector pipe

$$C_1 = P_3 - P_1$$

$$C_2 = \frac{1}{\rho\,C_D{}^2\,A_0{}^2}\,|\dot{m}_2|$$

If neither exist then

$$C_1 = 0$$

$$C_2 = 0$$

Related Topics

Junction Loss Models

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

Related Topics

K Factor Loss Model

AFT Impulse models K factor component losses according to the following equation:

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

where *K* is commonly referred to as the loss factor.

Table 1 lists the sources for the loss models used in AFT Impulse. The losses implemented directly in the code were chosen on the basis of ease of use. Many loss factor types are functions of the flow, and thus too general to be easily incorporated.

Table 1: Loss model references

Junction Type	References
Bend	Crane 1988, A-29 Miller 1990, 207-209
Area Change	Crane 1988, A-26 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 Impulse 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 only available in several junctions, including the General Component and Heat Exchanger junctions. It is useful when measured flow vs loss data is available for a component.

It is not uncommon to obtain from an equipment manufacturer a single data point. For example, the data might be 3 psi pressure drop at 1 lbm/sec. Along with this data point, another always comes along for free. The second data point is zero pressure drop at zero flow.

The pressure drop characteristics of most equipment follows the square law. As the flow rate increases, the pressure increases with the square of the flow rate.

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

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

Related Topics

Defining Junction Transients

This section contains information related to defining transient behavior for junctions in Impulse. Topics include discussions on standard junction transient information, transient initiation options, the infinite pipe boundary condition, and the Waterhammer assistant. For information on overall transient simulation controls see the Pipe Sectioning and Output group and Transient Control group.

Contents

Related Blogs

And, Not, Or - Starting Transient Events with Complex Logic

Junction Transient Data

Many junction types have an area where active transient data can be specified. **Active transient** in this context means that the behavior of a component over time is being directly defined by the user. It is also possible for components to react **passively** during the simulation as well - for example, a relief valve opening under excess pressure.

Table 1: Junctions that support transient data

Junction Type	Type of Transient
Branch	Flow rate
Reservoir	Surface Elevation
Assigned Flow	Flow rate
Assigned Pressure	Pressure
Control Valve	Control Setpoint
Valve	Loss
Surge Tank	Surface Pressure
Spray Discharge	Loss
Pump	Speed vs Time, Trip, Startup
Turbine	Gate Position

The active transient data is entered on the junction's Transient tab.

Figure 1: Transient tab for a Valve

In most cases, transient data can be entered as absolute values or as a percentage of steady-state, as defined by the first options in Transient Data.

The first data point always needs to match the steady-state value which is usually entered on the Loss Model tab. If, for example in Figure 1, the steady-state data for C_V of 1000 is changed to 750, the transient data will also need to be changed.

Note: If a junction does not have any active transient behavior, there is no need to enter any data on the Transient Data tab.

Repeat Transient

If the transient data is periodic, you can enter the data for one cycle of the period and then tick the check box for Repeat Transient. This will cause the one cycle of transient data to be repeated once it has reached the end. The repetition will continue until the end of the simulation.

Transient Special Condition

There are instances where you will only wish to use the defined transient data in certain cases or scenarios. Instead of deleting the transient data, specify the Transient Special Condition as Ignore Transient Data. The data will be left in the window for future use, but will not activate during the run.

A third type of Transient Special Condition is available for Assigned Flow and Assigned Pressure junctions. This is the No Reflections (Infinite Pipe) option. In such a case, the flow or pressure is used only for steady-state calculations. During the transient the flow or pressure is not fixed, but varies in accordance with the concept that no transient reflections occur. All transient data at this junction will be ignored. For more information see Infinite Pipe.

Initiation of Transient

There are multiple ways that the data entered in Transient Data can be applied during the simulation. The default selection is Time for most junctions, meaning that the behavior vs simulation time is directly defined by the user.

It is also possible to initiate a transient based on a system event. This will execute the Transient Data when a user defined parameter reaches a certain setting. For example, a valve can be set to close only when a specific flow rate becomes too low.

See Initiating a Transient for detailed information.

Related Topics

Related Blogs

And, Not, Or - Starting Transient Events with Complex Logic

Infinite Pipe

Infinite pipe is a special feature enabled with a Transient Special Condition in the Assigned Flow and Assigned Pressure junctions. This feature prevents the junction from reflecting pressure waves.

This is useful in modeling the part of a pipe system which is so long that its communication time prevents wave reflection interactions with the rest of the system during the simulation. During steady-state the flow or pressure is fixed, but during transient flow both the pressure and flow change in a manner consistent with the fact that no waves are reflected.

The infinite pipe feature can significantly reduce model run time. Any pipe long enough that it does not reflect any waves during a meaningful simulation time will also require a large number of sections. This may dramatically increase the number of calculations required, even if the behavior in that particular pipe is not of interest. Using an infinite pipe is a way of avoiding significant computations for the pipe but still maintaining its effect on the system.

Note: The connected pipe must have at least two sections.

The theory for an infinite pipe is given in Infinite Pipe Transient Theory. (Wylie et al., 1993, pp. 121-122)

Example

Figure 1: Example application of infinite pipe

Gasoline flows from a refinery to a tank farm 30 miles away. We want to know what the effect of closing the storage tank valve will be on the tank farm equipment.

The wavespeed of gasoline in these pipes is about 3500 ft/s - meaning that the communication time in P1 is about 90 seconds. If the transient simulation is less than this time, there is no need to model the entire pipe. Furthermore, P101 will require at least one section as the controlling pipe - this means that P1 will require almost **800 sections**.

Alternatively, we can model P1 as only 400 feet in length (infinite pipes must be long enough to contain 2 sections) and apply the infinite pipe feature. **This reduces the total number of sections in the model to about 5**. For the same time step, this would mean that the number of calculations required is reduced by a factor of more than 100.

The results from the second run will be identical as long as the simulation time is less than the communication time of P1.

Related Topics

Initiation of Transients

In general, transients can be initiated according to a certain time or according to certain events that happen in the system. These two categories are referred to as time based and event based transients. This is defined in the *Initiation of Transient* area of a junction's transient tab.
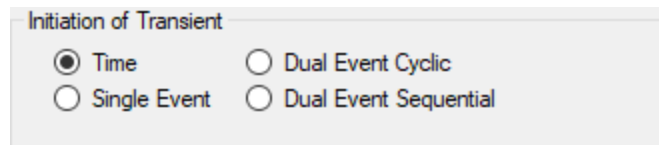
Figure 1: Type of transient initiation

It is helpful to consider the different time scales that are used when referring to the initiation of a transient, as shown in Figure 2.

Absolute and Simulation Time are often identical. The simulation Start Time is set in Simulation Mode/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.
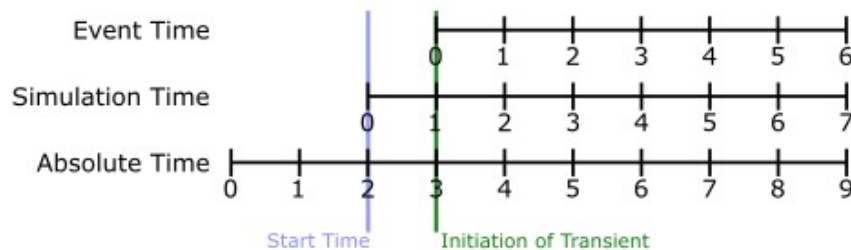
Figure 2: Time bases

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 P7 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 **	Mass Flow Rate in Pipe	Reservoir Surface Elevation (XTS)
Check Valve State	Pressure Stag. Diff. at Junction	Reservoir Surface Pressure (XTS)
Control Valve State	Pressure Stag. Diff. Between Pipes	Spray Discharge Outflow
Cv	Pressure Stagnation at Pipe	Surge Tank Gradeline (Impulse)
EGL at Pipe	Pressure Static at Pipe	Surge Tank Liquid Height (Impulse)
EGL Diff. at Junction	Pressure Static Diff. at Junction	Time Absolute
EGL Diff. Between Pipes	Pressure Static Diff. Between Pipes	Time Difference
HGL at Pipe	Pump Speed	Valve Open Percent (XTS)
HGL Diff. at Junction	Relief Valve State	Velocity in Pipe
HGL Diff. Between Pipes	Reservoir Liquid Height (XTS)	Volumetric Flow Rate in Pipe
Kv	Reservoir Liquid Volume (XTS)	

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. One common situation where this is useful is tripping and restarting a pump in AFT Impulse. Pump tripping is often a one-time event rather than a function of normal operation, so it may not be desirable to define this as a cyclic event.

Junctions With Inherent Event Logic

There are three junction types in AFT Impulse which have built-in, or inherent, event logic. These are the check valve, relief valve and air 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. If the user does not enter a closing or opening transient in the check valve window, 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 cracking pressure is reached. The second is that it closes again when the pressure falls back below the cracking pressure. If the user does not enter an opening or closing transient in the relief valve window, these transients are assumed to be instantaneous.

- **Air valve -** The air valve is very similar to the relief valve, and 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 rises above the set pressure and all gas is expelled.

Related Topics

Related Blogs

And, Not, Or - Starting Transient Events with Complex Logic

Multi-Condition Events

An available Event Type 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.
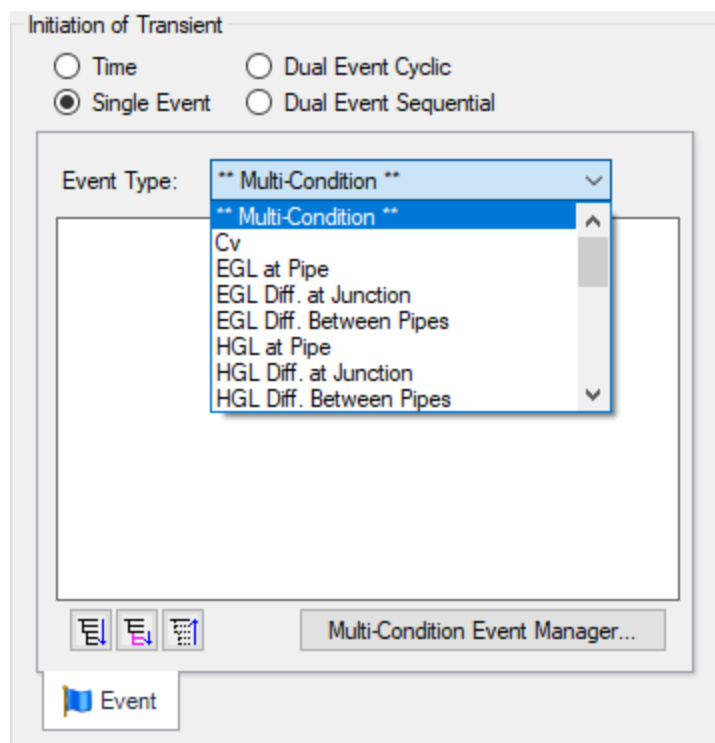
Figure 1: Selecting Multi-Condition Event Type

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.
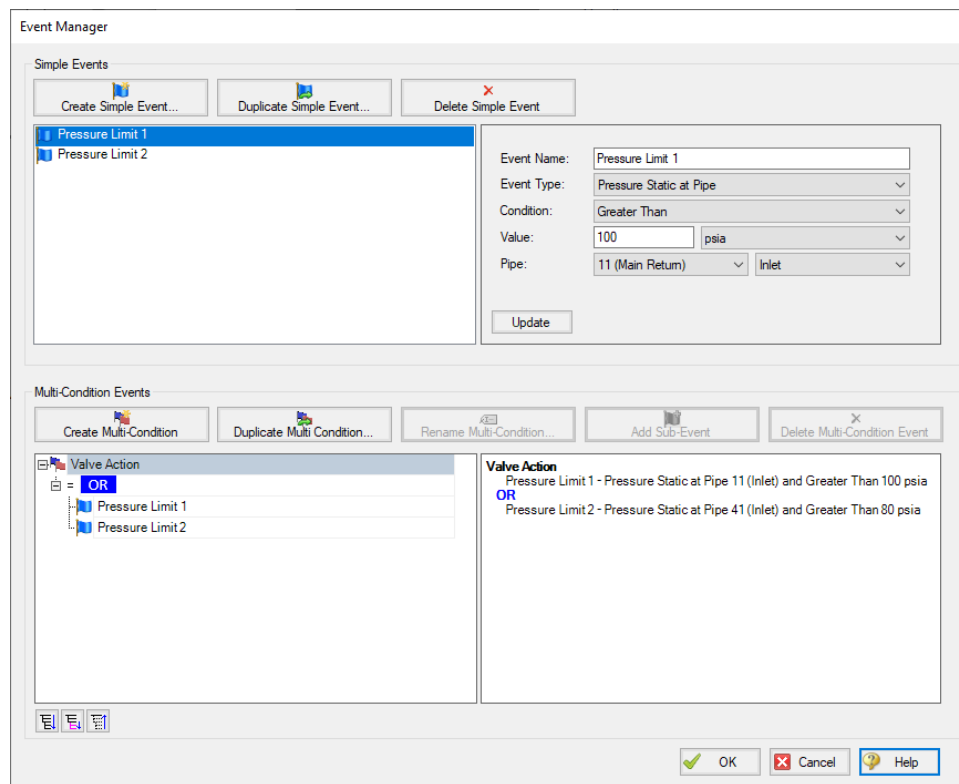
Figure 2: The Event Manager

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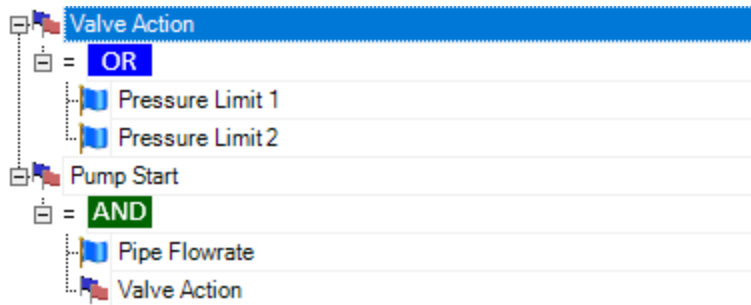
Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))

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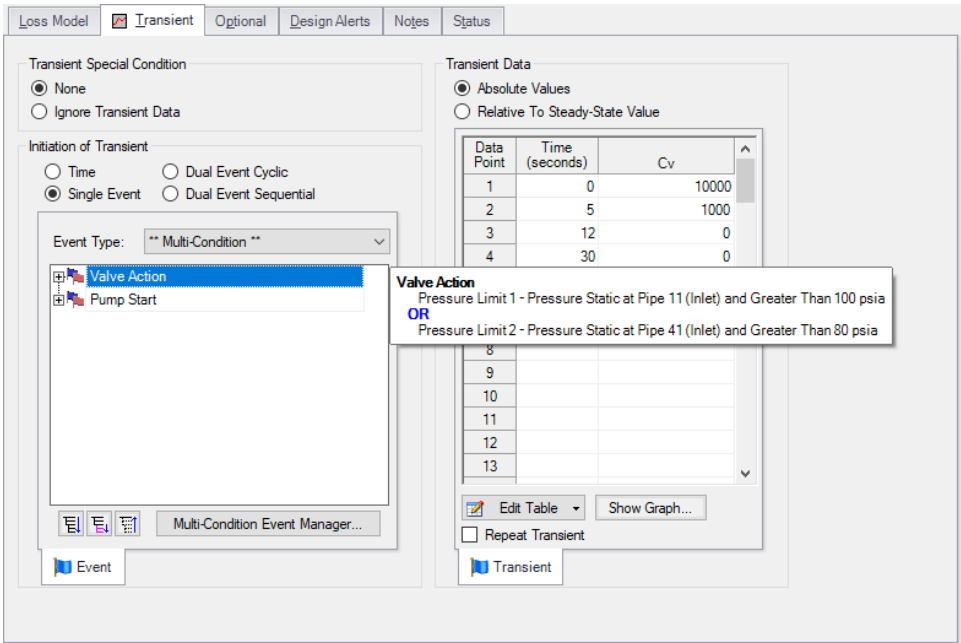
Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event.

Related Topics

Related Blogs

And, Not, Or - Starting Transient Events with Complex Logic

Air Valve

The Air Valve junction type (also called a vacuum breaker valve) requires either one or two connecting pipes, which can be specified on the bottom of the Valve Model tab.

The Air Valve 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 Air Valve offers protection for low pressure conditions. When the internal pressure drops below the cracking pressure (usually atmospheric), it opens to admit air (or whatever gas you specify). This keeps the pressure from dropping much below the atmospheric pressure. When the internal pressure rises above the atmospheric pressure, the air is discharged. To prevent large pressure spikes, the discharge flowrate is frequently kept lower than the incoming flowrate by using a smaller discharge flow area for outflow than inflow.

An air valve junction can connect with one pipe or two by changing this setting at the bottom of the Valve Model tab. When connected to one pipe, the valve is located at the end of the pipe. When connected to two pipes, the valve is inline and allows air to flow in and out at that location. During steady-state an air valve with one connecting pipe acts as a [dead end](#). If connected to two pipes, it acts as a lossless valve allowing liquid to flow between the two pipes.

Assumptions

- Gas volume entering the system stays at the junction which is a one dimensional point in space, and does not propagate down the length of the pipes.
- The Air Valve opens and closes instantaneously when the cracking conditions are met.
- AFT Impulse does not model two-phase flow; therefore, as the gas volume increases, so does the uncertainty in the simulation results.

Valve Type

There are four valve types available. The required information for Full Open Orifice CdA and the intermediate orifice will vary based on the selected Valve Type as described below.

- **Vacuum Breaker and Gas Release** - Default option. This valve type allows gas inflow when the pressure in the pipe drops below the cracking pressure, then allows gas outflow when the pressure in the pipe rises above the cracking pressure.
- **Vacuum Breaker** - This valve type allows gas inflow when the pressure in the pipe drops below the cracking pressure, but does not allow for gas outflow.
- **Gas Release** - This valve type allows gas outflow, but does not allow any gas inflow. This is mainly intended to be used to model a scenario where there is gas inside the pipe at the beginning of the simulation, which is allowed to exit the pipe during the transient. Note that Impulse cannot model gas pockets travelling through the pipes, so this case assumes the gas is directly at the air valve.
- **Vacuum Breaker and 2-Stage Gas Release** - This valve type works the same as the vacuum breaker valve type when gas is flowing in, but has two orifice sizes when gas

flows out. Pressure difference or volumetric flowrate criteria specify which orifice to use. When the actual value is less than the specified value, the normal Outflow CdA is used. When the actual value is higher, the intermediate orifice CdA is used. The intermediate orifice CdA values are typically much smaller than the normal outflow CdA.

Gas Properties

By default, Impulse chooses air as the default gas entering/exiting the Air Valve junction, and provides standard information for air including molecular weight and Cp/Cv for isentropic flow. These values can be adjusted by choosing Other for the gas type.

The gas external pressure, external temperature, and initial gas mass are set to typical values for an air valve. The Initial Gas Mass should generally be zero, unless the air valve will be open in the steady state.

Inlet Cracking Conditions

By default the inlet cracking conditions will be set equal to the external gas pressure/temperature defined in the Gas Properties section. The cracking conditions can be user specified if that option is enabled in the Advanced Parameters panel.

A cracking delay can also be specified to account for physical limitations of the valve, or distance from the line if the valve is modeled inline.

Liquid Temperature

Since the temperature of the liquid in the air valve is used for air valve calculations, the option is given to either use the temperature that was entered in the Fluid panel in the Analysis Setup window, or to explicitly define the temperature of the liquid at the valve.

Special Conditions

The Air Valve offers the special condition to Ignore calculations for the vacuum valve, meaning that it will be treated like a lossless valve. For more information, see Special Conditions.

Graphing Air Valve Data

By default, the air volume and mass flowrate in the junction will be saved to the output file, and the parameters can be graphed or reviewed in the Output window. The options for saving output are located in the Output Junctions panel of the Pipe Sectioning and Output group.

Related Topics

Related Blogs

Raise the bar using Vacuum Breaker Valves

Air Valve Waterhammer Theory

An air valve (sometimes called a vacuum breaker valve) allows air to flow into the pipe during low pressure events, and expels the air when the pressure rises again. Typically the cracking pressure is atmospheric. The theory is summarized in Wylie, et al., 1993, pp. 130-131, and detailed here.

There are four cases that can occur – two for inflow and two for outflow. The inflow can be subsonic or sonic, as can the outflow. These make up the four cases. The flowrate equations are given as follows:

- **Sonic Inflow -**

$$\dot{m} = \frac{C_d \, A_{in} \, P_o}{\sqrt{RT_o}} \sqrt{\gamma} \left(1 + \frac{\gamma - 1}{2}\right)^{(\gamma+1)/2(1-\gamma)}$$

- **Subsonic Inflow -**

$$\dot{m} = \frac{C_d \, A_{in} \, P_o}{\sqrt{RT_o}} \sqrt{\frac{2\,\gamma}{\gamma - 1}\left(\left(\frac{P_i}{P_o}\right)^{2/\gamma} - \left(\frac{P_i}{P_o}\right)^{(\gamma+1)/\gamma}\right)}$$

- **Sonic Outflow -**

$$\dot{m} = \frac{C_d \, A_{out} \, P_i}{\sqrt{RT_L}} \sqrt{\gamma} \left(1 + \frac{\gamma - 1}{2}\right)^{(\gamma+1)/2(1-\gamma)}$$

- **Subsonic Outflow -**

$$\dot{m} = \frac{C_d \, A_{out} \, P_i}{\sqrt{RT_L}} \sqrt{\frac{2\,\gamma}{\gamma - 1} \left(\left(\frac{P_i}{P_o} \right)^{-2/\gamma} - \left(\frac{P_i}{P_o} \right)^{-(\gamma+1)/\gamma} \right)}$$

where:

- P_i = pressure in the pipe at the junction node
- P_o = atmospheric pressure
- T_o = atmospheric temperature
- T_L = liquid temperature
- $C_d \, A_i$ = flow area for inflow (discharge coefficient times inflow orifice area)
- $C_d \, A_{out}$ = flow area for outflow (discharge coefficient times outflow orifice area)
- R = gas constant
- g = specific heat ratio cp/cv

Equation of State

Besides the compatibility equations 1 and 2 and , we need another equation to form a complete set. The ideal gas equation of state is used:

$$P\forall = m\,R\,T$$

Substituting terms one obtains

$$\frac{P_i}{RT_L} \left(\forall_{old} + \frac{\Delta t}{2} (Q_{down,new} + Q_{down,old} - Q_{up,new} - Q_{up,old}) \right)$$
$$= m_{old} + \frac{\Delta t}{2} (\dot{m}_{gas,new} + \dot{m}_{gas,old})$$

Eliminating the volumetric flowrate using the mass flowrate obtains

$$\frac{P_i}{RT_L} \left(\frac{2\,\forall_{old}}{\Delta t} + \frac{1}{\rho_L} (\dot{m}_{down,new} + \dot{m}_{down,old} - \dot{m}_{up,new} - \dot{m}_{up,old}) \right)$$
$$= \frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,new} + \dot{m}_{gas,old}$$

Using the Compatibility Equations to eliminate mass flowrate yields:

$$\frac{P_i}{RT_L}\left(\frac{2\,V_{old}}{\Delta t} + \frac{1}{\rho_L}\left(\frac{P_i - C_M}{B_M} + \dot{m}_{down,old} - \frac{C_P - P_i}{B_P} - \dot{m}_{up,old}\right)\right)$$
$$= \frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,new} + \dot{m}_{gas,old} \tag{1}$$

Now we can use the Flowrate equations for each of the four inflow and outflow cases to eliminate $\dot{m}_{gas,new}$ as follows:

Sonic Inflow

The pressure at the junction can be solved directly from Equation 1 as follows:

$$P_i^2\left(\frac{1}{\rho_L B_M} + \frac{1}{\rho_L B_P}\right) + P_i\left(-\frac{C_M}{\rho_L B_M} - \frac{C_P}{\rho_L B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} + \frac{2\,V_{old}}{\Delta t}\right)$$
$$- RT_L\left(\frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,old} + \dot{m}_{sonic}\right) = 0$$

where $\dot{m}_{sonic}$ is the sonic inflow which can be calculated from the Sonic Inflow Flowrate equation based completely on input data.

Subsonic Inflow

Equation 1 is modified using the Subsonic Inflow Flowrate equation and solved iteratively using Newton-Raphson. Defining $P_r = P_i/P_o$ obtains:

$$P_r^2\, P_o^2\left(\frac{1}{\rho_L B_M} + \frac{1}{\rho_L B_P}\right) + P_r\, P_o\left(-\frac{C_M}{\rho_L B_M} - \frac{C_P}{\rho_L B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} + \frac{2\,V_{old}}{\Delta t}\right)$$
$$- RT_L\left(\frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,old}\right) + C_d\, A_{in}\, P_o\, T_L\sqrt{\left(\frac{2\,\gamma\,R}{(\gamma-1)\,T_o}\right)\left(P_r^{2/\gamma} - P_r^{(\gamma+1)/\gamma}\right)} = 0$$

Defining terms for Newton-Raphson solution yields

$$C_1\left(P_r^a - P_r^b\right)^{0.5} + C_2\, P_r^2 + C_3\, P_r + C_4 = F$$

$$0.5\, C_1\left(P_r^a - P_r^b\right)^{-0.5}\left(a\, P_r^{a-1} - b\, P_r^{b-1}\right) + 2\, C_2\, P_r + C_3 = \frac{dF}{dP_r}$$

$$a = \frac{2}{\gamma}$$

$$b = \frac{\gamma + 1}{\gamma}$$

$$C_1 = C_d A_{in} P_o T_L \sqrt{\frac{2\,\gamma\,R}{(\gamma - 1)\,T_o}}$$

$$C_2 = P_o^2 \left(\frac{1}{\rho_L\,B_M} + \frac{1}{\rho_L\,B_P} \right)$$

$$C_3 = P_o \left(\frac{2\,V_{old}}{\Delta t} - \frac{C_M}{\rho_L\,B_M} - \frac{C_P}{\rho_L\,B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} \right)$$

$$C_4 = -R\,T_L \left(\frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,old} \right)$$

where the function F is solved for P_r.

Subsonic Outflow

Equation 1 is modified using the Subsonic Inflow Flowrate equation and solved iteratively using Newton-Raphson. Defining $P_r = P_i/P_o$ obtains:

$$P_r^2 P_o^2 \left(\frac{1}{\rho_L\,B_M} + \frac{1}{\rho_L\,B_P} \right) + P_r\,P_o \left(-\frac{C_M}{\rho_L\,B_M} - \frac{C_P}{\rho_L\,B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} + \frac{2\,V_{old}}{\Delta t} \right)$$
$$- R\,T_L \left(\frac{2\,m_{old}}{\Delta t} + \dot{m}_{gas,old} \right) + P_r\,(C_d A_{out}\,P_o) \sqrt{\left(\frac{2\,\gamma\,R\,T_L}{\gamma - 1} \right) \left(P_r^{-2/\gamma} - P_r^{-(\gamma+1)/\gamma} \right)} = 0$$

Defining terms for Newton-Raphson solution yields

$$C_1 \left(P_r^{-a} - P_r^{-b} \right)^{0.5} + C_2\, P_r + C_3 + C_4 \frac{1}{P_r} = F$$

$$0.5\, C_1\, P_r \left(P_r^{-a} - P_r^{-b} \right)^{-0.5} \left(-a\, P_r^{-a-1} + b\, P_r^{-b-1} \right) + C_2 - C_4 \frac{1}{P_r^2} = \frac{dF}{dP_r}$$

$$a = \frac{2}{\gamma}$$

$$b = \frac{\gamma + 1}{\gamma}$$

$$C_1 = C_d A_{out}\, P_o \sqrt{\frac{2\,\gamma\, R\, T_L}{\gamma - 1}}$$

$$C_2 = P_o^2 \left(\frac{1}{\rho_L\, B_M} + \frac{1}{\rho_L\, B_P} \right)$$

$$C_3 = P_o \left(\frac{2\, V_{old}}{\Delta t} - \frac{C_M}{\rho_L\, B_M} - \frac{C_P}{\rho_L\, B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} \right)$$

$$C_4 = -R\, T_L \left(\frac{2\, m_{old}}{\Delta t} + \dot{m}_{gas,old} \right)$$

where the function F is solved for P_r.

Sonic Outflow

Using the Sonic Outflow equation from above, the pressure at the junction can be solved directly from Equation 1 as follows:

$$P_i^2 \left(\frac{1}{\rho_L \, B_M} + \frac{1}{\rho_L \, B_P} \right) + P_i \left(-\frac{C_M}{\rho_L \, B_M} - \frac{C_P}{\rho_L \, B_P} + \frac{\dot{m}_{down,old}}{\rho_L} - \frac{\dot{m}_{up,old}}{\rho_L} + \frac{2 \, V_{old}}{\Delta t} \right)$$
$$+ P_i \left(C_d A_{out} \sqrt{\gamma \, R T_L} \left(1 + \frac{\gamma - 1}{2} \right)^{(\gamma+1)/2(1-\gamma)} \right) - R T_L \left(\frac{2 \, m_{old}}{\Delta t} + \dot{m}_{gas,old} \right) = 0$$

Related Topics

Related Blogs

Raise the bar using Vacuum Breaker Valves

Area Change

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

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

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

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

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

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

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

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

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 steady-state and/or transient flowrate 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 for steady flow, although they are for transient flow. If, for example, you have an outflow type junction and your system is physically flowing in, you cannot assign a negative flow rate to the junction. You must reverse the connecting pipe flow direction to be consistent with the actual direction and change from an outflow type to an inflow type.

Transient Data

On the transient data tab a flowrate transient can be entered. A special feature is the ability to model a sinusoidal flowrate transient. Input for the amplitude and frequency is required. If data is present in the transient data table, the sinusoidal transient is summed with the data in the transient data table.

A chopped sine wave takes the absolute value of the sine function. The end effect looks like doubling the frequency in the positive domain only, and having nothing in the negative domain.

A transient special condition also exist for assigned flow junctions. This is the No Reflections (Infinite Pipe) special condition.

For more information on transient data, including event transients, see Junction Transient Data.

Special Conditions

You can set a [Special Condition](#) for an Assigned Flow junction, which will turn the flow off and make it act like a Dead End during steady-state.

Special Features for Steady Flow

If the steady-state flow is zero at the junction, there may be occasions where you want the Assigned Flow junction to act like a pressure junction during the steady-state calculation. This would occur, for instance, if a known flowrate profile, initially zero, exists at the junction, and there are no other suitable pressure junctions in the system. In such a case, the pressure is assigned on the Optional tab.

Usually this feature would be used in conjunction with a Special Condition on the junction where its flow was turned off during steady-state.

Related Topics

SSL Module

Settling Slurry Module

An optional add-on module to AFT Impulse which allows users to simulate settling slurries.

Assigned Flow Waterhammer Theory

With an assigned flow, the flow rate is known at the pipe inlet or outlet ([Wylie et.al 1993](#), pp. 43). The unknown is then the pressure, which can be obtained by solving the appropriate compatibility equation. If the flow rate is known at the upstream end of the pipe, the [negative compatibility equation is used](#).

$$P_{i,new} = C_M + B_M \, \dot{m}_{known}$$

Conversely, if the pressure is known at the downstream end, the [positive compatibility equation is used](#).

$$P_{i,new} = C_P - B_P \dot{m}_{known}$$

If the known flow rate changes with time, then a current flow rate is obtained for the current time step and used in the equations.

Assigned Flow Vapor Cavitation Theory

When the calculated pressure at an Assigned Flow junction drops below vapor pressure, a vapor cavity forms at the junction.

Assuming the specified flow is an inflow, the vapor volume calculation is as follows:

$$V_{vapor,new} = V_{vapor,old} + (-\dot{m}_{up,new} - \dot{m}_{up,old} + \dot{m}_{down,new} + \dot{m}_{down,old}) \frac{\Delta t}{2 \, \rho}$$

The $\dot{m}_{up}$ terms are known because they are specified terms for inflow junctions. For outflow junctions the $\dot{m}_{down}$ terms are known. Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Assigned Pressure

The Assigned Pressure junction allows for the specification of a fixed boundary pressure. The Assigned Pressure window follows the second of the two basic Properties window formats. A table on the Loss Coefficients tab displays the connecting pipe information.

This junction is very similar to the Infinite 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.

Transient Data

On the transient data tab a flowrate transient can be entered. A special feature is the ability to model a sinusoidal flowrate transient. Input for the amplitude and frequency is required. If data is present in the transient data table, the sinusoidal transient is summed with the data in the transient data table.

A chopped sine wave takes the absolute value of the sine function. The end effect looks like doubling the frequency in the positive domain only, and having nothing in the negative domain.

A transient special condition also exist for assigned pressure junctions. This is the No Reflections (Infinite Pipe) special condition.

For more information on transient data, including event transients, see Junction Transient Data.

Related Topics

Related Online Content

- Static vs. Stagnation Pressure Video

Assigned Pressure Waterhammer Theory

With an assigned pressure, the pressure is known at the pipe endpoint. The unknown is then the flow rate, which can be obtained by solving the appropriate compatibility equation.

If the pressure is known at the upstream end of the pipe, the negative compatibility equation is used (Equation 2).

$$\dot{m}_{i,new} = \frac{P_{known} - C_M}{B_M}$$

Conversely, if the pressure is known at the downstream end, the positive compatibility equation is used (Equation 1).

$$\dot{m}_{i,new} = \frac{C_P - P_{known}}{B_P}$$

If the known pressure changes with time, then a current pressure is obtained for the current time step and used in the equations.

Assigned Pressure Vapor Cavitation Theory

Assigned Pressure junctions cannot reach vapor pressure because the pressure is specified and is therefore always above vapor pressure.

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.

The Bend Properties Window offers three standard types of bend losses: the Smooth Bend, the Standard Elbow (the standard short radius threaded elbow), and the Mitre Bend. The window shows a schematic of the selected geometry. Until you have input the information for the upstream pipe that allows flow area determination, the standard Bend junction loss factor is not calculated. For smooth radius bends, you also need to specify *r/D*, which is the radius of curvature over the pipe diameter. At *r/D* less than one the Reynolds number must be known to calculate the loss factor, and therefore cannot be displayed until output is available.

For bends that are not at a 90-degree angle, you can enter the angle and the standard loss factor will be calculated.

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

> **Note:** In principle, the upstream and downstream pipe diameters should be the same. However, the Bend junction only uses the reference area specified under Basis Area for Loss Model, even if the physical flow is in the opposite direction. If you are working with a user-specified loss, your geometry may have different diameter pipes connecting to the elbow. In this case, you would purposely define different upstream and downstream pipe areas. However, you would still want to be careful that the custom loss factor you assign is referenced to the correct area that is specified in Basis Area for Loss Model.

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* ≥ 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	$50\,f_T$
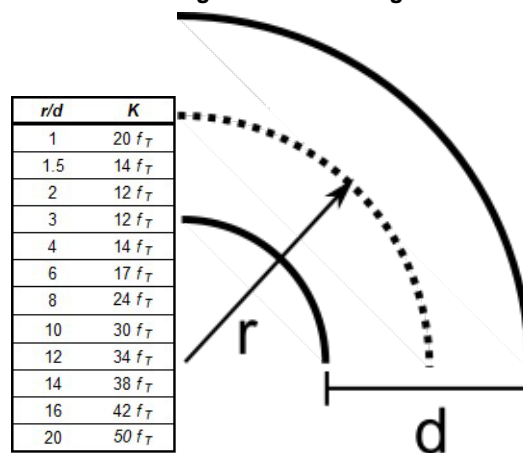
The **standard threaded elbow** is given by

$$K = 30\ f_T$$

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	$60 f_T$
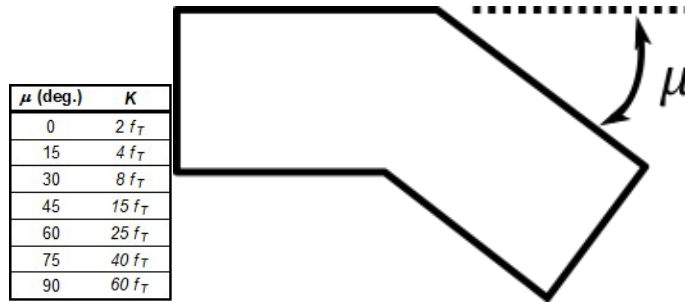
Smooth Flanged Bends, *r/D* < 1

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.

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

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 steady-state and/or transient 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 tab displays the connecting pipe information. This table grows in size to accommodate up to twenty-five connecting pipes. For each connecting pipe, the pipe table lists the reference flow direction and up to two loss factors. The first loss factor is for physical flow out of the branch and into the pipe. The second is for physical flow into the branch and out of the pipe. The loss factors can be specified independently or left as zero. The distance of each connecting pipe (if any difference exists) from the 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 Impulse'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.

Transient Data

On the transient data 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

Branch Waterhammer Theory

At a branch, there can be multiple pipes (Wylie et.al 1993, pp. 51-53). An additional relationship is needed, and that relationship is the conservation of mass. The total mass flow in and out of the junction must sum to zero. In addition, the branch has a single pressure solution, P_j, and this solution is common to all in-flowing pipes. The compatibility equation is written for each junction. For the pipes flowing into the branch, the positive equation is used (Compatibility Equations, Equation 1)

$$\dot{m}_{i,new} = \frac{C_P - P_j}{B_P}$$

where P_j is the branch junction pressure, as yet unknown.

For the pipes flowing out of the branch, the negative equation is used (Compatibility Equations, Equation 2)

$$\dot{m}_{i,new} = \frac{P_j - C_M}{B_M}$$

Now, sum all of the pipe flow rates into the junction

$$\sum \dot{m}_{i,new} = S_C - S_B \, P_J = 0 \qquad (1)$$

where

$$S_C = \sum \frac{C_P}{B_P} + \sum \frac{C_M}{B_M} \qquad (2)$$

$$S_B = \sum \frac{1}{B_P} + \sum \frac{1}{B_M} \qquad (3)$$

In general, there may be a known flow into the branch (a flow sink) or into the branch (a flow source). Therefore rewrite Equation 1 as

$$\sum \dot{m}_{i,new} = S_C - S_B \, P_j = -\dot{m}_{applied}$$

Now solve for the junction pressure

$$P_j = \frac{S_C + \dot{m}_{Applied}}{S_B} \qquad (4)$$

With the pressure known, substitute back into the compatibility equations to obtain the flow rate into the branch coming from each connecting pipe.

The applied flow rate may vary with time.

Branch Vapor Cavitation Theory

When the calculated pressure at a Branch junction drops below vapor pressure, a vapor cavity forms at the junction.

Assuming a flow source/sink is specified, the vapor volume calculation is as follows:

$$V_{vapor,new} = V_{vapor,old} + \left(\begin{array}{c} \sum \left(-\dot{m}_{in,new} - \dot{m}_{in,old} \right) \\ + \sum \left(\dot{m}_{out,new} + \dot{m}_{out,old} \right) \\ - \left(\dot{m}_{B,new} + \dot{m}_{B,old} \right) \end{array} \right) \frac{\Delta t}{2 \, \rho}$$

where $\dot{m}_B$ is the branch source/sink term defined as positive flow into the branch. The $\dot{m}_{in}$ terms are obtained from the positive compatibility equation, while the $\dot{m}_{out}$ terms are obtained from the negative equation.

Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Check Valve

The Check Valve junction type requires two connecting pipes. This junction type allows you to model the effect of a check valve which closes to prevent back flow.

The Check Valve Properties window follows the first of the two basic Properties window formats, displaying the connecting pipes in a fixed format. The Check Valve junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes for calculating loss. The user can define the base area for the loss model, though the default is set to use the upstream flow area.

A check valve is a device that allows flow in only one direction. AFT Impulse assumes that the Check Valve is initially open (unless modeled as Closed on the Optional tab). If the flow solution indicates that the system hydraulics are insufficient to keep the check valve open based upon the check valve input, AFT Impulse closes the valve. The valve can close during the steady-state or transient portion of the simulation.

AFT Impulse allows you to model the check valve by entering a User Specified closing velocity, using valve performance data to Estimate from Fluid Deceleration, or with a Force Balance calculation. Further details about each of these types of check valve are discussed below.

For all of these options the Loss Model for the open valve can be specified as a Cv, Kv or K factor by choosing an option on the Valve Model tab and entering the Full Open value.

User Specified

The User Specified option opens and closes based on user-defined values for velocity to close and a delta pressure to reopen. For steady-state analysis the valve is either fully open, or fully

closed. In a transient analysis, the valve opens and closes instantaneously.

The inputs are as follow:

- **Forward Velocity to Close Valve** - By default this value is zero. Some valves are designed to start closing at some reverse velocity - you can specify that here as a negative value. Entering a positive value in this field would imply that the check valve closes while the flow is still continuing forward.
- **Delta Pressure/Head to Reopen** - Frequently after a check valve closes, there is pressure across the valve at which it will open again. You can specify that delta pressure here.

Estimate from Fluid Deceleration (no reopen)

This option attempts to predict the maximum reverse velocity at which the valve will close. This maximum velocity is based on experimental predictions involving fluid deceleration and various check valve types from (pages 241-243). Once the fluid deceleration is determined, the chart for Maximum Reverse Velocity vs. Fluid Deceleration is used to determine the Forward Velocity to Close for the check valve. If the deceleration is above the range given in the chart for the specific valve type selected, the transient run will be terminated, and a different valve type must be selected. Once the check valve closes, it will not reopen.

There are three options provided to choose the Valve Performance Data Type that will be used for the Maximum Reverse Velocity vs. Fluid Deceleration chart.

- **Non-dimensional -** The non-dimensional data sets are provided from and . If this option is chosen, the user must provide the Minimum Velocity Required to Fully Open Valve, which is then used to find the Dimensionless Maximum Reverse Velocity.
- **Dimensional -** This option uses the original dimensional data from and , eliminating the need to convert to dimensionless values. This option may not be appropriate if the valves being modelled are not similar to the valve sizes used to obtain the experimental data. The dimensional data from Thorley was obtained for 600 mm swing check valves and 500 mm nozzle check valves. The Ballun data uses 8 in valves.
- **User Defined -** If dimensional data is available for the Maximum Reverse Velocity vs. Fluid Deceleration from the manufacturer, this option allows the user to directly define the curve to be used.

The **Deceleration Calculation Interval** is an optional input which allows the user to specify the time interval used to calculate the fluid deceleration. See the Deceleration Calculation Interval section below for further explanation. The resultant deceleration values can be viewed in the Output window.

Force Balance

Force Balance data can be used to model two valve types, Swing Check and Translating Nozzle/Plug valves. Both models are based on work published in .
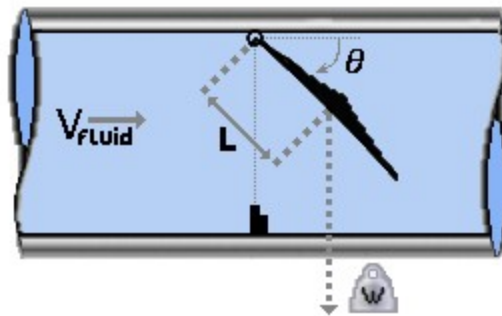
Figure 1: Swing Check valve diagram

The Swing Check Valve model uses a torque balance to estimate rate and position over time for a swing check valve. The inputs for the swing check valve are as follows (reference Figure 1):

- **Disk Properties**
    - **Distance to CG (L)** – The distance from the pivot point to the disk assembly center of gravity.
    - **Area (A)** – The surface area of the disk.
    - **Weight (submerged, W)** – The submerged weight of the disk assembly. If you do not know this value for the check valve you are trying to model, refer to the Approximating Submerged Weight for the Swing Check Valve section.
    - **Inertia (I)** – The moment of inertia of the disk about the pivot point. If you do not know this value for the check valve you are trying to model, refer to the Approximating Disk Inertia for the Swing Check Valve section.
- **Mass of Displaced Fluid** - By default, a mass of fluid equal to the mass of a sphere of fluid the same diameter as the valve disk is added to the valve inertia term. The user can specify a custom fluid mass to be included by choosing User Specified, or choose to neglect the fluid mass entirely by choosing None.
- **Valve Loss vs. θ Data** - The fully open (minimum) angle and the fully closed (maximum) angle of the disk must be defined, as shown in the diagram. Additionally the Cv vs. θ data must be defined in the table. A warning will be given to the user if the data in the table is inconsistent with the specified fully open and fully closed angles, or with the specified fully open Cv.
- **Include Spring**
    - **Spring Constant (k)** – The torsional spring stiffness, if a spring is present. The value is zero if there is no spring.
    - **Spring Torque With Valve Closed (T)** – The residual torque exerted by the spring when valve is fully closed. The value is zero if there is no spring torque when the valve is closed.
- **Custom Orientation** - By default, the orientation will be assumed to be horizontal which is shown in parentheses when the box is unchecked. Enabling this box allows the user to either specify the valve orientation as vertical, or for a user specified angle as measured from horizontal.
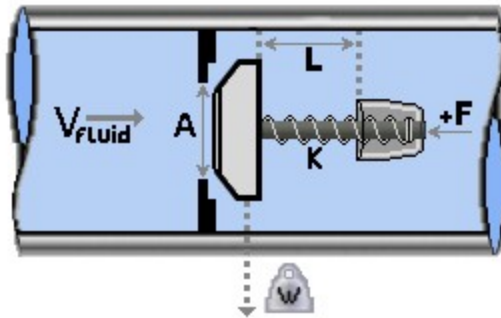
Figure 2: Translating nozzle/plug check valve diagram

The translating disk/plug check valve inertia model is solved using the force balance equation, the valve equation, and the positive and negative characteristic equations. The inputs are as follows (reference Figure 2):

- **Nozzle/Plug Properties**
    - **Max Travel Length (L)** – The distance the disk/plug travels from fully closed to fully open.
    - **Area (A)** – The cross-sectional surface area of the disk/plug perpendicular to the flow.
    - **Weight (W)** – The weight of the disk/plug assembly.
- **Mass of Displaced Fluid** - By default, the displaced fluid mass will be calculated using the distance travelled by the nozzle/plug. The user can specify a custom fluid mass to be included by choosing User Specified, or choose to neglect the fluid mass entirely by choosing None.
- **Valve Loss vs. Travel Length** – Variation of valve Cv in terms of Travel Length to Fully Open.
- **Spring Properties**
    - **Spring Constant (k)** – The linear spring stiffness, if a spring is present. The value is zero if there is no spring.
    - **Spring Force with Valve Closed (F)** – The residual force exerted by the spring when valve is fully closed. The value is zero if there is no spring force when the valve is closed. This is used to calculate Xs, the spring length at which the spring force is zero.
- **Custom Orientation** - By default the orientation will be assumed to be horizontal, which is shown in parentheses when the box is unchecked. Enabling this box allows the user to either specify the valve orientation as vertical, or for a user specified angle as measured from horizontal.

Deceleration Calculation Interval

This optional input is only applicable when using the *Estimate from Fluid Deceleration* check valve model. AFT Impulse needs to use the fluid deceleration to calculate the velocity at which

the check valve will close (and therefore the time step of closure). Due to certain transient behavior in a model, the deceleration may vary over time. For example, see Figure 3 below where the slope of the blue line changes over time. However, the Valve Performance Data (either the aforementioned Thorley/Ballun data or a user defined curve) will need a single deceleration value to calculate the fluid velocity at closure (Max Reverse Velocity). Therefore, Impulse will approximate the deceleration using the slope of a linear line from the time of initial deceleration (point 1) to the time when reverse velocity starts (point 2), as seen in the black dashed line. The default deceleration calculation might not accurately predict the velocity at which the check valve closes due to the slope of the blue and black lines differing at point 2.

The Deceleration Calculation Interval allows the user to manually specify a value for the time interval over which the slope (deceleration) is calculated. This effectively moves point 1 to a different time step. Impulse will then use the slope of the red dashed line as the value for deceleration. In this case shown in Figure 3, the default method likely provides a more conservative estimate because higher deceleration (steeper slope) would result in a larger Max Reverse Velocity, which then can result in a larger hydraulic force from check valve slam.

Note: This option is best used after you have already run the model, analyzed a Velocity vs Time graph, and determined that the deceleration (slope of the Velocity curve) differs significantly over time. In some cases using this setting may produce a less conservative estimate for a pressure/force spike (ex. pump trips), while other times it will produce a more conservative estimate (ex. valve closure elsewhere in the system where the effective closure time is short).
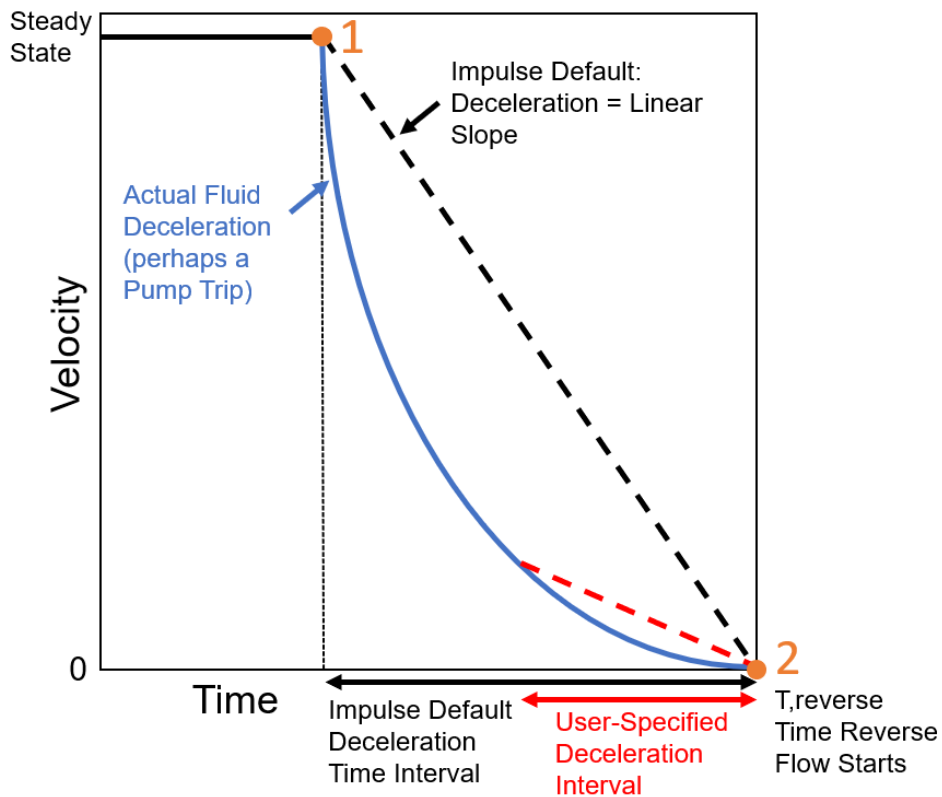
Figure 3: Example plot demonstrating how Impulse approximates the fluid deceleration for the Estimate From Fluid Deceleration Check Valve Model

Approximating Disk Inertia for the Swing Check Valve

The swing check valve model requires the user to input the disk inertia and the submerged disk weight of the check valve disk. If you do not know these values for the check valve you are trying to model, you can use the following equations to approximate these.

To approximate the disk inertia:

$$I_O = I_C + m\,d^2$$

Moment of Inertia of shape about point O

Moment of Inertia of Shape about Centroid (C)

Added Moment of Inertia due to distance between O and C
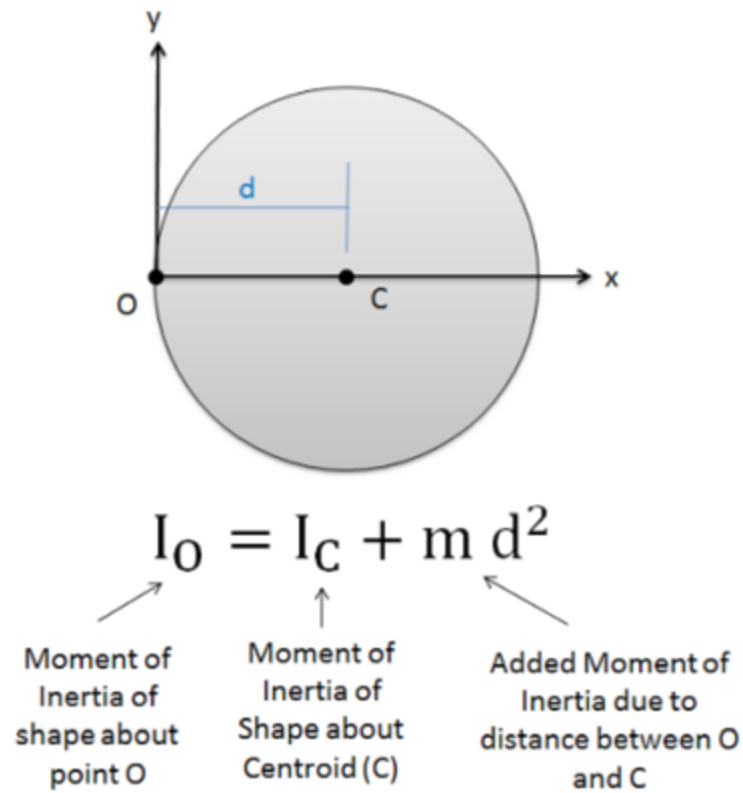
Figure 3: Approximate disk inertia

Where:

$$I_c = \frac{1}{4} * Mass\ of\ Disk * (Radius\ of\ Disc)^2$$

Approximating Submerged Weight for the Swing Check Valve

You can use Archimedes' Principle to approximate the submerged disk weight:

$$W_{submerged} = W_{disk} - \rho_{fluid}\,V_{fluid}\,g$$

Special Conditions

A check valve can have a Special Condition of None, Closed (stays closed or allowed to reopen in transient), or Open (stays open).

Related Topics

Check Valve Waterhammer Theory

Fluid Velocity at Valve Closure

Because a C_V is known in the models using either a user specified or estimated fluid velocity, the waterhammer theory for these is identical to a regular valve, except that the check valve closes when reverse flow occurs.

For the "Estimate from Fluid Deceleration" check valve, the fluid deceleration is calculated from the initial steady-state fluid velocity, the time at which the fluid begins to decelerate, and the time and velocity when the fluid first reverses through the valve by default. If a Deceleration Calculation Interval has been entered, this value will be subtracted from the time when reverse velocity begins to determine the time of initial deceleration (*t,initial deceleration = t,reverse - Deceleration Calculation Interval*). The velocity at this time will then be used as the steady state velocity.

For the Non-dimensional performance data, see the equations below.

$$Dimensionless\ Deceleration = \left(\frac{V_{steady} - V_{reverse}}{t_{initial\ deceleration} - t_{reverse}} \right) \frac{D_{pipe}}{V_{steady}^{2}}$$

Where:

- V_{steady} = Steady state fluid velocity

- $V_{reverse}$ = Fluid Velocity upon reversal

- $t_{initial\ deceleration}$ = Time at initiation of fluid deceleration

- $t_{reverse}$ = Time at fluid velocity reversal

- D_{pipe} = Diameter of upstream pipe
- V_0 = Minimum fluid velocity to fully open the valve

$$Dimensionless\ Maximum\ Reverse\ Velocity = \left(\frac{V_{max,reverse}}{V_0}\right)$$

Where:

- $V_{max,reverse}$ = Maximum reverse velocity at which valve will close

The maximum velocity at which the valve will close is determined from the specified Maximum Reverse Velocity vs. Fluid Deceleration chart using the calculated fluid deceleration. As an example, see the chart in Figure 1.
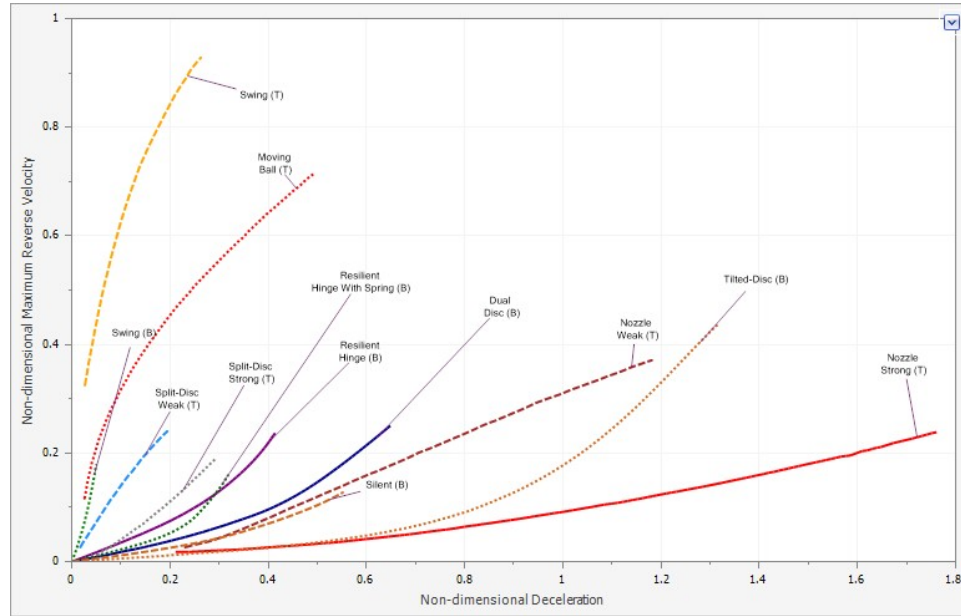
Figure 1: Non-Dimensional Fluid Deceleration vs. Maximum Reverse Velocity Chart with data from Thorley 2004(T) and Ballun 2007(B).

Force Balance (Swing Check Valve)

The swing check valve force balance model is solved using the torque balance equation, the valve equation, and the positive and negative characteristic equations.

From Wylie equation 6-42, neglecting friction torque and any torques exerted by external ballast or springs (torsion springs on the disk pivot are included):

$$T_w + T_e + T_p = I\frac{d^2\theta}{dt^2}$$

Where:

T_w = Torque due to the submerged weight of the disk

T_e = Torque due to spring torque applied at the disk pivot point

T_p = Torque due to hydrodynamic pressures on the disk

I = Moment of Inertia of the disk about the pivot point

$\dfrac{d^2\theta}{dt^2}$ = Angular acceleration of the disk about the pivot point

Wylie equation 6-43 for T_w :

$$T_w = W\,L\,\cos(\theta - \alpha)$$

Where:

W = submerged weight of the disk

L = distance from the pivot point to the disk center of gravity

α = Valve orientation angle (for inclined or vertical pipes)

From Wylie equation 6-44 for T_e :

$$T_e = S\left(\theta_s - \theta\right)$$

Where:

S = torsional spring constant

θ = angle of zero spring torque

θ_s = current disk angle

From Wylie equation 6-46 for T_p:

$$T_p = 144\,\Delta P\,L\,A$$

Where:

ΔP = pressure difference across valve, in psid

L = distance from the pivot point to the disk center of gravity

A = surface area of the disk

The valve equation (Equation 1) can be solved for ΔP as shown in Equation 2, which is substituted in Wylie 6-46:

$$\dot{m} = C_v\,\sqrt{\Delta P\,62.375\,\rho} \qquad (1)$$

$$\Delta P = \frac{\dot{m}^2}{C_v{}^2 \; 62.375 \; \rho} \tag{2}$$

Wylie equation 6-42 combined with the above equations becomes:

$$W \, L \cos\left(\theta - \alpha\right) + S\left(\theta_s - \theta\right) - \frac{144 \, \dot{m}^2 \, L \, A}{C_v{}^2 \; 62.375 \; \rho} - I \frac{d^2\theta}{dt^2} = 0$$

Along with the valve equation and the compatibility equations (Equations 3 through 5):

$$P_{up} - P_{dn} = \frac{\dot{m}^2}{C_v{}^2 \; 62.375 \; \rho} \tag{3}$$

$$P_{up} = C_P - B_p \, \dot{m} \tag{4}$$

$$P_{dn} = C_m + B_m \, \dot{m} \tag{5}$$

These four equations can be solved for θ, $\dot{m}$, P_{up}, and P_{dn}.

Force Balance (Translating Nozzle/Plug Valve)

Similar to the swing check valve, the translating nozzle/plug check valve force balance model is solved using the force balance equation, the valve equation, and the positive and negative characteristic equations.

From Wylie equation 6-51, neglecting friction force, and any force exerted by external ballast or springs (linear springs on the disk/plug are included), the force balance on the disk/plug is given by:

$$F_w + F_e + F_p = M \frac{d^2 X}{dt^2}$$

Where:

F_w = Force due to the weight of the disk/plug

F_e = Force due to spring applied to the disk/plug

F_p = Force due to hydrodynamic pressures on the disk/plug

M = Mass of the disk/plug and the mass of fluid that is moved

$\dfrac{d^2 X}{dt^2}$ = linear acceleration of the disk/plug

Force due to the weight of the disk/plug is given by:

$$F_w = W \sin \alpha$$

Where:

W = Weight of the disk/plug

α = Valve orientation angle (for inclined or vertical pipes)

The force due to spring applied to the disk/plug is given by:

$$F_e = k \left(X_s - X \right)$$

k = Linear spring constant

X_s = Length of spring at zero force

X = Current disk/plug position

The force due to hydrodynamic pressures on the disk/plug is given by:

$$F_p = 144 \, \Delta P \, A$$

Where:

ΔP = pressure difference across valve, in psid

A = cross-sectional area of the disk/plug

The valve equation, can be solved for ΔP as shown in the swing check valve section and substituted into the hydrodynamic force equation above:

$$F_p = \frac{144 \, \dot{m}^2 \, \rho \, A}{C_v^{\,2} \, 62.375}$$

The mass of the disk/plug and the mass of the fluid that is moved is given by:

$$M = \frac{W}{g} + \rho \, A \left(X_{prev} - X \right)$$

By substituting in the force equations and mass equation given above, Wylie equation 6-51 becomes:

$$W \sin \alpha + k \left(X_s - X \right) - \frac{\dot{m}^2 A}{62.375 \, \rho \, C_v^2} - \left[\frac{W}{g} + \rho A \left(X_{prev} - X \right) \right] \frac{d^2 X}{dt^2} = 0$$

Along with the valve equation and the compatibility equations shown above in equations 3-5 in the swing check valve section, with Cv expressed as a function of X, these four equations can be solved for X, $\dot{m}$, P_{up}, and P_{dn}.

Check Valve Vapor Cavitation Theory

The check valve vapor cavitation model is identical to a regular valve for all three types of check valve models, except that it closes when reverse flow occurs.

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's), Pressure Sustaining Valves (PSV's), Flow Control Valves (FCV's), and Pressure Drop Control Valves (PDCV's). Loss information for a control valve is not required, because control valves are dynamic devices that change their geometry in response to the pipe system behavior. The loss that results is that required to maintain the control parameter. You can, however, specify the full open loss. This is the loss that will occur should the valve fail to a full open state.

- A **PRV** is a device that controls the pressure in a pipe system. The PRV maintains a constant control pressure downstream of the junction as long as the upstream pressure exceeds the control pressure. If the upstream pressure is lower than the control pressure, the ability to control pressure is lost.

- A **PSV** is similar to a PRV in that it controls pressure in a pipe system. While the PRV maintains a constant downstream pressure, the PSV maintains a constant upstream pressure. If the downstream pressure rises higher than the control pressure, the ability to control pressure is lost.

- An **FCV** is a device that maintains a constant flow rate in a pipe system. By setting the junction to an FCV type and entering a flow rate, the junction will limit the flow through the connecting pipes to be equal to the control flow rate. The FCV can lose its ability to control flow when the pressure drop across it becomes zero or backward flow begins.

- A **PDCV** is a device that maintains a constant (stagnation) pressure drop. For this option, the valve is never allowed to not meet its setpoint. An indicator that an unrealistic pressure drop has been demanded is a failure to obtain a converged solution.

PRV/PSV Static vs Stagnation Pressure

The control pressure for a PRV or PSV can be either static or stagnation (HGL or EGL for controlled Head). The default selection is static (HGL), as this is the most frequent application in industry.

Action if Setpoint Not Achievable

- **Always Control During Steady State** - It is possible that a control valve cannot achieve a given set point. By default this option is off - this can be changed in the Parameters section of User Options. For additional information, see Control Valve Setpoint Not Achievable.

- **Use Default Actions** - The behavior of the control valve when it fails can be modified by unchecking this option. The default behaviors represent the normal behaviors for the standard valve types.

Loss When Fully Open

By default, the valve will be set to model the valve with no loss if it fails open. This can be changed by choosing a loss model on the Control Valve Model tab to use C_V, K_V, or a K factor. When C_V or K_V, is selected, the option becomes available to pull the value from an open percentage table on the Optional tab, and the option to apply an Opening/Closing Rate limit for the valve becomes available.

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 Impulse's flow solution. However, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary (in the Output window) always displays the valve C_V, and if data is specified for open

percentage and flow area it will also display open percentage and flow area at the operating point.

AFT Impulse can assist in creating this C_V vs Open percent curve. This is accessible in the optional by by selecting "Edit Table" and then "Create C_V vs. Open Percent...". This then opens a window in which a user defined, linear, equal percentage, or Pre-defined curve can be defined and transferred to the C_V. vs. Open Percentage table.

Opening/Closing Rate Limits

If the default options for "Action if Setpoint is Not Achievable" are being used and the loss When Fully open is defined in terms of C_V or K_V, the maximum valve opening and closing rate limits can be specified. Rate limits allow the modeling of valves that have some type of limiting device which prevents the valves from opening or closing too quickly. If, during the transient, the pressures would cause the valve to open or close faster than the rates specified in the rate limits table, the valve's loss value will be set to the value dictated by the rate limits. If no rate limits are defined Impulse will assume that the valve will close or open instantly if it fails.

Transient Data

Control Valve transients provide two Transient Parameter options, Control Setpoint and C_V.

The **Control Setpoint transient** can be applied to adjust the control setting of the valve during the transient. For instance, a change in control pressure for a regulator valve can be modeled.

The **Cv transient** is intended to model a case where the control valve transitions to a static valve at a fixed C_V. For example, a control valve in a feedback loop for a pump may transition to a fixed open percentage when a certain pressure is reached at the pump discharge. Another case would be if a control valve were to be fully closed or fully opened in an emergency situation. Once a C_V vs. Time transient begins, the control valve will not be able to regain control.

If the valve C_V when controlling is within the defined C_V vs. Time data when the control valve transient begins, the control valve will begin at the corresponding point on the Transient graph and open/close from that point, rather than starting at the defined time zero. If the valve C_V when controlling falls above or below the defined table, the valve will jump to the closest point in the table, then proceed with the defined transient. For example, consider a defined closing transient which goes from a C_V of 200 to 100 as shown in Figure 1. If the valve is controlling at a C_V of 150, the valve will begin in the middle of the line at C_V = 150, then decrease to 100 (green line). If the valve C_V is 300 when the valve is controlling, then at the beginning of the transient the valve C_V will jump from 300 to 200, then follow the defined table until it reaches 100 (red line). If the valve is controlling at C_V = 50, then the valve will jump to a C_V of 100 and remain at 100 (purple line).
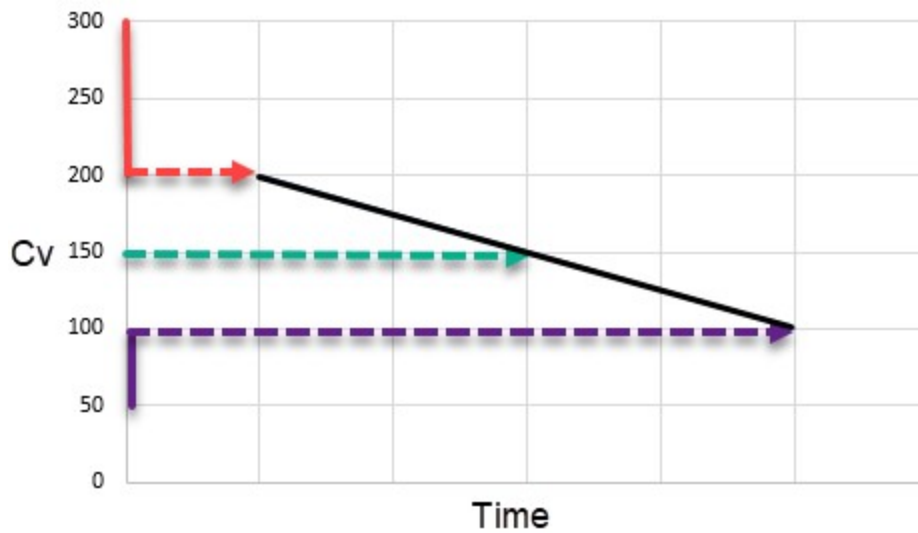
Figure 1: Example cases for Cv vs. Time defined for a closing valve

Control valve transients support single events. Thus the regulator pressure could be modified when some event occurred in the system.

For more information on transient data, including event transients, see Junction Transient Data.

Special Conditions

Control valves have two special conditions: to open fully and not control, or to close.

Related Topics

Related Blogs

Get in Control with Control Valves

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.

Always Control During Steady State

The Always Control During Steady State feature is turned off by default since this option is physically unrealistic, but it can be useful for troubleshooting 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 in the steady state. The default option for setting the control valve to Always Control can be adjusted in the Parameters section of User Options.

For transient analyses, if the Always Control is used during the steady state and the valve cannot meet the setpoint, the transient analysis will terminate.

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 in the steady state, whether open or closed, AFT Impulse will solve the model again, using the defined loss model if the valve is open.

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 Impulse 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.

It is possible for the valve to control properly during the steady-state, and then lose control during the transient part of the run. In fact, it can lose control and regain it repeatedly as long as a C_v vs. Time transient event is not triggered.

Related Topics

Control Valve Waterhammer Theory

We can use the methods for known flow or pressure to calculate the conditions at a control valve. AFT Impulse flow and pressure control valves allow the control setting to be changed with time. Whether constant or changing, the flow or pressure is known and can be used to solve the equations.

Flow Control Valves

With the known flowrate, one can substitute into Equation 1 and Equation 2 and solve for the upstream and downstream pressures, respectively, at the valve. These can then be compared to the Action if Setpoint not Achievable settings to see if loss of control occurs. For instance, if the upstream pressure was less than the downstream, that would indicate loss of control.

Once the valve loses control, it acts like a regular valve. However, it is possible for the control valve to later regain control if the upstream pressure raises sufficiently.

Pressure Control Valves

Pressure control can be specified on the upstream or downstream side of the valve. A discussion of one these control locations will apply to the other. Let's consider the downstream pressure control case – a Pressure Reducing Valve (PRV).

With the known pressure downstream, one can substitute the known pressure into Equation 2 and solve for the flowrate through the valve. This flowrate can then be substituted into Equation 1 to solve for the upstream pressure. The upstream pressure can then be compared to the control pressure and Action if Setpoint Not Achievable settings to see if loss of control occurs. For instance, if the upstream pressure was less than the control pressure, that would indicate loss of control.

Once the valve loses control, it acts like a regular valve. However, it is possible for the control valve to later regain control if the upstream pressure raises sufficiently.

Pressure Drop Control Valves

The flow through a constant pressure drop is obtained by subtracting Equation 1 and Equation 2 and solving for flowrate.

$$\dot{m}_{i,new} = \frac{C_P - C_M - \Delta P}{B_P + B_M}$$

Related Topics

Dead End

The Dead End junction must always have one connecting pipe. This junction type is one way to define flow in a pipe as truly zero.

In steady flow, dead ends have no flow and thus play no role in the hydraulic solution to the system. However, in transient flow pipes that terminate in dead ends can have flow. Thus dead ended lines can and do influence the transient response of the system. Unless you know otherwise for a specific system, dead ends should always be included in the model.

The Dead End Properties window follows the first of the two basic Properties window formats, displaying the connecting pipes in a fixed format. The only required information is the connecting pipe and the elevation.

Related Topics

Dead End Waterhammer Theory

A dead end is similar to an assigned flow, except the flow rate is zero (Wylie et.al 1993, pp. 44). The unknown is then the pressure, which can be solved by calling the appropriate compatibility equation.If the dead end is at the upstream end of the pipe, the negative compatibility equation is used with zero flow.

$$P_{i,new} = C_P$$

Conversely, if the dead end is at the downstream end, the positive compatibility equation is used with zero flow.

$$P_{i,new} = C_M$$

Dead End Vapor Cavitation Theory

When the calculated pressure at a Dead End junction drops below vapor pressure, a vapor cavity forms at the junction.

Assuming the dead end is at the end of a pipe, the vapor volume calculation is as follows:

$$V_{vapor,new} = V_{vapor,old} + (-\dot{m}_{up,new} - \dot{m}_{up,old}) \frac{\Delta t}{2 \; \rho}$$

since the $\dot{m}_{down}$ terms are always zero. For dead ends at the beginning of a pipe, the $\dot{m}_{up}$ terms are zero. Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Gas Accumulator

The Gas Accumulator junction allows up to twenty-five connecting pipes. This junction type is useful for modeling a gas volume designed as a surge protection device (Figure 1). The Gas Accumulator may be integral with the pipe system, separated by an orifice, or by a short connector pipe. During steady flow the gas accumulator typically acts a loss-less branch junction, although it can be made to act like a pressure junction. This is discussed later in the gas accumulator section.

The Gas Accumulator Properties window follows the second of the two basic Properties window formats. A table displays the connecting pipe information. The table grows in size to accommodate up to twenty-five connecting pipes.

Figure 1: Schematic of a gas accumulator

Tank Geometry

The tank geometry is an optional input which allows Impulse to account for variations in the liquid level within the gas accumulator.

Th vertical and horizontal tank types are programmed to allow Impulse to calculate the tank volume, height, or diameter based on two provided dimensions. For non-cylindrical tanks the other geometry option can be selected, which requires the user to directly input Accumulated Volume vs Liquid Height data.

Cap type selections are available to define the geometry of the tank cap/head as are defined in Crookston 2011 (see References for link).

Initial Gas Pressure

The Gas Accumulator provides two options for determining the steady state pressure at the accumulator.

- **Calculate From Steady State** - This option treats the gas accumulator junction like a lossless branch junction during the steady state. The pressure at the inlet of the gas

accumulator is determined based on the steady state pressure in the connected pipes.

- **Known** - This option fixes the gas pressure to the specified value. The gas accumulator junction then acts like an assigned pressure junction based on the defined gas pressure. If the tank geometry is defined the hydrostatic pressure from the liquid is also accounted for. The main purpose of this option is to model a pressurizer as is discussed further below in the Known Pressure for Steady Flow section.

Initial Gas Volume

The Gas Accumulator provides two options for determining the steady state gas volume in the accumulator.

- **Known Volume** - This option will fix the gas volume during the steady state to the specified value.
- **Calculate From Precharge Conditions** - This option will determine the steady state gas volume using the provided precharge conditions. The thermodynamic calculations for the steady state can be treated as isothermal or nonisothermal as described below:
    - **Isothermal** - Impulse assumes that the temperature is constant, so *PV* = constant. The steady state volume can be determined directly using the precharge volume and pressure, and the steady state pressure.
    - **Non-Isothermal** - Impulse assumes that the gas is an ideal gas in the steady state, so *PV/T* = constant. The precharge temperature must be provided in addition to the pressure and volume. The steady state volume can then be calculated using the precharge conditions, the steady state pressure, and the steady state temperature. The steady state gas temperature is assumed to be equal to the liquid temperature in the gas accumulator. The User specified liquid temperature option may be required if the fluid specified in the Fluid panel did not have a specified temperature.

Polytropic Constant in Transient

Regardless of the method used to calculate initial conditions the gas volume will change during the transient according to the thermodynamic law described in Gas Accumulator Waterhammer Theory. The law requires a polytropic constant to relate the pressure to the gas volume. An isentropic process should use a polytropic constant equal to the specific heat ratio of the gas (for air it is 1.4). An isothermal process should use a polytropic constant of 1. Typically the process falls in between the two and an average polytropic constant can be assumed (for air the average is 1.2).

The gas in the accumulator is assumed to behave as an ideal gas.

Maximum and Minimum Gas Volumes

If the gas is constrained in some way to a maximum or minimum volume, this can be entered in the optional fields. If a maximum or minimum exists, AFT Impulse will not allow the gas volume to exceed this value. If no data is entered, the volume can grow as large or small as necessary,

though the gas volume will still be constrained to be less than the tank volume if Tank Geometry information has been entered.

Interface with Pipe System

The Gas Accumulator can connect into the pipe system in three ways: it can be integral with the pipe, it can be separated by an orifice, or it can be connected by a short pipe.

These options are provided in the Flow Restrictor and Short Connector Pipe areas. If neither option is used, the accumulator is assumed to be attached directly to the pipe with no hydraulic loss.

- **Flow Restrictor** - The accumulator is assumed to be separated from the pipe system by an orifice, which causes a hydraulic loss as the liquid flows in and out of the accumulator. The loss is specified by entering a flow area and inflow/outflow discharge coefficient.
- **Short Connector Pipe** - This pipe is considered to be short in relation to the other pipes in the system after they are sectioned. If the pipe is short, it can be assumed to react instantaneously with the accumulator. The short pipe accounts for liquid inertia and friction. If a connector pipe exists which cannot be considered short, it should be modeled as an actual AFT Impulse pipe.

The Flow Restrictor and Connector Pipe can both exist in the same accumulator. That is, a flow restrictor can be modeled at the top of a connector pipe, and the effect of both will be included. The loss model for the Flow Restrictor and Short Connector Pipe options is discussed in the Lumped Inertia Pipe with Orifice topic.

Model Relief Valve

On the optional tab on the Gas Accumulator Properties window there is an additional option to model a relief valve on the gas accumulator. This option allows gas to be expelled from the accumulator to prevent pressures from exceeding the relief pressure specified. The relief valve on a gas accumulator is one way and gas will not flow into the accumulator. Once all of the gas has been expelled or the minimum volume is met the accumulator will essentially be a liquid full stagnant region and will not have dynamic system behavior. In other words, once all of the gas is expelled from the gas accumulator it will behave as a loss-less branch if the accumulator is inline or as a dead end if the accumulator is at the end of a pipe.

Special Condition

The special condition for the Gas Accumulator on the optional tab allows the user to set Impulse to ignore the accumulator. See Special Conditions for more information.

Known Pressure in Steady Flow

As mentioned previously, the gas accumulator acts like a branch junction during steady flow. However, the user can specify that it behaves like a pressure junction by using the Known Initial Pressure option. Why would one want to do this? The most obvious case is where the gas accumulator is modeling a pressurizer type of component in a closed system. A pressurizer is sometimes called an expansion tank or accumulator. The main purpose is to provide thermal expansion volume when the temperature of the process fluid changes. Sometimes these tanks are open to the atmosphere, in which case a Surge Tank would be a better choice than a gas accumulator. When a pressurizer is modeled as a gas accumulator, there will typically be no other pressure type junctions in the system. In fact, the pressurizer is the component which controls the static pressure on the system during steady-state operations. However, during transients the pressure will change. Thus there is a need to allow a gas accumulator to act as a pressure junction during the steady-state, and then like a normal gas accumulator during the transient, as can be done with the Known Initial Pressure option. When the accumulator is a pressurizer during steady-state, it has zero net flow. The pressure entered will be the steady-state static system pressure.

The gas accumulator junction can also be used with the Known Pressure in Steady Flow option to represent a closed tank with a known steady state pressure. In this case the steady state flow to the gas accumulator junction will be non-zero, and a caution will be given at run-time.

Graphing Gas Accumulator data

You can track the gas volume, liquid volumetric and mass flowrate, gas pressure, and liquid height in the accumulator by selecting these options in the Output Junctions panel of the Pipe Sectioning and Output group. These parameters can be graphed or reviewed in the Output window.

Related Topics

Gas Accumulator Waterhammer Theory

A gas accumulator (shown in Figure 1) is an enclosed tank filled with gas and has a free surface. It is used as a surge suppressor (Wylie, et al., 1993, pp. 125-128).The flow balance on the accumulator junction is similar to a branch junction, with the deficit equaling the flow out of the accumulator, $\dot{m}_A$.

Similar to a branch, it can be shown that for all connecting pipes to the junction,

$$P_{j,new} = \frac{S_C + \dot{m}_{A,new}}{S_B}$$

where S_C and S_B are given by Equations 2 and 3 from Branch Waterhammer Theory. The potential connector pipe and orifice effect is incorporated by the lumped inertia relationship as follows

$$P_{j,new} = P_{A,new} - C_1 - C_2\,\dot{m}_{A,new} + \rho g \Delta h_{liquid}$$

where C_1 and C_2 are defined in conjunction with lumped inertia. Refer to Equations 1 and 2 from the Lumped Inertia Pipe topic. The Δh_{liquid} is the height of liquid in the accumulator from the bottom of the accumulator, and is assumed to be zero if the tank geometry is not defined.
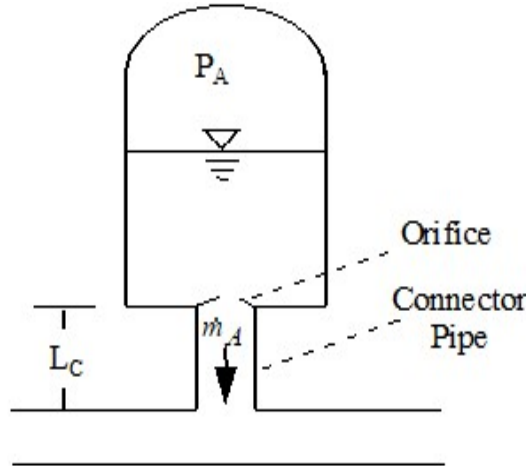
Figure 1: Gas accumulator schematic

The gas volume and pressure in the accumulator can be related by the thermodynamic law

$$P_A\,V_A^n = C_A$$

where n is the polytropic constant (1.4 for isentropic air, 1.0 for isothermal) and C_A is a constant. Therefore,

$$P_{A,new}\,V_{A,new}{}^n = C_A$$

and

$$V_{A,new} = V_{A,old} - \left(\dot{m}_{A,new} + \dot{m}_{A,old}\right) \frac{\Delta t}{2\,\rho}$$

By substitution

$$\frac{S_C + \dot{m}_{A,new}}{S_B} = P_{A,new} - C_1 - C_2\,\dot{m}_{A,new} + \rho g \Delta h_{liquid}$$

or

$$P_{A,new} = \frac{S_C + \dot{m}_{A,new}}{S_B} + C_1 + C_2\,\dot{m}_{A,new} - \rho g \Delta h_{liquid}$$

Combining equations and simplifying,

$$\left(C_3 + C_4\,\dot{m}_{A,new}\right)\left(C_5 + C_6\,\dot{m}_{A,new}\right)^n = C_A$$

where

$$C_3 = \frac{S_C}{S_B} + C_1 - \rho g \Delta h_{liquid}$$

$$C_4 = \frac{1}{S_B} + C_2$$

$$C_5 = V_{A,old} + \dot{m}_{A,old}\frac{\Delta t}{2\,\rho}$$

$$C_6 = \frac{\Delta t}{2\,\rho}$$

This is a non-linear equation that can be solved using Newton-Raphson iteration. Newton-Raphson has the following form:

$$x_{i+1} = x_i - \frac{F(x_i)}{F'(x_i)}$$

where x_i is the current value of x, x_{i+1} is the new value, F is the function of x to drive to zero, and F' is the derivative.

In this case

$$F = (C_3 + C_4\,\dot{m}_{A,new})(C_5 + C_6\,\dot{m}_{A,new})^n - C_A$$

When the value of $\dot{m}_{A,new}$ has been found that causes F to equal zero, the equation will be satisfied. To solve by Newton-Raphson the derivative of F is given by,

$$\frac{dF}{d\dot{m}_{A,new}} = F' = \frac{C_7}{C_5 + C_6\,\dot{m}_{A,new}} + C_4(C_5 + C_6\,\dot{m}_{A,new})^n$$

where

$$C_7 = -n\,C_A\,C_6$$

By iteration, $\dot{m}_{A,new}$ is calculated. This value can then be back-substituted to obtain the new junction pressure and new gas volume.

Gas Accumulator Vapor Cavitation Theory

Any short-lived vapor cavities that develop at a Gas Accumulator junction are assumed to have a negligible effect compared to the accumulator itself. Conceptually, vapor bubbles would rise by buoyancy to the accumulator surface and evaporate into the accumulator gas. Their effect on the system would thus be negligible compared to that of the accumulator gas. Therefore, vapor cavitation is ignored at gas accumulators.

Related Topics

General Component

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

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

When you select Variable K-Factor or 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 Impulse modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

No transient data can be entered for a General Component. From a transient point of view, the junction is a static element. For this reason the user should use General Components with caution. Where possible all static element hydraulic losses should be entered into pipes and not junctions.

Related Topics

Liquid Accumulator

The Liquid Accumulator junction allows up to twenty-five connecting pipes. This junction type is useful for modeling a liquid chamber that has more elasticity than the pipe itself. This may be a liquid-solid tank or a condenser box, for example.

The Liquid Accumulator Properties window follows the second of the two basic Properties window formats. A table displays the connecting pipe information. The table grows in size to accommodate up to twenty-five connecting pipes.

Initial Conditions and Elastic Process

The relationship between liquid accumulator volume and pressure is assumed to be linear. The amount of volume change for a given amount of pressure change is given by the elasticity of the accumulator. This input will need to be obtained from test data or approximated based on mechanical correlations.

The Initial Volume of the tank is a required input. This volume will change during the transient according to the linear law.

Special Condition

The special condition for the Liquid Accumulator on the Optional tab allows the user to set Impulse to ignore the accumulator. See Special Conditions for more information.

Related Topics

Liquid Accumulator Waterhammer Theory

A liquid accumulator, also called a lumped capacitance, models a liquid- filled container in a system (Wylie 1993, pp. 124-125). The interaction with the pipe system occurs as the container stretches and contracts in response to transient pressures. The liquid accumulator is similar to a branch, and thus the same equations are involved.

$$P_{j,new} = \frac{S_C + \dot{m}_{A,new}}{S_B}$$

where S_C and S_B are given by Equations 2 and 3 from Branch Waterhammer Theory. The relationship between accumulator pressure and volume is assumed to be linear,

$$K' = \frac{\Delta P}{\Delta V / V}$$

where K' is the effective bulk modulus of the container, and has units of pressure. The change in volume can be related to flow rate as follows:

$$\Delta V = - \left(\dot{m}_{A,new} + \dot{m}_{A,old} \right) \frac{\Delta t}{2 \, \rho}$$

Therefore,

$$P_{j,new} = P_{j,old} - \frac{K' \, \Delta t \, (\dot{m}_{A,new} + \dot{m}_{A,old})}{2 \, V \, \rho}$$

Substituting and eliminating flow rate

$$P_{j,new} = \frac{S_C + P_{j,old} \dfrac{2 \, V \, \rho}{K' \, \Delta t} - \dot{m}_{A_{old}}}{S_B + \dfrac{2 \, V \, \rho}{K' \, \Delta t}}$$

Liquid Accumulator Vapor Cavitation Theory

When the calculated pressure at a Liquid Accumulator junction drops below vapor pressure, a vapor cavity forms at the junction.

The vapor volume calculation is as follows:

$$V_{vapor,new} = V_{vapor,old} + \left(\begin{array}{c} \sum (-\dot{m}_{in,new} - \dot{m}_{in,old}) \\ + \sum (\dot{m}_{out,new} + \dot{m}_{out,old}) \\ -(\dot{m}_{A,new} + \dot{m}_{A,old}) \end{array} \right) \frac{\Delta t}{2 \, \rho}$$

where $\dot{m}_A$ is the accumulator flow out of the junction. The $\dot{m}_{in}$ terms are obtained from the positive compatibility equation, while the $\dot{m}_{out}$ terms are obtained from the negative equation.

Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

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 irrecoverable loss that occurs through a plate-type flow restriction. It also allows you to specify loss

factors as a function of a flow parameter and an orifice loss for pipes with an area change.

The Orifice Properties window follows the first of the two basic Properties Window formats, displaying the connecting pipes in a fixed format. The Orifice junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

Orifices that are located at 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/head.

Orifice Types

AFT Impulse 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 Impulse 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 Impulse modifies the loss factor in the Solver (the part of AFT Impulse 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.

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):

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}}\right)^2 \left[0.707\left(1 - \frac{A_{orifice}}{A_{up}}\right)^{0.375} + \left(1 - \frac{A_{orifice}}{A_{down}}\right)\right]^2$$
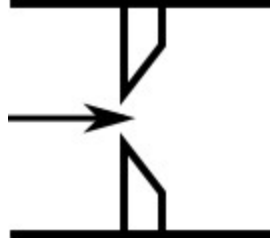
Figure 1: Sharp-edged orifice (in this case $A_{up} = A_{down}$)

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

$$K' = 0.03 + 0.47 * 10^{-7.7\, r_{edge}/D_{orifice}}$$
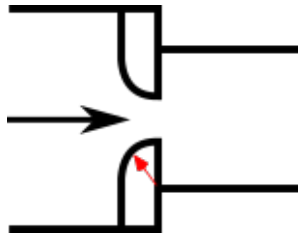
Figure 2: Round-edged orifice with different upstream and downstream pipe areas

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

$$K' = 0.03 + 0.47 * 10^{-7.7\, r_{edge}/D_{orifice}}$$
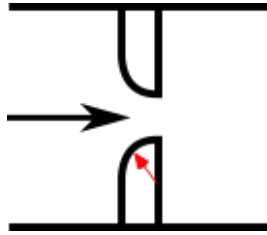
Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

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

Pumps

The Pump junction allows you to model the pressure added to a system by a pump, and model many transient events involving pumps. Because all pumps are different, AFT Impulse 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 Junction 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.

Pump Model

This tab requires the selection and definition of what type of pump the junction represents. There are two models currently 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.

- **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** - Under reverse flow, centrifugal pumps can be made to generate power and act as a turbine. This option represents such a situation.

Figure 1: Pump Models

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.

- **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.

Slurry De-Rating

(SSL Module Only)

Pump head degradation occurs when pumping solids. This is known as de-rating. This option is not available for the Pump as Turbine.

Transient

The pump junction allows several types of transients:

- Without Inertia
    - **None** - Do not apply any transient, maintain the speed set throughout the transient, or modify the speed to meet the Variable Speed setpoint.
    - **Speed vs. Time** - Directly specify how the speed of the pump changes over time. This can be used to model startups, trips, or control transients.
- With Inertia - Account for speed changes of a centrifugal pump impeller with an Inertial Model.
    - **Startup** - Along with the torque characteristics of the associated motor or driver, determine the speed change over time of the pump under an uncontrolled startup.

- **Trip** - Determine the speed change of the pump under instantaneous loss of motor or driver torque, such as during a power failure.
- PAT Transient
  - **Full Load Rejection** - Decouples the pump impeller from the motor and the PAT transitions from operating as a turbine to operating while decoupled.

For more information on transient data, including event transients, see Junction Transient Data.

All of the transient options need to know when to start with Initiation of Transient and allow the definition of a Transient Special Condition.
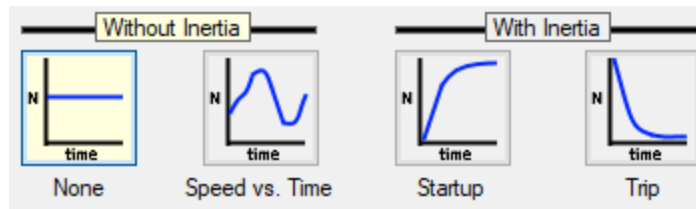
Figure 2: Pump Transients

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.
  - **Pump Off With Flow Through** - The pump is shut off but allows flow to continue through. This is identical to a lossless connection.
- **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. This option is not available for the Pump as Turbine.
- **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. This option is only available for the Centrifugal Pump model.

Related Topics

Related Blogs

What Does "Head (HGL)" Mean for Submerged Pumps and Exit Pressures?

Centrifugal Pump Overview

Standard Pump Curve

A Standard 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 these curves, click Enter Curve Data. Once this data has been defined, previews of the applied curves can be seen here.
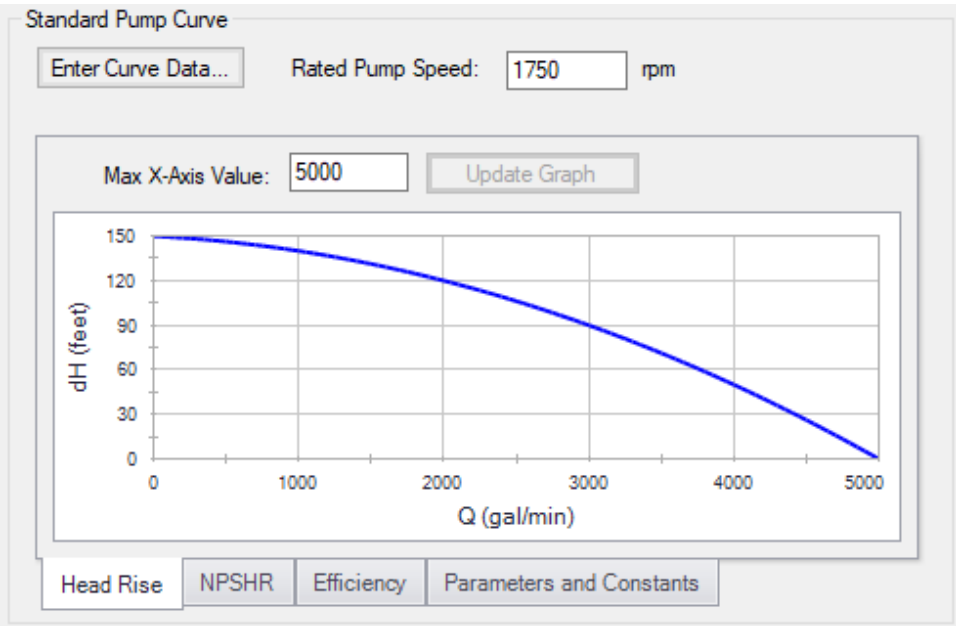
Figure 1: Standard Pump Curve

Performance Curve Used in Simulation

This section allows users to specify whether their pump will operate on the Standard Pump Curve they have defined, or on a Four Quadrant Curve. If it is known that the pump will never experience reverse flow, the Standard Pump Curve is sufficient. However, if it is possible for reverse flow to occur, the user should use the Four Quadrant Curve option, as it is important to accurately account for effects outside the Normal Pumping Zone (positive head, flow, and speed).

If the Standard Pump Curve is chosen, the only required input is the pump curve data, and that curve will always be used in the transient simulation. If reverse flow is encountered during the transient simulation, a warning will be presented, and the model should be adjusted accordingly. Impulse will attempt to estimate the effects of reverse flow by applying the zero-flow (deadhead) value to the fluid, but this is a rough estimate, and the users should modify the pump junction to use a four-quadrant curve if the pump sees large or sustained reverse flows.

If the Four Quadrant Curve is chosen, the user will be required to input the pump curve data, and select a Four Quadrant data set. To select a Four Quadrant data set, the user can either allow Impulse to Automatically Select the data set, or they can select a set manually. The Auto-matically Selected option will chose a data set based on the estimated specific speed of the user's pump.

When a Four Quadrant Curve is being used, the user has the option to allow Impulse to Prevent Reverse Rotation. This setting would reflect a physical device used to prevent the pump impeller from rotating in reverse, such as an impeller ratchet or similar. Reverse rotation can be pre-vented independently of reverse flow - enabling only this option will still allow reverse flow, but the impeller will not change directions under sustained reverse flow.
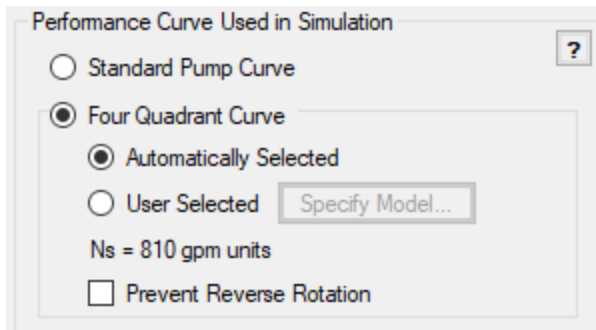
Figure 2: Performance Curve and related options

Model Combined Check Valve and Pump
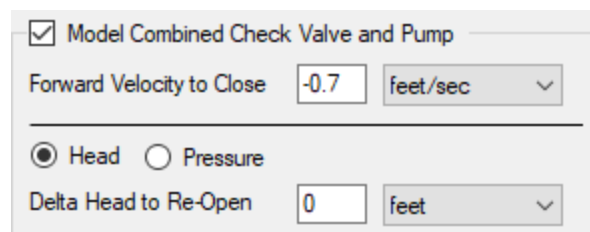
This option adds a simple check valve to the pump junction to limit the reverse flow.

- **Forward Velocity to Close** - The **forward** velocity under which the check valve closes. Once this velocity is reached during the simulation, the check valve will instantly close,

preventing further flow. A check valve that closes under reverse flow requires a **negative** value to be entered for the Forward Velocity to Close.

- **Delta Head/Pressure to Re-Open** - The check valve can re-open during the simulation. In order to do so, it must have a corresponding pressure or head difference across it defined. A positive value here indicates that the upstream pressure must be higher than the downstream pressure.

Note: The Combined Check Valve is limited in its ability to capture the potentially complex behavior of a check valve. Including the Check Valve within the Pump provides a convenient input method and also avoids modeling short pipes which lengthen the transient simulation. If the behavior of the check valve is critical to the system, it is recommended to use the Check Valve junction.
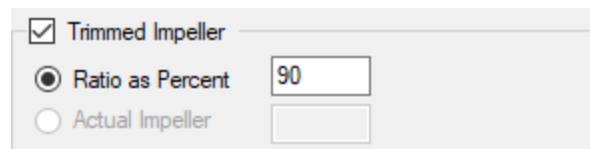
Figure 3: Model Combined Check Valve and Pump

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.

Figure 4: Trimmed Impeller

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See Submerged Pump.

Flow Directional Behavior

A visual representation of the modeled pump is presented in the lower right of the properties window. This provides immediate feedback on what the various options represent. In the pump depicted in Figure 5, a Four Quadrant Curve is being used, modeled to prevent reverse rotation and have a check valve that closes at a negative forward velocity.
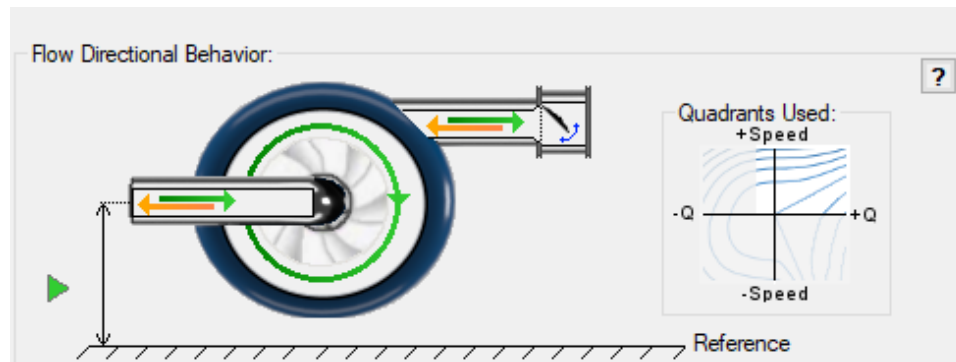
Figure 5: Visual diagram of model

Related Topics

Related Blogs

Determine Your Pumps BEP and Know API-610's Recommendations

Centrifugal 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.
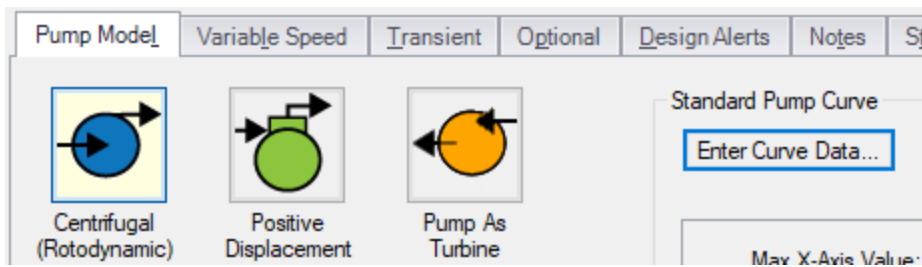
Figure 1: Enter Curve Data

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 Configuration Window

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 flowrate.

- **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 it's 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, <u>Reference Density</u> 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)** - % 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 based on the specific speed of the pump, from ANSI/API 610 11th Edition, or can be defined by the user.
- **Allowable Operating Region (AOR)** - Similarly, this flow range creates a warning in the output when the user specified flow range is not adhered to.

Graphs

The following tabs display the curves that have been fit to the given data. If multiple curves are defined, the Composite Graph tab becomes available. The other graph tabs show the individual curves, along with points representing the raw data. It is a good idea to check that these graphs to confirm the curve fit is reasonable for any potential operating region.
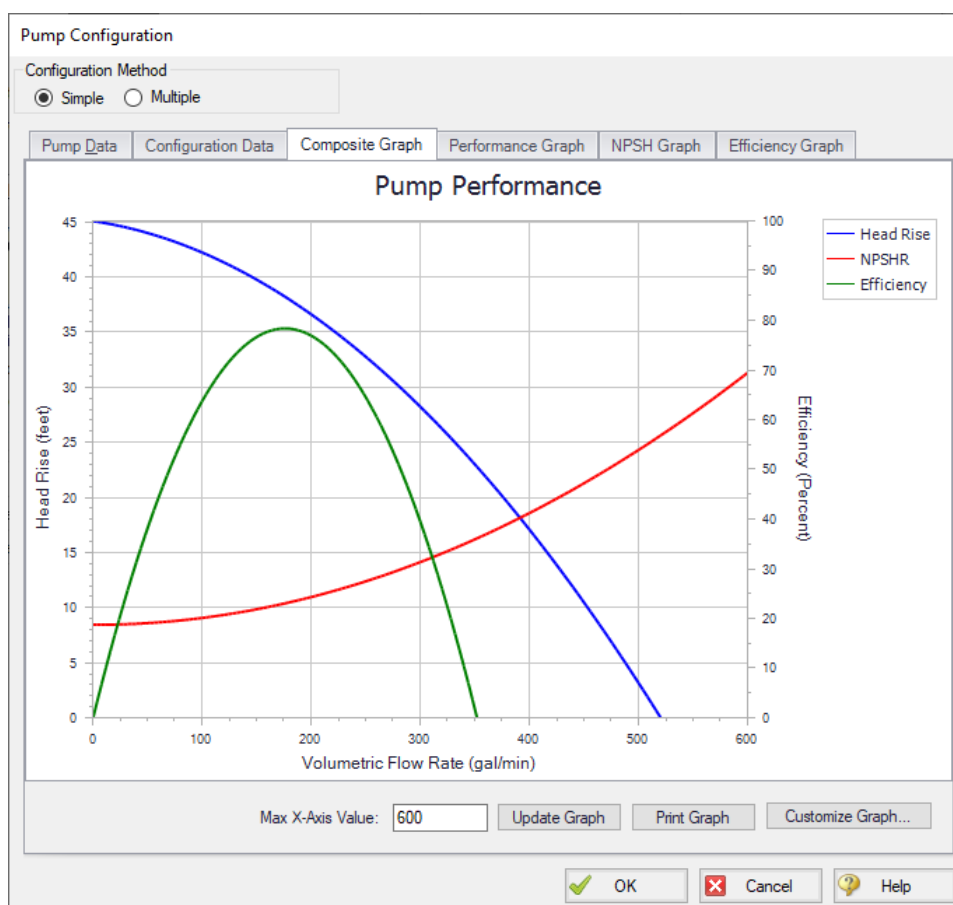
Figure 4: Composite Graph

Related Topics

Related Blogs

Determine Your Pumps BEP and Know API-610's Recommendations

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.

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

Standard Pump Curve Waterhammer Theory

A pump head curve is a function of flow and speed.

$$\Delta H = f(Q, N)$$

For any given fixed speed, the head curve is only a function of flow. The head curve can be directly adjusted for speed using the affinity laws.

Considering the case where the head/flow curve follows a polynomial relationship, and including a speed ratio term **s**:

$$\Delta H = s^2 a + s\, b\, Q + c\, Q^2 + \frac{d}{s}\, Q^3 + \frac{e}{s^2}\, Q^4$$

where

$$s = \frac{N}{N_{100\%}}$$

Converting volume flow to mass flow and head to pressure:

$$\Delta P = \rho\, g\, \Delta H \qquad Q = \frac{\dot{m}}{\rho}$$

$$\Delta P = \rho\, g\left(s^2 a + s\, b\, \frac{\dot{m}}{\rho} + c\left(\frac{\dot{m}}{\rho}\right)^2 + \frac{d}{s}\left(\frac{\dot{m}}{\rho}\right)^3 + \frac{e}{s^2}\left(\frac{\dot{m}}{\rho}\right)^4 \right)$$

Because **ρ**, **g**, and the curve constants are constant throughout the simulation, they can be combined and the equation simplified.

$$\Delta P = s^2 a' + s\, b'\, \dot{m} + c\, \dot{m}^2 + \frac{d'}{s}\, \dot{m}^3 + \frac{e'}{s^2}\, \dot{m}^4$$

Solving for **ΔP** with Compatibility Equations 1 and 2 :

$$\Delta P = P_{down,new} - P_{up,new} = (C_M + \dot{m}\, B_M) - (C_P - \dot{m}\, B_P)$$

and equating the two:

$$(C_M + \dot{m}\, B_M) - (C_P - \dot{m} B_P) = s^2 a' + s\, b'\, \dot{m} + c'\, \dot{m}^2 + \frac{d'}{s}\, \dot{m}^3 + \frac{e'}{s^2}\, \dot{m}^4$$

For a fixed speed, the only unknown is mass flow. For quadratic curve fits (d and e are zero), this equation is solved analytically. For higher order curve fits, it is solved for with Newton-Raphson iteration. Upstream and downstream pressures can then be determined from the compatibility equations.

For variable speed pumps, another parameter must be considered fixed - for example, suction or discharge pressure. In these cases, the above equations can be arranged to solve for the speed required.

If the speed and flow can vary throughout the transient without any parameter being fixed - for example, during a pump trip or startup - further equations and data are required. See Inertial Models.

Flows Outside the Normal Pumping Zone

The Standard Pump Curve is only valid for the Normal Pumping Zone - positive speed, flow, and head. To model such situations without additional four quadrant data, assumptions must be made.

Behavior from negative speed is not considered as the characteristics can be significantly different. This leaves two situations not normally covered - reverse flow and flow past the pump maximum.

- **Reverse Flow** - It is assumed that the head rise across the pump for all reverse flows is equivalent to the head rise at zero flow for the given curve at the given speed. This is a very rough estimate and will not be accurate for large or sustained flows, and a warning will be generated to this effect.
- **Flow Exceeding Maximum** - At high flows for a given speed there will be a point where head is actually *lost* across a pump. The Standard Pump Curve does not account for this - instead, it is assumed that the head rise is zero.

Submerged Pumps

If a Pump is defined as Submerged, the upstream pressure is known and fixed. The same process as before can be used to solve for flow.

Vapor Cavitation Theory

When the calculated pressure at the upstream side of a Pump junction drops below vapor pressure, a vapor cavity forms on that side of the junction.

Usually vapor cavities form on the upstream side of the pump only, but in cases where there is reverse flow through the pump cavities can also form on the downstream side.

$$V_{vapor,up,new}$$
$$= V_{vapor,up,old}$$
$$+ \left(-\dot{m}_{up,new} - \dot{m}_{up,old} + \dot{m}_{pump,new} + \dot{m}_{pump,old} \right) \frac{\Delta t}{2\,\rho}$$

$$V_{vapor,down,new} = V_{vapor,down,old}$$
$$+ \left(-\dot{m}_{pump,new} - \dot{m}_{pump,old} + \dot{m}_{down,new} \right.$$
$$\left. + \dot{m}_{down,old} \right) \frac{\Delta t}{2\,\rho}$$

When a vapor cavity forms upstream of a pump, the upstream pressure becomes fixed at the vapor pressure. The calculation then is similar to a submerged pump, which also flows from a fixed pressure. Similar to a pipe interior node, when the vapor volume is negative the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Specific Speed

Specific speed is an index which can be used to characterize the behavior of a pump. Pumps of identical specific speed will, in general, have similar behavior at the rated conditions. **It is important that specific speed be calculated for maximum impeller diameter, and at the pump's BEP, considering head per stage.** It is possible to use this formula with a trimmed impeller, or at operating conditions away from BEP, but the number will no longer have the same meaning.

The dimensionless form of specific speed is defined as:

$$N_s = \frac{\omega_{Rated}\,\sqrt{Q_{Rated}}}{(g\,H_{Rated})^{3/4}}$$

More commonly, a dimensional form is used. This equation is listed in both ANSI-HI 14.3.1.3.1-2019 and API 610 11th Edition

$$N_s = \frac{N_{Rated}\sqrt{Q_{Rated}}}{H_{Rated}^{3/4}}$$

Note: Specific speeds using this definition are often presented as "dimensionless" numbers - but the value returned depends on the unit system used. In such cases it should be made clear what unit system is involved - for example N_S = 4000 (gpm units) or N_S = 77.5 (metric units). There is a conversion factor of 51.6 between the two versions.

- N_S = Specific speed
- N_{Rated} = Rotational speed in revolutions per minute
- ω_{Rated} = Rotational speed in radians per second
- Q_{Rated} = Total pump flow rate in cubic meters per second (US gallons per minute)
- H_{Rated} = head **per stage** in meters (feet)

There are alternative methods used to calculate specific speed that include dividing the flow rate (Q) in the above equation by 2 if the impeller is of a double suction design. This will result in the calculated specific speed being less by a multiplying factor of $1/\sqrt{2}$, or approximately 0.707 times less. Therefore the user is urged to take caution and determine how the manufacturer of their pump determined specific speed.

Relation to Pump Design

Specific speed is common in pump design - it conveys information about the type of pump or impeller that will be required. Low specific speeds represent radial machines - delivering high head at relatively low flow. High specific speeds represent axial flow devices - delivering high flow at low head. Mixed flow devices have an intermediate specific speed. With specific speed alone it is possible to make an estimate of the physical construction of a pump.

Four Quadrant Model Overview

Four Quadrants of Operation

To accurately model a pump undergoing reverse flow or speed, or pumps past the runout flow, test data for those regions must be available. In certain cases rough approximations can be made without this data, but as with any model, good results require good data that reflect the actual characteristics.

Turbomachines can operate at any flow or speed, positive or negative. The speed-flow plane thus creates four quadrants - hence the name *Four Quadrant*.
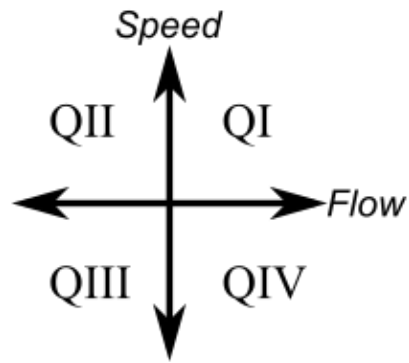
Figure 1: The speed-flow plane, showing the four possible quadrants of operation.

At any given speed and flow combination, there is an associated head and torque. Any given turbomachine must operate on these surfaces of points defined by head-speed-flow and torque-speed-flow in steady-state.
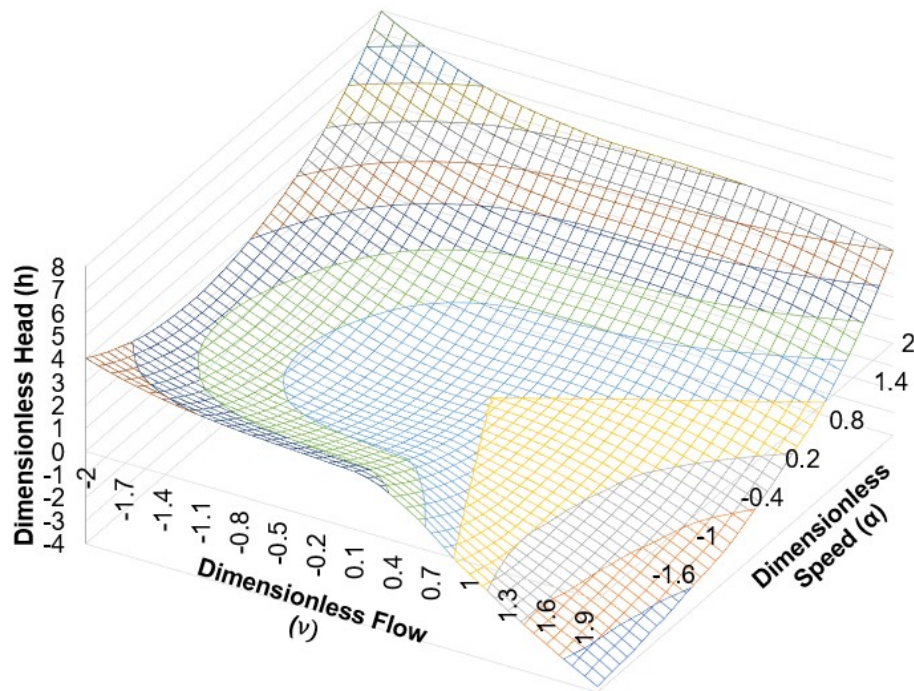
Figure 2: Relationship of head to flow and speed over all four quadrants

Such a surface is usually presented as a contour plot, with curves of constant head and torque drawn on a 2 dimensional speed-flow diagram.
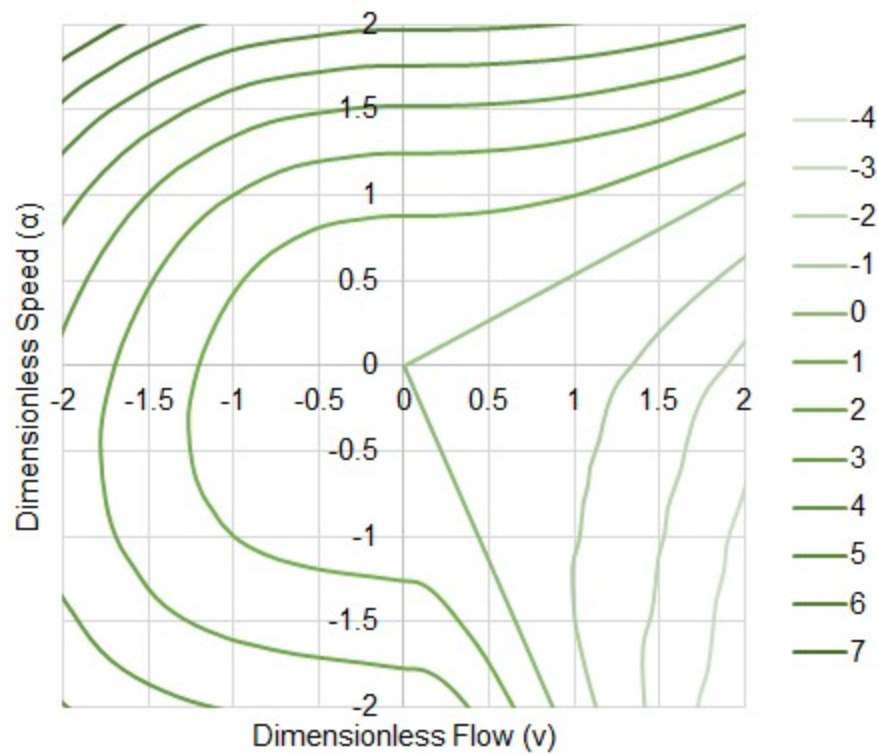
Figure 3: Contour plot showing lines of constant head. Same data as Figure 2.

Similar data for torque can be presented on the same plot, commonly referred to as a Karman-Knapp Circle Diagram.
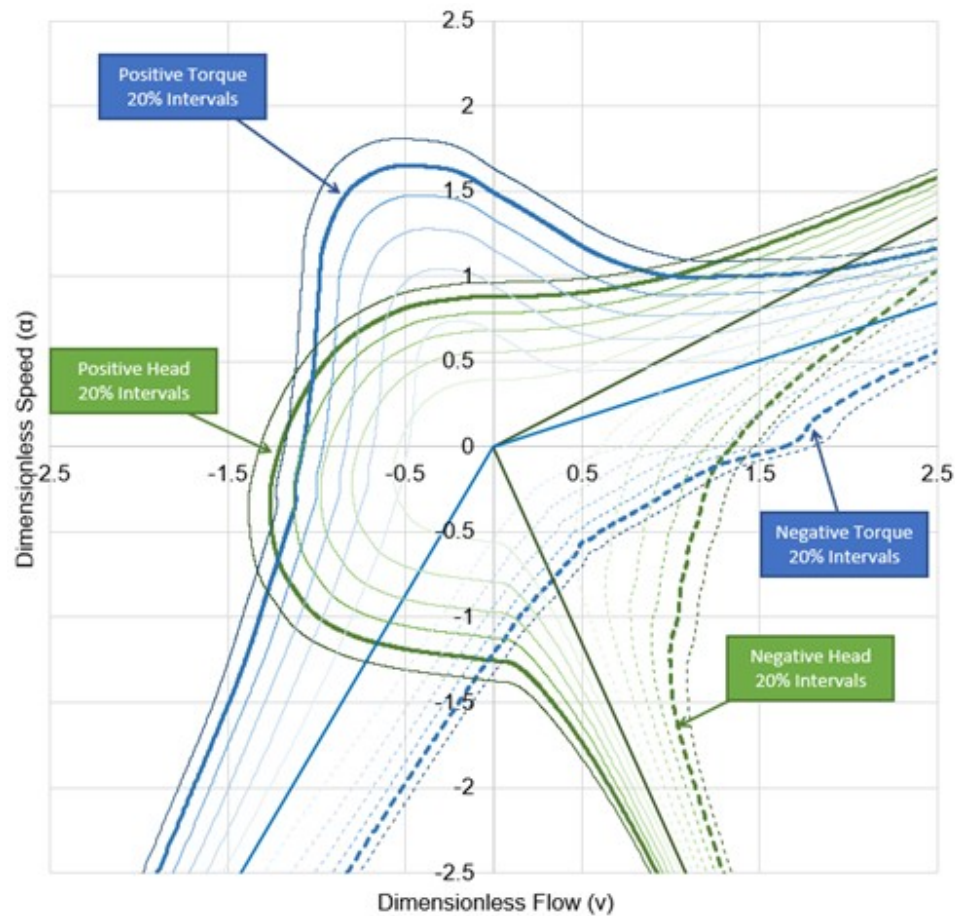
Figure 4: Constant contours of torque and head for a particular turbomachine. This is the same pump as Figures 2 & 3, but with different contours included.

This is the data required to model a turbomachine under reverse flow and speed. But how do we apply it? And what simplifications can be made?

Simplifying the Data with the Affinity Laws

A real turbomachine has characteristics unique to certain operating speeds or flows. For example, a very high flow or speed may induce transient cavitation, severely impacting head generated. On the other end of operation, a very low speed will be more impacted by friction of the various rotating components. If the flow is low enough, it will become laminar, changing the behavior of the fluid dramatically.

To fully account for these real effects, a simulation would need actual data for every possible situation. In other words, a set of curves such as that shown in Figure 4 would be the minimum data entry. Clearly, this would be an enormous amount of data to enter, if it was even possible to acquire. Fortunately, these effects - although real - do not need to be considered for the vast majority of pump studies.

To greatly simplify the data required, the pump can be considered as completely homologous - meaning that the affinity laws apply for all speeds, flows, heads, and torques. This means that instead of a large collection of curves, only a handful of curves are required, as the others can be appropriately "scaled" from the source curve.

In fact, the curves shown in the figures above represent exactly this scaling - they do not represent raw measurements for each combination of speed and flow. Instead, most of the contours were generated using the relationships defined by the affinity laws, with only full speed (positive and negative) values being measured, as represented by the bold contours.

Applying the Data in Numerical Simulation

It would be possible to use the raw data defined by a set of curves such as that shown above directly in a numerical simulation. One approach used in the early days was to store many curves in familiar forms such as head vs flow and torque vs flow at a variety of speeds. This approach, while valid, can pose great difficultly in computational analysis because every parameter - head, torque, speed, and flow - can become zero at any point during the simulation. Determining which curves to use and handling zero values dynamically is difficult.

An alternative method, proposed by Suter, 1966, involves transforming the data into a different coordinate system such that the values of numerical interest never become zero. This method, used by AFT Impulse, is commonly referred to as the "Suter Method."

Before proceeding with the transformation, the measured data from a particular turbomachine is made dimensionless by dividing through by a Dimensional Reference Point. It should be noted that these rated values are traditionally considered to be the rated values at the Best Efficiency Point. However, alternative references can be used in some situations.

Because there are four parameters, four reference values are required to make the data dimensionless, like the data presented in Figure 4 above.

$$h = \frac{H}{H_{Ref}} \qquad \beta = \frac{T}{T_{Ref}} \qquad \alpha = \frac{N}{N_{Ref}} \qquad v = \frac{Q}{Q_{Ref}}$$

where

h = Dimensionless head

H = Dimensional head

β = Dimensionless torque

T = Dimensional torque

α = Dimensionless speed (note that α is commonly used to denote angular acceleration. To maintain consistency with the literature, this represents the dimensionless speed here)

N = Dimensional speed

v = Dimensionless flow

Q = Dimensional flow

The data can now be transformed into a more useful coordinate system. For any given point on one of the head curves (as an example), there is a specific value of speed, flow, and head. Drawing a line from the origin to this point creates an angle to the exiting axis.
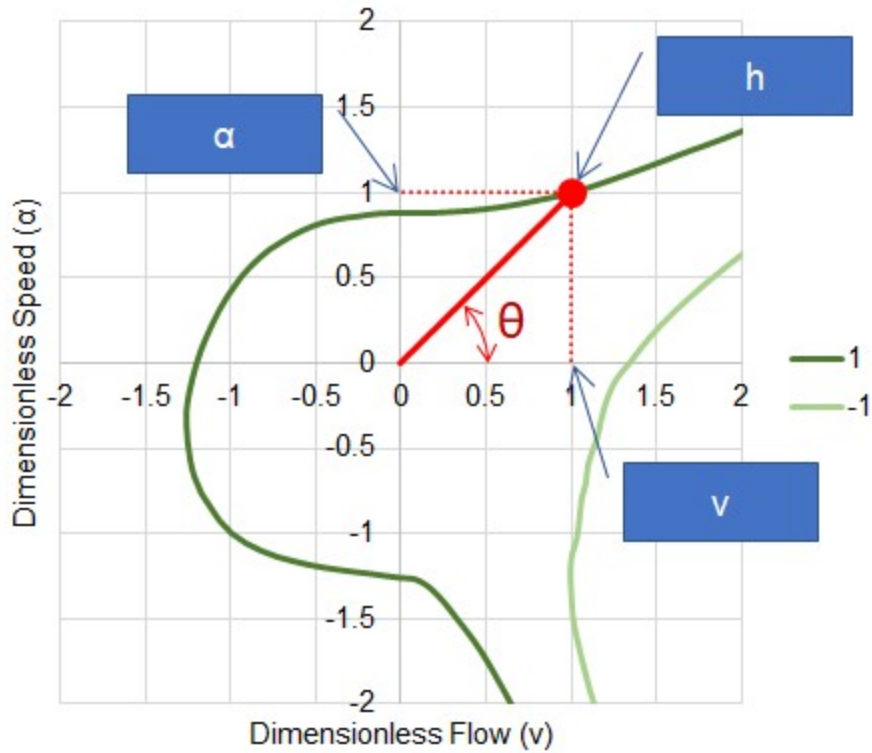
Figure 5: Transforming the data

The angle is simply defined with trigonometry:

$$\theta = \tan^{-1}\left(\frac{\alpha}{v}\right)$$

or in some definitions:

$$\theta = \pi + \tan^{-1}\left(\frac{v}{\alpha}\right)$$

To define the point itself, the following equations are used for head and torque, respectively:

$$F_H(\theta) = \frac{h}{\alpha^2 + v^2}$$

$$F_B(\theta) = \frac{\beta}{\alpha^2 + v^2}$$

For the point shown in Figure 5, F_H evaluates to 0.5, with a Θ of 45 degrees. Applying this transformation to all of the data in the speed-flow plane, we can construct values of F_H and F_B for all Θ. In other words, F_H and F_B are continuous functions of Θ.
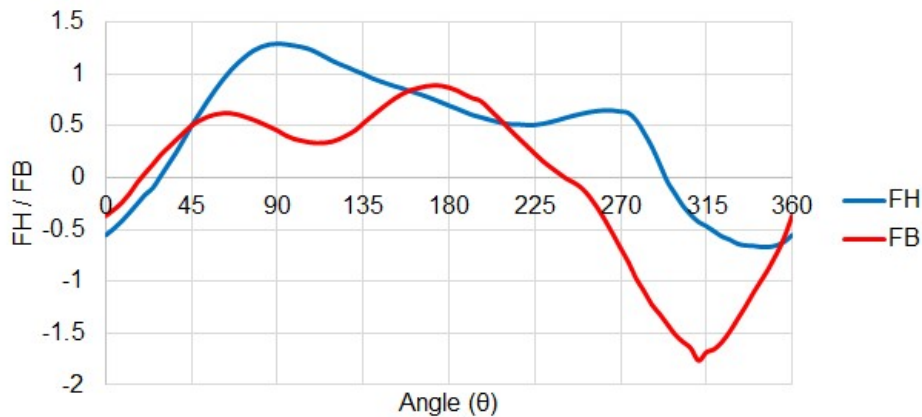
Figure 6: F_H and F_B values for all Θ. This is still representing the same data as all previous Figures.

By reversing the transformation, data in a standard form such as head vs flow can be recovered fully.

Note: All of the above example Figures represent the Ns = 0.66 (1800 gpm units) data set, available in AFT Impulse. For this reason Figures 2 - 4 do not show divergence from the affinity laws (it is not raw data, but back-calculated from the data shown in Figure 6).

Related Topics

Four Quadrant Zones of Operation

While a turbomachine can operate anywhere in the four quadrants of the speed-flow plane, it will only act as a pump in certain areas. It can also act as a turbine, or be in an "energy dissipation" zone.

The behavior of a pump changes significantly depending on its design - a radial (low [specific speed](#)) pump has different characteristics than an axial (high specific speed) pump.

Zones of Operation

- **Zone A - Normal Pumping** - This is the region represented by a Standard Pump Curve, generally available from the manufacturer. Energy is transferred from the driver to the fluid, imposing flow and positive head.

- **Zone B - Energy Dissipation** - Region exceeding maximum pump flow (or runout flow). Head is lost across the pump.

- **Zone C - Reverse Turbine** - Flow and speed are in the positive direction, driving the shaft. This is termed "reverse" because it is typical for turbines to run in the opposite direction.

- **Zone D - Energy Dissipation** - Energy entering the system is actively opposing the natural flow direction.

- **Zone E - Reverse Pump** - The impeller is rotating backwards, but pumping action is still taking place. Energy is transferred from the driver to the fluid, imposing flow and head. The behavior of this zone varies based on design - radial designs can still pump in the normal direction when rotating backwards, whereas axial machines can force the flow in the reverse direction.

- **Zone F - Energy Dissipation** - The impeller is rotating backwards, with negative flow. However, head is increased. This zone is similar to Zone B.

- **Zone G - Normal Turbine** - Flow and speed are in the reverse direction, driving the shaft, causing the machine to generate power.

- **Zone H - Energy Dissipation** - Energy entering the system is actively opposing the natural flow direction. This zone is similar to Zone D.

Table 1: Characteristics of Zones

Zone	Behavior	Flow	Speed	Head	Torque	Fluid Power (Sign of Q*H)	Shaft Power (Sign of N*T)	Useful Work (Powers same sign)
A	Normal Pumping	Positive	Positive	Positive	Positive	Positive	Positive	Yes
B	Energy Dissipation	Positive	Positive	Negative	Positive	Negative	Positive	No
C	Reverse Turbine	Positive	Positive	Negative	Negative	Negative	Negative	Yes
D	Energy Dissipation	Positive	Negative	Negative	Negative	Negative	Positive	No
E (Radial)	Reverse Pump	Positive	Negative	Positive	Negative	Positive	Positive	Yes
E (Axial)	Reverse Pump	Negative	Negative	Negative	Negative	Positive	Positive	Yes
F	Energy Dissipation	Negative	Negative	Positive	Negative	Negative	Positive	No
G	Normal Turbine	Negative	Negative	Positive	Positive	Negative	Negative	Yes
H	Energy Dissipation	Negative	Positive	Positive	Positive	Negative	Positive	No

Transient Behavior

The operation of a pump must move continuously from one point on the speed-flow plane to another. It can never "jump" to an entirely different zone.

For example, consider a pump lifting fluid over a hill under a trip event.

The pump will initially be operating in Zone A, but under loss of driver torque will immediately begin to change speed and flow.

The inertia of the device will maintain forward speed while flow reverses - in other words, the Zone will change from A to H. The energy dissipation Zone H is unstable - the act of reverse flow on the forward rotating impeller with no applied torque will slow the impeller down and eventually reverse its direction.

Once speed direction changes, the pump is now operating in Zone G. Energy could theoretically be extracted in this turbine zone, however, there is no connected driver. The energy from the still positive torque must go somewhere - into speeding up the impeller.

At some point, the impeller will speed up enough that operation crosses into Zone F. In Zone F, the torque changes sign - causing the impeller to slow down and return to Zone G. The device will oscillate around the zero torque line, eventually coming to a new steady-state - runaway conditions.
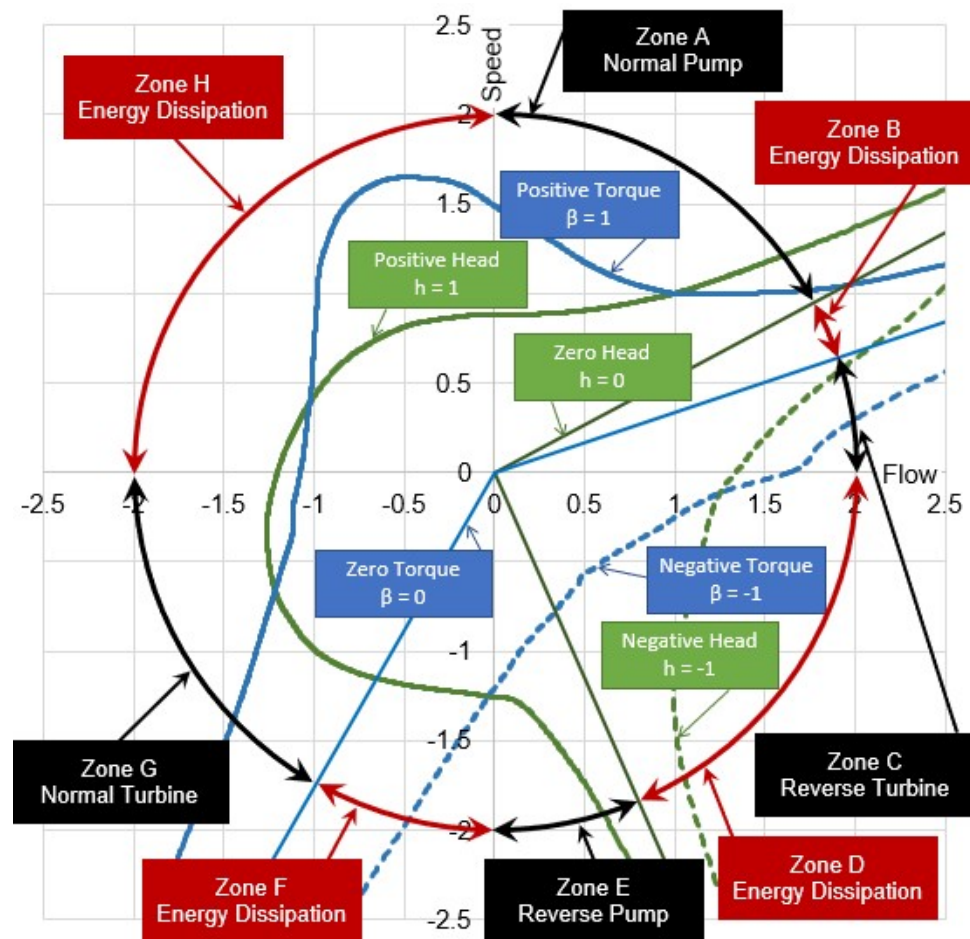
Figure 2: Zones of operation for a radial design pump, from data set Ns = 1800 gpm (34.9 m3/s)

Related Topics

Four Quadrant Data Sets

Four Quadrant data for a specific pump is rarely available. Manufacturers generally will only test pumps in the Normal Pumping Zone - positive flow, speed, head, and torque. Testing a pump outside of this range is expensive, and often requires testing apparatus beyond what is required for the standard range.

Even though four quadrant data specifically related to a certain pump may not be available, it is still important to account for the effects of reverse flow and speed when they occur. Making an estimation with a Standard Pump Curve is not sufficient in most cases, so another approach must be taken.

This approach - also an estimation - is to utilize four quadrant data from a **similar** pump. It is not unreasonable to assume that two pumps of similar construction would behave similarly under any given flow condition. It may be that the specific pump of interest may have only been tested in the Normal Pumping Zone, whereas the very similar pump may have been tested in all four quadrants. The test data may be used as a basis for the simulation of the pump of interest, with some adjustments.

What Makes a Similar Pump?

Accepting that data from a similar pump can be used for the simulation, we can proceed with selecting some data from an appropriate pump with all of the required data. However, how do we know what data is appropriate? It is highly unlikely that a physical comparison between the two pumps can be made. It is even unlikely that the rated parameters for the similar pump will be available, as the data is almost always stored in a dimensionless form as described in Four Quadrant Model Overview.

Fortunately, there is one convenient parameter that is traditionally used to characterize pumps of similar operation - specific speed. Specific speed attempts to describe the overall characteristics of a pump by relating its discharge and head. Pumps of an identical specific speed can generally be assumed to behave the same, even if they are of different design or size.

This means that if a set of four quadrant data has a specific speed similar to the pump being analyzed, this data can be applied without needing to know anything further about the pump that was actually tested.

Potential Issues Using Specific Speed to Measure Similarity

Turbomachines are complex devices with many components that affect their behavior. Breaking down this behavior into a single number - specific speed - necessarily has some drawbacks. Two pumps of identical specific speed may not show exactly the same behavior at all flows and speeds. Some references (Brown and Rogers, 1980) indicate that the correlation between overall behavior and specific speed is less than may be expected, especially for low specific speed (radial) devices.

It could be the case that a four quadrant data set developed for a pump of a *different* specific speed may be more appropriate in a given simulation, as it may better reflect the behavior of the pump in question.

Using Existing Data is an Estimation

When applying existing data to simulation, it is critical to remember that this method is an estimation. For accurate results, test data from the actual pump should be used. Experience shows that this estimation, when properly applied, is acceptable within the normal bounds of fluid

transient analysis. The accuracy of the estimation depends heavily on the specific pump, the data set used, the system, and how the estimation is applied.

Furthermore, because the four quadrant data sets are stored in a dimensionless form, to be used in simulation they must be made dimensional. The dimensional data can be forced to match in certain conditions, which will force the data *not* to match in other conditions. Because the data sources represent **two different pumps**, a perfect match under all conditions is impossible.

The data is made dimensional with a given reference point based on the Standard Pump Curve. This choice can have dramatic impacts on the simulation, as described in Four Quadrant Dimensional Reference Point.

Four Quadrant Data Sets Available in AFT Impulse

There are many four quadrant data sets available for use in AFT Impulse, from a variety of sources. Many of these data sets appear in multiple references throughout the technical literature on four quadrant methods. The available data sets are listed in Table 1 below.

Notice that each data set is labeled as Preferred, Average, or Use With Caution. By default only preferred and average data sets will be shown. These recommendations are based on Walters, Dahl & Rogers (2020) which uses a comparison of the four quadrant data sets to the stepanoff curves to evaluate the quality of the test data and the accuracy of the reported specific speed for each data set. Figure 1 which was adapted from Walters, Dahl & Rogers (2020) shows several of the four quadrant data sets used in Impulse compared to the stepanoff curves. The 13500(262) and 1800(34.9) curves, shown in red and blue on the figures, are listed as preferred in Impulse, while the 7600(147) curve, shown in yellow, is listed as average.

Table 1: Four Quadrant Data Sets Available

Specific Speed				
	metric units			
gpm units	(gpm units / 51.6)	Data Range	Status	Reference
810	15.7	Partial (45-220°)	Preferred	Brown and Rogers 1980
1030	20.0	Full	Preferred	Ayder, Ilikan et al. 2009
1140	22.1	Full	Preferred	Brown and Rogers 1980
1320	25.6	Partial (45-220°)	Use With Caution	Brown and Rogers 1980
1490	28.9	Partial (45-220°)	Use With Caution	Brown and Rogers 1980
1500	29.1	Full	Preferred	Kittredge 1956
1565	30.3	Partial (45-220°)	Use With Caution	Brown and Rogers 1980
1700	33.0	Full	Preferred	Ayder, Ilikan et al. 2009
1800	34.9	Full	Preferred	Donsky 1961/Knapp 1941
1935	37.5	Full	Preferred	Kittredge 1956
2160	41.9	Full	Average	Thorley 1996
3300	64.0	Full	Average	Thorley 1996
3725	72.2	Full	Preferred	Thorley 1996
3940	76.4	Full	Preferred	Thorley 1996
4400	85.3	Full	Preferred	Brown and Rogers 1980
5000	96.9	Full	Preferred	Thorley 1996
5200	101	Full	Use With Caution	Thorley 1996
5420	105	Full	Average	Ayder, Ilikan et al. 2009
6340	123	Full	Use With Caution	Thorley 1996
6790	132	Full	Use With Caution	Thorley 1996
6940	134	Full	Use With Caution	Thorley 1996
7600	147	Full	Average	Donsky 1961
8760	170	Full	Use With Caution	Thorley 1996
10800	209	Full	Average	Ayder, Ilikan et al. 2009
13500	262	Full	Preferred	Donsky 1961
13500	262	Full	Preferred	Ayder, Ilikan et al. 2009
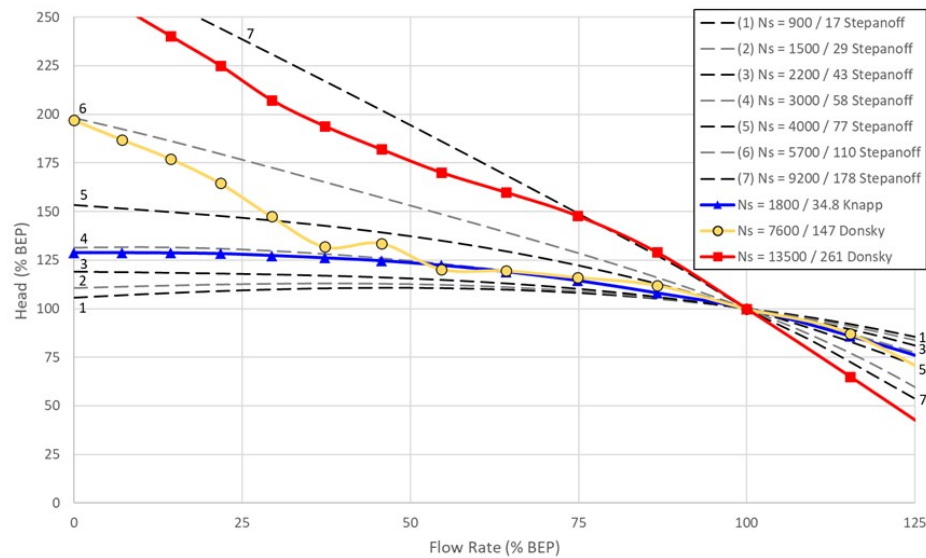
Figure 1: Comparison of head curves from several four quadrant data sets to Stepanoff curves
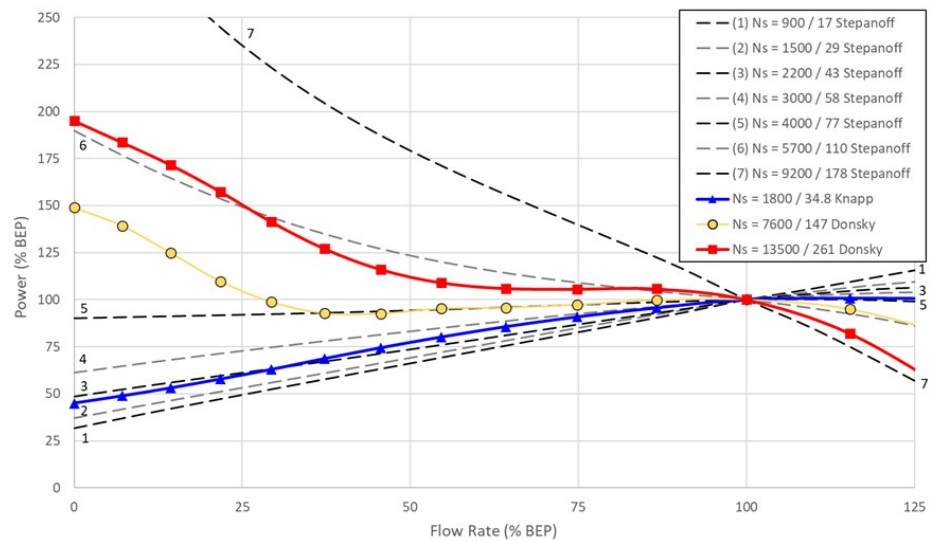
Figure 2: Comparison of power curves from several four quadrant data sets to Stepanoff curves

Related Topics

Four Quadrant Dimensional Reference Point

As described in Four Quadrant Data Sets, data specific to a given pump is rarely available. Instead, four quadrant test data from a different pump with similar specific speed is commonly used as an estimation. Because the Suter method stores the four quadrant data in dimensionless form, the data must be made dimensional.

As described in Four Quadrant Model Overview, the data is made dimensionless by dividing each parameter by some reference value:

$$h = \frac{H}{H_{Ref}} \qquad \beta = \frac{T}{T_{Ref}} \qquad \alpha = \frac{N}{N_{Ref}} \qquad v = \frac{Q}{Q_{Ref}}$$

To obtain the original values, the dimensionless parameters can simply be multiplied by the original reference values. This only works to recreate the values for the original pump - the pump of interest (the "similar" pump) - may be of a similar specific speed, but of different size. In this case, different values will be used in dimensionalizing the data. These four reference values are referred to as the **Four Quadrant Dimensional Reference Point**.

Multiple Choices for a Dimensional Reference Point (DRP)

Traditionally, the Dimensional Reference Point used to create the dimensionless data will be the rated conditions at the pump's Best Efficiency Point. Applying these same rated conditions to the dimensionless data will fully recover the original data. Due to the nature of using similar four quadrant data sets to estimate behavior, the DRP used to "unpack" the data will almost certainly be different than the DRP originally used in its creation. There are a couple of reasons for this.

First, the tested pump may have identical specific speed - and perhaps even identical *behavior* - to the pump under analysis, yet still have a different pump curve. This is because a pumps of different size but identical construction will behave in a similar way. This is at the root of the homologous pump laws, which are used to derive the affinity laws. The data from one pump can be made dimensionless with that pump's DRP, and made dimensional for the *other* pump with the *other pump's* DRP. If these two pumps are truly homologous, then the actual, dimensional behavior of the second pump will be fully and accurately represented. Of course, two pumps are very unlikely to be truly homologous and therefore this process will introduce some error.

Second, it is not necessarily required to use the same "type" of DRP for unpacking the data. By default, AFT Impulse uses the rated conditions at **Best Efficiency Point**. Also available is the option to use the **Steady-State Operating Point**. Both of these options are estimations, and both have certain advantages and disadvantages.

> **Note: The choice of Dimensional Reference Point should not be taken lightly**. The choice can have significant impacts on the results of a transient simulation, potentially indicating a system is safe when significant transient cavitation could occur, or perhaps indicating that cavitation will occur when it may in fact be unlikely. ([Walters, Lang, and Miller, 2018](#))
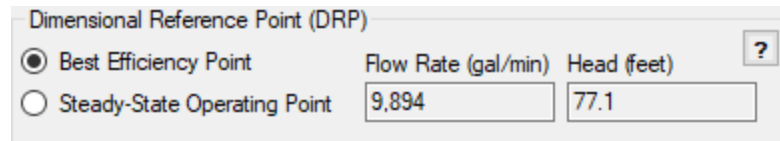
Figure 1: Specification of DRP in AFT Impulse.

Using Best Efficiency Point as the DRP

AFT Impulse can calculate the pump's rated parameters at the Best Efficiency Point (BEP) when the Standard Pump Curve is defined with the required efficiency/power data and a rated speed. With this option selected, these parameters become the DRP.

As mentioned, the Four Quadrant Data Set in use was likely created with the BEP of the test pump as the DRP. If the two pumps are similar, it is therefore reasonable to use the BEP of the new pump as the DRP. If the Four Quadrant Data Set is well matched, the resulting Four Quadrant Curves will closely match the Standard Pump Curve. If the curves appear to be well matched in the Normal Pumping Zone - having similar shape, shutoff, and runout head - it is likely, although not guaranteed, that the Four Quadrant Curve is reasonably accurate under reverse flow and speed conditions as well.

Because the BEP of the pump is an unchanging value, the same Four Quadrant Curve will always be generated regardless of where the pump is running on its curve. This means that if the pump is not running exactly at BEP, even a well matched Four Quadrant Data Set will likely not agree exactly with the Standard Pump Curve. A "jump" from one curve to another can not be allowed during a transient simulation as this would induce an Artificial Transient. To avoid this, **the Standard Pump Curve is not used in simulation when BEP is selected as the DRP**. The Standard Pump Curve is only used for the purpose of generating the required DRP.

This means that the **steady-state results will not agree with the the Standard Pump Curve** unless the data was perfectly matched. This is an area where the estimation begins to have an impact. If the steady-state results are of critical importance in the transient simulation this may be unacceptable. Typically, variations in steady-state behavior are acceptable when analyzing a transient event, as the magnitude of pressure and velocity changes from transient events far outweigh small steady-state differences.

Figure 2: Using Best Efficiency Point as the DRP. The Four Quadrant Curve is static and disagrees in some areas with the Standard Pump Curve. However, the curve matches reasonably, especially near BEP.

Using Steady-State Operating Point as the DRP

If the decrease in accuracy of the steady-state results is unacceptable, it is possible to use the Standard Pump Curve in the steady-state simulation to obtain accurate results. If this is done, another approach must be taken to avoid "jumping" curves and inducing an Artificial Transient. Instead of using the BEP, it is possible to use the Steady-State Operating Point as determined directly with the Standard Pump Curve during the steady-state simulation.

When dimensionalizing the Four Quadrant Data Set with the DRP, the resulting Four Quadrant Curves will be forced to pass through the DRP. In other words, the Standard Pump Curve and the Four Quadrant Curve will be made to intersect exactly where the pump is operating. This has the effect of eliminating any Artificial Transient, while allowing the steady-state simulation to use one curve (the Standard Pump Curve) and the transient simulation to use a different curve (the Four Quadrant Curve).

Keep in mind that the method itself is identical - only the DRP used to carry it out has changed. In fact, if the pump happened to be operating exactly at BEP, the two methods would be completely identical.

Unlike the BEP option, the generated Four Quadrant Curve will change based on where the pump is running on its curve. In effect, the original Four Quadrant Data Set is being *distorted* to match at the steady-state condition. This means that despite the increase in steady-state accuracy we are not able to get away from the fact that this is still an estimation. By reducing error in one area, it has been amplified in another.

What does it mean to say the curve has been distorted? Because the Four Quadrant Data Set represents a certain "character" of pump, the overall shape and trends of the data cannot be changed. By using a different DRP, the data is instead "shifted" as necessary in the appropriate directions. **For example, if the Steady-State Operating Point is calculated to have lower flow and higher head than the BEP, the *entire* Four Quadrant data set will be *compressed* in the "flow direction" and *stretched* in the "head direction" to force the original reference point to meet the new reference point**.

If the pump is operating far from BEP, this can potentially result in **severe** distortion of the Four Quadrant Pump Curve. In such cases, the pump behavior is accurate during the steady-state simulation, but may differ wildly from the Standard Pump Curve under different flow conditions.
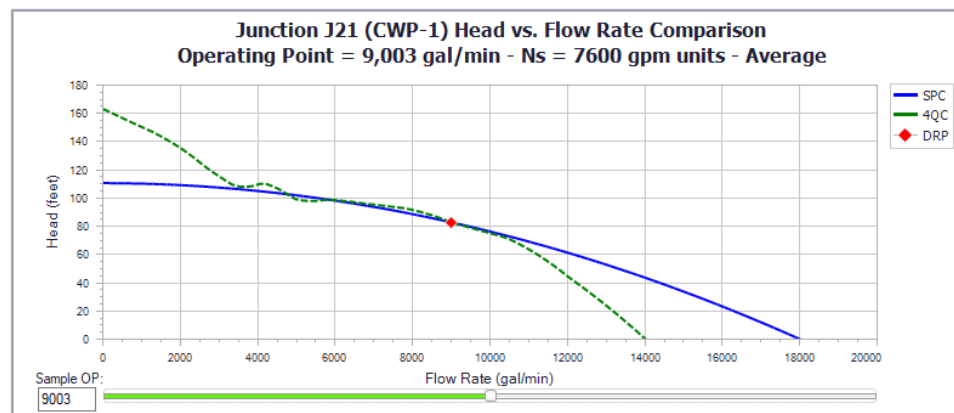
Figure 3: Using a Sample Steady-State Operating Point as the DRP for the same Standard Pump Curve shown in Figure 2. The point is near BEP so the curve still matches reasonably well.
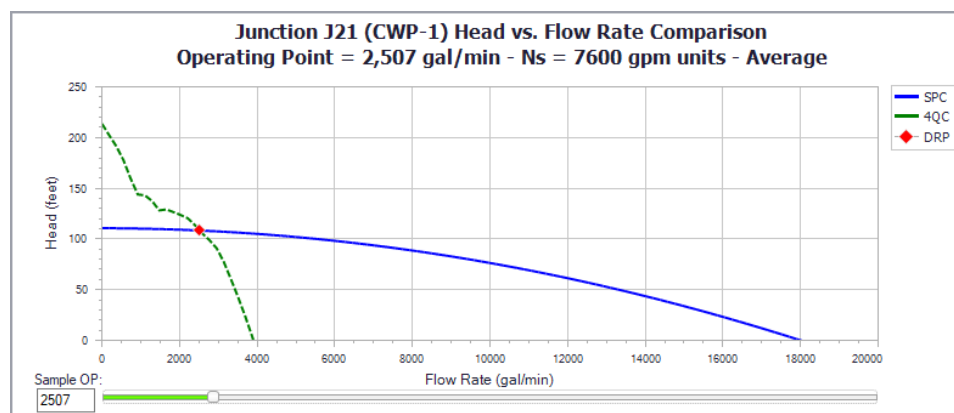
Figure 4: Using a Sample Steady-State Operating Point far from BEP introduces significant distortion. The curves still match at the DRP, but the differences are severe at other flows.

Which DRP should I use?

The question of which point should be used may be obvious, but the answer is not. The best approach is to try and ensure that the Four Quadrant Curve reasonable matches the Standard Pump Curve for the expected range of operation.

If it is not clear which option is better, it is recommended to do a sensitivity analysis and run cases of the model with both options. Note that because *both* methods are estimations, this does not "bound" the possible solutions. Instead, it will give some idea of how impactful the choice is. Large differences in simulation results should encourage close attention and high safety factors. See the Four Quadrant Data Set Selection topic for more details.

Related Topics

Four Quadrant Data Set Selection

As discussed in the Four Quadrant Model Overview, four quadrant data sets are measured data sets intended to approximate the pump behavior outside of the Normal Pumping Zone (i.e. pump operating at positive speed and positive head with forwards rotation). In cases where the pump operates beyond these boundaries, four quadrant data may be used as a tool to predict the head and flow at the pump when measured data is not available. Thus, when the pump is operating in the first quadrant, results generated using the pump performance data provided by the manufacturer (the manufacturer curve) will provide the most accurate results. Once the pump begins operating beyond this region, such as when reverse flow begins, the dimensionalized four quadrant data set (four quadrant curve) will likely provide more accurate results than the manufacturer curve. However, different four quadrant sets can provide widely varied results. Caution should be used to ensure that the chosen four quadrant data set provides reasonable results.

In Impulse, a four quadrant curve can either be selected automatically, or may be selected directly by the user.

Automatically Selected Four Quadrant Curve

When the Four Quadrant Curve option is first selected, a four quadrant curve will automatically be selected by Impulse. Impulse selects a four quadrant curve by doing the following:

1) The specific speed for the pump is estimated using the BEP calculated from the user-specified pump curve, and assuming that the pump includes only a single stage and the impeller is untrimmed.

2) Using the estimated specific speed from step 1, the four quadrant data set with the nearest specific speed is selected. By default Impulse will only choose a Preferred or Average data set.

3) The BEP, which is either specified directly by the user in the Pump Configuration window or calculated based on the user-specified pump curve, is used as the Dimensional Reference Point to dimensionalize the four quadrant curve

This automatically selected four quadrant curve will provide a reasonable estimate for most cases. However, it is advised that the user check the validity of Impulse's selection by using the considerations and procedure described below.

Considerations for Selecting a Four Quadrant Curve

The following recommendations from section 6.2 of Walters, Dahl, and Rogers, 2020 should be considered when selecting a four quadrant data set for the waterhammer analysis.

1) Comparison to the Manufacturer Curve

The manufacturer curve is measured data from the actual pump being used in the system, and thus will provide the most accurate prediction for the pump behavior. Though it cannot be used to predict behavior beyond the range of the measured data, it can be used for comparison to the four quadrant curve(s) for the steady state simulation. In some systems, the transient results can be highly sensitive to the steady state of the system, thus accuracy of the pump curve used in the steady state can be an indicator of accuracy during the transient. In addition, the final steady state for the system once the transient response has concluded can be compared as an indicator of how close the four quadrant curve will behave to the manufacturer curve during the transient.

2) Pump Specific Speed

As discussed in a previous topic on Four Quadrant Data Sets, it is typically a good practice to select a four quadrant data set based on the specific speed of the pump, as pumps with similar specific speeds will typically be constructed similarly. However, this does not always hold true. Thus, comparing multiple data sets at specific speeds close to the specific speed specified by the pump manufacturer is advised.

3) Dimensional Reference Point

As was previously discussed, the Four Quadrant Dimensional Reference Point can be defined as either the Best Efficiency Point (BEP) or the steady state operating point (SSOP) from the manufacturer's curve. Using the BEP will result in a different operating point during the steady state, but may provide a closer match for the final steady state using the manufacturer's curve. Using the SSOP will allow the steady state operating point to exactly match the manufacturer's curve, but may diverge widely from the final steady state operating point predicted by the manufacturer's curve due to distortions in the dimensionalized curve.

Procedure for Four Quadrant Data Set Selection

A generic procedure for creating a model and analyzing the results when using four quadrant data is given below:

1. Create a child scenario of the scenario that is defined using the manufacturer's curve

2. Open the Pump Properties window and select Four Quadrant Curve as the Performance Curve Used in Simulation

3. Choose User Selected and click Specify Model to launch the Specify Four Quadrant Model window

4. Choose the desired four quadrant data set from the drop-down list at the top of the window, and select Best Efficiency Point as the Dimensional Reference Point.

5. Select Ok to accept the changes

6. Clone the current scenario and change the Dimensional Reference Point to the Steady-State Operating Point in the cloned scenario

7. Repeat steps 1 - 6 for each data set to be analyzed

8. Run all of the newly created scenarios and compare the transient results. Note that frequently results will not differ greatly among scenarios. In that case, results are not very sensitive to the specific choice of four quadrant curve. In order to be most conservative, use the worst case predictions from among the various four quadrant scenarios for design purposes.

9. If the worst case predictions lead to excessive and/or expensive implications in design and/or operations, take a closer look at results. Compare each four quadrant pump curve to the manufacturer's curve at the initial steady state and, if applicable, the final steady state operating points to determine which scenario(s) appears to give the most accurate results. Criteria used to decide the most reliable four quadrant curve(s) include:

   1. Similarity of specific speed, N_S – when making this comparison, be sure you have an accurate N_S value for your pump

   2. Overall agreement between the manufacturer's curve (Standard Pump Curve) and dimensionalized four quadrant curve for head and power vs. flow rate in the normal zone of pump operation (Impulse compares these curves for you – review the comparisons)

   3. Give more weight the four quadrant curves which show closer agreement within the range of flow rates that the manufacturer tested the pump at (i.e. the range of flow rates that the pump curve is graphed for on the specification sheet)

   4. Give more weight to four quadrant curves which generate transient results similar to the Standard Pump Curve during time frames when the pump is operating in the normal zone of operation (positive flow, head and rotation)

   5. Give more weight to four quadrant curves Impulse categorizes as Preferred

   6. Good agreement in initial steady-state flow rates between SPC and four quadrant curve

   7. Good agreement in final steady-state flow rates between SPC and four quadrant curve – this not as important as agreement in initial steady-state flow rates, but should be considered

Note that there are several reasons why the engineer may choose to skip step 9 above. One reason would be if the transient results for the different scenarios show only minor differences as noted in Step 8. In that case it would be unnecessary to determine which scenario is most accurate, as any scenario will produce similar results. Another reason would be if the engineer's analysis involves a project where highly conservative results are desired, such as for a nuclear system, or another such instance where less accurate results would be costly (e.g., force installation of an expensive surge relief system that may not truly be needed). In that case the engineer

could choose to simply use the most extreme results for the analysis for their design purposes. Note this means that if different scenarios produce the worst-case pressures in different areas of the model, then the engineer may want to use the most extreme results (e.g., highest or lowest pressure) regardless of scenario, rather than only using the results from one scenario.

Four Quadrant Data Set Selection Example

Several walk-through examples describing the basics of four quadrant modeling can be found in AFT Impulse's walkthrough examples.

For a basic introduction to using four quadrant curves see the "Pump Trip With Backflow - Four Quadrant Modeling" example.

An advanced example on performing a sensitivity study to select a four quadrant curve is also available, titled "Selecting a Pump Four Quadrant Curve".

Related Topics

Four Quadrant Waterhammer Theory

The waterhammer theory when using four quadrant data in SuterSuter form follows a similar approach to the Standard Pump Curve Waterhammer Theory.

First, we know from the compatibility equations that the pressure change across the pump is related to the mass flow:

$$\Delta P = C_M - C_P + (B_M + B_P)\,\dot{m}$$

We also know that the dimensionless four quadrant parameters are related to the values used for the Dimensional Reference Point:

$$h = \frac{H}{H_{Ref}} \qquad v = \frac{Q}{Q_{Ref}}$$

From the definition of **h** we have a relationship between the curve $F_H(\Theta)$, speed and flow.

$$\Delta H = R_{Ref}\, h = H_{Ref} \left(\alpha^2 + v^2 \right) * F_H(\theta)$$

Multiplying through by **ρg** we can attain a similar expression in pressure terms:

$$\Delta P = P_{Ref} \left(\alpha^2 + v^2 \right) * F_H(\theta)$$

Similarly, there is a relationship for volumetric flow that can be converted to mass flow by multiplying by density:

$$Q = Q_{Ref}\, v \qquad \dot{m} = \dot{m}_{Ref}\, v$$

The previous equations can be combined with the compatibility equation for the pump:

$$C_P - C_M - \dot{m}_{Ref}\, v \left(B_M + B_P \right) + P_{Ref} \left(\alpha^2 + v^2 \right) * F_H(\theta) = 0$$

This equation contains three potential unknowns - speed (**α**), flow (**v**), and **F_H**(**Θ**). **F_H** and **Θ** are related via tabulated data. The value of **Θ** is related to the speed and flow, bringing the unknown variables down to just speed and flow:

$$\theta = \tan^{-1}\left(\frac{\alpha}{v} \right)$$

As with the Standard Pump Curve, for a fixed speed there is only one unknown - flow. The equations and tabulated data can be iterated on to find a solution for flow and therefore pressure rise across the pump.

During a pump trip or startup, when neither speed nor flow are known, additional equations are required. The necessary torque balance terms are described in Transient - Inertial Models resulting in the below equation for speed change:

$$\omega_{new} = \omega_{old} + \frac{\Delta t}{I_{total}} \left(\frac{T_{driver,old} + T_{driver,new}}{2} - \frac{T_{pump,old} + T_{pump,new}}{2} \right)$$

It is helpful to modify this equation so that it is directly applicable in four quadrant terms. The following relations are required:

$$\omega = \frac{2\,\pi}{60} N \qquad N = N_{Ref}\, \alpha \qquad T = T_{Ref}\, \beta$$

resulting in the following:

$$\alpha_{new} = \alpha_{old} + \frac{30\,\Delta t}{I_{total}\,\pi\,N_{Ref}}\left(\frac{T_{driver,old} + T_{driver,new}}{2} - T_{Ref}\frac{\beta_{pump,old} + \beta_{pump,new}}{2}\right)$$

This equation represents the full torque balance with unknown driver torque, pump torque, and pump speed in four quadrant form. For a pump trip, the driver torque is zero leaving only two unknowns. For a startup, the driver torque vs speed must be specified. Like the head term above, the dimensionless torque **β** can be solved for in terms of speed (**α**), flow (**v**), and **F_B** (**Θ**).

$$\beta = (\alpha^2 + v^2) * F_B(\theta)$$

These equations can then be solved for speed, flow, torque and head.

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 for pump performance - even if the fluid reference density is the same as water. AFT Impulse employs the methods described in ANSI/HI 9.6.7-2021 to correct for losses in performance due to viscosity.

Correction factors referred to as C_H, C_Q and C_E are defined. These correction factors are defined as:

$$C_H = \frac{H_{viscous}}{H_{water}} \qquad C_Q = \frac{Q_{viscous}}{Q_{water}} \qquad C_E = \frac{\eta_{viscous}}{\eta_{water}}$$

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

The Positive Displacement pump option allows users to model the behavior (either steady or periodic flow) of a reciprocating positive displacement pump.

Parameter

- **Volumetric Flow Rate** - Fix the volumetric flow rate through the pump.

- **Mass Flow Rate** - Fix the mass flow rate through the pump.

Flowrate

There are three ways to define the flowrate:

- **Steady** - The flowrate will be fixed at the specified value throughout the transient. This is a good approximation for many Positive Displacement Pumps.

- **Periodic (simple)** - Some Positive Displacement Pumps have a distinct pulsing behavior depending on the number of Heads and speed of the pump. This can be defined here, with a preview of the imposed flow being shown on the right of the tab.

- **Periodic (detailed)** - If detailed information is known about the positive displacement pump, a more accurate periodic behavior can be defined by providing the bore diameter, pump stroke length, and connecting rod length of pump slider crank.
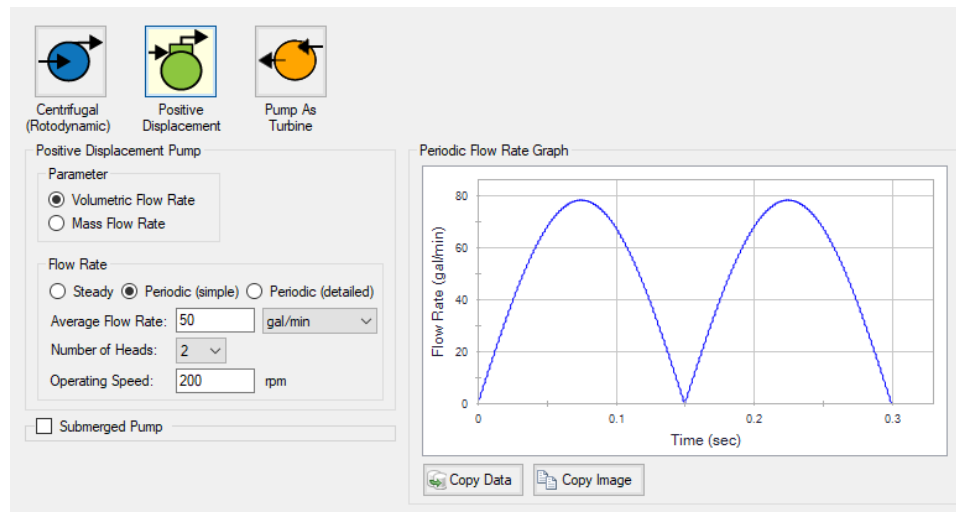
Figure 1: Simple Periodic Positive Displacement Pump
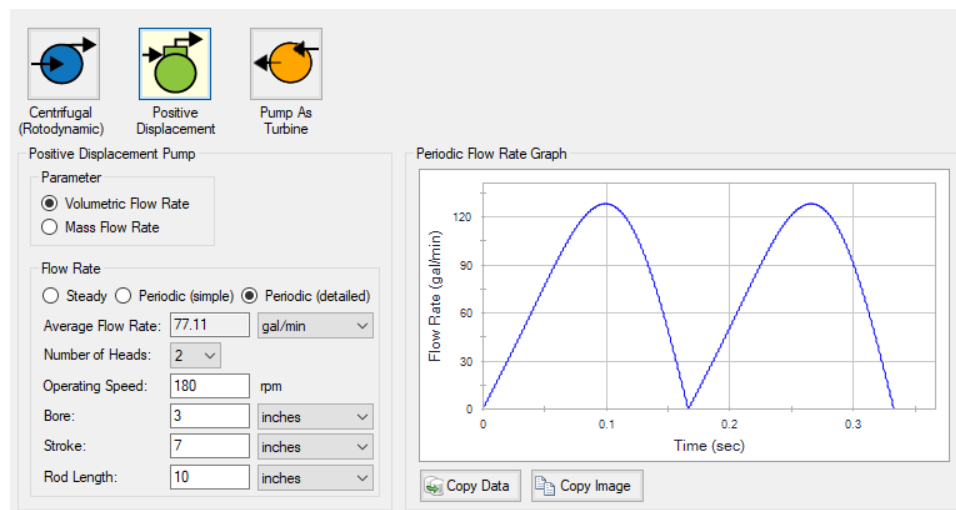
Figure 2: Detailed Periodic Positive Displacement Pump

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See Submerged Pump.

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 Curve frame once the data has been defined.
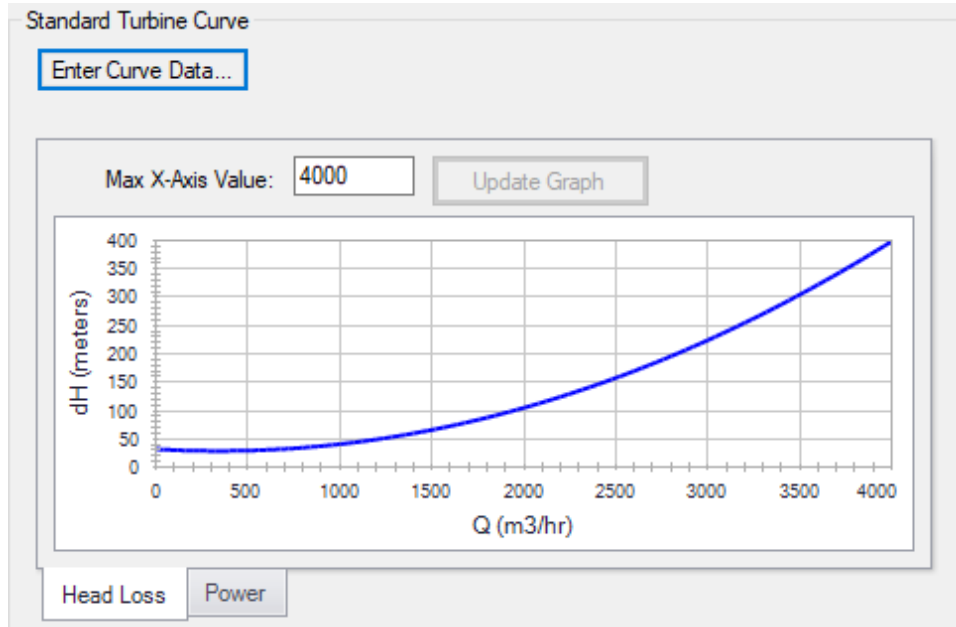
Figure 1: Standard Turbine Curve

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.
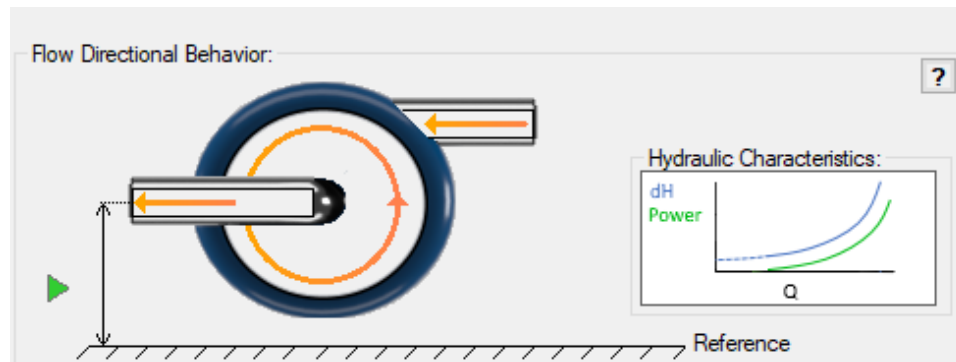
Figure 2: Flow Directional Behavior image for an example PAT.

Related Topics

Related Blogs

It's a Pump! It's a Turbine! It's a Pump as Turbine!

Pump As Turbine 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.
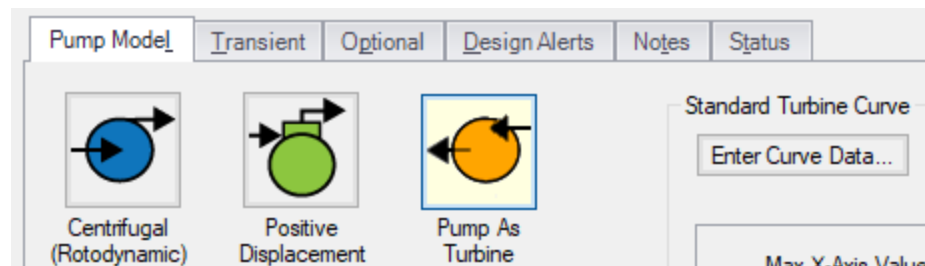
Figure 1: Enter Curve Data

Turbine Data

The first tab of the PAT configuration defines the PAT's most fundamental behavior - the turbine curves.

Figure 2: PAT Data

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.

Related Topics

Related Blogs

It's a Pump! It's a Turbine! It's a Pump as Turbine!

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 will likely see 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.
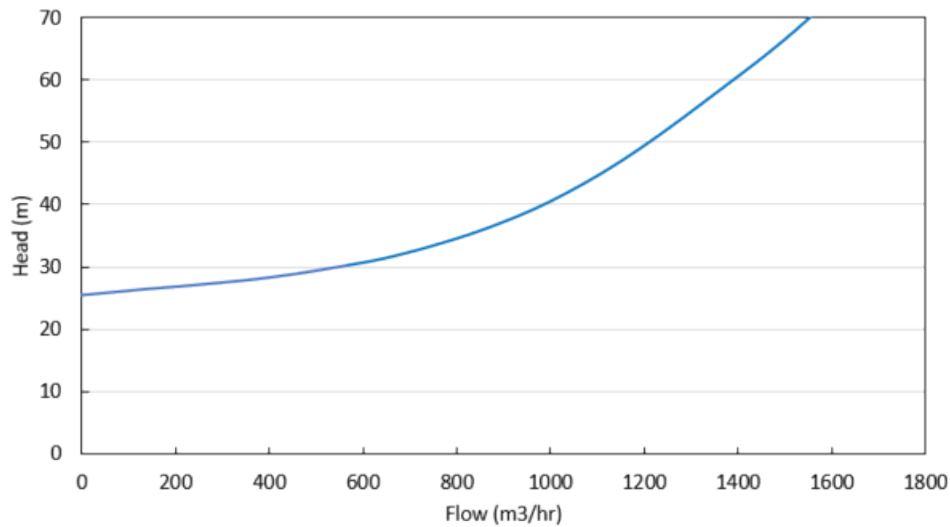
Figure 1: Example Turbine Performance Curve

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.
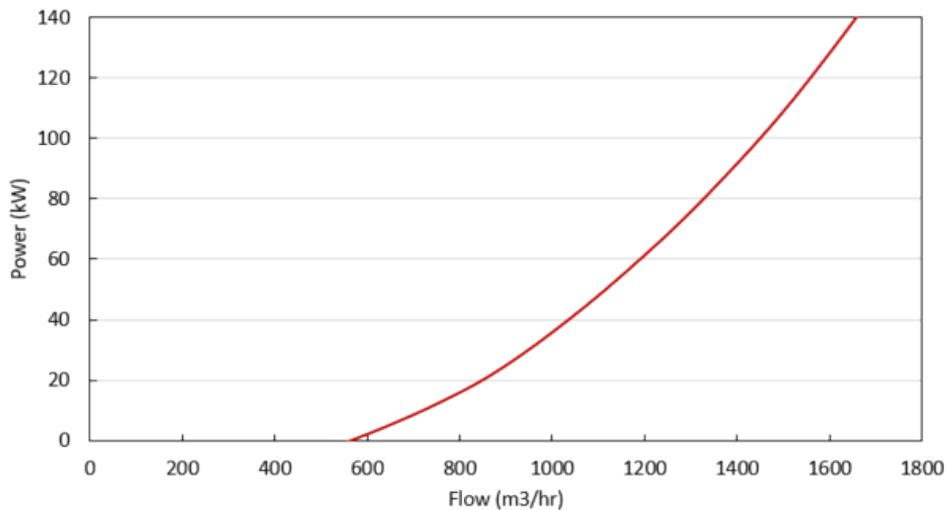
Figure 2: Example Turbine Power Curve

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.
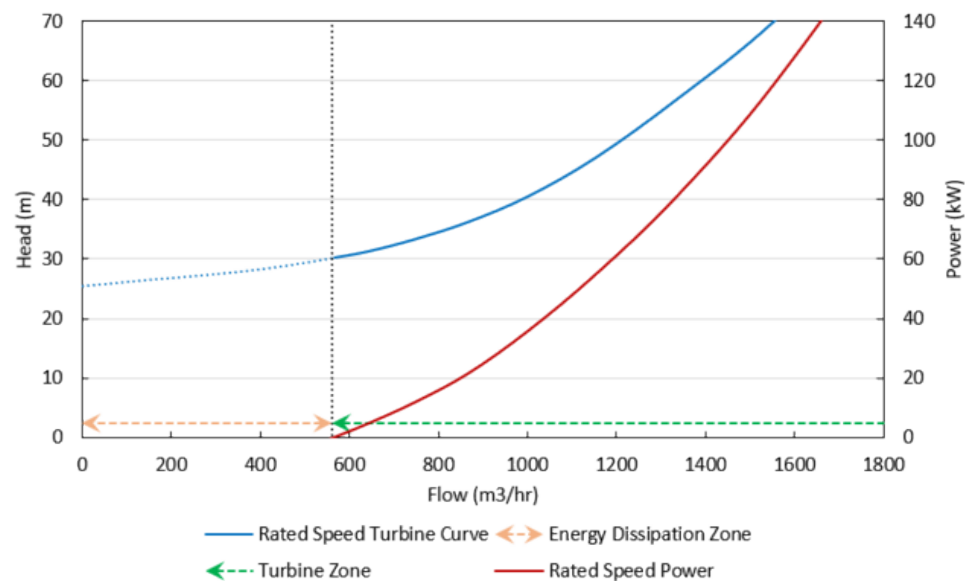
Figure 3: General turbine operating conditions

Related Topics

Related Blogs

It's a Pump! It's a Turbine! It's a Pump as Turbine!

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. When the free-spinning impeller

reaches a steady-state speed, no torque is imparted on the impeller by the fluid. This speed is called the runaway speed.
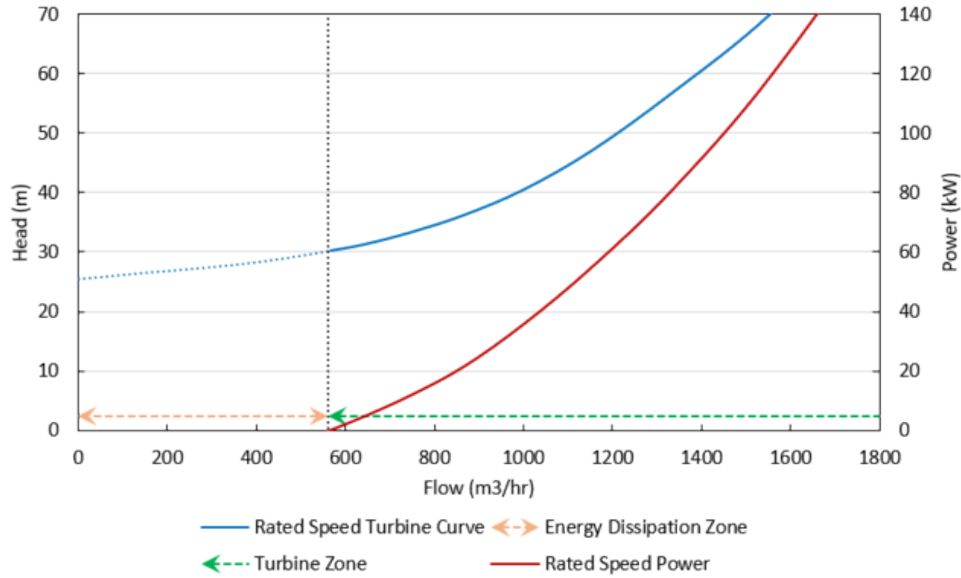
Figure 1: General Turbine Operating Conditions

The pressure loss of a free-spinning PAT is determined by the Turbine Performance curve, adjusted by the pump 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 following equation. Here time step *i* is the current time step, *i+1* is the next time step, *N* is the impeller speed (in RPM), *P* is the power imparted on the impeller by the fluid, and *I* is the total rotating inertia of the PAT impeller and shaft.

$$N_{i+1} = N_i - \frac{900\,\Delta t}{2\,g\,\pi^2\,I}\left(\frac{P_i}{N_i} + \frac{P_{i+1}}{N_{i+1}}\right)$$

The power term is always dictated by the Turbine Power Curve, adjusted using the Affinity Laws for speed. When the decoupled PAT nears runaway speed, the PAT approaches the No Load point (flow rate where the power curve is equal to zero) and power approaches zero. Thus, the speed does not change from time step to time step.

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 Impulse 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.
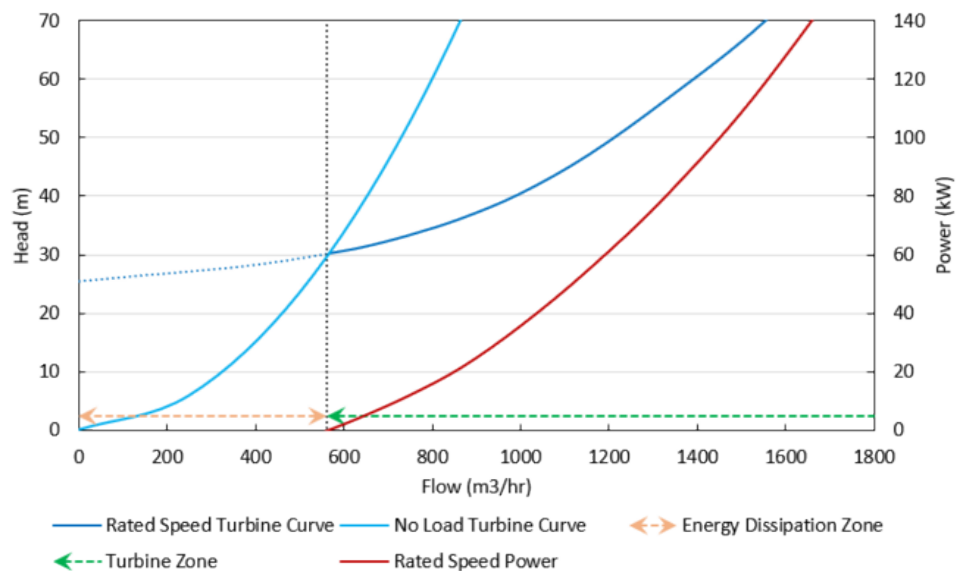
Figure 2: Example No Load Curve showing general turbine operating conditions.

Related Topics

Related Blogs

[It's a Pump! It's a Turbine! It's a Pump as Turbine!](#)

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. In Impulse, the PAT can never recouple after becoming decoupled.

The first transition from coupled to decoupled is referred to as [Full Load Rejection](#) and can be set as a transient event in the Transient tab of the Pump Properties window.
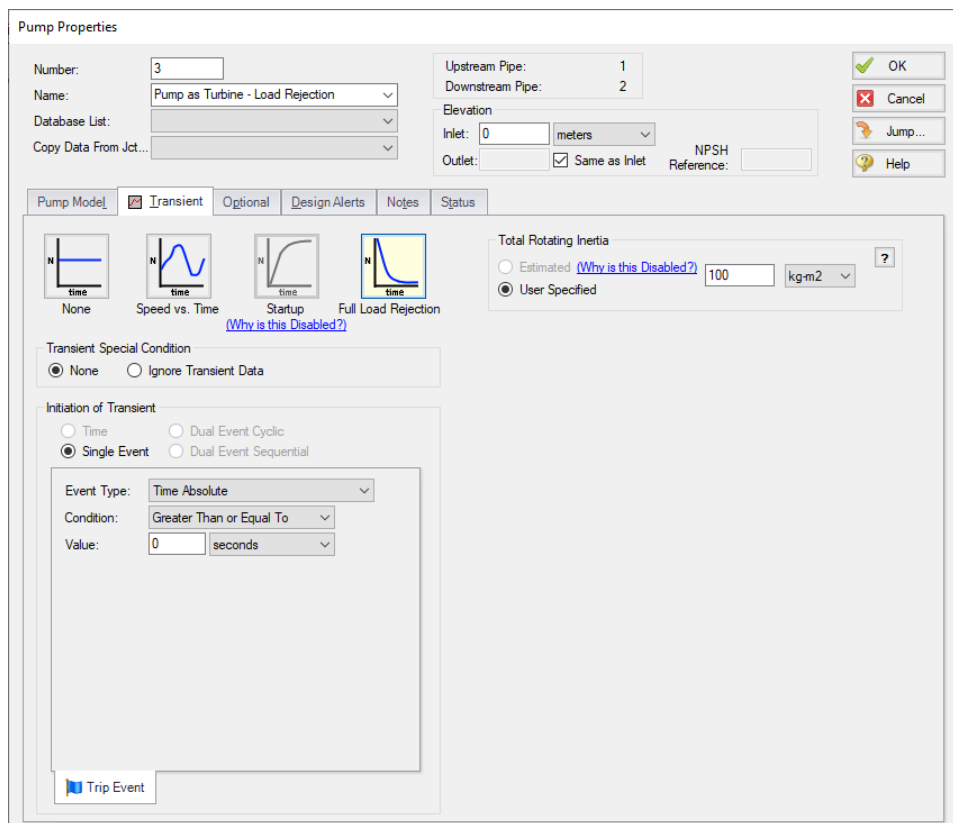
Figure 1: Transient tab of the Pump Properties window for a PAT showing the Full Load Rejection option.

When this transient event is initiated, the PAT is immediately uncoupled from its motor, the speed is allowed to change as dictated by the fluid, and the PAT speed is determined for each time step using the below equation. Here time step i is the current time step, i+1 is the next time step, N is the impeller speed (in RPM), P is the power imparted on the impeller by the fluid, and I is the total rotating inertia of the PAT impeller and shaft.

$$N_{i+1} = N_i - \frac{900\,\Delta t}{2\,g\,\pi^2\,I}\left(\frac{P_i}{N_i} + \frac{P_{i+1}}{N_{i+1}}\right)$$

This equation is the same equation used to predict PAT speed when decoupled.

During a full load rejection, the PAT will rapidly spin up as the generator is no longer resisting the torque applied on the impeller by the fluid. The flow rate through the PAT also slows down as the PAT shifts from operating in the Turbine Zone to operating at the No Load point.

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 pump, the pump will immediately become uncoupled from the motor and become free-spinning with its speed determined by the equation described above.

Related Topics

Related Blogs

It's a Pump! It's a Turbine! It's a Pump as Turbine!

Affinity Laws

Pumps can operate at variable speeds during either steady operation or a transient event. Affinity laws (also called homologous laws) allow an estimation of the pump performance at speeds other than the design speed. For the Centrifugal (Rotodynamic) Pump model, these Affinity laws are used to model a variable speed pump during steady operation and to model the performance of a pump during any of the available transient models. For the Pump as Turbine model, these affinity laws are used to determine power generation or head loss while decoupled.

To a somewhat lesser degree, impeller diameter changes 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. This

data should be used if available. When not available, 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 N2.

- **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 -**

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

Application to Transients

During a transient event, the speed of the pump may change. In such cases, the affinity laws are used to adjust the head and power curves appropriately for the new speed. This is a process that often requires additional iteration.

Related Topics

Related Blogs

To Affinity and Beyond!

Submerged Pump

A pump junction can be defined as Submerged, meaning the process fluid directly enters the pump without any entrance piping. This definition is convenient to avoid the need of modeling suction piping when the suction HGL or pressure is a known and fixed value. The submerged pump feature is only available for the Centrifugal and Positive Displacement models.

There are three options to choose from to define a submerged pump: Head (HGL), Pressure, and Surface Conditions.

Head (HGL)

With the Heat (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.
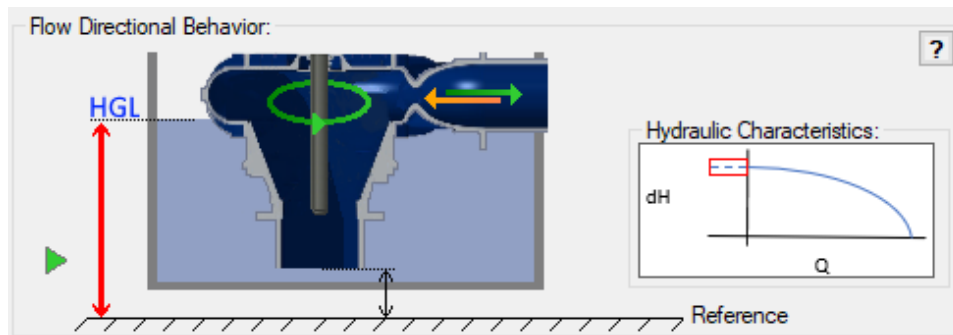
Figure 1: Diagram of Submerged Pump, Liquid Surface HGL input

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.
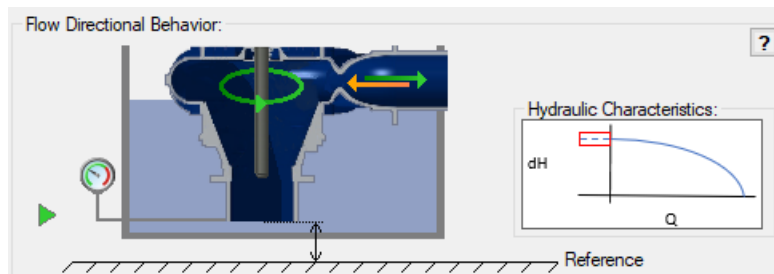
Figure 2: Diagram of Submerged Pump, Pressure input

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

Related Blogs

Back to the Basics: Calculating and Defining Head

What Does "Head (HGL)" Mean for Submerged Pumps and Exit Pressures?

Pump Transients

The Pump junction offers several transient event options on the Transient tab of the Pipe Properties window. Depending on the pump configuration, the available transient events will change. Each transient event is described below including: the situation in which it is available, the required inputs, and how to define the transient.

Related Blogs

Where to Start with Pump Startups

Start your engines! Modeling an Inertial Pump Startup

Speed vs. Time

Pump configurations other than pumps defined with a fixed setpoint allow the direct specification of Speed vs. Time. This is not an Inertial Model - it is possible to specify a speed change that would be unrealistic without an external force on the Motor or Pump.

Transient Data

The Speed vs. Time data must be directly defined in Transient Data. The speed of the pump will be forced to meet this data at each time step of the transient simulation, with the pump curves being adjusted every time step according to the affinity laws.
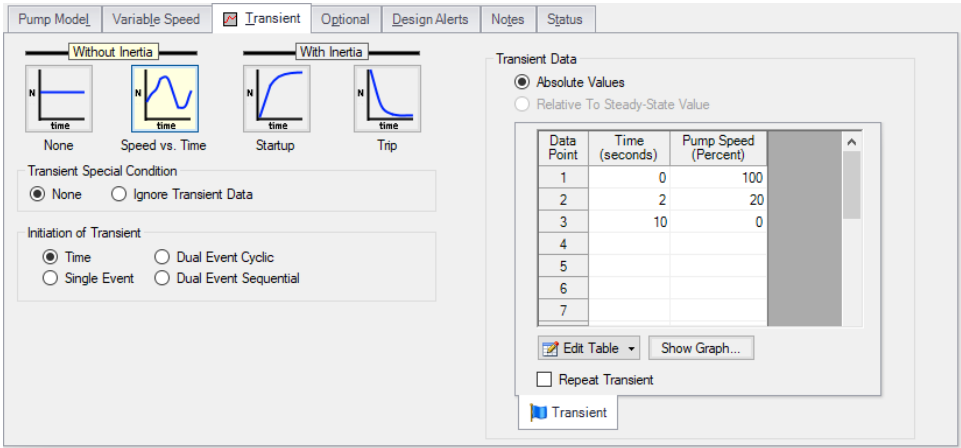
Figure 1: A Speed vs. Time transient

Related Topics

Related Blogs

Where to Start with Pump Startups

Start your engines! Modeling an Inertial Pump Startup

Setpoint vs. Time

If modeling a Controlled Pump (Variable Speed) or a Positive Displacement Pump with a steady setpoint, users can define a transient setpoint vs time profile. This replaces the Speed vs. Time option, as defining both the speed and the setpoint would over constrain the system.

Transient Data

The Setpoint vs. Time data must be directly defined in Transient Data. The selected parameter will be forced to meet this data at each time step of the transient simulation, with the pump curves being adjusted every time step following the affinity laws. The resulting speed variation over time will be visible in the output.
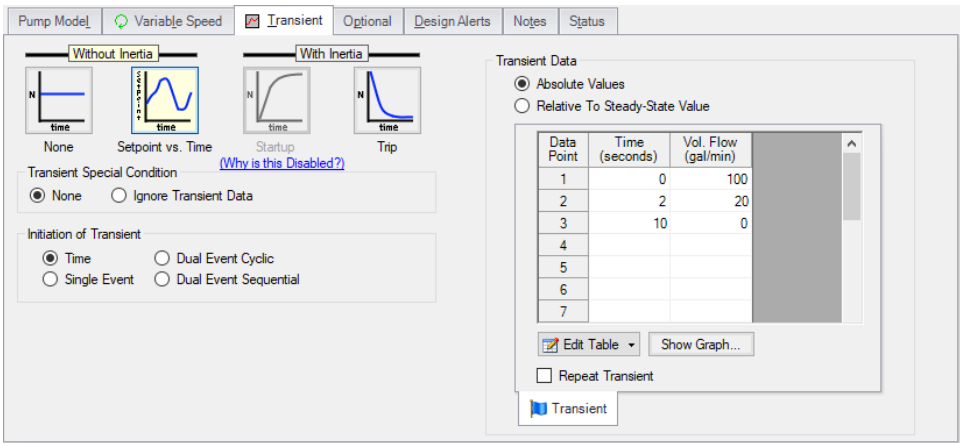
Figure 1: A Setpoint vs. Time transient

Related Topics

Startup with Inertia

The Startup with Inertia transient is the first of two inertial transients that can be defined for the Centrifugal (Rotodynamic) Pump. Modeling an inertial pump startup simulates a pump which is connected to a motor that is initially supplying no torque, and is suddenly switched on. The inertial startup is an *uncontrolled* start - meaning the torque characteristics of the motor alone will drive the speed change behavior. The inertial startup does not capture a controlled start, where there may be complex equipment such as a PID-controlled VFD slowing down the startup of the pump, and cannot be used in combination with the Variable Speed Controlled Setpoint options for a pump.

Inertial Characteristics

The **total rotating inertia** of the pump/motor system must be available. **This must include all rotating components - the motor rotor, shaft, impeller, entrained liquid, and any other connected components.** This value can be estimated or directly specified.

Transient Data

To solve the torque balance equations, the torque on the shaft at any given speed must be known. The torque exerted by the pump on the shaft (that is, from the fluid) can be determined with the Standard Pump Curve power curve. The torque characteristics of the device driving the pump must be defined by the user. Typically, this is an induction motor. It could also be a synchronous motor, a DC motor, or another device entirely such as a shaft-connected turbine or diesel engine.

Induction motors generally follow one of a few standard torque vs speed profiles. Shown below is torque profile of a motor showing typical NEMA Design B characteristics. Also shown is the torque behavior of the pump, as determined with the affinity laws.

Note: The pump torque characteristics can be considered ideal for the user defined power curve. A real pump will have higher torque during startup due to friction and other startup effects. These are not represented here, as including them would make the model and required input much more complicated for little benefit.
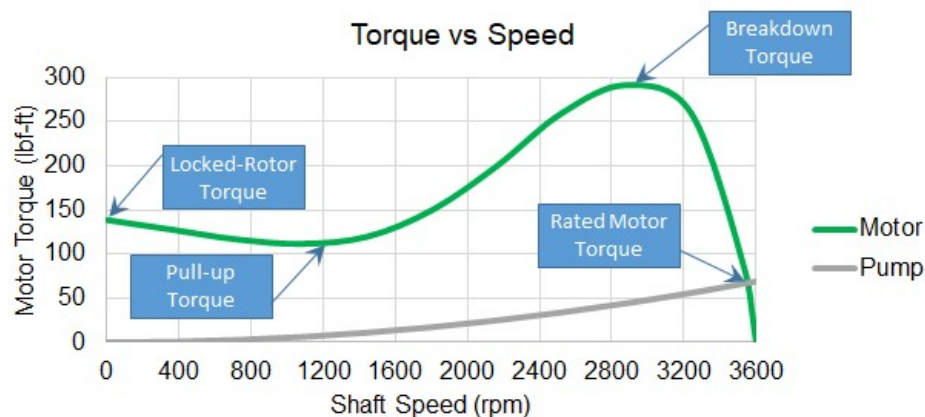
Figure 1: A typical NEMA Design B motor torque curve and ideal pump torque curve.

AFT Impulse allows the definition of the Motor or Driver Torque curve vs. relative or absolute speeds.
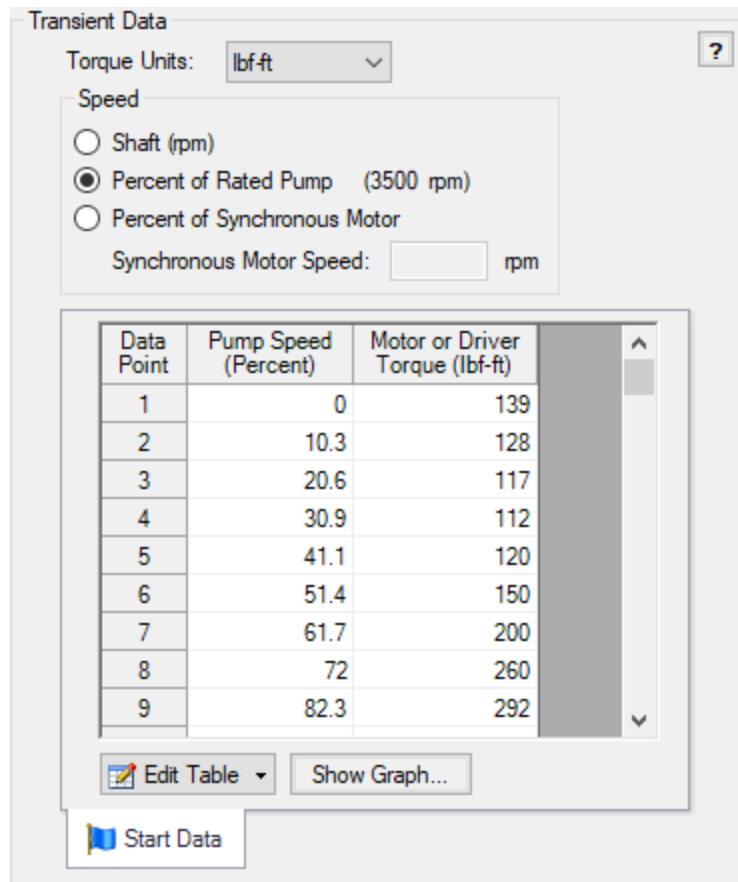
Figure 2: Motor or Driver Torque curve entered as Percent of Rated Pump Speed vs Torque.

It is important to note that **Rated Pump Speed, Rated Motor Speed, and Synchronous Motor Speed are different values**. If the motor is perfectly matched to the pump, the two rated speeds may be the same. If entering the data as Shaft Speed (rpm), then no concern over a rated speed is required. However, they are important for relative (%) data entry.

To clarify the difference between these options consider the following values (which correspond to Figure 1):

- **Rated Pump Speed - 3500 rpm** - Rated by the pump manufacturer
- **Rated Motor Speed - 3540 rpm** - Rated by the motor manufacturer, the speed at which the motor is designed to handle its rated torque
- **Synchronous Motor Speed - 3600 rpm** - A characteristic of most electric motors - the speed at which the electrical field rotates. Available from the motor manufacturer, and often the torque characteristics are plotted against percent of synchronous speed.
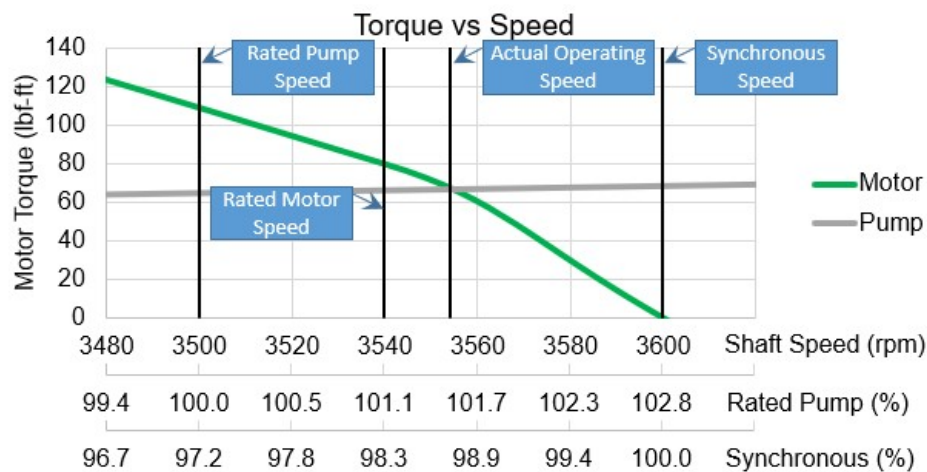
Figure 3: Various speeds for the Figure 1 characteristics. Note that the corresponding torque value for 100% Synchronous and 100% Rated Pump are different.

Of interest in Figure 3 is that not only are there three different commonly referenced speed references, when placed into a system the actual operating speed is not any of these values! Why is this? The actual operating point depends on how well matched the motor is to the pump and its operating point. In this instance, the motor is providing enough torque that the rated pump speed is exceeded. Additionally, the system is configured in such a way that the resistive torque is lower than the motor's rated torque at rated motor speed. This causes the motor to speed up further until the pump torque and motor torque match.

Special Condition

The pump will need to have its special condition set to Pump Off With Flow Through to model a Startup.

Related Topics

Related Blogs

Where to Start with Pump Startups

Start your engines! Modeling an Inertial Pump Startup

Trip with Inertia

The Trip with Inertia transient is the second of two inertial transients that can be defined for the Centrifugal (Rotodynamic) Pump. Modeling an inertial pump trip allows the representation of a situation where a pump is operating normally, but all motor or driver torque is instantaneously lost. This trip is uncontrolled - meaning that the torque characteristics of the pump alone will drive the speed change behavior.

Inertial Characteristics

The total rotating inertia of the pump/motor system must be available. **This must include all rotating components - the motor rotor, shaft, impeller, entrained liquid, and any other connected components.** This value can be estimated or directly specified.

Transient Data

No further data is required from the user to define a trip.

To solve the torque balance equations, the torque on the shaft at any given speed must be known. The torque exerted by the pump on the shaft (that is, from the fluid) can be determined with the Standard Pump Curve, which is already defined. The torque of the motor or driver is assumed to instantly fall to zero.

Related Topics

Inertial Models Detailed Theory

Any device has an inertial resistance to change in motion. Pumps are no different - in order to change their speed, a torque must be applied. Instantaneous speed change would require infinite torque. When modeling fluid transients, a more realistic expectation of pressures and flows can be determined when taking this into account. As expected, accounting for inertia requires some additional information.

Inertial Model Theory

Pumps in AFT Impulse are considered completely homologous. This means that the pump curves used are considered to be scalable to any speed with the affinity laws - real pumps will deviate somewhat from this behavior. Accounting for this behavior would require much more data for little benefit. Fortunately, this simplification makes approaching something like an inertial model much more straightforward.

Due to the above simplification we can use standard relationships for power, torque, rotational speed, and rotational acceleration.

$$P_{shaft} = T_{shaft}\, \omega$$

$$T_{shaft} = I_{total}\frac{d\omega}{dt}$$

where

P_{shaft} = power required to drive the shaft at a given torque and speed

T_{shaft} = the torque applied to the impeller shaft.

ω = the angular speed of the shaft, in rad/s

I_{total} = the **total rotational inertia** of the assembly. This must include the inertia of the pump impeller, entrained fluid, the shaft, and the motor rotor - all of the connected rotating components.

The shaft is acted on by the driver (typically a motor), and the pump (the impeller). Positive torque and speed are defined to be in the same direction, with positive relating to normal pumping conditions.

$$T_{shaft} = T_{driver} - T_{pump}$$

Under steady conditions, there is no torque imbalance meaning that T_{shaft} and angular acceleration are zero.

There are a couple of common ways the torque becomes imbalanced - when starting up a pump a large torque is applied by the driver, while the pump torque is initially zero. When tripping a pump, the driver torque drops to zero. In both cases the shaft torque is not zero, meaning the speed must change.

We can integrate the torque balance equation over a time step to get a form that is useable in numerical simulation. Torque is a function of time, so we consider the average torque over the time step.

$$\int_{0}^{\Delta t} (T_{driver} - T_{pump})\, dt = \int_{\omega_{old}}^{\omega_{new}} I_{total}\, d\omega$$

$$\left(\frac{T_{driver,old} + T_{driver,new}}{2} - \frac{T_{pump,old} + T_{pump,new}}{2}\right) \Delta t = I_{total}(\omega_{new} - \omega_{old})$$

Solving for the new speed:

$$\omega_{new} = \omega_{old} + \frac{\Delta t}{I_{total}} \left(\frac{T_{driver,old} + T_{driver,new}}{2} - \frac{T_{pump,old} + T_{pump,new}}{2} \right)$$

There are three unknowns in this equation: ω_{new}, $T_{driver,new}$, and $T_{pump,new}$. We need two more equations for a solution.

To represent the driver, we need raw data relating speed ω and torque T_{driver}.

This leaves two unknowns: ω_{new} and $T_{pump,new}$. A function (curve fit) relating power and flow is required input - with the affinity laws, this is further a function of speed.

$$P = f(Q, \omega)$$

We know that the torque is related to power by a factor of speed from the first equation above, which means torque is also a function of speed and flow.

$$T = f(Q, \omega)$$

We have a relationship for ω_{new} and $T_{pump,new}$, but have introduced one more variable - flow. This is solved for with the pump head curve and the Method of Characteristics, as its magnitude depends on the system as well.

With iteration, we can now determine a new speed and new torque for the new time step. The only additional input required is the behavior of T_{driver} and the value of I_{total}. A pump trip is defined as an event where all driver torque is immediately set to zero. A pump startup is represented as an "uncontrolled" start - driver speed vs torque provides the information required.

Total Rotating Inertia

To model an inertial startup or trip, the total inertia of the rotating system must be known. **This must include all rotating components - the motor rotor, shaft, impeller, entrained liquid, and any other connected components.**

Obtaining this information is often quite difficult. Determining the rotational inertia of a single material component with a complex shape like an impeller is challenging enough on its own, and this is only part of the puzzle. Instead of calculating a value analytically, it is possible to use test data to determine a value for inertia. The best test data will come directly from the pump/motor assembly of interest, but again, this can be difficult to get.

An alternative method is to use statistical analysis of industry pumps and motor to determine a likely inertia value for a given pump and motor. Thorley (2004) presents just such analysis of pumps and motors across industries, and AFT Impulse employs this estimate when the appropriate option is selected.
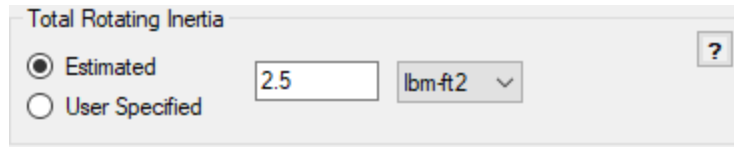
Figure 1: Estimated Total Rotating Inertia

For pumps, Thorley presents the following correlation derived from a linear regression of 284 data points (the formula has been adjusted to use rpm instead of 1,000's of rpm):

$$I_{pump} = 1.5 \cdot 10^7 \left(\frac{P}{N^3} \right)^{0.9556}$$

where

I_{pump} = The rotational inertia of only the pump in kg-m^2

P = shaft power at rated conditions and BEP in kW. To determine the power at rated conditions, the system fluid must be defined and a Standard Pump Curve with Power/Efficiency data must be available.

N = shaft speed in rpm

Thorley notes that this correlation has a very good correlation coefficient of 0.96. However, he further notes that the actual values of inertia range from +100 to -50% from the predicted value.

Note: Thorley presents two correlations - the one above is applicable to most pumps, however, one type of pump from a particular manufacturer in the study followed a slightly different trend. This is not presented here, and AFT Impulse uses the above correlation in its estimate.

For motors, Thorley presents the following, from 272 data points (the formula has been adjusted to use rpm instead of 1,000's of rpm):

$$I_{motor} = 118 \left(\frac{P}{N} \right)^{1.48}$$

where

I_{motor} = The rotational inertia of only the motor in kg-m^2

P = shaft power at rated conditions and BEP in kW

N = shaft speed in rpm

Similar to pumps, this has a good correlation of 0.97 with a similar spread.

The total rotational inertia is then considered to be the combination of the two estimates:

$$I_{total} = I_{pump} + I_{motor}$$

Note: Remember that this value is **only an estimate**. As Thorley notes, there is significant spread in the statistical data. If pump speed change is a critical parameter, total rotational inertia is also a critical parameter. **In such cases, a sensitivity study should be completed by doubling and halving the estimated inertia value.**

Related Topics

Full Load Rejection

The Pump as Turbine (PAT) model has one unique transient definition: a Full Load Rejection. This option is shown in the Transient tab of the Pump Properties window.
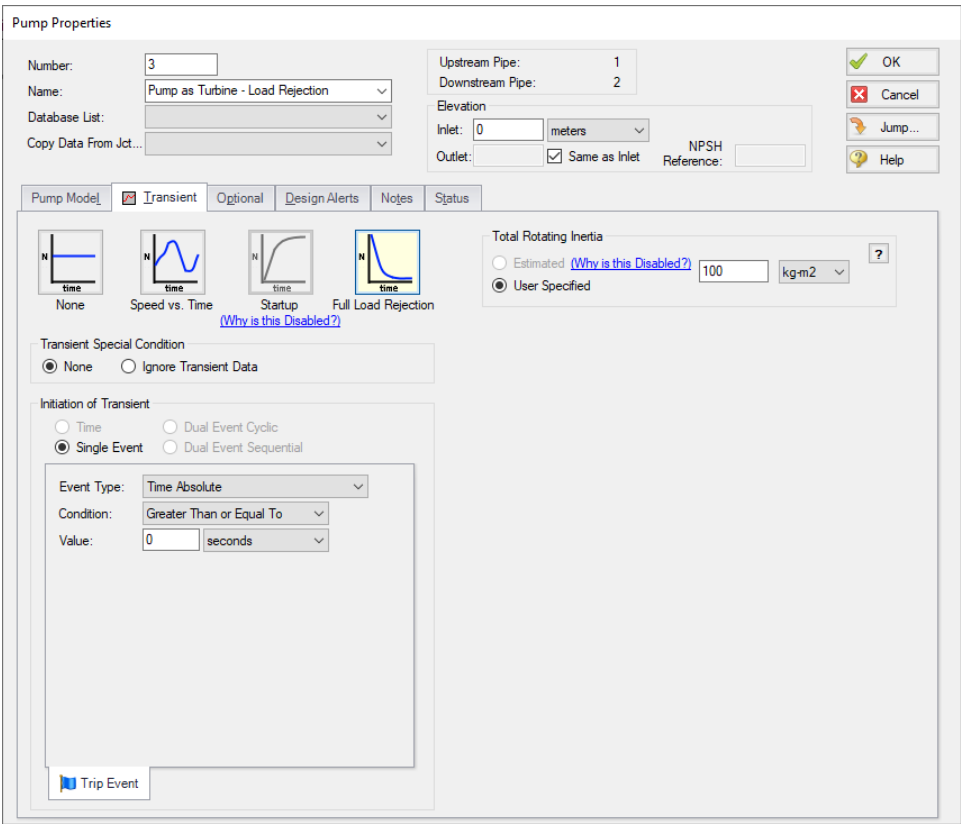
Figure 1: Transient tab of the Pump Properties window for a PAT showing the Full Load Rejection option.

During this event, the PAT is decoupled from its motor at the specified Transient Initiator, and the PAT undergoes decoupling. The PAT then spins with the fluid until it reaches runaway speed.

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 four different methods for the valve to open or close. There are also four preset valve profiles with a fixed combination of these methods that the user can choose from. These options are listed below.

The four methods by which the valve can open or close 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.
- **Time** - The valve opens or closes according to a Cv vs. Time profile as the opening or closing conditions are met. The Cv vs. Time profiles are entered on the Transient tab, as needed.
- **Pressure** - The valve passively opens or closes according to a Cv vs. Pressure profile as the opening or closing conditions are met. The Cv vs. Pressure profiles are based on the Valve Setpoints and the data entered in the Loss Model tab. Maximum Cv vs. Time rate limits can be applied.
- **Never** - The valve never closes. This method cannot be applied to the valve opening.

The four preset valve profiles are:

- **Passive** - The valve is set to open and close based on pressure setpoints.
- **Pilot Operated** - The valve is set to open based on time and close based on pressure setpoints. Users must define an Opening Transient in the Transient tab.
- **Rupture Disk** - The valve is set to open instantly when the set pressure is exceeded and never close.
- **Surge Anticipator** - The valve is set to open and close based on pressure setpoints. This

profile also sets the valve to use remote sensing with high and low pressure setpoints and with opening and closing Cv vs. Time rate limits.

The relief valve also has a General profile. This profile allows the user to customize the method by which the valve opens and closes. The only restriction placed on combinations of opening and closing methods is that if the valve opens based on pressure, it can only close based on pressure.

Finally, it is important to note that if the valve opens based on time and closes based on pressure, the valve cannot open faster than pressure at the valve allows. If the opening transient were to open the valve to a Cv larger than what the pressure at the relief valve dictates, the valve would immediately start to close. Thus, a time opening transient is effectively rate limited to what the relief valve inlet pressure dictates.

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 and Low 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.

Note: The Relief Valve junction operates exclusively using stagnation pressure. All setpoints are defined as stagnation pressure and all calculations are based on stagnation pressure. In most cases, the assumption that stagnation pressure is equal to static pressure is valid.

Pilot Operation, Remote Sensing, and Surge Anticipation

The Relief Valve junction gives users the ability to model Pilot Operated Relief Valves (PORVs), Remote Sensing Relief valves, and Surge Anticipation Relief Valves. The PORV is enabled with the Pilot Operated preset profile, while the other options are available anytime the valve operates as a Passive pressure/pressure valve.

A PORV is a relief valve that can more rapidly respond to pressure surges. This rapid response is accomplished with two changes from a passive pressure relief valve. First, the pressure on both sides of the PORV's plug is maintained at the upstream pressure via a pressure sense line (see Figure 1). Since the area above the plug is larger than the area below the plug, the net force on the plug acts to keep the valve closed regardless of pressure. Second, the valve is controlled by a Pilot. When the set pressure is reached upstream, the pilot acts to vent the dome of the valve (above the plug), rapidly changing the net force on the plug and opening the relief valve.
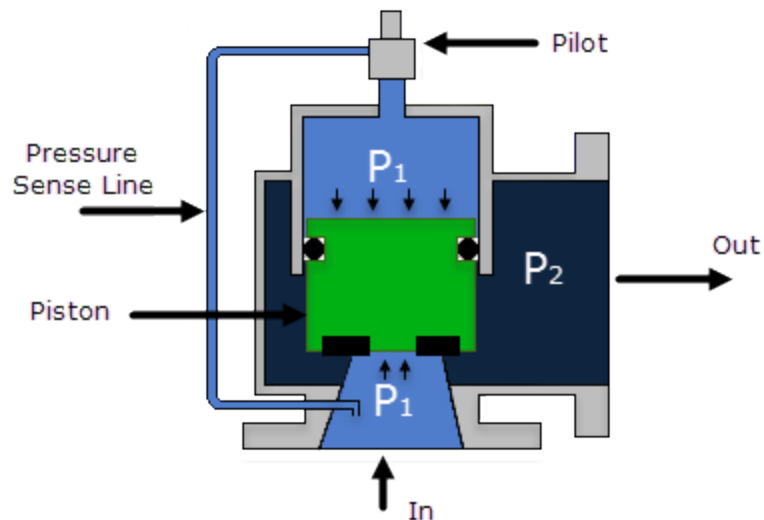
Figure 1: Pilot Operated Relief Valve Schematic

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. 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. The degree to which the relief valve is open is still determined by the pressure at the valve, but the pressure wave will be immediately relieved by the valve instead of first having to open the valve.

A surge Anticipation Relief Valve is a relief valve that opens when the pressure at a specified remote location in the system drops below a low pressure set point. These valves are typically paired with a pump and help mitigate high-pressure surges that are preceded by a low-pressure front. When the remote location's pressure falls below the Low Pressure setpoint, the low-pressure pilot partially vents the dome of the valve above the plug. Then, if the remote location's pressure rises above the High Pressure setpoint, the Surge Anticipation Relief Valve will behave the same as the PORV, opening based on the pressure at the relief valve inlet.

For the Surge Anticipation Relief Valve, the valve will only close when one of two conditions is met. The first condition is when the pressure at the valve falls back below the blowdown pressure and the valve is reset. The second condition is when the pressure rises above the High Pressure setpoint, the valve opens based on pressure, and then the pressure falls again, closing and resetting the valve. Effectively, if the remote pressure rises above the High Pressure Setpoint, the valve will behave as if it is a PORV and will not return to Surge Anticipation until the valve closes again.

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/Time Profiles**
  - **Cv (Constant)** - The relief valve instantly changes from closed to a fixed Cv.
  - **Kv (Constant)** - The relief valve instantly changes from closed to a fixed Kv.
  - **CdA (Constant)** - The relief valve instantly changes from closed to a fixed CdA. This CdA is defined as the product of an Orifice Effective Area (A) and a Discharge Coefficient (Cd).
    - The Orifice Effective Area can come from an API 526 letter designation or it can be User Specified.
    - The Discharge Coefficient must always be specified by the user. The value is typically specified by the manufacturer.
  - **K Factor** - The relief valve instantly changes from closed to a fixed K Factor.
- **Pressure Profiles**
  - **Cv (Variable)** - The relief valve moves between the closed and open states with the Cv varying between 0 and the Fully Open Cv value.
  - **Kv (Variable)** - The relief valve moves between the closed and open states with the Kv varying between 0 and the Fully Open Kv value.

The Variable Data option for the Relief Valve Loss Model allows users to model the valve losses based on where the valve is between fully open and fully closed. The Linear based on setpoints option has Impulse determine those losses solely based on the setpoints the user has specified. If the user knows enough about the valve to describe a detailed loss profile, they can select the Non-linear option. The first data point must have a loss value of 0 at the Blowdown pressure. The

final data point must have a loss value equal to the Fully Open loss value, at the Overpresure. The entered data is interpolated during the transient simulation.

Users can also define Rate Limits for valves that open or close based on pressure. These limits allow the user to restrict how fast the valve can change position. Some relief valves may have hardware in place to limit valve movement, other relief valves may have enough mass that their inertia prevents them from immediately responding to pressure changes.

Transient Data

Transient Cv, Kv, or K Factor vs. Time data can be entered on the Transient tab when the valve opens or closes based on time.

For a valve that opens and closes based on time, the transient data will be used exclusively to determine the valve's position. If the valve opens based on time but closes based on pressure, the valve position will be limited by the pressure at the valve during the opening transient. The valve will switch to passive pressure control and remain at passive pressure control if the Loss Factor vs. Time data would cause the valve to open past what the pressure conditions dictate.

For more information about how to create Loss Factor vs. Time tables based on Open Percent data, please see the Valve Properties Window page.

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

Related Blogs

Looking to relieve pressure? Let's talk about Relief Valves in AFT Impulse

Relief Valve Waterhammer Theory

Relief Valve junction models a valve that opens at a specified pressure in the system. The valve can relieve either to an outside ambient pressure or into another pipe in the system. The Relief Valve junction has three different types.

Internal Relief Valve

An internal relief valve behaves similarly to a regular valve. When it is closed it behaves identically to a closed Valve junction. When it is open, the loss factor at the valve is first determined based on the pressure conditions at the valve. Then, the pressure loss and flowrate at the valve are determined as if the relief valve is a regular valve junction with the loss factor assigned for that time step.

Exit Relief Valve

An exit relief valve behaves similarly to a regular exit valve. When it is closed it behaves identically to a closed exit valve. When the valve is open, it relieves to the user-specified Exit Pressure.

$$P_{j,new} = \frac{S_C - \dot{m}_{exit,new}}{S_B}$$

where S_C and S_B are given by Equations 2 and 3.The pressure drop across the exit is

$$\Delta P = \frac{1}{2} \rho V^2$$

or,

$$\Delta P = P_{j,new} - P_{exit} = \frac{K}{2 \rho A_{exit}^2} \dot{m}_{exit,new}^2$$

Substituting

$$\Delta P = \frac{S_C - \dot{m}_{exit,new}}{S_B} - P_{exit} = \frac{K}{2 \rho A_{exit}^2} \dot{m}_{exit,new}^2$$

This is a quadratic equation in which $\dot{m}_{exit,new}$ can be obtained from the quadratic formula.

$$\dot{m}_{exit,new}{}^2 + \frac{1}{S_B}\frac{2\,K}{A_{exit}{}^2}\dot{m}_{exit,new} + \frac{2\,K\,\rho}{A_{exit}{}^2}\left(P_{exit} - \frac{S_C}{S_B}\right) = 0$$

Inline Relief Valve

An inline relief valve behaves similarly to a Branch or Spray Discharge junction, depending on if it is closed or open. When the valve is closed, it acts like a lossless Branch junction with two pipes. When the valve is open, it acts like a Spray Discharge junction with two connected pipes.

Relief Valve Vapor Cavitation Theory

Cavitation at a relief valve can be complicated to model and understand. If cavitation is present while the valve is closed, the valve will behave as described below. If cavitation is present while the valve is open, the user should closely examine the results to ensure the results are accurate and reasonable.

When the relief valve is closed, cavitation is modeled differently for each type of relief valve:

- **Internal Relief Valve:** the cavitation at the valve is identical to that of a regular valve. Note that for this type of relief valve, cavitation is modeled like a regular valve even when open.
- **Exit Relief Valves:** the cavitation at the valve is identical to that of a dead end junction.
- **Inline Relief Valves:** the cavitation at the valve is identical to a branch junction connected to two pipes.

Related Topics

Related Blogs

Looking to relieve pressure? Let's talk about Relief Valves in AFT Impulse

Reservoir

The Reservoir junction type allows you to connect up to twenty-five pipes. One connecting pipe is required.

The Reservoir Properties window follows the second of the two basic Properties window formats. A table on the Loss Coefficients tab displays the connecting pipe information. This table grows to accommodate up to twenty-five pipes. After you add a fifth connecting pipe, a scroll bar appears, allowing you to review and enter loss factors for all pipes in the table. The pipe table displays the reference positive flow direction of each connecting pipe. To enter loss factors, select the cell in the table and edit the value. Each pipe can use a different loss model or custom value.

Tank Model

There are two possible Tank Models available in the Reservoir junction, Infinite Reservoir and Finite Open Tank.

- Infinite Reservoir - 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. The Infinite Reservoir model is convenient for specifying a fixed pressure in your system. This tank model applies a defined pressure at the junction location in the model. When solving a pipe flow system, an Infinite Reservoir causes the rest of the system to distribute the flow in a manner consistent with the defined pressure.

- Finite Open Tank - 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 open reservoirs are open to the atmosphere or some other fixed surface pressure. 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.

For both models the Liquid Surface Elevation, Liquid Surface Pressure, and Pipe Depth/Elevation will be required to be defined by default.

If the Finite Open Tank model is chosen, additional information for the Cross-Sectional Area, Tank Height, and Tank Bottom Elevation will be required. The cross-sectional area can be defined as a constant area if it is a standard cylinder, or can be modeled as variable. If the variable option is chosen, an additional Tank Geometry input will be required on the Pipe Depth & Loss Coefficients tab to enter the Accumulated Volume in the Tank vs the Liquid Height.

Known Parameters Initially

For the Finite Open Tank option, the options become active to specify whether some parameters are defined by the user for the steady state. 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. Impulse will only allow one of these parameters to be left unknown at a time.

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. 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.

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.

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 Impulse applies the proper boundary condition for above the liquid surface.

If the pipe is above the liquid surface, the only appropriate condition is for the fluid to be flowing from the pipe into the reservoir. Fluid cannot flow from the reservoir into the pipe as the pipe is above the liquid surface. However, AFT Impulse will assume the fluid flowing into the pipe is the same as the reservoir fluid, solve the system, then issue a warning to the user in the output that this is unrealistic.

Finite Tank Overflow/Drain

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.

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 Impulse. 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 Impulse assumes all pipes are liquid full, and cannot model draining pipes. A warning will be displayed in the output tab if this occurs.

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.

Transient Data

On the transient data tab the surface elevation can be changed with time. For more information on transient data, including event transients, see Junction Transient Data.

Graphing Reservoir Data

You can graph various parameters such as tank liquid height and total mass or volume inflow by selecting to save the transient output for these junctions in the Output Junctions panel of the Pipe Sectioning and Output group. These parameters can be graphed or reviewed in the Output window.

Related Topics

Reservoir Waterhammer Theory

A reservoir is similar to an assigned pressure. With the known reservoir height, the pressure can be obtained at the junction and the flow can be solved directly (Wylie 1993, pp. 43):

$$P_{known} = P_{i,new} = H_R \, \rho \, g + P_{surface}$$

Then the flow rate is obtained by employing the compatibility equations.

If the pressure is known at the upstream end of the pipe, the negative compatibility equation is used (Equation 2).

$$\dot{m}_{i,new} = \frac{P_{known} - C_M}{B_M}$$

Conversely, if the pressure is known at the downstream end, the positive compatibility equation is used (Equation 1).

$$\dot{m}_{i,new} = \frac{C_P - P_{known}}{B_P}$$

If the reservoir height changes with time, then a current height is obtained for the current time step and used in the equations.

Reservoir Vapor Cavitation Theory

Infinite Reservoir junctions cannot reach vapor pressure because the pressure is specified and is therefore always above vapor pressure.

Any short-lived vapor cavities that develop at a Finite Open Tank junction are assumed to have a negligible effect compared to the tank itself. Conceptually, vapor bubbles would rise by buoyancy to the open tank surface and evaporate at the surface.

Therefore, vapor cavitation is ignored at finite open tanks.

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 Impulse 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 Impulse 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.

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):

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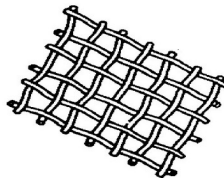
Figure 1: Circular metal wire screen

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).
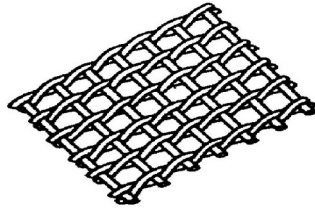
Figure 2: Silk thread screen

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 flow occurs across a user specified hydraulic loss such as a discharge coefficient. This can be varied with time to open or close the Spray Discharge junction.

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.

Two geometries can be modeled. The first is a normal, one hole spray, where the discharge flow area of the hole is entered. The second is a sparger which, in principle, works in the same way as a spray. For the sparger, the flow area of a single hole and the number of holes are entered. This assumes that the holes are hydraulically close in proximity and are the same area.

For each connecting pipe, the pipe table on the Pipe Losses tab lists the reference flow direction and up to two loss factors. The first loss factor is for physical flow out of the pipe and into the junction, the second is for physical flow into the pipe and out of the junction. The loss factors can be specified independently or left as zero. To edit within the pipe table, click in the appropriate column and row.

The distance of each connecting pipe (if any difference exists) from the 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.

Transient Data

On the transient data tab the discharge flow area 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 Junction Transient Data.

Graphing Spray Discharge Data

You can track the discharge flowrate and total mass or volume discharged at the spray discharge junction by selecting to save the transient output for these junctions in the Output Junctions panel of the Pipe Sectioning and Output group. These parameters can be graphed or reviewed in the Output window.

Related Topics

Spray Discharge Fire Sprinklers

The definition of the *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 gpm for flowrate and psid for pressure. In Europe it is frequently liter/min for flow rate and bar for pressure. Note that $K_{sprinkler}$ 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 lbm/ft3 or 1000 kg/m3. 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 Impulse, 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 K Factor Loss Model section and used extensively throughout AFT Impulse.

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 Waterhammer Theory

Spray discharge junctions (shown in Figure 1) model spray locations in a system. The spray junction discharges to a known pressure through an orifice or nozzle of some kind.
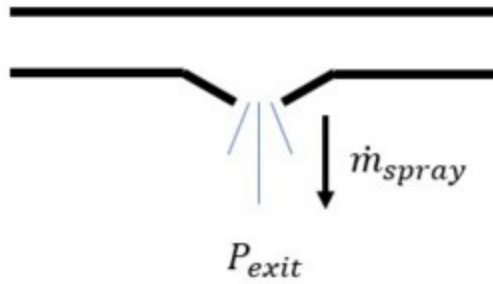
Figure 1: Spray discharge schematic

The relationship for pressure drop across the nozzle is as follows

$$\Delta P = \frac{1}{2}\,\rho\,V^2$$

$$\Delta P = P_{j,new} - P_{exit} = sign\,(\dot{m}_{spray})\,\frac{1}{2\,\rho\,(C_D\,A_{spray})^2}\,\dot{m}_{spray}^{\,2}$$

where, for clarity, $\dot{m}_{spray}$ is defined positive flowing out of the system.

In addition, the spray junction is similar to a branch junction, so the following equation applies with the flowrate defined positive out.

$$P_{j,new} = \frac{S_C - \dot{m}_{spray}}{S_B}$$

where S_C and S_B are given by Equations 2 and 3 and .Combining the preceding equations and eliminating flow rate,

$$sign\,(\dot{m})\,\frac{1}{2\,\rho\,C_D^{\,2}\,A_{spray}^{\,2}}\,\dot{m}^2 + \frac{1}{S_B}\,\dot{m} + P_{exit} - \frac{S_C}{S_B} = 0$$

or

$$sign(\dot{m})\,\dot{m}^2 + \frac{2\,\rho\,C_D^{\,2}\,A_{spray}^{\,2}}{S_B}\,\dot{m} + \frac{2\,\rho\,C_D^{\,2}\,A_{spray}^{\,2}\,(S_B\,P_{exit} - S_C)}{S_B} = 0$$

This is a quadratic equation, and the mass flowrate can be obtained by use of the quadratic formula.

> **Note:** If the spray junction pressure falls below the exit pressure, reverse flow will occur. If the exit pressure is the atmosphere, the reverse flow will be air. However, Impulse assumes the inflow is at the pipe system density. This can lead to erroneous results if ignored. Reverse flow can be determined by graphing the flow at the spray junction or looking at the summary of maximum and minimum flows in the Output window.

Spray Discharge Vapor Cavitation Theory

When the calculated pressure at a Spray Discharge junction drops below vapor pressure, a vapor cavity forms at the junction. This is similar to a Branch junction. The vapor volume calculation is as follows:

$$V_{vapor,new} = V_{vapor,old} + \left(\begin{array}{c} \sum \left(-\dot{m}_{in,new} - \dot{m}_{in,old} \right) \\ + \sum \left(\dot{m}_{out,new} + \dot{m}_{out,old} \right) \\ + \left(\dot{m}_{Spray,new} + \dot{m}_{Spray,old} \right) \end{array} \right) \frac{\Delta t}{2 \rho}$$

where $\dot{m}_{spray}$ is the spray outflow term. The $\dot{m}_{in}$ terms are obtained from the positive compatibility equation, while the $\dot{m}_{out}$ terms are obtained from the negative equation.

Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Surge Tank

The Surge Tank junction allows up to twenty-five connecting pipes. This junction type is useful for modeling a small open tank vessel designed as a surge protection device (see Figure 1). The Surge Tank may be integral with the pipe system, separated by an orifice, or by a short connector pipe.

The Surge Tank Properties window follows the second of the two basic Properties window formats. A table on the Pipes tab displays the connecting pipe information, including flow direction and loss values. The table grows in size to accommodate up to twenty-five connecting pipes.

Figure 1: Surge Tank Properties Window

Tank Geometry

The Tank Height is the physical top of the tank in relation to the elevation. If there is a Connector Pipe, then the Tank Height is relative to the top of the Connector Pipe. This height is used in the case of overflow. If the surge tank overflows, spillage will occur and the liquid will exit the pipe system not to be recovered. If you do not want to model the overflow, specify a very tall tank or do not enter a Tank Height.

If the Tank Cross-Sectional Area is specified as Constant, the surge tank is assumed to be a vertical tank with constant cross-sectional area. If the tank 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 tank during a time step.

Initial State of Liquid Height

The Initial State of Liquid Height must be defined based on the application of the surge tank. There are three available options.

- **Liquid Height Unknown (Calculated by Impulse) -** This is the default option which allows Impulse to calculate the liquid height based on the static head at the junction, and is typical for most applications. During the transient, flow will go in and out of the tank, and the liquid level will change.

- **Liquid Height Known (Zero Inflow/Outflow) -** This option allows the surge tank to act as a pressurizer, similar to a reservoir junction. In this case it is assumed that the surge tank is acting as a reference pressure for a closed loop, in which no inflow/outflow is possible in the steady state. The defined initial liquid height in the tank will be converted to a reference pressure. During the transient, flow will go in and out of the tank, and the liquid level will change.

- **Liquid Height Known (Possible Inflow/Outflow) -** This option again allows the surge tank to act as a pressurizer. In this case it is assumed that the surge tank is providing a reference pressure for an open system, in which steady state inflow/outflow would be possible. The defined initial liquid height in the tank will be converted to a reference pressure. During the transient, flow will go in and out of the tank, and the liquid level will change. This option is similar to using the Finite Open tank model in the Reservoir junction.

Interface with Pipe System

The Surge Tank can connect into the pipe system in three ways: it can be integral with the pipe, it can be separated by an orifice, or it can be connected by a short pipe.

These options are provided in the Flow Restrictor and Connector Pipe areas. If neither option is used, the surge tank is assumed to be attached directly to the pipe with no hydraulic loss.

If the Flow Restrictor is used, the surge tank is assumed to be separated from the pipe system by an orifice, which causes an hydraulic loss as the liquid flows in and out of the surge tank. The loss is specified by entering a flow area and discharge coefficient. Further, this loss can be different for inflow vs. outflow.

Lastly, the surge tank can be connected by a short pipe which is considered to be short in relation to the other pipes in the system after they are sectioned. If the pipe is short, it can be assumed to react instantaneously with the surge tank. The short pipe accounts for liquid inertia and friction. If a connector pipe exists which cannot be considered short, it should be modeled as an actual AFT Impulse pipe.

The Flow Restrictor and Connector Pipe can both exist in the same surge tank. That is, a flow restrictor can be modeled at the top of a connector pipe, and the effect of both will be included.

One-Way Surge Tank

One drawback of surge tanks is that they must be located in low pressure parts of the system because high pressure locations require a very tall surge tank (a water surge tank located where the pipe system is at 100 psig must have a liquid surface at least 230 feet higher than the pipe), unless there is a check valve connecting the surge tank to the pipe. Since the tank can only flow out, it is only useful to limit low transient pressures by flowing liquid into the system when the pressure drops. However, when the pressure rises, the check valve closes and the tank cannot accept any fluid and thus decrease the system pressure.

Such a design is called a One-Way Surge Tank, and it can be modeled by clicking the check box next to One-Way Surge Tank on the Tank Model Tab, then defining the check valve characteristics and initial liquid height which subsequently become active.

Model as Dipping Tube Vessel

A dipping tube vessel is a combination surge tank and gas accumulator (see Figure 2). As long as the liquid level remains below the Height at Tube Bottom as shown in Figure 2, it acts like a conventional surge tank. If the liquid level rises above the bottom of the tube,gas gets trapped in the upper region and the device acts like a gas accumulator.

The dipper tube capability can be enabled by checking the box on the Tank Model tab, which will update the window with the required additional inputs of polytropic constant, height at tube bottom, and tank height.
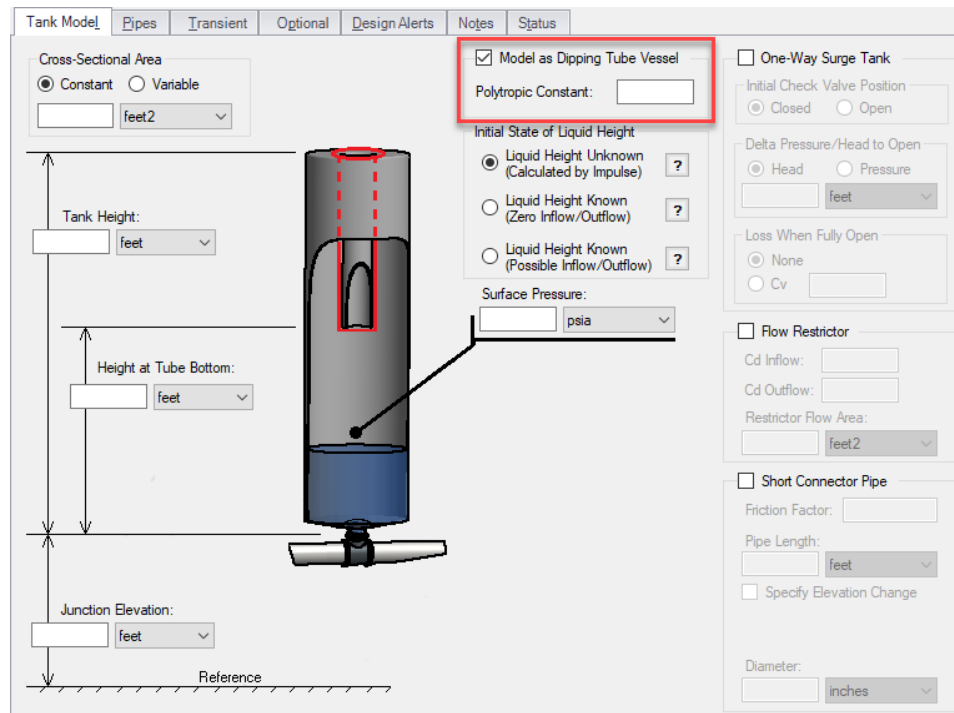
Figure 2: Dipping tube vessel geometric representation. When the liquid level rises above the height of the tube bottom, it acts like a gas accumulator

Transient Data

On the transient data tab the surface pressure can be varied with time to simulate a tank pressurization. For more information on transient data, including event transients, see Junction Transient Data.

Special Condition

The special condition for the Surge Tank on the optional tab allows the user to set Impulse to ignore the surge tank. See Special Conditions for more information.

Graphing Surge Tank Data

You can track the surge tank flowrate, total mass or volume, and liquid surface level pressure by selecting these options in the Output Junctions panel of the Pipe Sectioning and Output group. These parameters can be graphed or reviewed in the Output window.

Related Topics

Surge Tank Waterhammer Theory

A surge tank is an open tank to the atmosphere (Figure 1) and is used as a surge suppression device (Wylie 1993, pp. 128-129).The flow balance on the surge tank junction is similar to a branch junction, with the deficit equaling the flow out of the surge tank, $\dot{m}_T$

Similar to a branch, it can be shown that for all connecting pipes to the junction

$$P_{j,new} = \frac{S_C + \dot{m}_{T,new}}{S_B}$$

where S_C and S_B are given by Equations 2 and 3 from Branch Waterhammer Theory.
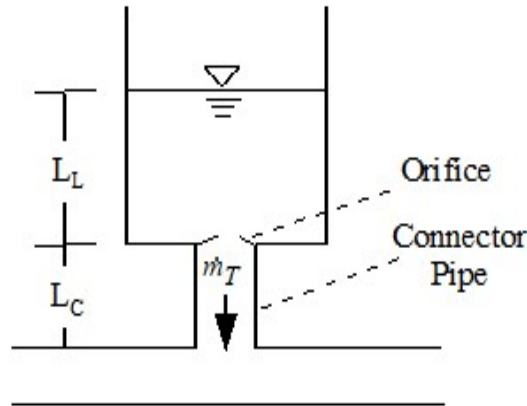
Figure 1: Surge tank schematic

The optional connector pipe and orifice effect is incorporated by the lumped inertia relationship as follows:

$$P_{j,new} = P_{surface} + \rho\, g\, L_{L,new} - C_1 - C_2\, \dot{m}_{T,new}$$

where C_1 and C_2 are defined in conjunction with lumped inertia. Refer to Equations 1 and 2 for lumped pipe inertia. The new tank liquid height can be related to the old height by the following:

$$L_{L,new} = L_{L,old} - \frac{\dot{m}_{T,new} + \dot{m}_{T,old}}{2 \, \rho \, A_T} \Delta t$$

Substitution yields

$$\frac{S_C + \dot{m}_{T,new}}{S_B} + P_{surface} + \rho \, g \left(-L_{L,old} + \frac{\dot{m}_{T,old}}{2 \, \rho \, A_T} \Delta t \right) + g \frac{\dot{m}_{T,new} \, \Delta t}{2 \, A_t} - C_1 - C_2 \, \dot{m}_{T,new} = 0$$

Solving for $\dot{m}_{T,new}$

$$\dot{m}_{T,new} = \frac{\dfrac{S_C}{S_B} - P_{surface} - \rho \, g \left(L_{L,old} + \dfrac{\dot{m}_{T,old} \, \Delta t}{2 \, \rho \, A_T} \right) + C_1}{- \left(C_2 + \dfrac{g \, \Delta t}{2 \, A_T} + \dfrac{1}{S_B} \right)}$$

This flow rate can be back substituted to obtain the junction pressure and surge tank liquid height.

Surge Tank Vapor Cavitation Theory

Any short-lived vapor cavities that develop at a Surge Tank junction are assumed to have a negligible effect compared to the surge tank itself. Conceptually, vapor bubbles would rise by buoyancy to the surge tank surface and evaporate at the surface.

Therefore, vapor cavitation is ignored at surge tanks.

Related Topics

Tee/Wye

The Tee/Wye junction type requires three connecting pipes.

Loss Factors

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, Impulse will automatically define the straight and branch run pipes based off of the arrangement in the Workspace. If Impulse 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 Impulse incorporates the best available 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 an in-depth discussion of Detailed Tees.

If you have specific loss factors that you want to use for a tee/wye junction, you should use a Branch junction instead.

The Tee/Wye junction functions the same during the steady-state and transient.

Related Topics

Related Blogs

Having model convergence problems? Start here!

Give me some details about Detailed Tees!

Tee/Wye Loss Model

The loss factors calculated for tee and wye junctions involve complicated correlations that depend on the flow split, the ratio of flow areas, and the angle of the connecting pipes. The models used by AFT Impulse are taken mostly from Chapter 7 of Idelchik's Handbook of Hydraulic

. The equations will not be shown in this topic, but the references are given in the table below.

All tees and wyes have either diverging or converging flow streams. Two sets of equations are required for each configuration.

In the following discussion, the parameter $K_{c,s}$ represents the loss factor based on the combined flow, stream flow area, and velocity for the side branch. Similarly, $K_{c,st}$ is for the straight-through pipe (the run).

Figure 1 depicts the nomenclature. K factor subscripts refer to base area reference and location in the Tee. For example, $K_{c,s}$ represents the K factor for the side branch (s) referenced to the combined (c) pipe flow area.
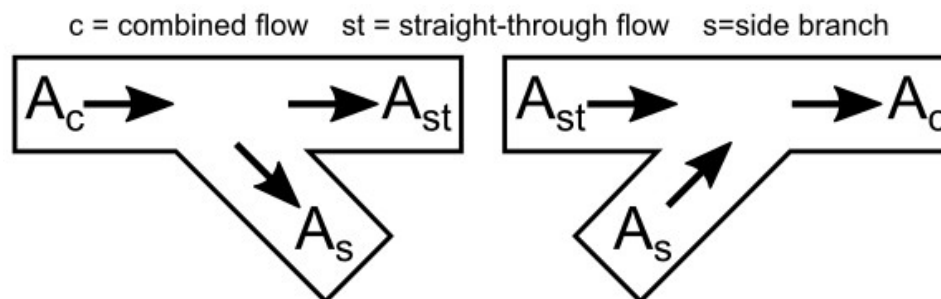
Figure 1: Idelchik's nomenclature for diverging and converging flow through Tee/Wye junction

In the AFT Impulse Tee/Wye window, a simpler and more general nomenclature is used. AFT Impulse refers to the *a*, *b* and *c* pipes. This allows AFT Impulse to cover all the possibilities in Table 3 without specifying exactly which pipe is combined flow pipe. In some cases, you may not know all the flow directions in a system, and might not be able to specify which is the combined flow pipe.

AFT Impulse's input method allows you to specify which pipe is the branch pipe. Idelchik calls this the "s" pipe for "side branch". AFT Impulse calls it the *c* pipe. If the tee angle is 90 degrees, it is not important which pipe you call the *a* pipe and which the *b* pipe. The distinction between *a* and *b* takes on more importance for non-90 degree pipes. In this case, AFT Impulse allows you to specify the angle of the branch pipe.

For tees/wyes the correlations used to calculate loss assume that the angle between the side branch and straight through flow pipes (labeled as As and Ast in Figure 2 above) can be at maximum 90 degrees. If the angle entered in the tee/wye junction exceeds 90 degrees between the side branch and straight through flow pipe, then the loss factor will be assumed to be equivalent to the loss with a 90 degree angle.

Figure 2 shows the Tee/Wye Properties window, with the Arrangement information at the bottom of the window.. The α angle itself is specified on the Tee/Wye Model tab, where there are other options for how to model the tee.

Figure 2: Tee/Wye Properties window allows you to specify the a, b and c pipes. The c pipe is always the branch pipe. This general method means you do not need to know the flow direction ahead of time.

Table 3: References for Tee/Wye junction correlations

Geometry	Type	Direction	Reference	Page
Constant Area, Sharp	Diverging Tee	Branch	Idelchik 2007	523
	Diverging Tee	Run	Idelchik 2007	525
	Converging Tee	Branch	Idelchik 2007	501-504
	Converging Tee	Run	Idelchik 2007	501-504
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	316*
Constant Area, r/D = 0.1	Diverging Tee	Branch	Idelchik 2007	531-533
	Diverging Tee	Run	Idelchik 2007	526
	Converging Tee	Branch	Idelchik 2007	512-514
	Converging Tee	Run	Idelchik 2007	512-514
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	315*
Area Sum, Sharp	Diverging Tee	Branch	Idelchik 2007	524
	Diverging Tee	Run	Idelchik 2007	526-527
	Converging Tee	Branch	Idelchik 2007	505-509
	Converging Tee	Run	Idelchik 2007	512-514
	Splitting Tee	From Branch	Miller 1990	323*
	Combining Tee	To Branch	Miller 1990	315*
Wye	Diverging Wye	Branch	Idelchik 2007	523
	Congering Wye	Run	Idelchik 2007	545
*Estimates have been made on available data				

Related Topics

Related Blogs

Give me some details about Detailed Tees!

Tee/Wye Waterhammer Theory

The Tee/Wye junction is a static element, and its solution method is essentially the same as the Branch junction. The only difference is that the loss factors are updated dynamically similar to steady flow if the Detailed model is in full use during the transient.

Generally it is not recommended to use the detailed loss model during the transient run for multiple reasons. One is that the complexity of iterations required for the detailed tee can increase the run time. The iterations can also easily become unstable, especially in cases where the direction of the flow is changing or oscillating at the tee/wye. Additionally, the detailed tee model

was not developed with the intention of calculating transient loss, so this model should be applied in the transient run with care.

In most cases, the loss at the tee is small enough that there will not be much difference between the simple and detailed models in the steady state, making it wise to just use the simple model. However, in some cases it may be required to use the detailed tee model. In that case, there are several options available on the Optional tab in the Transient K Factor section.

- **Calculated** - This is the default option, which dynamically updates the loss factors throughout the transient.
- **Calculated but Limited -** This option allows the K factor to be dynamically updated, but defines a Maximum K factor for the calculations, which may be useful in the case of large oscillations in the transient run
- **Keep Steady State Values** - This option allows the loss to be calculated normally in the steady state, then behaves like a branch junction with static K factors during the transient. This may be a good assumption in the case of small changes at the tee junction.

Related Topics

Related Blogs

Give me some details about Detailed Tees!

Turbine

The Turbine 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.

The AFT Impulse turbine junction is intended to model a hydroelectric turbine, in which hydraulic power is converted to shaft power, which drives an electric generator. There are many different hydro-turbine designs, but they are generally placed into one of two categories:

- Impulse-Type Turbine (e.g. Pelton Wheel)
- Reaction-Type Turbine (e.g. Francis or Kaplan)

AFT Impulse models only the reaction-type turbine; specifically, the Francis turbine.

The conventional method for modeling Francis turbine behavior employs four quadrant characteristic data. Four quadrant data relates flow, head loss, torque, and speed in a format which is convenient for digital computers. Similar four quadrant data is used to model pump behavior; however, the turbine does not generally involve reverse flow or reverse rotation.

Typical turbine head loss data will look something like Figure 1:
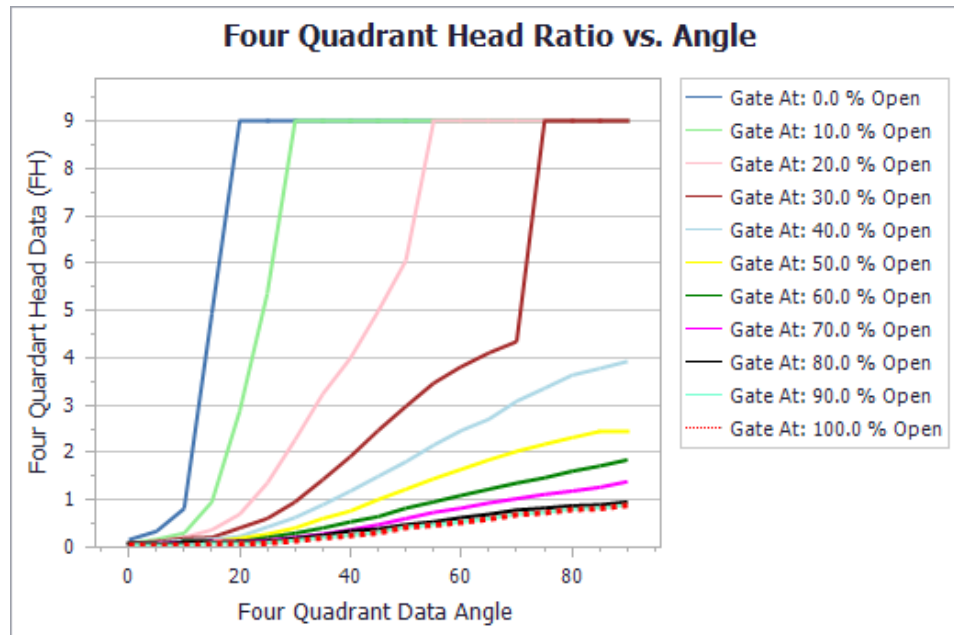
Figure 1: Typical turbine head loss data

Notice in Figure 1 that the turbine head loss data is represented by an array of curves, one for each wicket gate position. A similar array of curves is used to describe turbine net-torque (see Figure 2):
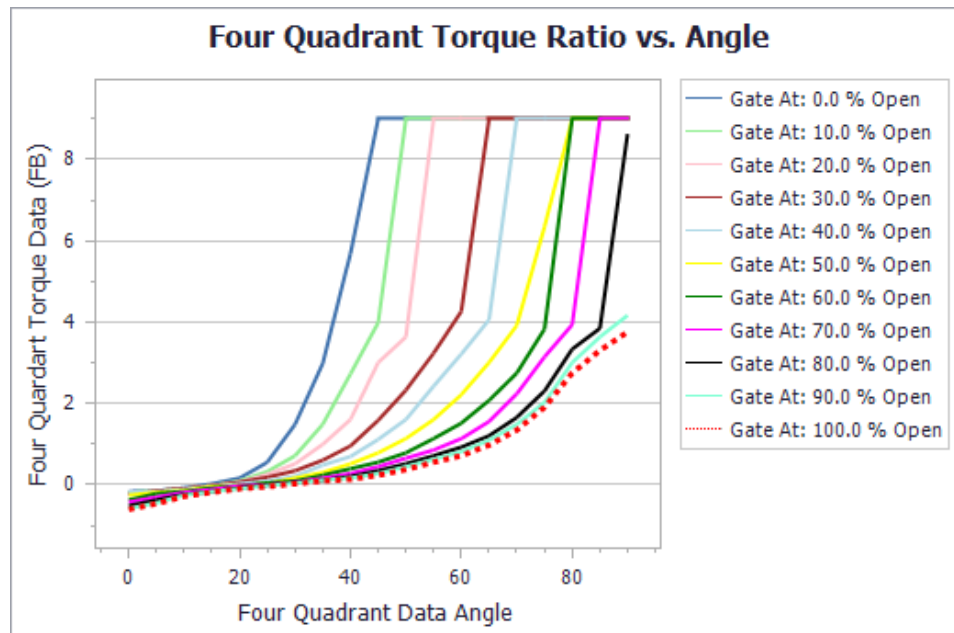
Figure 2: Turbine net-torque curve

The user must specify how many wicket gate positions are to be modeled, and how many four quadrant angles are to be modeled. With these parameters given, complete FH and FB data can be entered in the Turbine Four Quadrant Data table, as seen in Figure 3.

Figure 3: Sample FH data entered in the Turbine Four Quadrant Data table

The turbine four quadrant data relies on having a complete set of rated turbine performance parameters. Examples of these parameters entered in the properties window are shown in Figure 4. The importance and use of these parameters can be seen in the Turbine Waterhammer Theory help topic.

Figure 4: Sample turbine rated conditions

Turbine Transients

Generally speaking, there are three kinds of turbine transients which can be modeled in AFT Impulse.
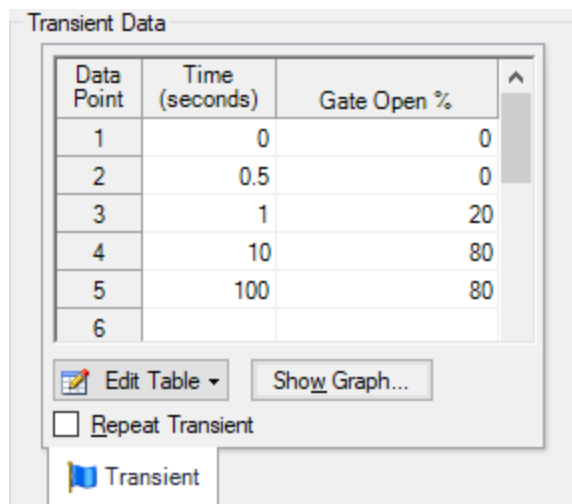
- **Turbine Trip -** Gates Initially Open, Closing During Transient Simulation
- **Turbine Startup -** Gates Initially Closed, Opening During Transient Simulation
- **No Transient -** Reactive Only, No Transient Gate Behavior

Turbine Trip

The turbine trip assumes the generator torque drops out instantly, between steady state and the first time step of the transient solution. Generator torque is not specified. Instead, the initial speed and four quadrant torque data provide an initial torque imbalance The initial torque imbalance is immediately resolved by an appropriate speed change (damped by inertia) when the transient simulation begins.

Turbine Startup

The turbine initial conditions may be specified as zero speed and zero flow. When this is the case, the Steady State Special Condition, in the Optional tab, must be set to "Closed". The turbine will have zero flow in steady state, and will remain at zero flow until the gates are opened, according to the Transient Data, specified in the Transient tab. See Figure 5 for an example of this data entered in the Transient Data tab.

Figure 5: Sample input for a turbine startup

As the wicket gates open, flow and speed will increase accordingly.

No Transient

No transient data is required to simulate turbine behavior, as it reacts to circumstances. Specify only the parameters required in the Turbine Model tab. The turbine speed and flow will continuously adjust to changes in supply pressure and or discharge pressure. Keep in mind; this is essentially the same as a turbine trip, but without commanding the gates to close. The turbine speed will immediately find equilibrium with the specified initial conditions. It can be seen in the torque data that there is a crossover point, where the torque ratio goes from positive to negative. If the initial specified conditions produce an initial Theta which lies in the region of this crossover point, the turbine will be initially stable. However, if the initial torque ratio, as determined by the initial speed and flow, is NOT in the region of the crossover point, the turbine will have to resolve the non-zero net-torque with a speed change, until equilibrium is established. This speed change will occur as soon as the transient simulation begins.

Related Topics

Turbine Waterhammer Theory

Four quadrant data is defined by the following dimensionless parameters ("R" subscript refers to rated values). Reference .

- Head Loss Ratio:

$$h = \frac{H_L}{H_R} \qquad (1)$$

- Torque Ratio (unbalanced torque):

$$\beta = \frac{T}{T_r} \qquad (2)$$

- Flow Ratio:

$$\nu = \frac{Q}{Q_R} \tag{3}$$

- Speed Ratio:

$$\alpha = \frac{N}{N_R} \tag{4}$$

- Function Relating Head Loss to Flow and Speed:

$$Fh\left(n(\theta)_{int}\right) = \frac{h}{\alpha^2 + \nu^2} \tag{5}$$

- Function Relating Torque to Flow and Speed:

$$Fb\left(n(\theta)_{int}\right) = \frac{\beta}{\alpha^2 + \nu^2} \tag{6}$$

- Four Quadrant Data Angle:

$$\theta = \tan^{-1}\left(\frac{\nu}{\alpha}\right) \tag{7}$$

- Integer function for Fh and Fb indices:

$$n = int\left(\frac{\theta}{\Delta\theta}\right) + 1 \tag{8}$$

For the Speed change related to net torque, we start with Eq. 7 - 47 from Wylie (similar to Eq. 7 - 20, without the negative sign)

$$T = I\frac{d\omega}{dt} \tag{9}$$

$$\omega = \alpha \, N_R \frac{2\pi}{60} \tag{10}$$

$$T = \left(N_R \frac{2\pi}{60}\right) \frac{d\alpha}{dt} \tag{11}$$

$$T = \beta \, T_R \tag{12}$$

$$\beta \, T_R = \left(N_R \frac{2\pi}{60}\right) \frac{d\alpha}{dt} \tag{13}$$

$$\int_t^{t+\Delta t} \beta \, T_R \, dt = \int_t^{t+\Delta t} \left(N_R \frac{2\pi}{60}\right) \frac{d\alpha}{dt} dt \tag{14}$$

$$\int_t^{t+\Delta t} \beta \, T_R \, dt = T_R \frac{(\beta + \beta_0)}{2} \Delta t \tag{15}$$

$$\int_{t}^{t+\Delta t} \left(N_R \frac{2\pi}{60} \right) \frac{d\alpha}{dt} dt = \left(N_R \frac{2\pi}{60} \right) (\alpha - \alpha_0) \qquad (16)$$

$$T_R \frac{\beta + \beta_0}{2} \Delta t = \left(N_R \frac{2\pi}{60} \right) (\alpha - \alpha_0) \qquad (17)$$

$$(\beta + \beta_0) = \frac{2 \left(N_R \frac{2\pi}{60} \right)}{\Delta t \, T_R} (\alpha - \alpha_0) \qquad (18)$$

$$\beta = \frac{2 \left(N_R \frac{2\pi}{60} \right)}{\Delta t \, T_R} (\alpha - \alpha_0) - \beta_0 \qquad (19)$$

This is a modified version Eq 7-22 from Wylie. Wylie integrates over two time steps, to be consistent with the "staggered grid" solution of the Method of Characteristics grid. The above result is based on integration over only one time step for use in the "complete grid".

It's convenient, at this point, to lump constants together and simplify.

$$C_T = \frac{2 \left(N_R \frac{2\pi}{60} \right)}{\Delta t \, T_R} = \frac{N_R \frac{\pi}{15}}{\Delta t \, T_R} \qquad (20)$$

$$\beta = C_T (\alpha - \alpha_0) - \beta_0 \qquad (21)$$

$$Fb\left(n(\theta)_{int}\right) = \frac{\beta}{\alpha^2 + \nu^2} \tag{22}$$

$$Fb\left(n(\theta)_{int}\right) = B_1 + B_2\,\theta \tag{23}$$

$$B_2 = \frac{Fb\left(n+1\right) - Fb\left(n\right)}{\Delta\theta} \tag{24}$$

$$B_1 = Fb\left(n+1\right) - n\,\beta_2\,\Delta\theta \tag{25}$$

$$\frac{\beta}{\alpha^2 + \nu^2} = B_1 + B_2\,\theta \tag{26}$$

$$\beta = (B_1 + B_2\,\theta)(\alpha^2 + \nu^2) \tag{27}$$

$$(B_1 + B_2\,\theta)\left(\alpha^2 + \nu^2\right) = C_T\left(\alpha - \alpha_0\right) - B_0 \tag{28}$$

$$0 = C_T\left(\alpha - \alpha_0\right) - B_0 - (B_1 + B_2\,\theta)\left(\alpha^2 + \nu^2\right) \tag{29}$$

$$F_T = C_T\left(\alpha - \alpha_0\right) - B_0 - (B_1 + B_2\,\theta)\left(\alpha^2 + \nu^2\right) \tag{30}$$

Eq. 30 is a modified version of Wylie's Eq. 7-26. Wylie uses Eq. 7-20 (for pumps) but Equation 30 above uses Eq. 7-47 (for turbines).

Head loss can be described using a similar procedure, along with the characteristic equations.

$$H_{Up} = C_P - B_P\,Q \tag{31}$$

$$H_{Dn} = C_P + B_M\,Q \tag{32}$$

$$Fh\left(n(\theta)_{int}\right) = \frac{h}{\alpha^2 + \nu^2} \tag{33}$$

$$Fh\left(n(\theta)_{int}\right)\left(\alpha^2 + \nu^2\right) = h \tag{34}$$

$$h = \frac{H_L}{H_R} \tag{35}$$

$$Fh\left(n(\theta)_{int}\right)\left(\alpha^2 + \nu^2\right) = \frac{H_L}{H_R} \tag{36}$$

$$Fh\left(n(\theta)_{int}\right)\left(\alpha^2 + \nu^2\right)H_R = H_L \tag{37}$$

$$Fh\left(n(\theta)_{int}\right) = A_1 + A_2\,\theta \tag{38}$$

$$H_L = (A_1 + A_2\,\theta)(\alpha^2 + \nu^2)\,H_R \tag{39}$$

$$H_L = H_{Up} - H_{Dn} \tag{40}$$

$$H_{Up} - H_{Dn} = (A_1 + A_2\,\theta)(\alpha^2 + \nu^2)\,H_R \qquad (41)$$

$$C_P - B_P\,Q - (C_m + B_m\,Q) = (A_1 + A_2\,\theta)(\alpha^2 + \nu^2)\,H_R \qquad (42)$$

$$C_P - C_m - Q(B_P + B_m) = (A_1 + A_2\,\theta)(\alpha^2 + \nu^2)\,H_R \qquad (43)$$

$$F_H = (A_1 + A_2\,\theta)(\alpha^2 + \nu^2)\,H_R - C_P + C_m + Q(B_P + B_m) \qquad (44)$$

Eq. 44 comes from Wylie Eq. 7-19. Wylie includes a valve loss term in Eq. 44, which is not included here. Also, signs are reversed to be consistent with turbine head loss, as opposed to pump head rise.

At this point, FH and FT can be solved using Newton Raphson, in 2x2 matrix form.

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

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

Valve Data Source and Loss Model

In order to completely define a valve junction, you must specify the valve data source by clicking on the appropriate radio button under Valve Data Source. For this data source, you can use handbook data from the Impulse library by clicking the radio button next to Handbook Data and then specifying what type of valve you are modeling to use a loss factor from handbook data, or you can choose User Specified and separately input the valve loss information.

The valve loss model can be defined using one of three different valve parameters. These four parameters include specifying the valve: C_V, K_V, or K factor.

A special reduced port configuration can be specified by selecting User Specified under Valve Data Source, and then checking the box next to Reduced Port Losses in the center of the window under the Loss Model tab. Once this section has been enabled, you will need to specify the valve type and complete any required input for the specified valve.

The upstream pipe is selected by default as the Base Area for Loss Model. This can be changed by selecting the downstream pipe instead, or choosing to define a User Specified Area or Diameter.

Exit Valves

Valves that are used as exit flow control mechanisms are specified as exit valves. These valves require an associated back pressure definition. Exit valves can only be connected to a single upstream pipe. To specify an exit valve, check the Exit Valve check box, located at the bottom of the Loss Model tab. After this box is checked, you will be required to input the exit pressure.

Transient Data

On the transient data tab a transient C_V, K_V, or K, or Open Percent table can be entered to simulate the opening or closing of the valve. For a closed valve, the K factor is infinite. To account for this, enter a K factor of –1 (negative one) in the transient data table. For more information on transient data, including event transients, see Junction Transient Data.

AFT Impulse can also create a C_V vs. Time profile based off of the open percentage table, using the assumption the valve is linearly actuated. To access this in the transient tab select "Edit Table" and then "Create C_V vs. Open Percent...". This option enables the C_V vs. Open percent to be defined from several options:

- **Transferred -** If a table has already been defined on the Optional tab of the Valve Properties window, it can be copied over
- **User Defined -** Data can be entered directly into the table, or imported from existing data
- **Linear** - A linear curve is generated based on the user defined full open 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 profiles can be chosen and adjusted for a defined full open C_V value based on data from Val-Matic 2009.

After the C_V vs. Open Percent is defined a transient can be defined based on duration, increment for the curves data points, and whether the valve is opening or closing. A linear C_V vs. Time curve will then be generated for the given data.

Open Percentage Table

The Optional tab allows data to be entered for special 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 Impulse's flow solution. However, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary (in the Output window) always displays the valve C_V, and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point.

AFT Impulse can assist in creating this C_V vs Open percent curve. This is accessible in the optional by by selecting "Edit Table" and then "Create C_V vs. Open Percent...". This then opens a window in which a user defined, linear, equal percentage or pre-defined curve can be defined as described in the Transient Data section and transferred to the C_V vs. Open Percentage table. If a C_V vs. Open Percent table was already defined on the transient tab in the window to create a C_V vs. Time profile as described in the Transient Data section, then this curve will be automatically transferred.

Special Conditions

The Special Condition for a valve always closes it.

Related Topics

Related Blogs

That Valve's Got Character – Applying Pre-defined Characteristic Curves

A Clever Way to Mitigate Waterhammer During a Valve Closure

Modeling Valve Closures using K Factors

A closed valve would have an infinite K-factor. Since infinity is not a possible input, AFT Impulse uses the convention of $K = -1$ (negative one) to represent a closed valve.

This input is only valid when defining a valve transient. To represent a valve that is closed in steady state, use the valve Special Conditions in the Optional tab.

Related Topics

Related Blogs

[That Valve's Got Character – Applying Pre-defined Characteristic Curves](#)

[A Clever Way to Mitigate Waterhammer During a Valve Closure](#)

Valve Waterhammer Theory

A valve junction has a pressure loss across the valve. When the valve closes this loss becomes infinite ([Wylie 1993](#), pp. 57). With two connecting pipes, the upstream and downstream pipe pressures are obtained from Compatibility [Equations 1 and 2](#):

$$P_{i,new,up} = C_{P,up} - B_{P,up}\,\dot{m}_{i,new,up}$$

$$P_{i,new,down} = C_{M,down} + B_{M,down}\,\dot{m}_{i,new,down}$$

From a mass balance:

$$\dot{m}_{i,new,up} = \dot{m}_{i,new,down} = \dot{m}_{valve}$$

The pressure drop is related to the flow rate according to the following:

$$P_{i,new,up} - P_{i,new,down} = R_{valve}\,\dot{m}_{valve}^{2}$$

where

R_{valve} = Valve Resistance

Substituting the compatibility equations (Equations 1, 2) in and eliminating the pressures

$$(C_{P,up} - B_{P,up}\,\dot{m}_{valve}) - (C_{M,down} + B_{M,down}\,\dot{m}_{valve}) = R_{valve}\,\dot{m}_{valve}^{2}$$

Rearranging terms,

$$\dot{m}_{valve}{}^2 - \frac{1}{R_{valve}}\left(C_{P,up} - B_{P,up}\,\dot{m}_{valve}\right) + \frac{1}{R_{valve}}\left(C_{M,down} + B_{M,down}\,\dot{m}_{valve}\right) = 0$$

Using the quadratic equation, the solution for positive flow through the valve is

$$\dot{m}_{valve} = -\frac{1}{R_{valve}}\left(B_{P,up} + B_{M,down}\right) + \sqrt{\frac{1}{R_{valve}{}^2}\left(B_{P,up} + B_{M,down}\right)^2 + \frac{4}{R_{valve}}\left(C_{P,up} - C_{M,down}\right)}$$

For negative flow, the solution is

$$\dot{m}_{valve} = \frac{1}{R_{valve}}\left(B_{P,up} + B_{M,down}\right) - \sqrt{\frac{1}{R_{valve}{}^2}\left(B_{P,up} + B_{M,down}\right)^2 - \frac{4}{R_{valve}}\left(C_{P,up} - C_{M,down}\right)}$$

After solving for flow rate, $\dot{m}_{valve}$, the upstream and downstream pressures can be obtained from the compatibility equations.

If the valve closes then $1/R_{valve} = 0$ and $\dot{m}_{valve} = 0$.

Exit Valves

If the valve is an exit valve, then the downstream pressure is known and

$$\dot{m}_{valve} = -\frac{1}{R_{valve}}B_{P,up} + \sqrt{\frac{1}{R_{valve}{}^2}B_{P,up}{}^2 + \frac{4}{R_{valve}}\left(C_{P,up} - P_{exit}\right)}$$

Valve Vapor Cavitation Theory

When the calculated pressure at the upstream or downstream side of a valve junction drops below vapor pressure, a vapor cavity forms on that side of the junction.

Vapor cavities can form on either side of the valve or on both sides.

$$V_{vapor,up,new} = V_{vapor,up,old} + \left(-\dot{m}_{up,new} - \dot{m}_{up,old} + \dot{m}_{valve,new} + \dot{m}_{valve,old}\right)\frac{\Delta t}{2\,\rho}$$

$$V_{vapor,down,new} = V_{vapor,down,old} + \left(-\dot{m}_{valve,new} - \dot{m}_{valve,old} + \dot{m}_{down,new} + \dot{m}_{down,old}\right)\frac{\Delta t}{2\,\rho}$$

As an example, when a vapor cavity forms downstream of a valve, the downstream pressure become fixed at the vapor pressure. The calculation then is similar to an exit valve, which also flows to a fixed pressure.

$$\dot{m}_{valve} = -\frac{1}{R_{valve}}B_{P,up} + \sqrt{\frac{1}{R_{valve}^2}B_{P,up}{}^2 + \frac{4}{R_{valve}}(C_{P,up} - P_{sat})}$$

When a vapor cavity occurs on both sides of the valve, the pressure is at vapor pressure on both sides and thus there is no pressure drop across the valve. Accordingly, the flow rate goes to zero.

Similar to a pipe interior node, when the vapor volume is negative, the cavity collapses and the fluid pressure then rises above the vapor pressure.

Related Topics

Related Blogs

That Valve's Got Character – Applying Pre-defined Characteristic Curves

A Clever Way to Mitigate Waterhammer During a Valve Closure

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 Impulse 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.

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 will likely show a Caution message about the density change. This does not necessarily mean that a Volume Balance should be used at the mentioned location.

The Volume Balance junction type is internal to the 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. Impulse 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. AFT Impulse assumes the interface does not move during the transient.

Volume Balance junctions should be used in conjunction with the variable fluid properties model (see Variable Fluids panel).

The usage of a Volume Balance junction is illustrated with the example below. Figure 1 below provides a summary of the four snapshots in time that occur in the example. In reality, injecting slurry into a pipeline where there are two separate fluids with an interface between is a transient situation with a moving interface. In Impulse, this junction can be used to represent an interface which does not move rapidly in the transient. There is no option to define transient data for this junction.
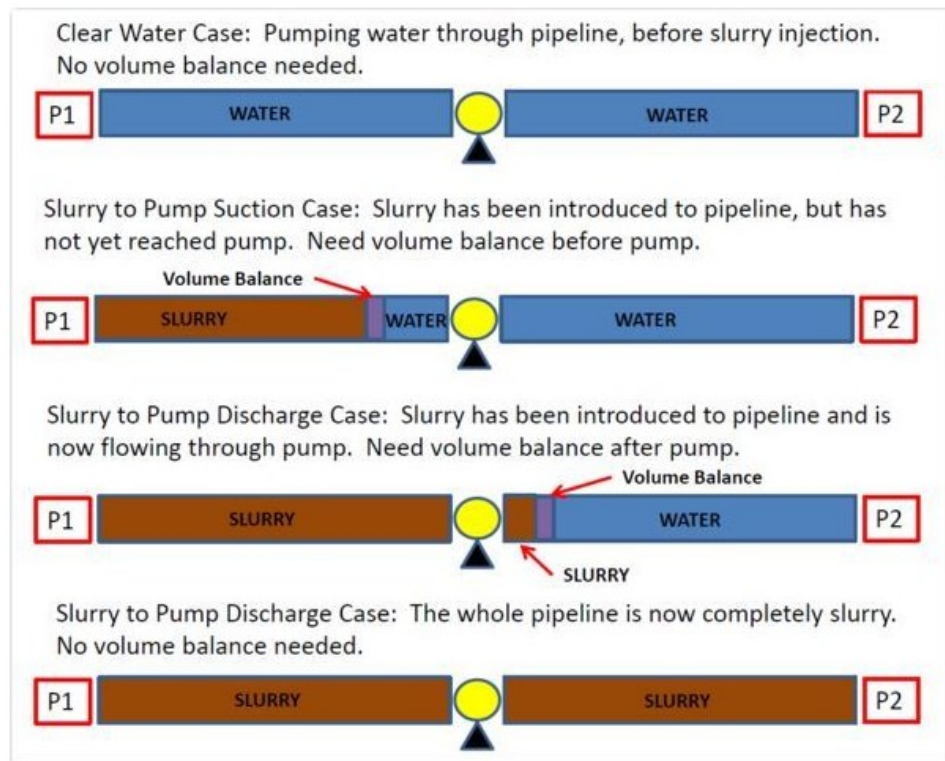
Figure 1: Four snapshots in time and when to use the volume balance junction is shown for slurries and variable fluid properties systems

Related Topics

Weir

Note: The Weir junction must be used with great caution. A weir inherently is for open-channel flow. However, Impulse 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 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.

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
  - Cross-Sectional Area for Upstream and Downstream Area
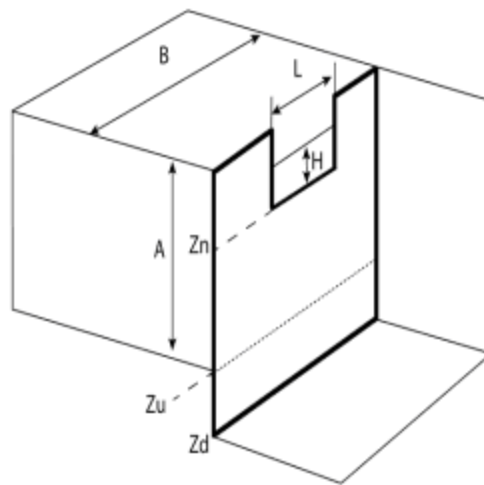
Figure 1: Rectangular Sharp Crested Weir Model

- **V-Notch Sharp Crested -**
  - Surface Pressure
  - Notch Elevation (Z_n)
  - Notch Angle (theta)
  - Channel Width (B)
  - Cross-Sectional Area for Upstream and Downstream Area
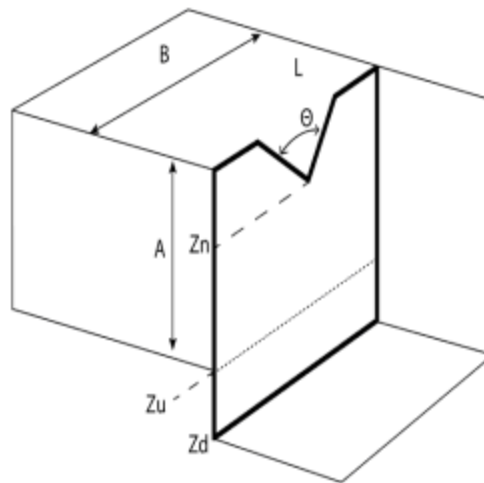
Figure 2: V-Notch Sharp Crested Weir Model

- **Cipolletti, where the vertical angle is a 1:4 ratio -**
    - Surface Pressure
    - Notch Elevation (Z_n)
    - Notch Width (L)
    - Channel Width (B)
    - Cross-Sectional Area for Upstream and Downstream Area
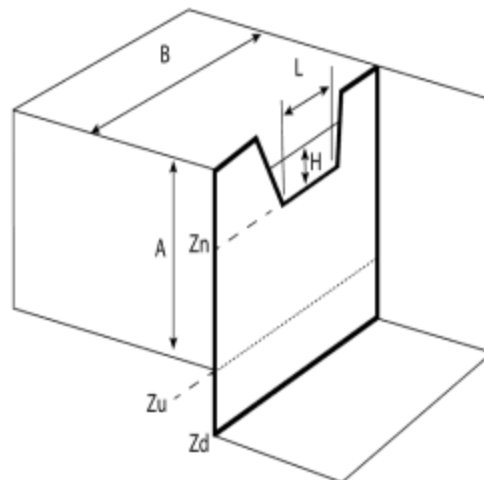
Figure 3: Cipolletti (1:4 ratio) Weir Model

Notch elevation (*Zn*)

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 (*Zu* and *Zd*)

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 Impulse can be used to customize the user's experience within the software. Topics discussed include the options present in each of the AFT Impulse drop-down menus, the tools available in the 5 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 Impulse model can be completed from the various menus located at the top of the window. Additionally, these menus can provide helpful features

such as saving, importing and exporting files, and accessing various help documents such as the product help files and examples. These menus include

- [The File menu](#)
- [The Edit menu](#)
- [The Analysis menu](#)
- [The Tools menu](#)
- [The Library menu](#)
- [The Arrange menu](#)
- [The Window menu](#)
- [The Help menu](#)

This topic also gives a high level overview of [Toolbars](#), including the Main Toolbar as well as each Primary Windows toolbar.

File Menu

- **New** - Opens new model.
- **Open** - Calls Open file common dialog window so a pre-existing model can be selected for opening.
- **Save** - Saves input model to disk using current model file name.
- **Save As** - Opens Save As common dialog so you can specify a file name to save model to disk.
- **Save Graph As** - (Graph Results Only) - Opens Save Graph As common dialog so user can specify a file name to save a graph as a graphics image file to disk.
- **Save Graph Data As** - (Graph Results Only) - Opens Save Graph Data As common dialog so user can specify a file name to save graph x-y data to text file on disk.
- **Import Graph Data** - (Graph Results Only) - Allows for data as .txt and .dat to be imported for creating graphs
- **Export Model Data** - (Model Data Only) - Opens formatting window and Save As dialog where you can specify data from the Model Data window to be saved to a delimited file appropriate for spreadsheet use. Available in Model Data window.
- **Export Output** - (Output Only) - Opens formatting window and Save As dialog where you can specify data from the Output window to be saved to a delimited file appropriate for spreadsheet use.
- **Export Force File -** (Output Only) - Opens [Export Force File](#) window for the selected file type from those shown below.
    - **CAESAR II Force File**
    - **TRIFLEX Force File**
    - **ROHR2 Force File**
- **Import Excel Change Data** - (Workspace and Model Data Only) - Imports data from Excel file which modifies certain input data.
- **Excel Export Manager -** Opens the [Excel Export Manager](#).

- **Merge** - (Workspace Only) - Opens a Merge dialog where you can specify another model file to merge into the current model.

- **Print Preview** - Opens Print Preview which allows you to specify special formatting for printed graphics and text, as well as preview printed content before sending it to the printer.

- **Start Batch Run** - The Start Batch Run button on the File Menu opens a Batch Run dialog where you can select a different scenarios or models and initiates a process whereby Impulse runs the multiple scenarios/models automatically. This tool can also be used to export output to Excel using the settings already configured in the Excel Export Manager.

- **Load Background Picture** - (Workspace Only) - Loads a background picture into the Workspace. Formats supported include bitmap, windows metafile, enhanced metafile, JPEG, GIF and RLE. Available in Workspace window.

- **Import Piping Layout From**
    - **Piping Component File (PCF)** - Import a Piping Component File
    - **Caesar Neutral File (CII)** - Import a Caesar Neutral File
    - **GIS Shapefile (SHP)** - Allows for Importing GIS Shapefile (geospatial maps) as .shp file.
    - **EPANET File (INP)** - Import an EPANET File

- **Export Piping Layout To**
    - **EPANET File (INP)** - Export an EPANET File

- **Exit** - Exits AFT Impulse.

Related Topics

Edit Menu

- **Undo** - (Workspace Only) - Undoes the previous workspace manipulation action. Impulse will keep track of a historical list of changes, and users can revert the workspace back to any of those previous states.

- **Redo** - (Workspace Only) - Reverts the workspace following Undo actions. Impulse will keep track of undo actions, and users can redo any changes that have been undone.

- **Cut** - (Workspace Only) - Cuts the selected pipes and junctions on the Workspace and places them on the clipboard.

- **Copy** - (Workspace, Model Data, and Graph Results Only) - Copies the selected window contents to the clipboard.

- **Copy Graphics** - (Workspace and Visual Report Only) - Copies selected Workspace or Visual Report content to the Windows clipboard. Formats supported include bitmap, windows metafile, and enhanced metafile.

- **Copy X-Y Data -** Copies numerical data from graph to the Windows clipboard. Available in the Graph Results window.

- **Paste** - (Workspace Only) - Pastes clipboard pipes and junction contents (from previous cut or copy operation) into the Workspace model. Paste preview mode allows the pasted pipes and junctions to be moved and specifically placed where desired and will indicate interference with other junction by changing color. A green box indicates that there is no interference and a red box indicates that the junctions will be overlapping and are not in a proper location. Available in the Workspace window.

- **Delete** - (Workspace Only) - Deletes the selected pipes and junctions on the Workspace.

- **Duplicate** - (Workspace Only) - Duplicates the selected pipes and junctions on the Workspace.

- **Duplicate Special** - (Workspace Only) - Opens Duplicate Special dialog window where you can specify the increment by which all pipes and junction numbers will be increased when the model is duplicated, the creation of a new group from the duplicated objects, and whether to hide the duplicated objects in the output.

- **Select All** - (Workspace Only) - Selects all pipes and junctions on the Workspace.

- **Select Last Selection** - (Workspace Only) - The most recent pipes and/or junctions selected may be reselected by selecting Select Last Selection from the Edit menu.

- **Select Special** - (Workspace Only) - Opens Select Special window where group of pipes and junctions can be built up.

- **Select Flow Path** - (Workspace Only) - Selects the sequence of pipes and/or junctions in the flow path connected to the selected pipe(s) on the Workspace.

- **Select Shortest Path** - (Workspace Only) - Selects the sequence of pipes and junctions that create the shortest path between any two junctions selected on the Workspace.

- **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/Junctions** - (Workspace Only) - A continuous series of connected pipes and junctions can be combined into a single pipe object to simplify the Workspace. See Combining and Splitting Pipes.

- **Split Pipe at Intermediate Elevations** - (Workspace Only) - A pipe with intermediate elevations can be split into separate pipes at the intermediate elevation points. See Combining and Splitting Pipes.

- **Global Pipe Edit** - (Workspace and Model Data Only) - Opens Global Pipe Edit window where editing of multiple pipes can be performed.

- **Global Junction Edit** - (Workspace and Model Data Only) - Opens Global Junction Edit window where editing of multiple junctions can be performed.

- **Delete Annotation** - (Visual Report Only) - Deletes the selected pipes and junctions on the Visual Report.

- **Create Annotation** - (Visual Report Only) - Allows you to create an annotation on the Visual Report window.

- **Transfer Results to Initial Guesses -** Copies solution results into initial guesses for pipes and junctions.

- **Make Selection Same As Parent** - (Workspace Only) - Replaces all input data for the selected pipes/junctions with data from the pipes/junctions in the current scenario's parent.

- **Clear Initial Guesses -** Clears all initial guesses in pipes and junctions.

Related Topics

Related Blogs

AFT Responds to 2020 Customer Survey Feedback

Five Fantastic New Features for Fast Fluid Transients: Impulse 8

View Menu

- **Last View -** (Workspace Only) - Switches to the most recent Workspace view. The view is changed by scrolling, zooming or using the Workspace Find. The Last View in the View Menu will take you back to the previous view of the Workspace. You can toggle between two views by continuously clicking Last View.

- **Select Objects -** (Workspace Only) - Changes cursor to selection tool to select junctions, pipes, etc.

- **Overview Map -** (Workspace Only) - Opens overview map in the Quick Access Panel.

- **Workspace Pan -** (Workspace Only) - Changes cursor to pan tool to move around workspace.

- **Show Object Status -** (Workspace Only) - Toggles Show Object Status, which checks status of all pipes and junctions in model.

- **List Undefined Objects -** (Workspace Only) - Opens Undefined Objects window which displays undefined pipes and junctions and their properties.

- **Extended Model Check -** Performs and exhaustive check of model consistency and completeness.
- **Highlight in Pipe and Jct Windows -** (Workspace Only) - Toggles Highlight feature for pipe and junction Properties Windows.
- **Bookmarks -** (Workspace Only) - Allows creation and management of bookmarks.
- **Find -** (Workspace Only) - Opens window whereby specific pipes and junctions can be found.

Related Topics

Analysis Menu

- **Time Simulation -** Allows selection of either steady-state run or transient run.
- **Slurry Modeling -** (SSL Module Only) - Allows selection of whether to use normal fluid calculation or settling slurry calculation in the current run.
- **Pulsation Frequency Analysis -** (PFA Module Only) - Allows pulsation analysis settings to be ignored or used in the model.
- **System Properties -** Opens the Fluid panel in Analysis Setup.
- **Pulsation Setup -** (PFA Module Only) - Opens the Pulse Setup panel in Analysis Setup.
- **Section Pipes -** Opens the Sectioning panel in Analysis Setup.
- **Transient Control -** Opens the Simulation Mode/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 -** This allows you to toggle between two scenarios by loading the most recently loaded scenario. To do this, load one scenario using the Scenario Manager or the Quick Access Panel, then load the other. Use the Last Scenario in the Tools Menu to then toggle between the two scenarios by continuously clicking Last Scenario.
- **Compare Scenarios** - This option quickly sets up a comparison between the current scenario and a related scenario.
- **Scenario Comparison Tool** - This tool opens the Modify Comparison window, allowing users to compare differences between any two or more scenarios in a model.
- **Integration Query -** Opens Integration Query window, allowing users to have Impulse perform a numerical integration of results over specified time frames.
- **Output Control -** Opens the Output Control Window.
- **Output Print Content** - (Output Only) - Prints content from Output Window.
- **Output Language -** Selects language for display of output results.
- **Model Data Control -** (Model Data Only) - Opens Model Data Control Window.
- **Model Data Print Content -** (Model Data Only) - Prints content from 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.
- **Graph Options** - (Graph Results Only) - Allows user to adjust various graph options.
- **Visual Report Control -** Opens Visual Report Control window.
- **Design Alerts -** Opens the Design Alerts window.
- **Excel Export Manager** - Opens the Excel Export Manager for exporting model results to Excel.
- **Annotation Manager -** (Workspace and Visual Report Only) - The Annotation Manager, available on the Tools menu, allows you to specify whether an annotation will be shown or hidden, and in which scenarios it should appear.
- **Fluid Property Viewers** - Gives options to open either the SteamCalc Viewer or Chempak Viewer.
- **Activate Modules -** Opens Activate Modules window where optional modules can be activated or deactivated.
- **eLicense Information** - Visible if AFT Impulse 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.
- **User Options -** Opens User Options window.

Related Topics

Related Blogs

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

Library Menu

- **Add Junction to Library -** (Workspace Only) - Prompts you to enter the name to be used in the Junction Library for the currently selected junction. After naming, adds junction to the library.
- **Edit Junctions -** Opens Edit Junctions panel where you can delete or rename entries in the Component Library.
- **Edit Fluids -** Opens Edit Fluids panel where you can add, delete, modify or review custom fluids.
- **Edit Pipe Materials -** Opens Edit Pipe Materials panel where you can add, delete, modify or review custom pipe materials.
- **Edit Fittings & Losses -** Opens Edit Pipe Fittings & Losses panel where you can add, delete, modify or review custom pipe fittings and losses.
- **Edit Solids -** (SSL Module Only) Opens the Edit Solids panel where you can add, delete, modify or review solids properties for slurry modeling.
- **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).
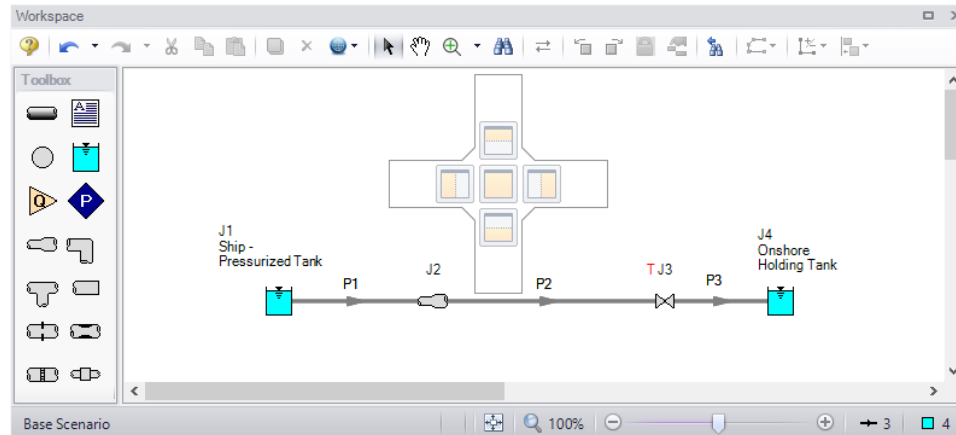
Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time

- **Sort Table By -** (Output Only) - Allows user to order tables in order of any variable and in ascending or descending order. Available in the Output window.
- **Rotate Icon Right or Left -** (Workspace Only) - Rotates icons for all selected junctions on the Workspace. Available in the Workspace window.
- **Rotate Selection Right or Left -** (Workspace Only) - Rotates selected pipes and junctions as a group. Available in the Workspace window.
- **Auto-rotate Icon -** (Workspace Only) - Rotates selected junction to line up with the connected pipes in the workspace. Available in the Workspace window.
- **Align -** (Workspace Only) - Aligns selected pipes and/or junctions on the Workspace. You can choose to align pipes and junctions, just pipes, or just junctions horizontally or vertically on the workspace. Available in Workspace window.
- **Pipe Drawing Mode -** (Workspace Only) - Select the type of grid layout that the model will use.
    - **2D Freeform -** Junctions can be placed anywhere on the Workspace.
    - **2D Orthogonal -** Toggles orthogonal drawing capability on the Workspace. This toggle is a shortcut to enabling the feature in User Options window. See Orthogonal Drawing Overview for more information.
    - **Isometric -** Uses an isometric grid.
- **Snap to Grid -** (Workspace Only) - Toggles Snap to Grid feature on the Workspace. When ON, all Workspace pipes, junctions and annotations snap to an grid specified in User Options window. Available in the Workspace window.

- **Show Grid -** (Workspace Only) - Toggles Show Grid feature on the Workspace. Shows or hides grid that is specified in User Options window. Available in the Workspace window.

- **Make Selected Pipes and Jcts Orthogonal -** (Workspace Only) - Makes all selected pipes and pipes connected to selected junctions on the Workspace orthogonal. Available in the Workspace window.

- **Flip Vertex -** (Workspace Only) - Available when orthogonal drawing is enabled. This will flip the vertex of all selected pipes on the Workspace. Available in the Workspace window.

- **Scale/Flip Workspace -** (Workspace Only) - Scales (shrinks or expands) or flips either selected or all pipes and junctions on the Workspace. Available in the Workspace window.

- **Set Pipe and Junction Locations to Parent -** (Workspace Only) - Sets locations of pipe and junction labels to the same position as the parent scenario. Available in the Workspace window.

- **Reset Label Locations -** (Workspace Only) - Sets locations of pipe and junction labels to their default locations. Available in the Workspace window.

- **Disable Pipes Snapping to Isometric Grid** - (Workspace Only) - Toggles Snap to Grid feature on the Workspace for the selected pipe. When Snap to Grid is disabled for that pipe, the pipe end point no longer snaps to grid points and pipe segments do not follow the Isometric lines. Available in the Workspace window.

- **Add/Remove Segments -** (Workspace Only) - Allows user to Add Segment (split single into two pipes), Add Segment at Right Angle, Add Multiple Segments, Remove Segment, or Remove all Segments. Available in the Workspace window.

- **Lock Object -** (Workspace Only) - Locks selected pipes and junctions on the Workspace so they cannot be moved. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction. Available in the Workspace window.

- **Reverse Direction -** (Workspace Only) - Reverses the reference positive flow direction of all selected pipes on the Workspace. Available in the Workspace window.

Related Topics

Related Blogs

AFT Responds to 2020 Customer Survey Feedback

Window Menu

- **Workspace -** Makes the Workspace window active.
- **Model Data -** Makes the Model Data window active.
- **Output -** Makes the Output window active.
- **Graph Results -** Makes the Graph Results window active.
- **Visual Report -** Makes the Visual Report window active.

Related Topics

Help Menu

- **Help File -** Opens the help site home page
- **Use Offline Help -** Indicates that the PDF help file will be used instead of the Online AFT Help site when the help buttons in interface windows are clicked. Note that the PDF help file must be downloaded from the PDF Downloads page on the online help site prior to using this option.
- **Quick Start Guide -** Opens quick start guide on aft.com in English or metric units
- **Quick Start Video -** Opens ~15 minute quick start video on aft.com in English or metric units
- **Watermammer Assistant** - Opens the Waterhammer Assistant window, which displays tips to increase the efficiency of the model
- **Keyboard Shortcuts -** Opens keyboard shortcut help page in local help system
- **Show Examples -** Opens help system with links to various example problems
- **Video Tutorials -** Opens video tutorials on aft.com in English or metric units and English or Spanish language
- **Technical Seminars -** Opens information on training courses on aft.com
- **Flow Expert Package -** Opens information on aft.com for individualized help from AFT engineers for training, modeling, consulting, and more
- **Online Learning Center -** Opens learning center on AFT website with links to resources such as blogs, video tutorials, webinars, etc.
- **AFT Home Page -** Opens home page on aft.com
- **AFT Blog -** Opens blog 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 Impulse release
- **AFT Notifications -** Opens page on aft.com where you can sign up for email notifications for new software updates and releases
- **About AFT Impulse -** Opens window in AFT Impulse with information on current AFT Impulse version, license number, etc.

Related Topics

Toolbars

The Toolbars offer quick access to the features used most frequently in AFT Impulse. The figures below show the Toolbars. There are actually six toolbars in all. One is the Common Toolbar, and it is always shown. Its features are common to all five Primary windows. In addition, there is one Toolbar for each Primary window, offering features appropriate for work in that window. You only see two Toolbars at a time in AFT Impulse. The function of each icon in a given Toolbar appears when you pass the mouse over the icon button without clicking on the button. Each of the functions on the Toolbar is also available through the menu system.
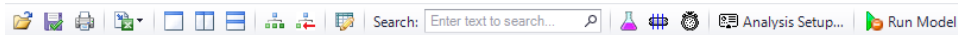
Figure 1: Common Toolbar

Figure 2: Workspace Toolbar

Figure 3: Model Data Toolbar

Figure 4: Output Toolbar

Figure 5: Graph Results Toolbar

Figure 6: Visual Report Toolbar

Related Topics

Print Preview

The Print Preview window allows you to access special print features such as Print Preview and Fit to One Page. It is accessed from the File menu or using the shortcut CTRL + P.

The Print Preview window also offers you a preview of the print content and format for all Primary Windows. By clicking the Print Preview button you can review your printed material at several zoom states. Individual pages can be printed.

The Fit to Page(s) feature causes the graphic image generated from the Workspace or Visual Report windows to be compressed onto the specified number of pages. This is a convenient feature for creating reports of your model. AFT Impulse supports whatever printer page sizes are supported by your printer, so use of legal paper or larger may allow you to get more visible details. The graphics can also be centered vertically and/or horizontally. You also can fit your content onto multiple pages.

The Page Setup option in the Print Preview window allows you to set the page orientation, paper size/type, and margins.

Additional special printing features affect the Workspace and Visual Report. The Selected on Workspace option works with Workspace printing by selectively printing the pipes and junctions currently selected on the Workspace.

By default, AFT Impulse prints the junction icons for the Workspace and Visual Report with the same colors as displayed on the screen. It also prints the background color as the same color as your Workspace or Visual Report background. You can also specify that the junction icons be printed in black and white (still with the color background) or that the entire printout be black and white.

If you added a background image from the File menu, you can include this picture in the printout.

Finally, if you have a grid displayed on the Workspace or Visual Report, you can include the grid in the printout.

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

Related Topics

Related Blogs

Quick Guide to Custom Output

Primary Window Overview

AFT Impulse has five Primary Windows. The Primary Windows are tabbed and can be pulled out of the main window onto separate screens. The Primary Windows are permanent in that you work in one of these windows at all times. The Primary Windows work together to provide tools for entering model input, analyzing results for accuracy, and preparing results for documentation.

Of the five primary windows, two are input windows, two are output windows, and one displays both input and output information. Figure 1 shows the relationship between workflow using the primary windows.

- The Workspace window allows you to build the model visually and see the model layout.

- The Model Data window is the text-based complement to the Workspace window. The Model Data window shows input data in text form. This window works hand-in-hand with the Workspace window to provide exceptional flexibility in manipulating data.

- The Output window displays the results of the analysis in text form and lets you produce attractive, effectively organized printed output.

- The Graph Results window provides full-featured Windows plotting capability. Here results can be viewed in a variety of ways for evaluating system transients or identifying important trends. The Graph Results window also allows graph customization.

- The Visual Report window merges the output data with the pipe system layout from the Workspace to present a unique perspective of the results. This window also allows customized layout of the information for documentation and presentation purposes. An alternate use for the Visual Report window is to show textual input data combined with the input schematic.

You can change between the five Primary Windows by selecting them from the Window menu, the toolbar, or by clicking the appropriate tab within the AFT Impulse window.
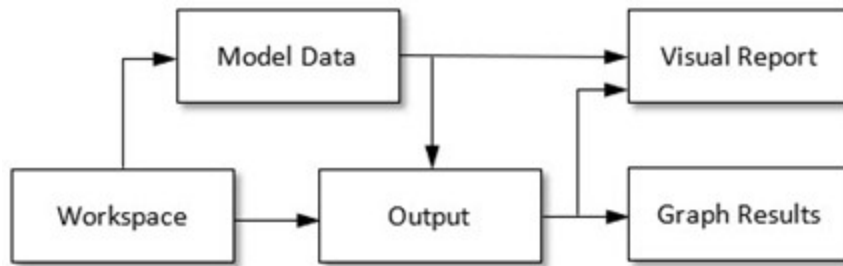
Figure 1: Primary window workflow in AFT Impulse

Related Topics

Status Bar

AFT Impulse's Status Bar displays contextual information about the current scenario or operation.
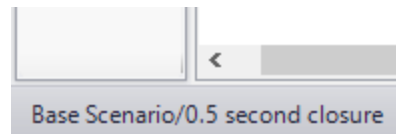
Figure 1: Current scenario displayed in the Status Bar

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.
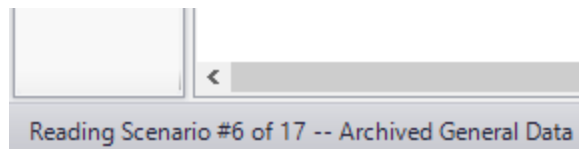
Figure 2: Status Bar displaying progress when opening a model

Related Topics

Quick Access Panel

The Quick Access Panel as shown in Figure 1, contains several useful features. The Quick Access Panel allows you to manage your scenarios in a timely fashion, navigate significantly large models with the Workspace Overview Map, activate modules, monitor the Analysis Setup window, view a pipe or junctions input properties and output results, control what graph parameters are displayed, and manage Graphs using the Graph List Manager.
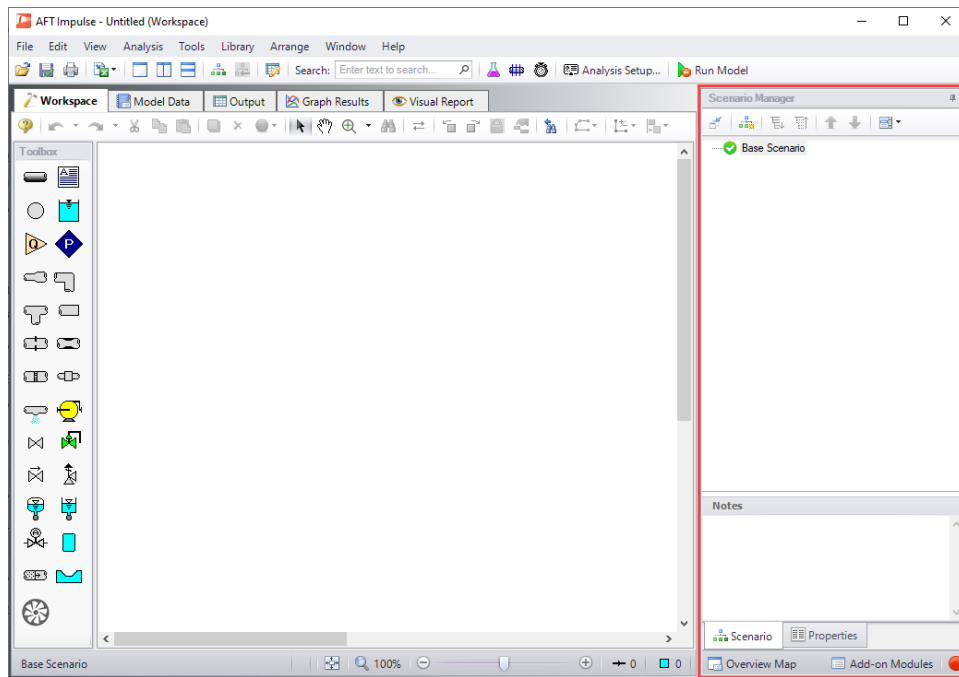
Figure 1: The Quick Access Panel

By clicking on the "Scenario" tab in Figure 2 below, you have the full scenario customization ability as you would with the Scenario Manager. You can create child scenarios, expand or collapse various families of scenarios, and utilize many other Scenario Manager features here as well.
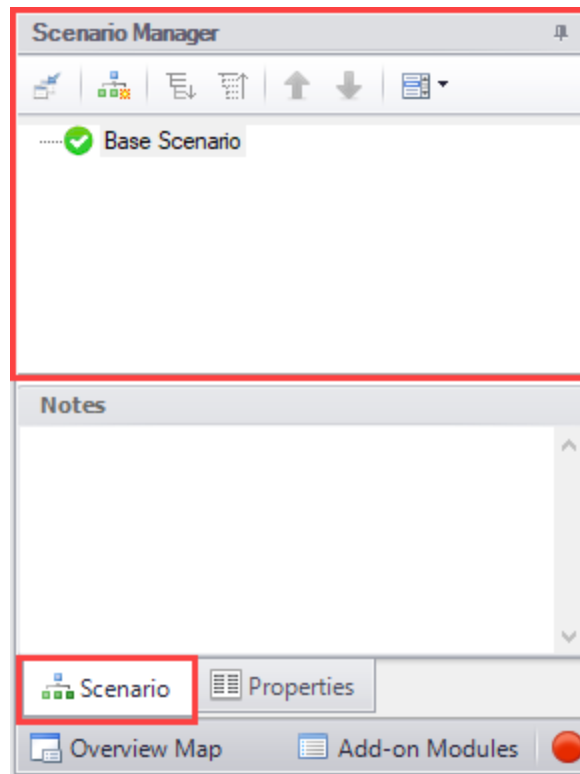
Figure 2: The Scenario Manager

When a model is significantly large in size, it can be troublesome to have to constantly drag the model around on the Workspace to get to a specific area of the model you want to view. By clicking on the "Workspace Overview Map" button in the red box in Figure 3, you will bring up a map that is a bird's eye view of the model. Drag the small square in the Workspace Overview Map over various parts of the model, then zoom in and out of the model. You can also click the "Fit" button to see the full model zoomed out on the Workspace, and use the mouse wheel to zoom in and out.
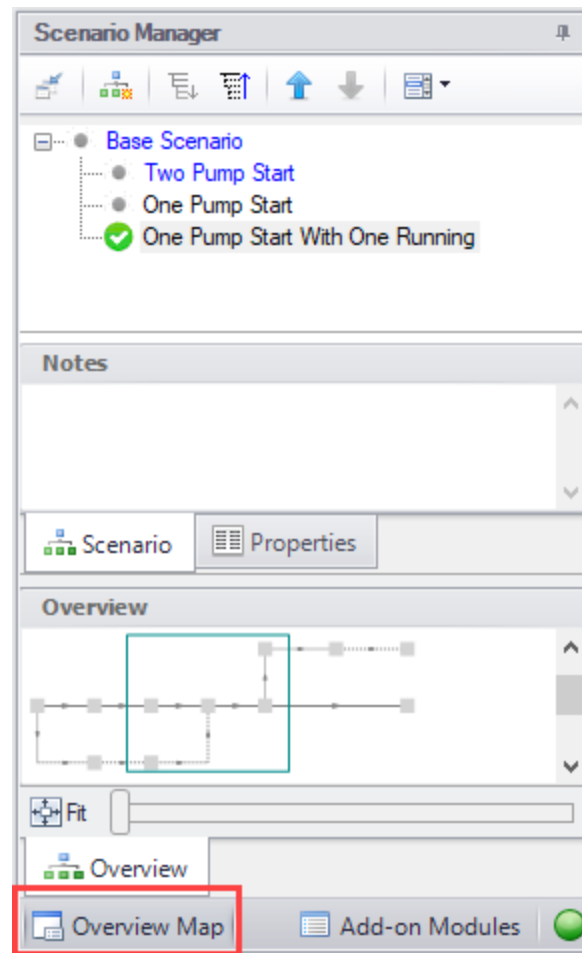
Figure 3: The Workspace Overview Map

Clicking the Modules button in the bottom right of the Quick Access Panel in Figure 4 allows you to quickly activate or deactivate the SSL and PFA modules.
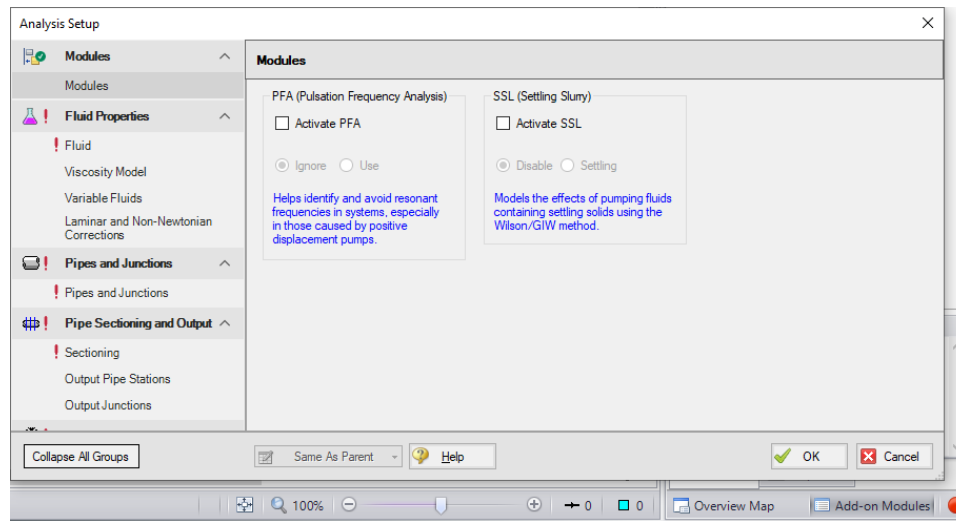
Figure 4: Activating the Modules from the Quick Access Panel

In the bottom far right of the Quick Access Panel in Figure 5 is the Status Light. Clicking on the Status Light will display the Analysis Setup window. When each item in the Analysis Setup window is fully defined, it will have a green check mark. When all necessary items in the Analysis Setup window are completed, the Status Light will turn from red to green. Depending on the Analysis type and module selection, additional items will appear in the Analysis Setup window.
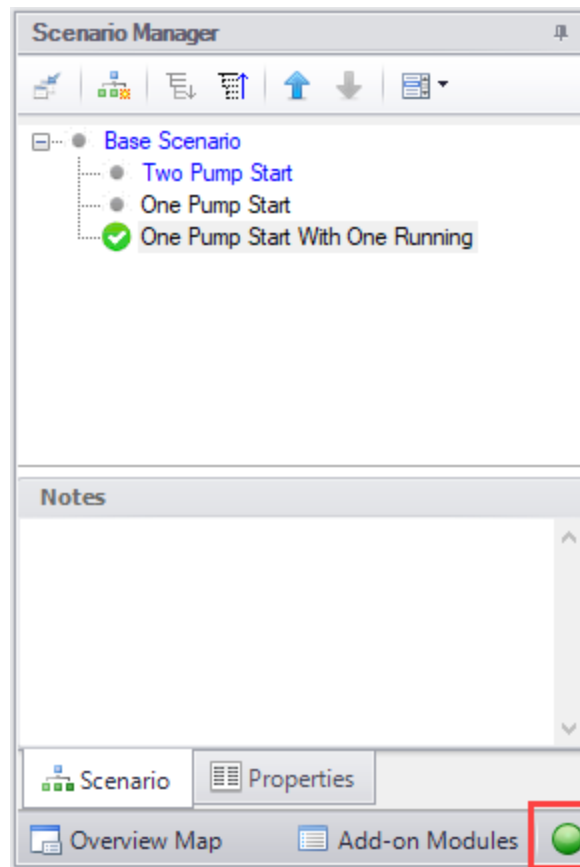
Figure 5: The Status Light in the QAP

As seen in Figure 1, there is a "Properties" tab in the Quick Access Panel. When pipes and junctions are drawn on the Workspace, you can quickly view the Pipe or Junction Properties by first selecting a pipe or junction on the Workspace, then clicking on "Properties" tab. This will display the Object Property data as well as any Output data if the model has been run. Figure 6 below shows an example of what the Properties tab would look like when clicking on pipe P1 on the Workspace after it has been defined and the model was run.
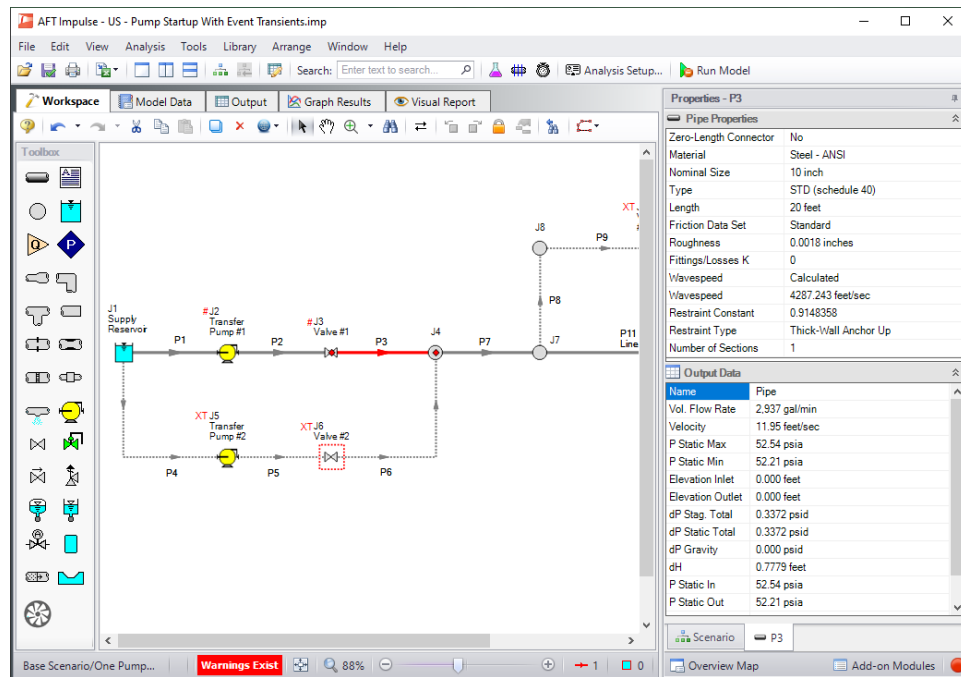
Figure 6: Viewing the Pipe (or Junction) Properties and Output Data from the Quick Access Panel

Minimizing the Quick Access Panel

By Default, AFT Impulse opens with the Quick Access Panel automatically expanded. The Quick Access Panel can be easily collapsed and pinned to the side of the Workspace to allow for a larger usage area of the Workspace. Figure 7 shows the Quick Access Panel initially expanded. By clicking on the thumbtack pin button highlighted with the red box, the Quick Access Panel will then become collapsed as shown in Figure 8. After the Quick Access Panel is collapsed, it can be temporarily expanded again simply by hovering over the "Scenario" sidebar with the mouse.
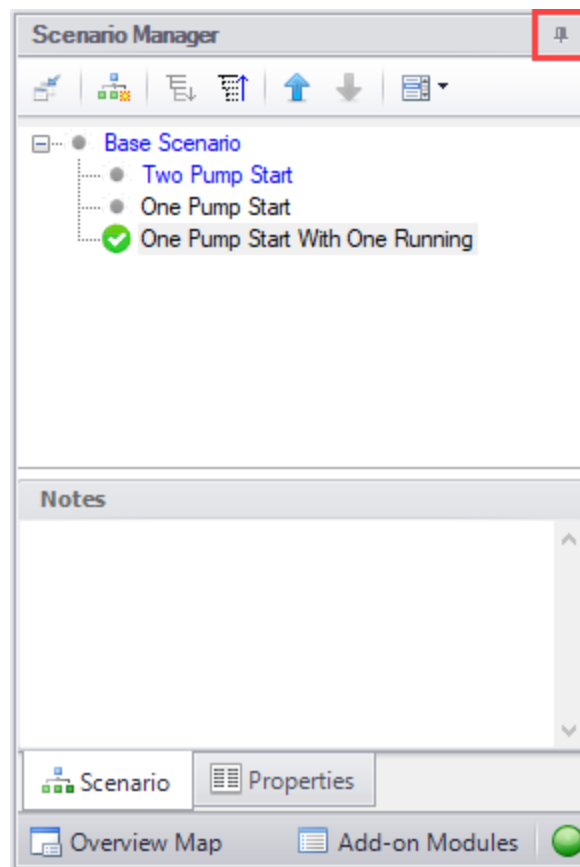
Figure 7: The Workspace with the Quick Access Panel Expanded
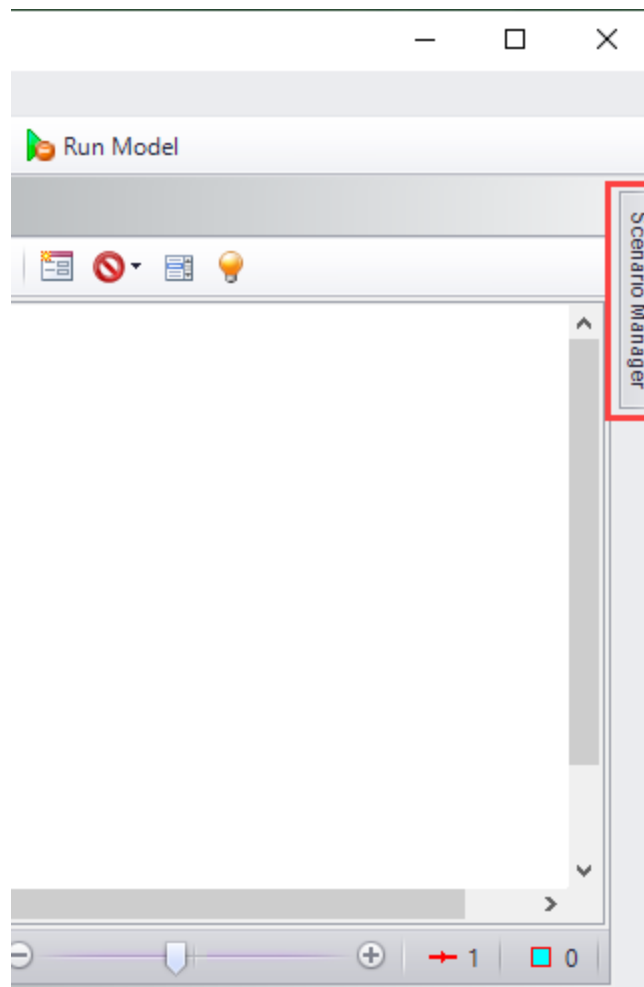
Figure 8: The Workspace with the Quick Access Panel collapsed

Related Topics

Model Status Light

The light at the right of the Status Bar located at the bottom of the AFT Impulse window, which indicates whether or not the model can be run.

Clicking on the Model Status Light will show the Analysis Setup window.

Related Topics

Solution Progress Window

Solution Progress indicates the current state of the steady-state solution iterations and the transient simulation.

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 Solution Progress Bar shows how far the transient simulation has progressed.

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 simulation is complete, options to view Output, Graph Results, and Visual Report become available.

Pausing thesimulation 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

Waterhammer Assistant

AFT Impulse checks the model with the Waterhammer Assistant before a transient simulation to see if it has any advice. If it does, the Waterhammer Assistant is displayed. The user then has the option of canceling the run to take the suggested actions or ignoring the advice and continuing. If you do not want the Waterhammer Assistant to be consulted before each run, you can specify this choice in the User Options window on the Tools menu. The Waterhammer Assistant can also be opened from the Help menu.

An example of something the Waterhammer Assistant will check for is if there are any pipes that could be combined to reduce runtimes using the Combine Pipes tool.

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).
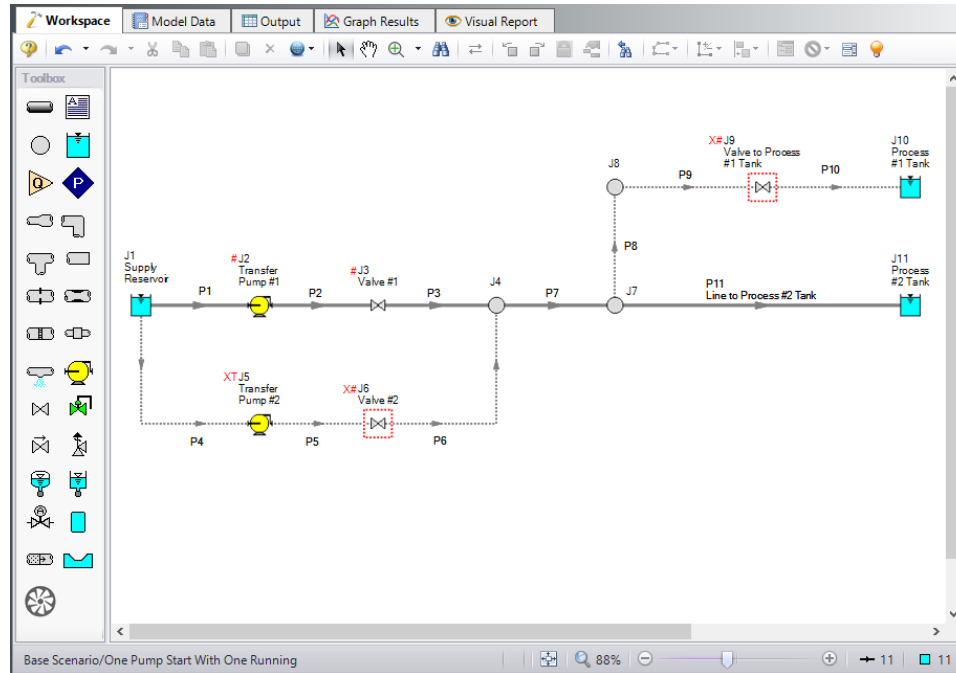
Figure 1: The Workspace window

Display and Printing Features

The Workspace image can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The Print Preview window allows the above outputs to be applied to all or selected Workspace objects.

Related Topics

Merging Models

Two models can be merged together using the command on the File menu. Here's how:

- Open the first model
- Choose Merge on the File menu
- From the file list, select the model to be merged

If there are duplicate numbers, the objects in the second model will be changed. The second model will be selected (highlighted) so that you can move it into position (be careful not to click off the selected portions or the second model may be deselected).

Tip: To move a selected group of objects, use a junction icon, instead of a pipe, to drag the group. This will avoid missing the pipe and clicking on an empty area of the Workspace which will deselect your group.

Merging Models with Multiple Scenarios

If the model you are merging from has multiple scenarios, only the base scenario can be merged. If you want to merge one of the children of that model, do the following:

1. Open the model which you want to merge.
2. Open the Scenario Manager.
3. Select the child scenario you want to merge.
4. On the Other Actions button, choose the Save Scenario to File Without Children.
5. Enter a name and click OK. This process creates a new Impulse 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

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.

Impulse 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

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 two options available to adjust the Magnifier view:

- **Zoom Level** - Adjusts the level of magnification used
- **Magnifier Size** - Adjusts the size of the magnifier circle
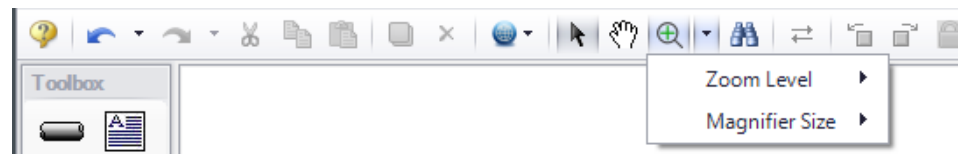
Figure 1: Magnifier tool on the Toolbar

The Magnifier tool can also be used in the Visual Report window.

Related Topics

Annotations

The Toolbox tool to the right of the Pipe Drawing Tool is the Annotation Tool. With this tool, you can draw a box that will contain an Annotation displayed on the Workspace.

The Annotation Tool allows you to create a text message, show an outline around the message, insert pictures in the message box, and draw lines with pointers to certain areas of the model. Different style, color and font options are provided.

Annotations can be "attached" to the Workspace, a pipe, or a junction. When attached to the Workspace, the annotation always remains in the same location until you move it. When attached to a pipe or junction, the annotation will move when the pipe or junction is moved. The Scenario Options tab allows you to specify in which scenarios the annotation appears.

Annotations can be layered to be either "Always in Front" or "Always in Back".

Annotations print out with the model.

Annotations can be displayed or hidden among different scenarios using the Annotation Manager from the Tools Menu..

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.

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.
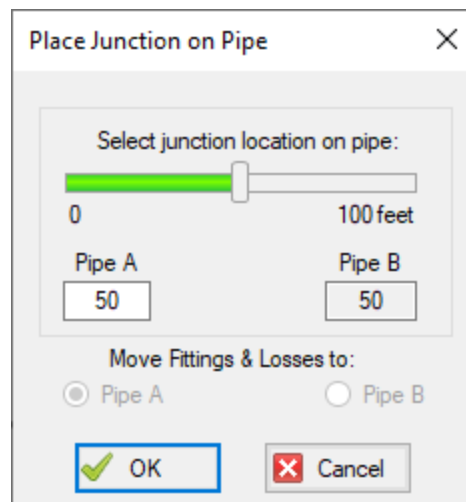
Figure 1: Split pipe keyboard shortcut

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
- Go to the Visual Report window using CTRL + I

Other

- Many commands use common windows shortcuts. Some of the most common are listed here:
- Run the model: CTRL + R
- Save the model: CTRL + S
- Open a file: CTRL + O
- Undo the last action: CTRL + Z
- Cut: CTRL + X
- Copy: CTRL + C
- Paste: CTRL + V
- Other available shortcuts can be seen listed by their command on the menus

Related Topics

Undefined Objects Window

This window is launched from the View menu by selecting List Undefined Objects, or can be viewed in the Pipes and Junctions panel.

This window displays all undefined pipes or junctions in the model. Clicking on the undefined object will display the undefined properties in the list below. All undefined data for all objects can be saved to the file or copied to the clipboard.

Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.

Related Topics

Raw Data Changes

When other objects in the model use the same base raw data as the current object and the data is changed, you can decide whether to use the new data on the other model objects.

This frequently happens when you have multiple junctions based on the same x-y data for 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.
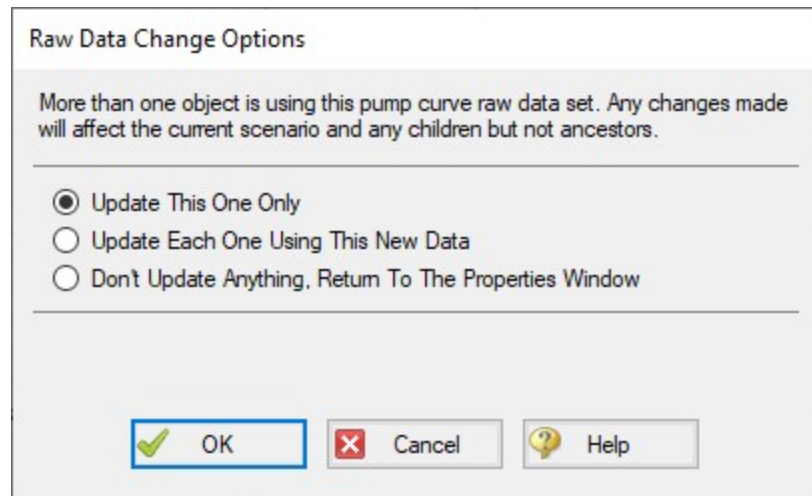
Figure 1: Raw Data Change Options

Related Topics

Duplicate Special

Duplicate Special allows pieces of the model to be copy & pasted, while simultaneously applying other helpful settings as explained below.

- **Increment All Pipe and Junction Numbers By** - Adds the specified number to all pipe/junction numbers for the created object duplicates. Useful for maintaining consistent numbering

- **Create Group** - Adds the duplicates to a group.

- **Hide in Output** - Automatically hides the pipe and junction duplicates in the Output window so only the original pipes and junctions will be seen.

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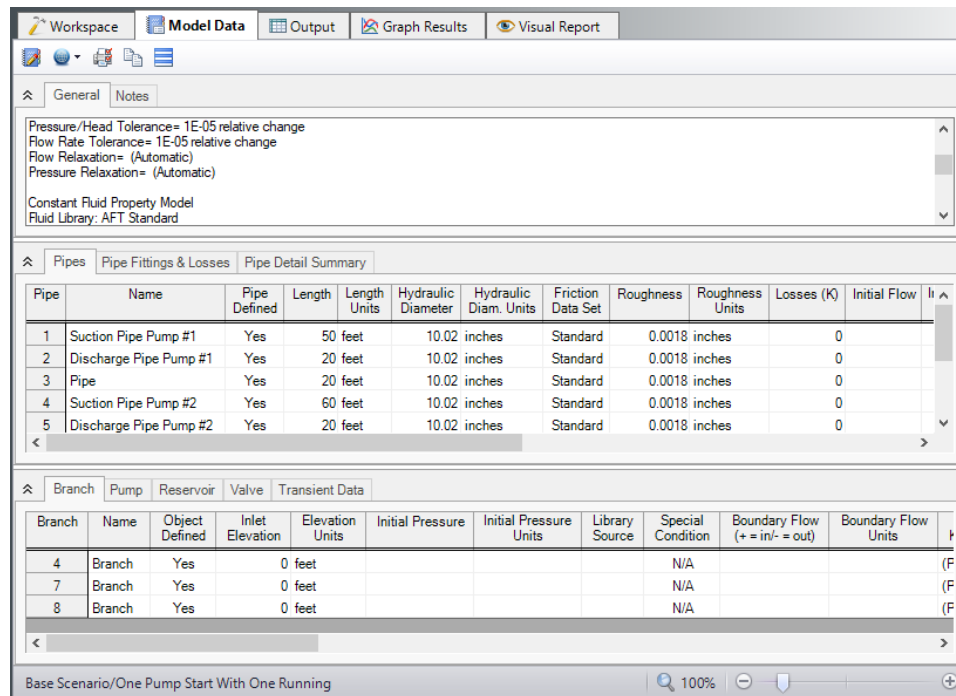
Figure 1: The Model Data window displays input in text form

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. All transient data entered for the junctions can be seen in the Transient Data tab on the far right.

To edit information in the Model Data window, you can open the Properties Window for the pipe or junction. This can be done by double-clicking anywhere in the row for the pipe or junction to be edited. Alternatively, you can use the Global Edit windows opened from the Edit menu.

Display and Printing Features

Model Data Control lets you specify the combination of pipe, junction, and general information you want to see in the Model Data Window.

The Model Data Print Content window lets you specify the content to include in the printed Model Data report and select the font to be used for printing. This window is opened from the Tools menu (or Toolbar).

The Model Data window contents can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Saved directly to an Adobe PDF file
- Sent to the printer

Related Topics

Related Blogs

Having model convergence problems? Start here!

Model Data Control Window

The Model Data Control allows you to customize what data to show for the pipes and for each junction type and also which pipes and junctions to show (see Figure 1). It can be opened from the Tools menu (or Toolbar).

Pipe Data Display

The data to be shown for the pipes is selected on the left side of the Model Data Control window. Only the items with check marks will be shown in the Model Data window. All pipes shown will report the selected items.

Junction Data Display

The data to be shown for the junctions is selected on the right side of the Model Data Control window. The data is grouped by junction type. However, some data is general and applied to each of the junction types. Select the junction type from the list and then select the data for that junction type. Each type also includes the general data which may be selected/deselected.

You can select/deselect the general data for all junction types simultaneously by clicking Set Common.
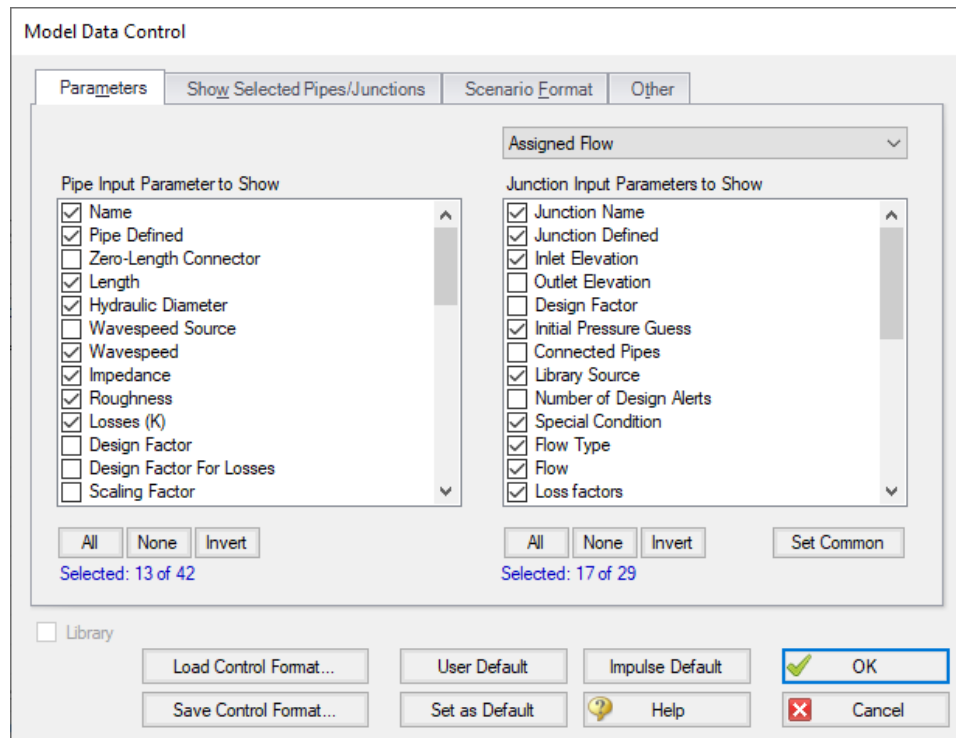
Figure 1: The Model Data Control window allows selection of the parameters to show in the Model Data window

Model Data Format

The Model Data Control format can be saved to a file so that the format can be quickly applied to other model files. There are multiple ways to do this. The Model Data Control format can be saved to the internal User library by clicking the Set As Default button, then choosing User Default in a new model file to apply the settings.

The Model Data format can also be saved to an external file, which can then be shared with different users. The external file can be generated by clicking the Save Control Format button at the bottom of the window. The format can then be applied to other model files by clicking the Load Control Format button.

Show Selected Pipes and Junctions

Specific pipes and junctions can be shown in Model Data. Click the Show Selected Pipes/Jct tab, then click the items in the lists to select the pipe(s) and junction(s) to be shown.

Other Data

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 junction curve data is to be displayed with the Model Data, whether to display Design Alerts, and whether to display Force Sets.

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

Output Window

The Output window is the primary vehicle for communicating the results of an analysis in text form (see Figure 1). The Output window follows the same general window organization as the Model Data window.

When a model is run, the Solution Progress window appears. After the solution is obtained, you can view the results in the Output window by clicking the View Output button.

If the information in the Output window is not in the desired format, you can choose Output Control from the Tools menu (or Toolbar) to modify the format. The Output Control window lets you specify the parameters, their units, and the order of their display in the Output window.

The Output Print Content window available on the Tools menu (or Toolbar) lets you specify the content to include in the printed Output report and select the font to be used.

The Sort window opened from the Arrange menu (or toolbar) offers the ability to sort the steady-state Pipe Results or Junction Results tables according to the values in any of the columns. Transient results in the Transient Max/Min tables can be sorted using the right-click context menu.

The Transfer Results to Initial Guess feature on the Edit menu (and Toolbar) sets all pipe and junction initial steady-state flows and pressures equal to the converged results, which causes future runs to converge much faster.

You can view the input data for individual pipes or junctions directly from the Output window in one of two ways. First, if you press down the right mouse button on the pipe or junction number in the far left column of the table you will see the Inspection window. Second, if you double-click the pipe or junction number you will open the Properties window in read-only form where you can review input in detail.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to 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 displays output in text form

Right-click Context Menu Actions

There are several options available when right-clicking on a cell, row header, column header, or entire table in the Output window:

- **Column Header**
  - **Change Units -** Quickly redisplay the selected column in a different unit.
  - **Sort** - Sort the table in Ascending or Descending order on the selected column.
  - **Add *"Parameter"* for all *Table* Objects to Excel Export Manager** - Add this column to the Excel Export Manager.
  - **Open Excel Export Manager** - Directly open the Excel Export Manager.
  - **Copy Selected Cell Data** - Copies all of the values in the column to the clipboard.
- **Row Header**
  - **Add all *Object* Parameters to Excel Export Manager** - Add this row to the Excel Export Manager.
  - **Open Excel Export Manager** - Directly open the Excel Export Manager.
  - **Find on Workspace** - Quickly locate an object on the Workspace.
  - **Copy Selected Cell Data** - Copies all of the values in the row to the clipboard.
  - **Open Properties Window** - Opens the properties window for the associated pipe or junction.
- **Table (upper left cell)**
  - **Add *Object* Table to Excel Export Manager** - Add the entire table to the Excel Export Manager.
  - **Open Excel Export Manager** - Directly open the Excel Export Manager.
  - **Copy Selected Cell Data** - Copies all of the values in the table to the clipboard.
- **Cell**
  - **Add *Object "Parameter"* to Excel Export Manager** - Add this cell to the Excel Export Manager.
  - **Open Excel Export Manager** - Directly open the Excel Export Manager.
  - **Create Design Alert** - Create a Design Alert related to the selected object and value.
  - **Copy Selected Cell Data** - Copies the value of the cell to the clipboard.

Output Window Updates

Whenever you change the input model, all output windows are erased. This prevents you from changing your input model, forgetting to rerun it, and mistaking the previous Output window for the current model's results.

Steady Flow Results

Figure 2 shows the steady flow results in tables for pipes and junctions as specified in the Output Control window.

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

Transient Flow Results for Pipes

You can review the transient results for any pipe station for any time which was saved to file as specified in the Output Pipe Stations panel. These results are shown in the Output window Transient Output table (Figure 3)

Figure 3 shows the transient flow results at the initial time step (which is equal to zero here). The columns of data shown in Figure 3 are specified in the Output Control window. Using the Time Slider will change what time step results are displayed.

Note that the pipe stations shown are those saved to the output file.

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

Transient Max/Min Results

The Transient Max/Min table in the Pipe Output section shows the maximum and minimum values for each parameter in each pipe, the time it occurred at, and the station it occurred at. Thus, for each transient parameter added in the Output Control window, 6 columns will be shown in the Transient Max/Min table. Figure 4 below shows an example.

The default format of the Transient Max/Min table is the summary version, showing the maximum and minimum values for the whole pipe, as described above. A detailed format is also available, where the maximum and minimum values for each parameter at each station are shown, along with the time they occurred at. Thus, for each transient parameter added in the Output Control window, 4 columns will be shown in the Detailed Transient Max/Min table. Figure 5 below shows an example.

The Output Control window, under the Format & Action tab can be used to switch between the detailed and summary views.

The information shown in the Transient Max/Min tables is not dependent upon what pipe station and time step output is saved for a given simulation.

Figure 4: Summary version of the Transient Max/Min table

Figure 5: Detailed version of the Transient Max/Min table

Transient Event Messages

When events occur at junctions during the transient simulation, a log of all events is displayed in the two Event Message lists. In the first list events are sorted by junction number. In the second list they are sorted by time. The list includes when check valves close (and re-open), relief valves open (and re-close), control valves lose control (and regain it), as well as all standard event transients.

Results Diagram

Results diagrams are available for steady state pump results (see Figure 6). These diagrams allow you display the information in a more intuitive manner, copy the image of the diagram to your clipboard, or print the diagram. The option to show the results diagram can be found in the pump summary tab.
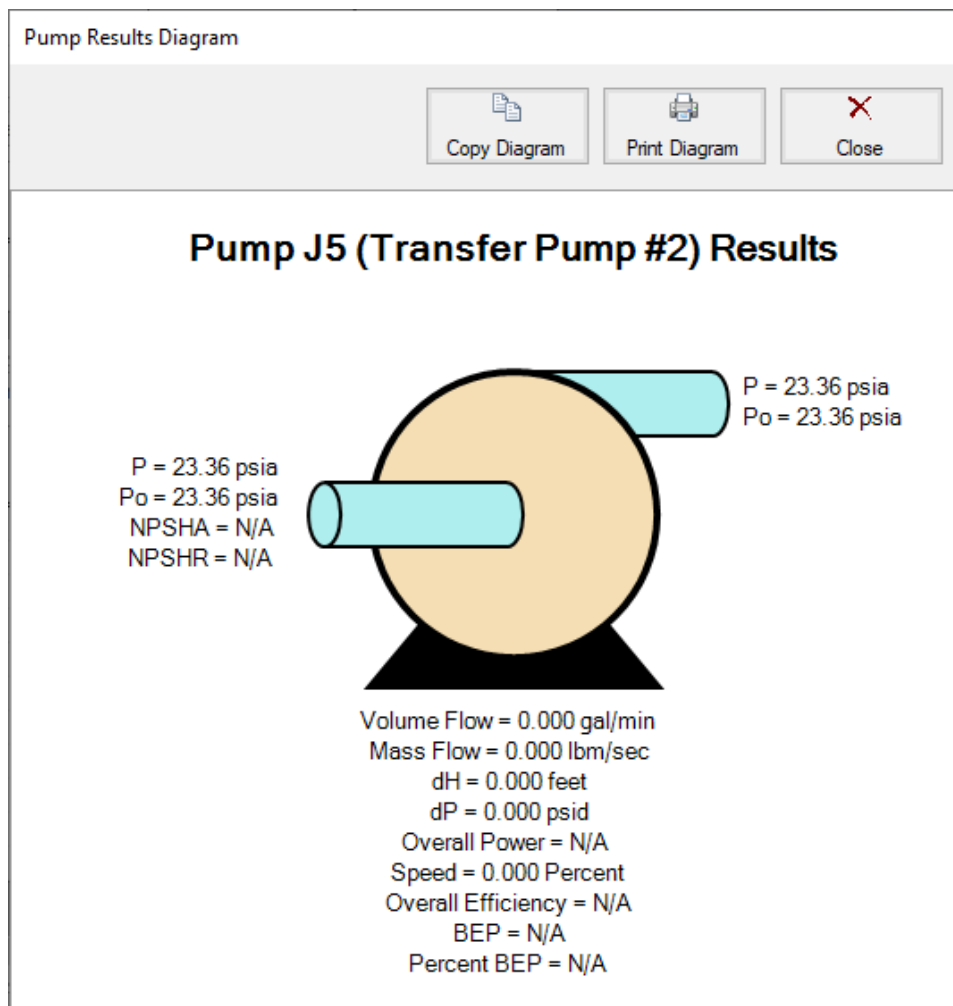
Figure 6: Pump Results Diagram

Printing and Exporting

Data in the Output window can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Exported directly to Excel
- Saved directly to an Adobe PDF file
- Sent to the printer

The printed and Adobe PDF output follows the same flexible, organized, and accessible format as the Output window itself.

Related Topics

Output Control Window

The Output Control window (see Figure 1) allows you to customize the parameters shown on the Output Window.
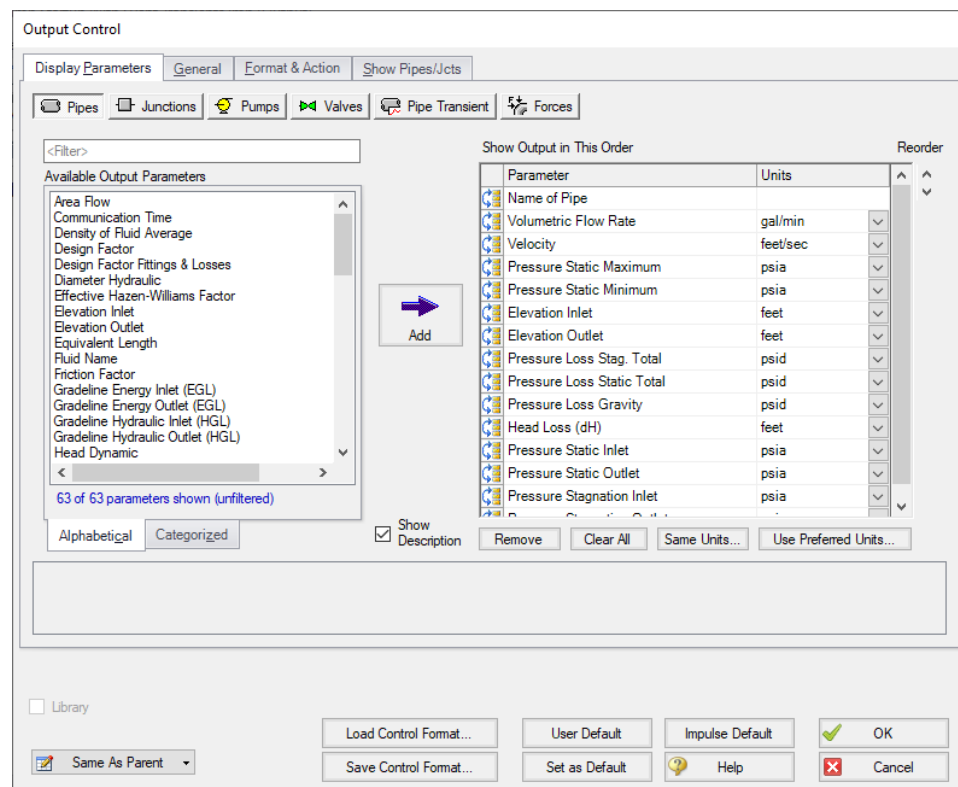
Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe Output parameters are shown. The other output parameters function the same way

Display Parameters

The Display Parameters tab controls the parameters to be shown for the pipes, junctions and summary reports. The output type is selected from the group of buttons at the top. For each

output type, the list on the left side shows all available parameters for the selected report. For the pipe and junction output, this list can be viewed in alphabetized or categorized mode, which makes it easier to find a parameter.

Choose a parameter to be shown by selecting it on the left-hand list and clicking the Add button, or double-clicking the parameter name. The selected item will appear in the list on the right either at the bottom or at the previously selected location. The list on the right shows the order of the parameters to be displayed in the output and the units (if any) in which to display the results.

You can change the order of the output by selecting a parameter on the right-hand list, then clicking the Reorder scroll bar on the far right or by clicking and dragging the row using the handle on the left. The list will be reordered accordingly. The output is presented from left to right in the order defined in this list.

The units used for a parameter can be changed by selecting the parameter then choosing the desired units from the adjacent dropdown list. By selecting the units of interest, you can obtain your results in whatever units you find most convenient and meaningful. This means that you can enter all your input parameters in one set of units (or a variety of units, for that matter), and have all your output parameters in a completely different set of units.

To set selected units for both pipes, junctions and summary reports to the preferred units (set in the User Options window), click the Use Preferred Units button. If you change the units of a parameter and you would like that unit to be used for all similar parameters in the list or entire output, select the Same Units button.

You will note that some output parameters do not offer the ability to specify output units. These parameters either do not have units associated with them (such as Reynolds Number or friction factor), or are in fact echoes of input parameters (such as Pipe Nominal Size).

Several Special Summary Reports can be displayed in the Output window. These include the following:

- **Pump Summary**
- **Valve Summary**
- **Pipe Transient Summary**
- **Forces Summary**

In addition, if you are using the SSL module, there will be a slurry report available.

General

The General Section is the text area located at the top of the Output window. The project title, reference information and solution balance display are controlled here.

You can enter a descriptive Analysis Title which will be used in the Model Data, Output and Visual Report windows. A title is required and can have up to 100 characters.

You can keep a lengthy explanation or any other documentation about your model in the Reference Information section. Names of projects, individuals, and assumptions can all be kept with the model. This information can then be included in your reports to improve tracking.

There is an option "Show Solution Balance Summary" that is defaulted to off. This option toggles the display of a general output tab showing the total flow entering and exiting each junction, as

well as the sum of those flows. This can be helpful in verifying the results as well as in some troubleshooting cases.

Format & Action

The results can be automatically sent to the printer or saved to a file when the model has reached convergence. Additionally, the results can be automatically saved as initial conditions when the model is run. By saving the results as initial guesses, the iteration time for AFT Impulse 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.
- 0.09876
- **Align Only When Max/Min < Limit-** aligns the data on the decimal point only if the maximum value in the column divided by the minimum value is less than a limiting order of magnitude that you set. If this ratio is greater than the limit, then each value in the column will be exactly the number of digits specified. Using the above example with a limit set to 4 then 1234.5/.09876 = 12500 or an order of magnitude of 5. This is greater than the limit. The entire column will not be aligned and will be displayed as:
  - 1235.
  - 0.09876
- **Use Exponential When Max/Min>Limit -** similar to the above method except that if the ratio is greater than the limit then the entire column will be shown in exponential format. Using the above example, the column would be shown as
  - 1.235E+03
  - 9.876E-02

These formatting options are for the entire Output window; however, the last two are calculated and applied on a column basis. In other words, if you choose the third option one column may be aligned but another may not be.

The output can be forced to always display in exponential notation by choosing the checkbox.

The Limit Event Options box can be checked to set maximum number of events which will be displayed in the Event Messages tabs that appear for cases such as valve transients. This can be useful in cases such as valve chatter.

For transient results, the maximum and minimum values for all selected parameters are shown in the Output window. The results can show the max/min values for all pipe stations, or only the max/min values for anywhere in the pipe. You can choose which type of report you want by selecting Detailed or Summary in the Max/Min Transient Table Contents area. For models with a large number of pipe sections, the Summary report is generally preferable because of the time required to display the Detailed table.

The Grid Headers and Design Alert highlight colors can be configured to create a visually appealing output for ease of reading.

The box next to Show Results Diagram Column can be unchecked to hide the column which allows easy access to the Pump Diagram for Steady State pump output.

Show Pipes/Jcts

You can choose which pipes and junctions are to be displayed in the Output window. Clicking a selected item in the list on the Show Pipes/Jcts folder tab will deselect it. Clicking a deselected item will select it. This feature is especially useful when you are interested only in a certain area of the model.

You can quickly define the output you want by doing the following:

1. Select the pipes and junctions on the Workspace
2. Open the Output Control window and select the Show Pipes/Jcts folder tab
3. Click the Workspace button for both the pipes and junctions
4. Click the Show button

Library Connections

In the lower left of the Output Control window there is a Library checkbox. If this box is checked, your Output Control parameters are setup as determined by the library to which you are connected. This is referred to as an *active* library. To make it inactive, 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. Impulse has built-in default parameters, units and settings which you can choose by clicking the Impulse Default button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of Impulse's defaults). To make your own default, first select the output parameters, units and settings you would like to use then click the Set As Default button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the User Default button. The saved settings are updated only when you click Set As Default.

You can save the output settings (except for the selected pipes/junctions) to a file by pressing the Save Control Format button and entering a file name. These settings are loaded again by pressing the Load Control Format button and choosing the file name. For example, you may have a final report format that is always desired. You can load this format before generating final results. If you have another format you use for reviewing model accuracy, you may want a larger number of parameters in the output.

The format files you create can be placed on a network for sharing among a group or company, 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

Related Blogs

Quick Guide to Custom Output

Back to the Basics: Calculating and Defining Head

AFT Responds to 2020 Customer Survey Feedback

Applied Standards

When an applicable standard is used in a calculation, an additional tab appears in the general area of the Output Window. This tab indicates what standard(s) were used for what calculations in the model.
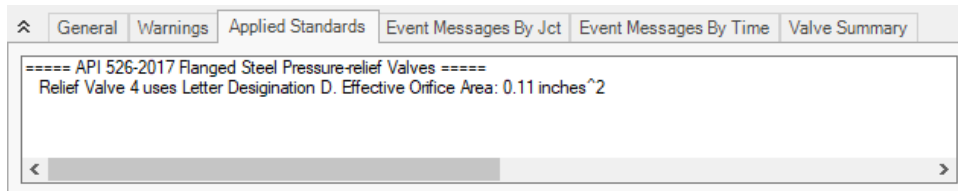
Figure 1: Applied Standards when applied for Relief Valve Orifice Area

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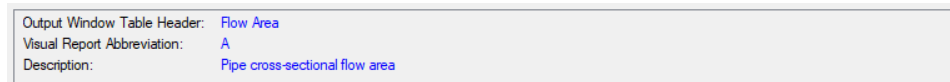
Figure 1: A description is provided at the bottom of Output Control for the selected parameter

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

Graph Results Window

The Graph Results window, shown in Figure 1, allows users to generate high quality printed graphs.

The Graphing Panel offers several types of standard graphs for transient pipe or junction, profile, and force information. Users can also control graph content, plotting information for multiple parameters, junctions, pipes, paths, or scenarios simultaneously.

Results can be graphed as linear or log versus linear.
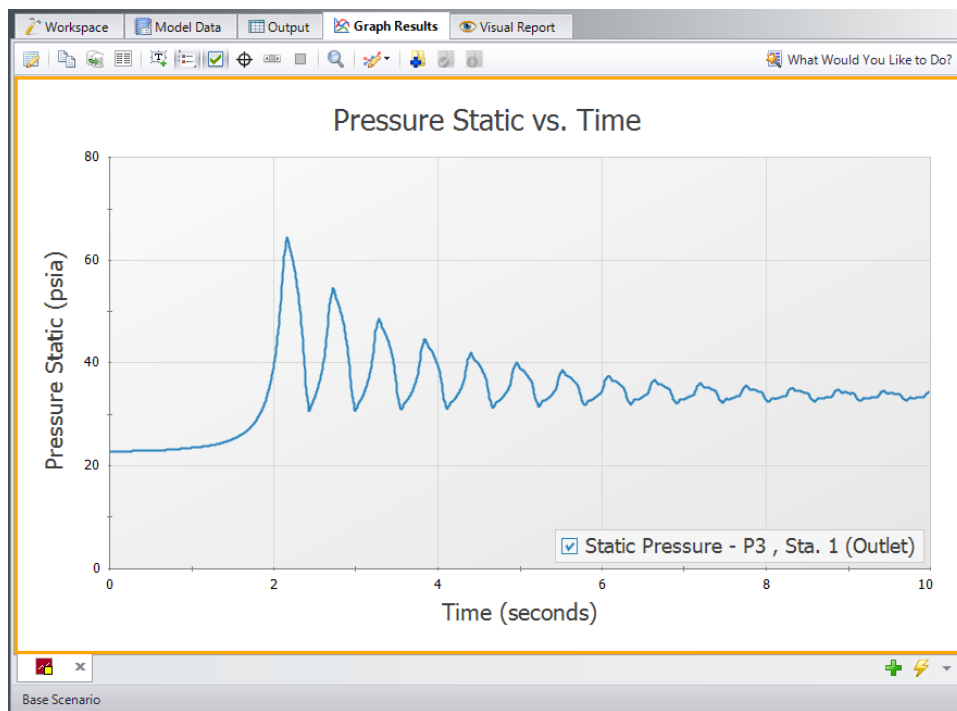
Figure 1: The Graph Results window offers full-featured plot generation

Graph List Manager

Allows the user to create, update, and even replace unique graphs that can be regenerated with new data. Multiple graphs can be displayed at the same time and organized into separate Folders. The default folder is named "My Graphs" and more folders can be created using the "New Folder…" icon on the Graph List Manager Toolbar or by right clicking on "My Graphs" and selecting "New Folder…"

Once a folder is created and selected, a Graph can be added to its list by selecting the "Add Graph to List" icon found on the Graph Results Toolbar. This can also be done by right-clicking

anywhere on the graph. Once Graphs are in Folder individual Graphs can be generated in unique tabs or each Graph in a Folder can be displayed in a single tab by right clicking on the Folder and selecting "Load Current Folder Graphs in New Tabs" or "Load Current Folder Graphs in Same Tabs". These options are available in the Graph List Manager toolbar as well. Other options that can be found by right clicking on a Folder or Graph in the Graph List Manager or on the toolbar are:

- **Load Graph in Current Tab** – Regenerates the graph in the currently active tab.
- **Load Graph in New Tab** – Creates a new tab and regenerates the graph there.
- **Load Current Folder Graphs in New Tabs** - Regenerates all the Graphs in a Folder in separate tabs.
- **Load Current Folder Graphs in Same Tab** – Regenerates all the Graphs in a Folder in a single tab
- **New Folder** – Creates a new Folder where more Graphs can be saved to.
- **Move Graph Up** – Moves the selected Graph up one level.
- **Move Graph Down** - Moves the selected Graph down one level.
- **Rename Graph…** - Changes the name of the Graph
- **Duplicate Graph** – Creates a copy of the selected Graph with a new name.
- **Delete Graph** – Removes Graph from the Graph List Manager
- **Move To Folder** – Moves the selected Graph to a previously existing Folder or a new Folder.

Graph Results Toolbar

The Graph Results Toolbar also has many options that are available by right clicking on the Graph Results space at any point. These options are:

- **Select Graph Parameters** – Select the type of Graph to create and its parameters.
- **Add Graph To List** – Saves the Graph to the currently active list allowing it to be regenerated in the future.
- **Update Current Graph In List** – Saves any changes to the current graph that have been made. (e.g. Changing the Graph Title or Line Style). Note that it is preferred to use the Replace Graph in List option if changes to the parameters or objects/paths being shown have been made.
- **Replace Graph In List** – Replaces the graph highlighted in the Graph List with the current graph that is shown.
- **Copy Graph Image** – Copies a .jpeg of the Graph to the clipboard
- **Copy X-Y Data** – Copies the X-Y used by the Graph to the clipboard
- **Show Data** – Displays the X-Y data used to generate the graph in a column to the right of the graph
    - 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 junction labels on the graph to be shown/hidden individually.

- **Zoom to 100%** - Resets the focus of the graph to 100%

- **Format** – Allows the user to change the Graph's format. Including
    - Header, Axis, and Legend names and fonts.
    - X and Y scaling
    - Line color and thickness

Graph Outline Colors

The active graph will always have an outline which indicates what state it is in.

- **Orange** – Graph not added to list
- **Dark Green** – Graph added to list
- **Light Green** – Graph added to list, but changes have occurred. These can be saved and make they graph dark green by selecting "Update Current Graph in List".

Graph Tab Icons

Each Graph Type has a specific color associated with it. This color will display in the box on the tab for the Graph. The colors are below:

- **Transient Pipe** – Dark Red
- **Transient Junction** - Dark Gray
- **Profile** – Purple
- **Forces** – Yellow

- **Frequency** - Light Brown
- **Blank Graph** – Light Gray

These tabs also display if the Graph in the tab is not saved or has been changed from the original Graph List item by showing a small yellow square over the tab's icon. If the current graph has been saved, there will be no yellow square.

Graph Tab Actions

- **New Tab** – Creates a blank tab in which a new type of Graph can be generate and added to the Graph List Manager
- **Tab Actions**
    - Close Active Tab
    - Close All Tabs
    - Close All But The Active Tab
    - Two Vertical Tab Groups
    - Two Horizontal Tab Groups
    - Reset to Single Tab Group
    - Tabs on…. – Selects the location of tabs (defaults to the bottom of the Graph)
- **Graph Tab Selection** – Selects a Graph for the Graphs currently in the tabs based on name.

Printing and Exporting

The Graph Results image can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The x-y data in the Graph Results graph can be:

- Copied to the clipboard
- Exported to a delimited file suitable for direct importing into spreadsheet software

Animations of graphs can be saved as .wmv or .mp4 files.

Data Point Limitations

By default, the maximum number of data points Impulse will plot for an individual graph is 500,000, though users can change this limit under the General -> Other section of the User

Options window. This limit is in place to help ensure users do not encounter memory issues related to plotting large numbers of data points. The allowed range of data points plotted is 10,000 - 1,000,000. The data point limit is applied to an individual graph. Two parameters plotted as stacked graphs count as two separate graphs. Two parameters plotted with one as a secondary y-axis graph count as one graph. One parameter plotted for two scenarios on the same graph counts as two graphs (i.e. the data point limit is essentially doubled from a single-scenario graph to a two-scenario plot graph).

When a user attempts to generate a graph that exceeds the maximum number of data points, Impulse will skip data points in order to remain under that limit. The frequency of skipped data points is proportional to the amount by which the generated graph exceeds the maximum number of data points. When data points are skipped, there is the potential for important data points to be excluded from the graph such as maximum or minimum values.

Users can take the following steps to avoid running into this issue, and to double-check that the graph gives an accurate representation of the results and does not skip any maximum or minimum data points:

- Increase the limit in the User Options window (the time required to load a graph will increase)
- Plot fewer curves in the graph
- Graph data for limited amounts of time rather than the entire simulation time
- Save fewer output data points from the Simulation Mode/Duration panel
- Double check results shown in the graph with transient output shown in the Output window. Users can either reference the Transient Output, showing output values for every time step and pipe section data is saved for, or examine the Transient Min/Max values shown for the parameters selected in the Output Control window.

Related Topics

Related Blogs

Quick Guide to Custom Output

SSL Module

Settling Slurry Module

An optional add-on module to AFT
Impulse which allows users to sim-
ulate settling slurries.
PFA Module
**Pulsation Frequency Analysis
Module**
An optional add-on module to AFT
Impulse which allows users to
determine resonant frequencies in
their system.

Quick Graph Formatting Window

The Quick Graph Formatting 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 thickness and basic color sets
- Axis min and max values

You can save the settings you set up by clicking the Save button. If you make changes to the settings, and want to get back to previous settings, click the Load button.

Related Topics

Graph Types

AFT Impulse offers several types of graphs to help users better understand and interpret their data. Each type of graph is given a brief description below. Starting with AFT Impulse 8, users are also able to plot output for multiple scenarios in a single graph, which is also discussed below.

Transient Pipe

Transient pipe results can be plotted against time for any pipe station that is saved to the output file. By default, only the inlet and outlet stations for each pipe will be saved to the output file. The

Output Pipe Stations panel allows the user to specify which pipe station data will be saved. Each pipe is shown in an expandable list. When a pipe is expanded, the pipe stations for which data was saved are shown. These pipe stations can be added to the list on the right to be included in the graph by either double clicking the parameter, or highlighting the parameter and using the right arrow button.

The user can plot the data for the entire simulation, or for a user specified time frame.

One or more parameters can be graphed by using the Add button with the green plus sign. The Remove button can be used to eliminate extra parameters. Parameters can either be plotted in stacked graphs, or two parameters can be plotted on the same graph using the secondary axis. To move a parameter to the secondary access, add it below the parameter that will be on the primary access, then click the right blue arrow button, or double-click on the gray square to the left of the parameter. This will cause the parameter to be indented, as shown in Figure 2.

Further, if Design Alerts are specified for the pipe (e.g., max/min operating pressures), these can be cross-plotted against the transient data by selecting the check box under "DA".

Select Parameter		Add ▾ X Remove ↓ ↑ ←		
Parameter		Units		DA
☑ Pressure Static		bar ▾	▾	☐
◀ ☑ Mass Flowrate		kg/sec ▾	▾	☐
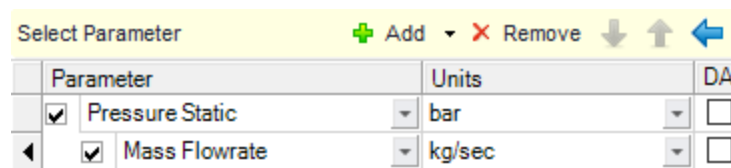
Figure 1: Creating a graph with a parameter on the secondary axis.

Transient Junction

Similar to transient pipe results, transient junction results can be plotted vs. time for any junction that is saved to the output file. Junction data includes items such as pump speed and accumulator volume. These are specified in the Output Junctions panel.

Profile

This graph category allows users to plot the selected parameter(s) along a flow path. In order to generate this graph, the specified pipes in the Select Graph Parameters window that comprise the path must form a single, continuous path. If they do not, an error message will appear listing the pipes which are preventing the path from being continuous. The independent variable in these graphs is the length along the flowpath.

A path of pipes can be specified in one of two ways. First, the user can select the pipes in the displayed list. Additionally, you can select the path on the workspace, and then use the Workspace button on the Select Graph Parameters window to automatically select all pipes currently selected on the Workspace. These methods permit that only a single path to be specified.

One or more flow paths can be specified using groups. Only those groups that consist of a continuous pipe sequence will be displayed as an option in the Select Graph Parameters window.

The results can be the values at a particular point in time, or the overall maximum and/or minimum values. You can also cross-plot the max/min values, the steady-state results, and Design

Alerts. This is done clicking the check boxes underneath "Mx" for maximum, "Mn" for minimum, "SS" for steady-state, and "DA" for design alerts.

All profile graph types support animation. There are two animation options: Animate Using Output File, and Animate Using Solver. These are discussed in Animating Transient Results.

Forces

If transient force sets have been defined in the Force Definitions panel, the resulting force data can be plotted vs. time. Force sets selected from the list of defined force sets will be plotted. The unbalanced transient force data can be plotted for the entire transient simulation, or the user has the option of selecting a specific time frame to display.

This force data can also be exported for analysis in pipe stress analysis software.

Frequency

(PFA module only) As part of the pulsation frequency analysis, the tab will appear to plot the frequencies identified for the system. More information on this can be seen in Pump RPM Evaluation in the PFA Module.

Multi-Scenario Graphing Features

Starting with AFT Impulse 8, users can plot information from multiple scenarios in the same plot. In the Select Graph Parameters window, the button for 'Multiple Scenarios' allows users to select the scenarios in the model they wish to plot for a particular graph. The scenarios being plotted in the graph must have the same pipe layout for all scenarios, but can have different sectioning in AFT Impulse 9. In AFT Impulse 8, all scenarios must have the same Analysis Setup settings including sectioning, start time, stop time, and number of data points saved for all scenarios being plotted in a graph.

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, Impulse will add the active scenario into the Graph List Item, but will not save the active scenario into it.

Scenarios can be selected using the Multi-scenario... button as shown in Figure 2 below.
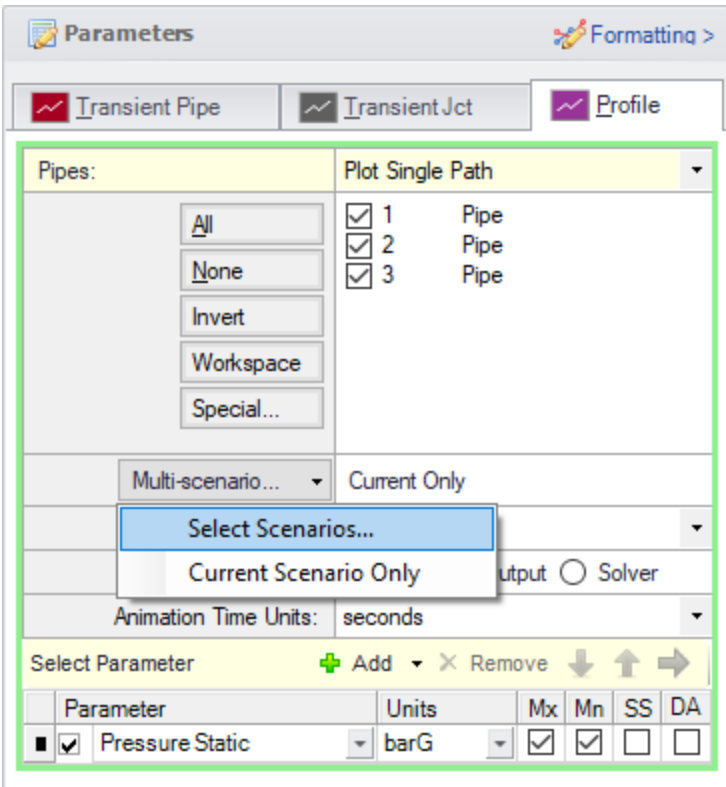
Figure 2: Graph Parameters window showing Multi-scenario selection options

Related Topics

Related Blogs

Five Fantastic New Features for Fast Fluid Transients: Impulse 8

PFA Module

Pulsation Frequency Analysis Module

An optional add-on module to AFT Impulse which allows users to determine resonant frequencies in their system.

Basic Graph Tutorial

Generating Graphs is a powerful tool for analysis. Below are step by step instructions about how to make and manage graphs from one of the example models. Many of the graphs created in this example will be multi-scenario graphs. However, the information discussed within is the same regardless of which scenarios are selected.

For this example, the "US - Ammonia Transfer with Cavitation.imp" file was used. The file should be at this location by default C:\AFT Products\AFT Impulse 9\Examples\US - Ammonia Transfer with Cavitation.imp. It is recommended to "Save As" the file under a different name and location to prevent altering the example files.

Open the model and run all the 0.5 Second Closure, 1 Second Closure, and 2 Second Closure children scenarios. Return to the "0.5 Second Closure" scenario.

To skip to different sections of this tutorial, use the links below:

- Transient Junctions and Saving Graphs
- Transient Profiles and Animations
- Displaying multiple graphs simultaneously
- Creating graphs from the Workspace

Transient Junctions and Saving Graphs (return to top)
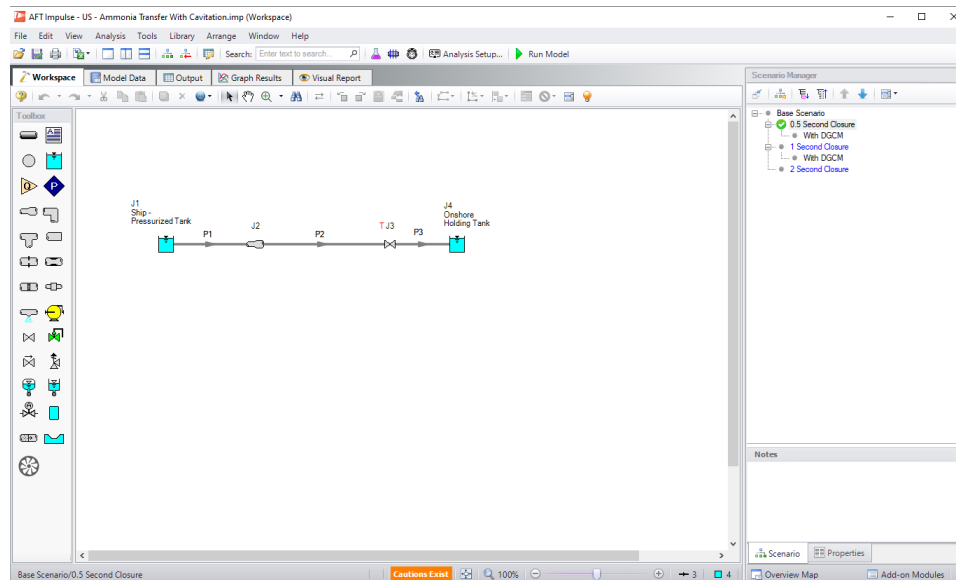
Figure 1: Valve Closure Example

Go to the Graph Results Primary Window and determine what type of graph you want to generate. The first graph created in this example will be a Transient Junction Graph. The Transient Junction is one of several different types of Graphs that can be made. This graph is created by first selecting the Transient Jct tab and selecting the desired parameters. The only junction with

saved transient data will be the valve J3. Add this junction to the "Graph These Junctions:" column by double-clicking the junction ID number, or selecting the ">" symbol. Select "Cv" as the parameter. Select the three scenarios using the Multi-scenario... button, and then select "Generate". This creates a plot of "Cv vs. Time" for all three scenarios on the left.

Note that the Graph Guide, accessed by clicking on the "What Would You Like to Do?" icon on the upper right of the graph, can help guide you through the process of creating a graph
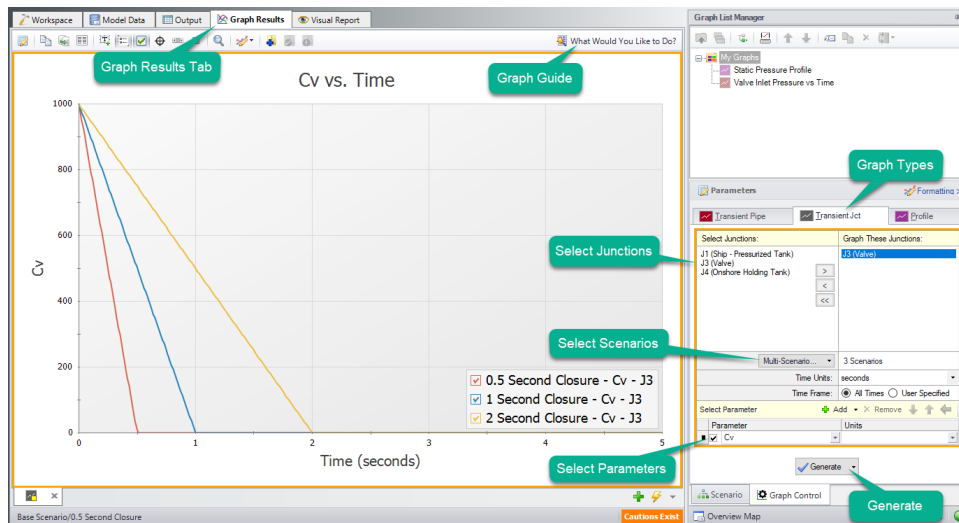
Figure 2: Valve Cv vs. Time

The Graph has been created, but has not yet been added to the list under "My Graphs". To add this Graph to the list, first select the "Add Graph to List" icon on the Graph Results toolbar (this option and many others can be accessed by right-clicking on the Graph at any time). Name the Graph "Valve Cv vs. Time". The Graph has now been saved and will be easily regenerated with the same format if the output changes. The name of the saved Graph can be seen in the Graph List Manager, with two small squares in the icon associated with it indicating it is a multi-scenario graph. The border of the Graph has also changed to dark green along with borders of the parameters and the name of the Graph in the Graph List Manager, indicating the Graph is current, saved, and that the displayed parameters are the ones used to generate the Graph.
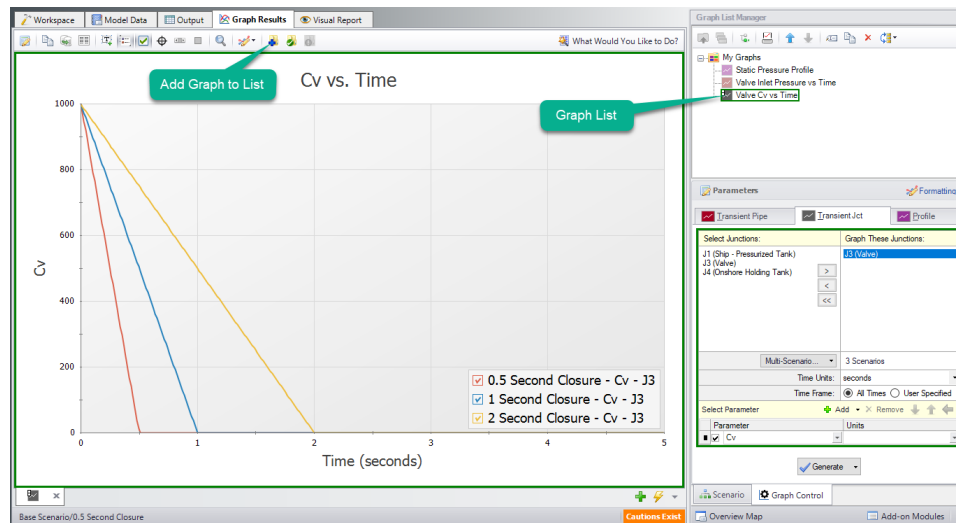
Figure 3: Adding a Graph to My Graphs

Formatting options are available by selecting "Formatting" above the different graph types. When formatting changes occur the border of a Graph will change from dark green to light green, indicating a change has been made to a graph in the Graph List. Another indicator that the Graph is not saved is the small yellow square over the icon on the tab. Here, the title has been changed from "Cv vs. Time" to "Valve Cv vs. Time". Once the formatting settings have been changed, This change and other formatting changes are not retained to the Graph unless "Update Current Graph in List" is selected.
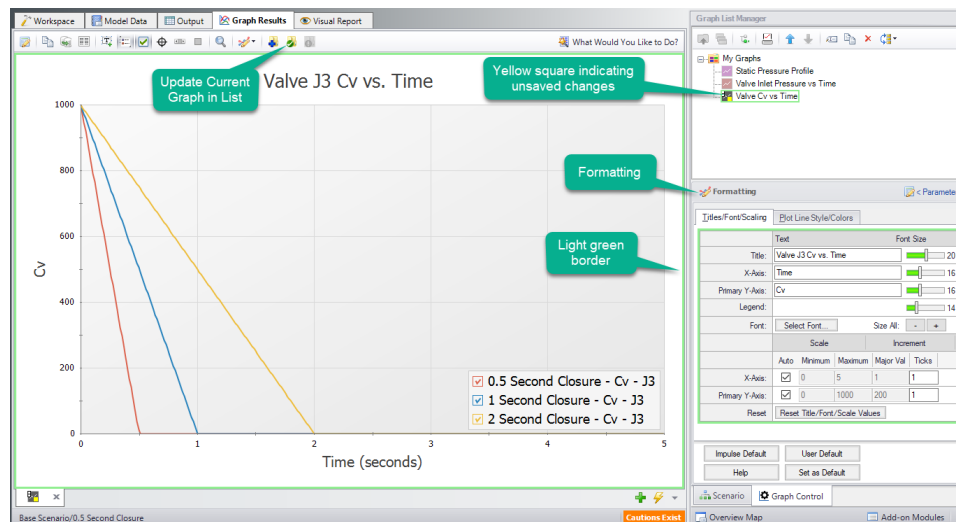
Figure 4: Update Formatting Changes to Graphs

After selecting "Update Current Graph in List" the border has changed back to dark green and the yellow square has disappeared. This indicates that this is the current formatting of this graph,

and that regenerating this graph will generate the same graph (according to the current model output).

Figure 5: Valve Cv graph with updated formatting changes

Transient Profiles and Animations (return to top)

The next step in the example is to make a new Graph in a separate tab. This is done by selecting the "New Tab" icon in the lower right. This Graph will be a "Profile" Graph that displays Static Pressure along the length of the flow path. Select pipes 1, 2, and 3 in the system and change the parameter to "Pressure Static" with units of length in feet and units of pressure in psig. Make sure that the Mx and Mn boxes are checked to display the maximum and minimum stagnation pressures.

Note: An easy way to select the pipes in a longer path would be to select them in the Workspace, then click the Workspace button under Pipes to automatically select the desired path. This method is discussed further below.
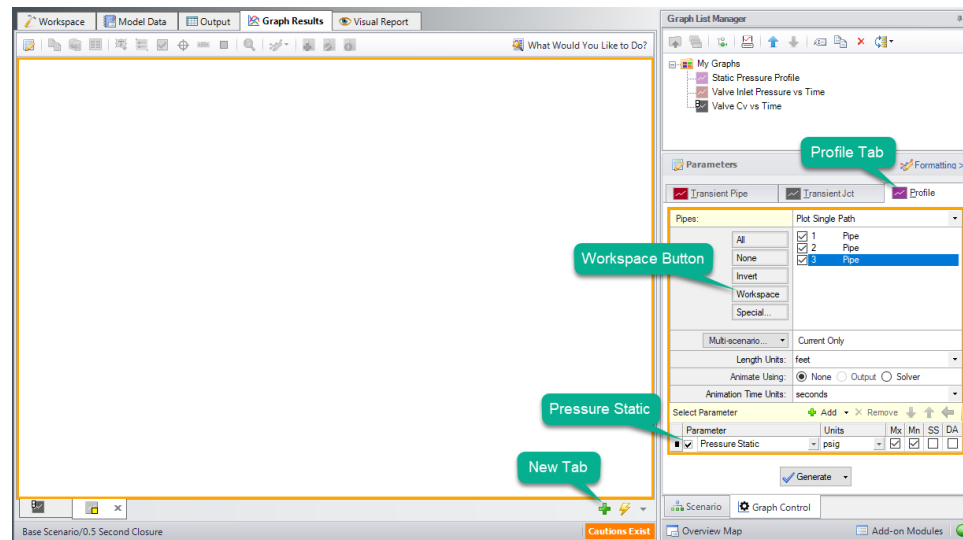
Figure 6: Generating a Profile plot with Min and Max Pressure Stagnation

The graph shows the minimum and maximum static pressures during the transient simulation for each pipe station along the path. This Graph is the same as the saved "Min/Max Static Pressure Profile" Graph List Item. We could have generated the same graph by loading that graph instead. Save this new graph to replace the saved Graph List Item, or create a new entry. The graphs we have made can be switched quickly between by either selecting the Graph from the list, by clicking on the corresponding tab, or using Graph Tab Selection.
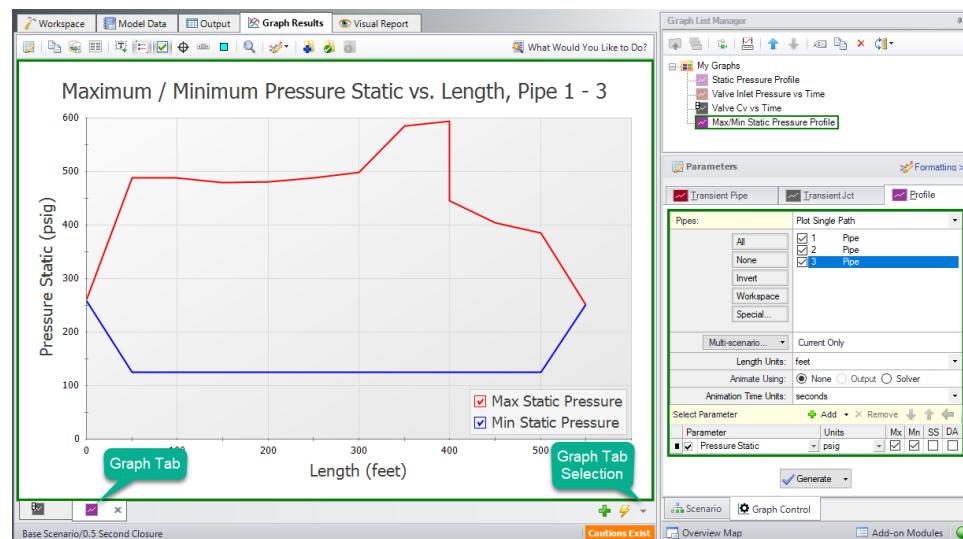
Figure 7: Pressure Stagnation vs. Flow Length Profile added to Graph List

This is a useful graph, but animating the pressure profile over time can provide even more information. There are two animation options available. Animating from the Output file requires that all data for the pipe stations being animated to be saved to the Transient Output File using the Output Pipe Stations panel. This gives more flexibility to move the animation forwards and

backwards in time, but makes the run-time longer. Animating using the Solver option does not require output for all stations, but instead reruns the solver. This takes longer to graph and has less accuracy, but allows for a shorter model run-time. This model was set to only save data for pipe inlet and outlet stations, so we will use the "Solver" option. To use the Animate via Output option, return to the Output Pipe Stations panel and change which Pipe Station output is being saved.

After selecting generate the Graph will now allow for animation with options to Play, Record, Stop, and manipulate the animation in other ways. The recorded animation can be exported to a video file for convenience.
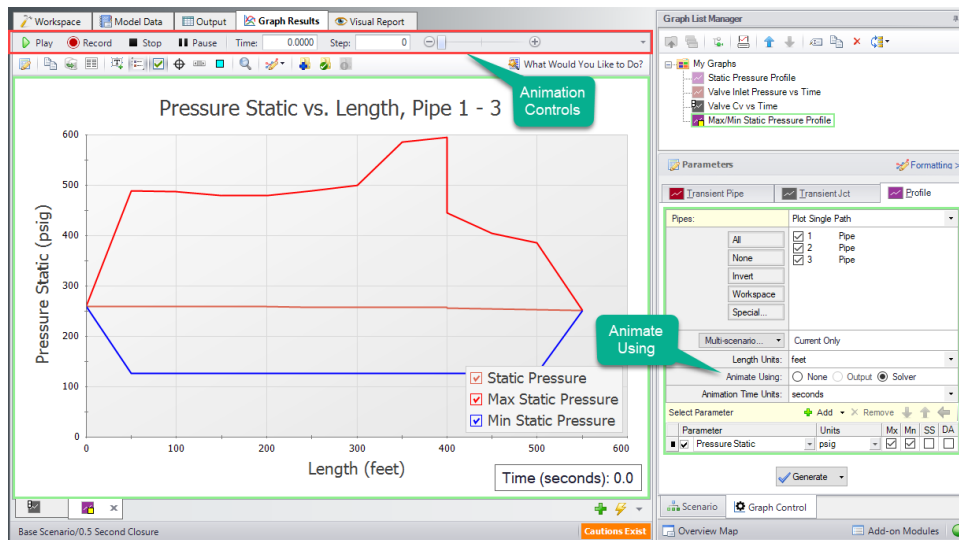
Figure 8: Max/Min Profile Graph with Animation

Displaying Multiple Graphs Simultaneously [(return to top)](return to top)

Another option that is available is to add another parameter to the graph. This is called generating stacked graphs. To do this select "Add" next to "Select Parameter". This will graph an additional parameter that shares the same X-Axis as the original parameter. For this example, select "Pressure Static" as the parameter, but with units in barG rather than psig.
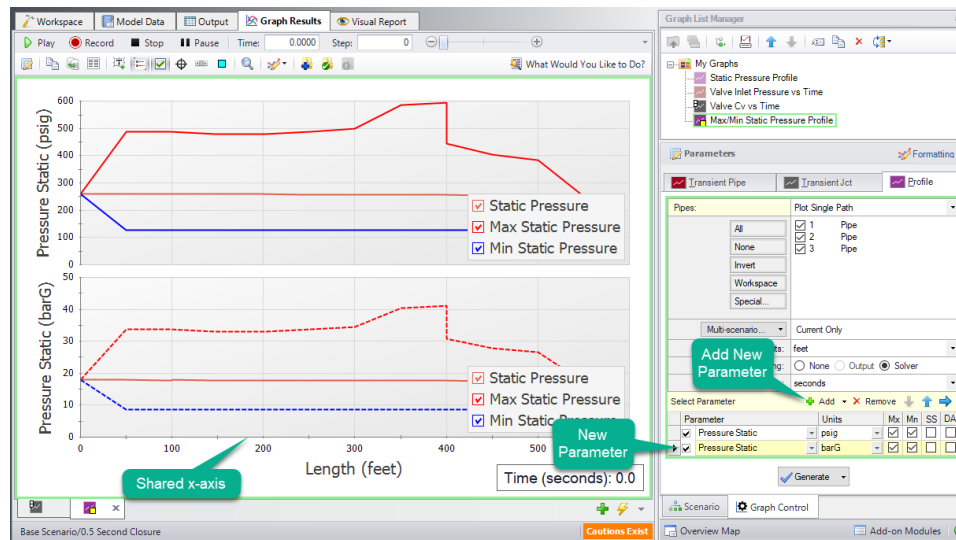
Figure 9: Stacked Pressure Profile with different units

We now have the same plot twice, which is not the most efficient way to view this data. A better way would be to have the secondary parameter (which could be any of the available options for a profile plot) on a secondary Y-Axis. This can be done by double clicking on the arrow to the left of "Pressure Stagnation: bar" or by selecting the blue right arrow on the "Select Parameter Row". This will visibly indent the secondary parameter. When the Graph is regenerated it will now only be one plot of Pressure Static vs. Length, but it will have units displayed in psig on the primary Y-axis and units of barG on the secondary y-axis.
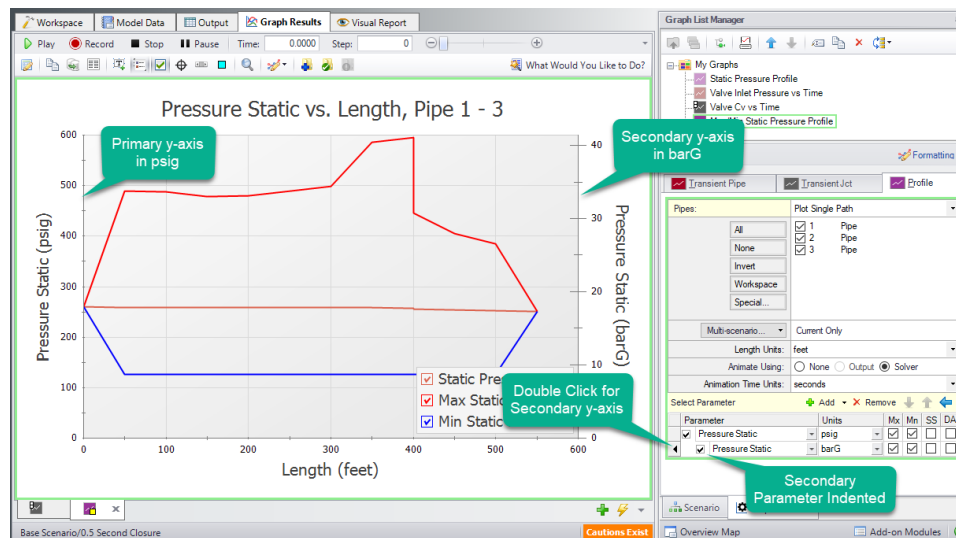
Figure 10: A Single Graph with Alternate Units on a Secondary Y-Axis

Update Current Graph in List button. The Graph is now complete and can be easily regenerated if input data is changed on the Workspace and the model reran. It can also be used to generate the same graph for data from other scenarios.
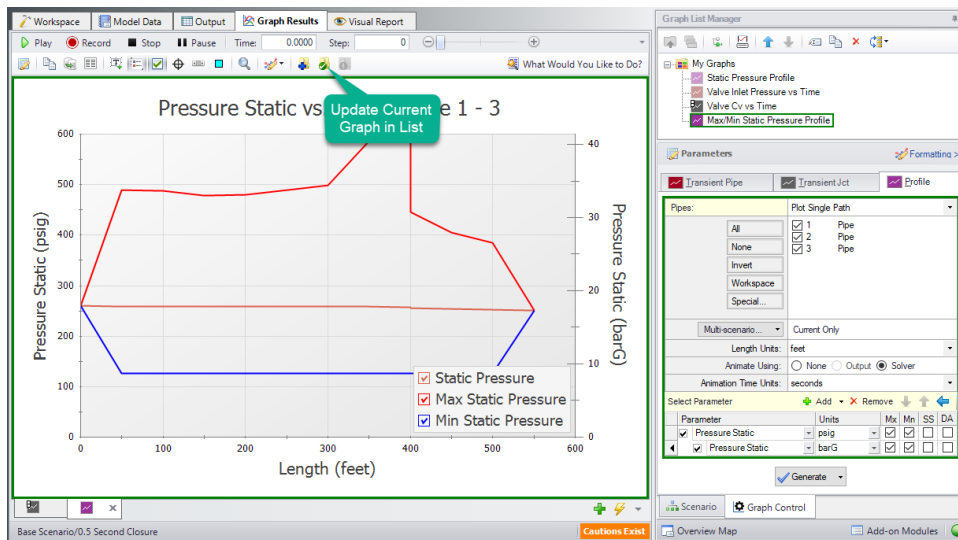
Figure 11: The Resulting Graph when it is Updated

It is now possible to close the Graphs and regenerate them either in their own tabs or all graphs in a folder in the same tab. To test this out, close both Graph tabs and then select the "My Graphs" folder. Right-click on the "My Graphs" folder and then choose "Load Folder Items" and then "Same Tab". After the Graphs have been loaded in the same tab, the tab icon changes to an orange square, indicating that multiple Graphs are being displayed on one tab. All graphs will now be animated, with a vertical red line appearing on the graphs with a time axis. This red line will move over the animation time to compare values more readily.

Depending on the monitor display being used, or the number of graphs being displayed, this view may be easy to view or difficult to see data. For the sake of this example, we've kept the display size relatively small, meaning it is more difficult to view the data on all the graphs. However, with larger monitors available, it is likely easier to view this data.
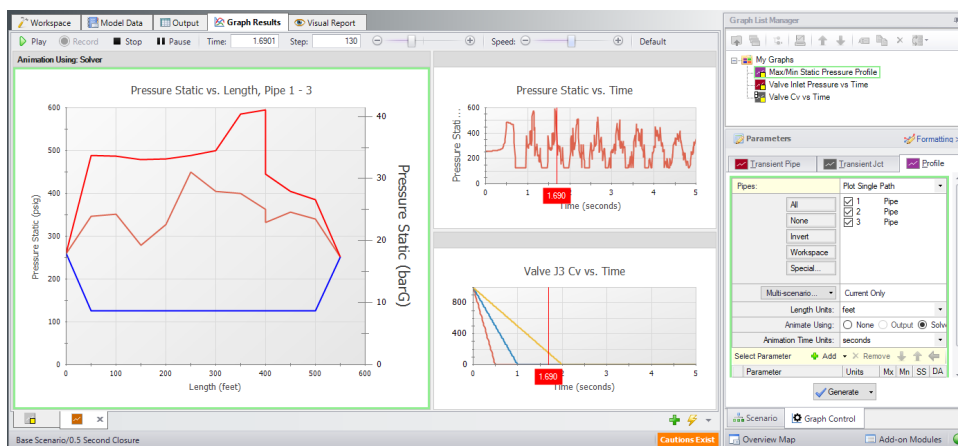
Figure 12: All graphs loaded into the same tab

Other ways to view multiple graphs simultaneously are to plot them as stacked graphs, as we did previously for our pressure profile in psig and barG, or to plot saved graph items in two tabs either stacked or side-by-side. Right-clicking on a Graph-Tab at the bottom of the window shows the options for displaying two separate tabs.

Since we have two Time graphs with the same X-axis, we can display them in a single, stacked graph. This approach to viewing graphs can be especially helpful in examining how a junction transient is affecting output parameters nearby. To create a stacked graph of multiple Graph List Items, first make sure both items have the same time scale. Both of our example graphs do. Then, move them into their own folder. Here, we've created a new "Profile" folder, and moved our Max/Min Static Pressure Profile graph into it. Finally, right-click the My Graphs folder, select "Load Folder Items", and then the "Stack Common X-Axis in Same Tab" option.
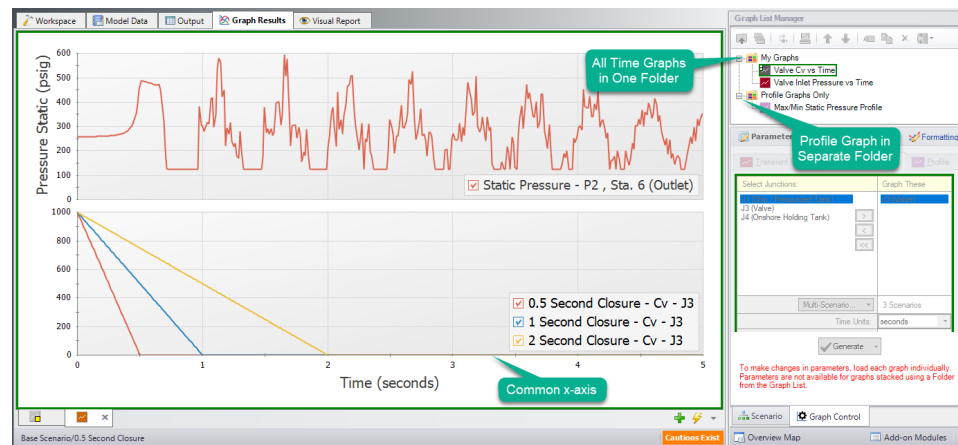
Figure 13: Load Current Folder Graphs in Same Tab - Stack Same X-Axis

As we just saw with the Profile Graph Folder, it is possible to create more Graph Folders other than "My Graphs". Saved Graph List Items can then be organized into different folders and can independently be generated into their own tabs or a whole folder of Graphs can be generated into one tab. Using multiple folders will allow the user to create many unique variations of animations or other Graphs that can be used to convey transient output in a meaningful and intuitive manner.

Creating Graphs from the Workspace [(return to top)](#)

Some graphs can also be generated directly from the Workspace after running the model. To create a graph for the valve inlet and outlet stagnation pressures from the Workspace right click on the valve J3, choose "Create Transient Graph for connected pipes (Inlet/Outlet)", and select "Pressure Static".
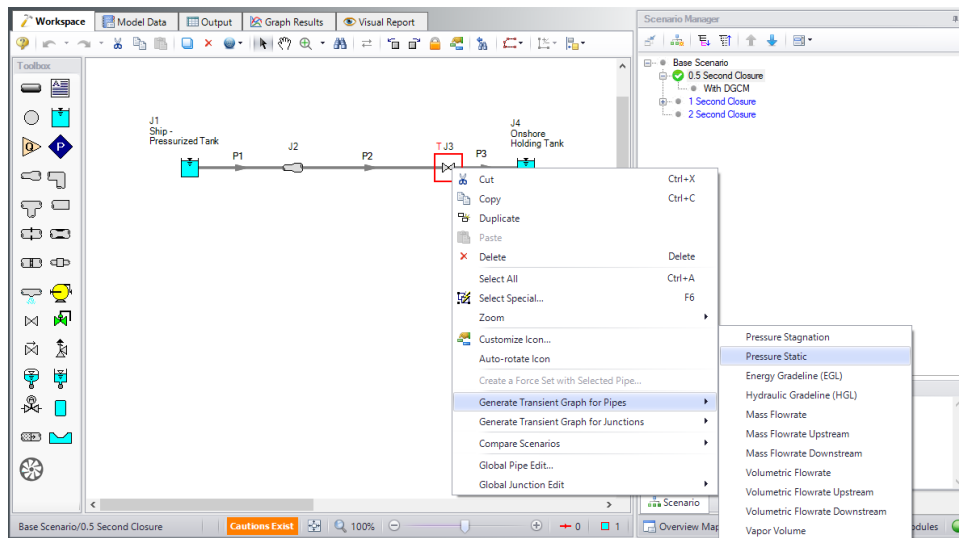
Figure 14: Creating Transient Graphs of the Inlet/Outlet Pipes Connected to a Junction

This will automatically generate the requested Graph with default units. Add the graph to the "My Graphs" folder using the Add Graph To List button and name it "Valve Inlet/Outlet Pressures". This is a quick and easy way to see the pressure difference across the valve. Notice that this graph will be a Transient Pipe graph, since the pipe inlet/outlet stations are being used to graph the equivalent valve inlet/outlet values.
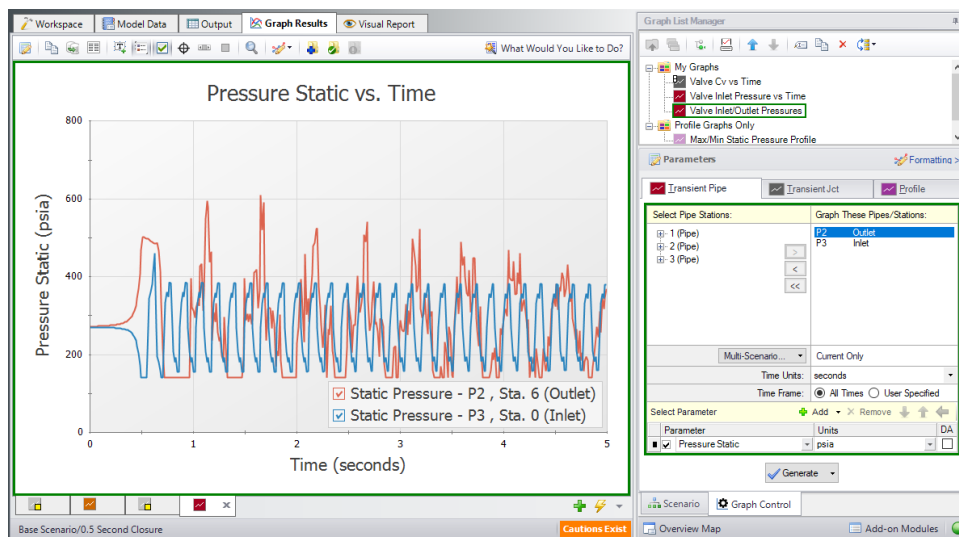
Figure 15: The Pressure Stagnation vs. Time of the Two Pipes Connected to Valve J3

Make one more Graph from the Workspace by right-clicking pipe 3, selecting "Create Transient Pipe Graph", "Volumetric Flowrate Downstream", and then "Outlet Only".
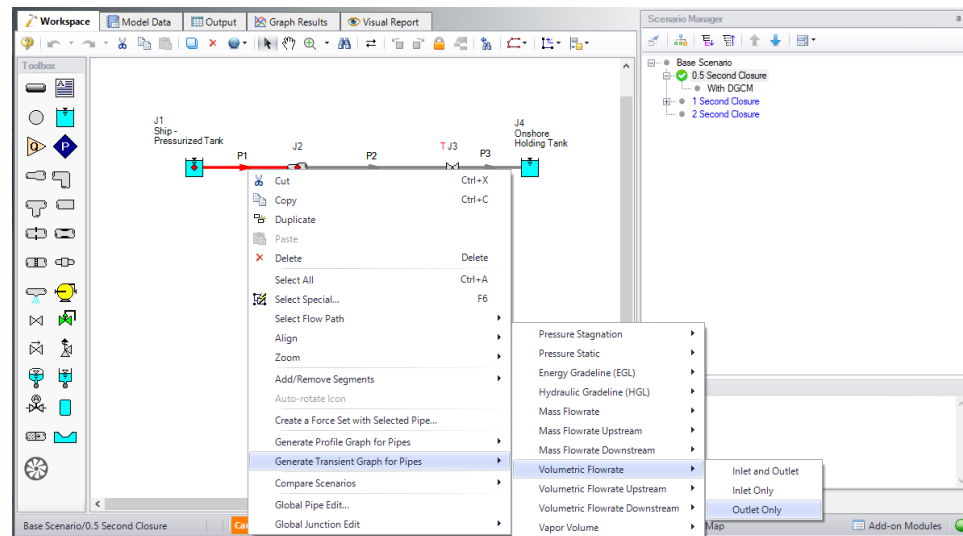
Figure 16: Graph the Changes to Volumetric Flowrate vs. Time from the Workspace

This will add a Graph of the Volumetric Flowrate at the Outlet Station of pipe 1. Add the graph to the Graph list and name it "Pipe 1 - Outlet Volumetric Flowrate".
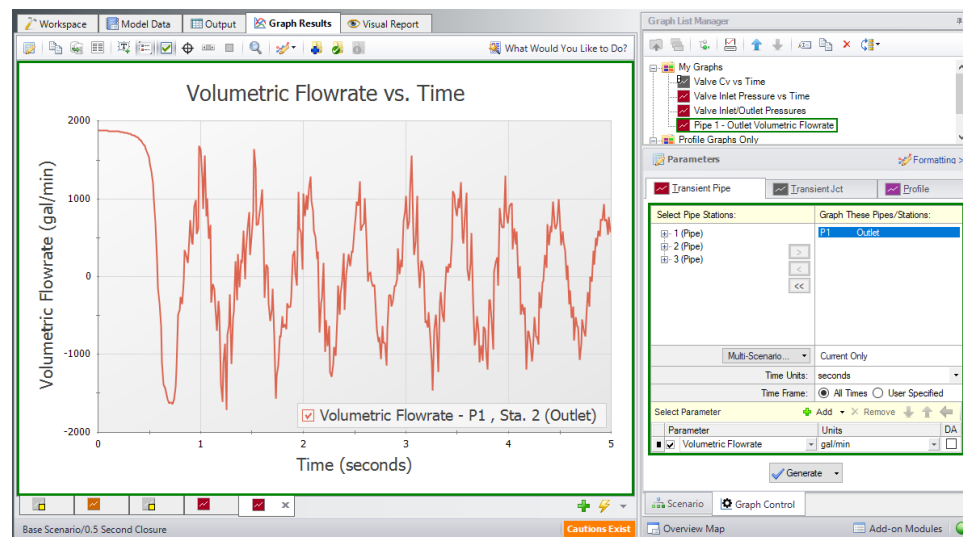
Figure 17: Volumetric Flowrate Downstream vs. Time at the Outlet of pipe P1

Related Topics

Visual Report Window

The Visual Report will help you visualize the model data and flow solution in relationship to the model layout. The Visual Report window displays the pipe system layout from the Workspace window with information in text form (see Figure 1). Additionally, relative magnitudes of parameters may also be shown with color variations using color maps, or animated using Visual Report Animation.

The values to be shown may come from the input data (when in input mode), the steady-state output or transient output results. You can change between these by selecting the option at the top of the Visual Report Control window.

In the Visual Report window, the positions of the junctions and pipes are fixed based on their locations on the Workspace. While the pipe system layout cannot be edited in the Visual Report window, the integrated text results for each pipe or junction can be rearranged by dragging and dropping to improve the final appearance of the window.

The contents of the Visual Report window can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

Visual Report animations can be recorded to a .wmv or mp4 file.

The controls in the lower-right corner allows you to zoom in and out, as well fit the image to the size of the screen. The printed size is consistent with the zoom state. That is, if you zoom out to 50%, the Visual Report screen prints at 50% of its normal size.

The Visual Report window is always erased when you make changes to model input. If viewing output results, rerun the model to view the new results.
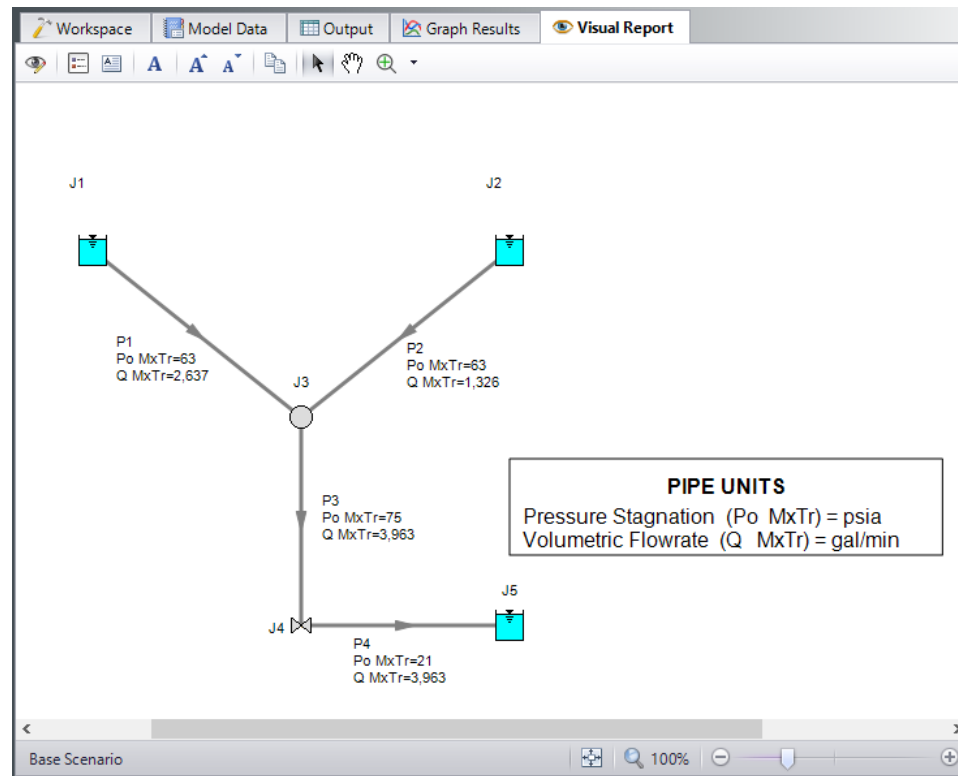
Figure 1: Visual Report window integrates input data and output results with model layout

Related Topics

Related Blogs

Quick Guide to Custom Output

AFT Responds to 2020 Customer Survey Feedback

Visual Report Control

The Visual Report Control window lets you control the content and appearance of the Visual Report window (Figure 1). You can also save custom Visual Report window layouts to disk for later recall, so you don't have to reformat this window repeatedly.

When the Visual Report window is opened and no output results are currently displayed, Visual Report Control will be in the Input Display Mode. If transient output results are displayed, it will be

opened in the Transient Output Display Mode. Parameters available for display vary depending upon the mode, i.e. input, steady output, transient output, or animate. If output results exist the display mode may be changed at any time.
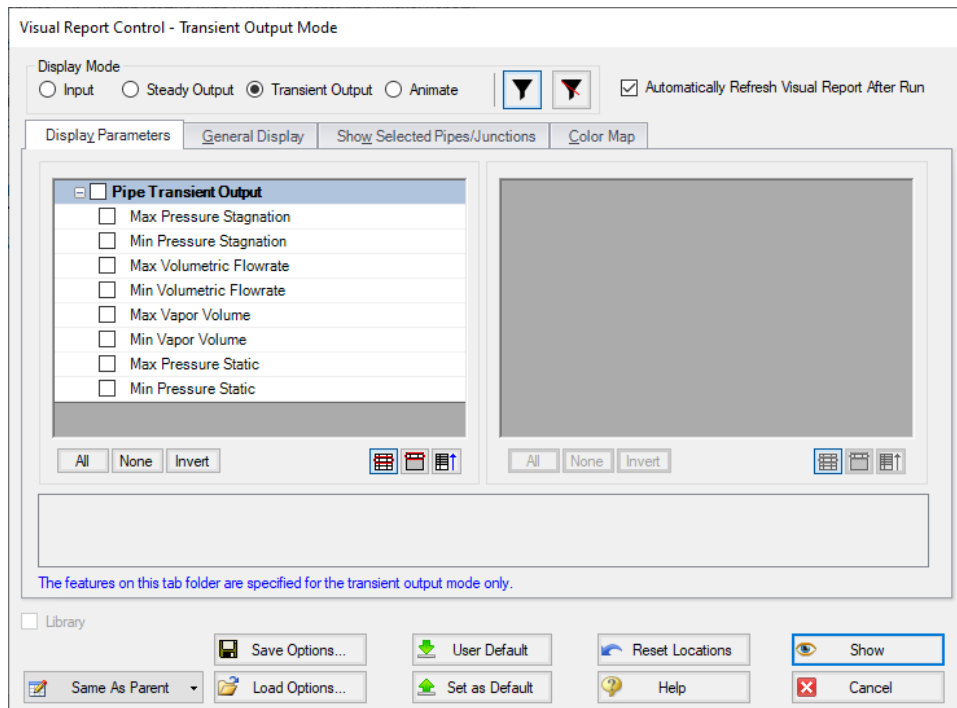
Figure 1: The Visual Report Control window allows you to configure the Visual Report display

Display Parameters

The parameters to be shown in the Visual Report window are selected from the Display Parameters folder tab. The left list selects the pipe parameters and the right list selects the junction parameters. In these lists you can show only the parameters you have selected in the Output Control window, or all output parameters. You can select more than one parameter; however, the Visual Report window may get crowded if too many are selected.

General Display

The General Display tab allows you to configure several items.

The title, defined in the Output Control window, filename and scenario name can be shown or not shown depending on your preference.

The units for all parameters can be shown in a legend instead of next to the parameter value. This will reduce the amount of text and crowding shown in the Visual Report window. The legend can be moved to any location in the Visual Report window. If this option is chosen while in input only mode, all values will be converted to a consistent set of units.

You can change various font sizes by clicking the Change button beside any one of the three types of text-containing categories (Data/Text, Title, and Legend) under the Fonts header and choosing the font, font style, and font size desired.

Tip: The font and font size in the Visual Report can also be changed outside of the General Display in Visual Report Control by clicking on the Fonts button and the Font Size Up or Font Size Down buttons, respectively, on the Toolbar.

The number of digits to show in the numerical display can be configured using the provided drop-down list.

Normally on the Workspace, closed objects are displayed using special graphics (dashed lines, X's, etc.). The number of a junction which has special conditions set, by default, has an X pre-ceding the J (e.g. XJ56). If you want these symbols to be used in the Visual Report window also, choose the Show Closed Symbols checkbox. Normally, pipes in closed sections are shown as dashed lines, and junctions outlined with a dashed line. If you want these symbols to be used in the Visual Report window also, choose the Use Closed Object Style checkbox.

Show Selected Pipes/Junctions

You can choose on an individual pipe and junction basis what you would like to show in the Visual Report window. This display customization feature is useful for large models where only some of the data points are of interest.

Each pipe and junction is listed in their respective Display Options frames. Every object can dis-play four pieces of information on the Visual Report - Number, Name, Data, and the Object itself. The display of these items is indicated by the existence of a checkmark in the corresponding box. To toggle the display of a particular item for a particular object, simply click the checkbox.

To restore the settings to what is shown in the Workspace and the data, click the Reset to Work-space Defaults button.

Multiple objects can have items toggled on or off at once by selecting the desired objects with the left-most checkboxes and clicking the corresponding checkbox at the top of each column. Selec-tions can also be made with the standard selection buttons at the bottom of these panels.
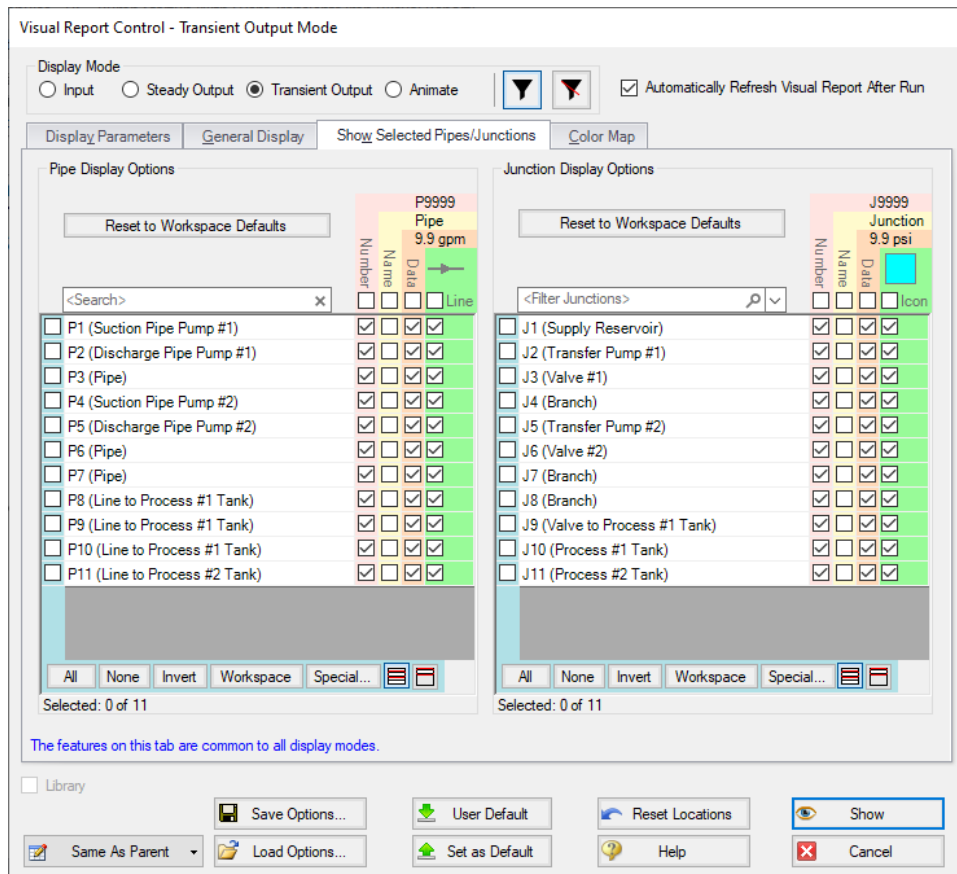
Figure 2: Show Selected Pipes/Junctions

Color Map

The Color Map feature allows you to visualize magnitudes of various parameters by coloring the pipes. There is one Color Map for the steady output mode, one for the transient output mode, and another for the input only mode. A Color Map is useful to show graphically where, for example, high velocities or reverse flows are located or to visualize the pipe diameters or lengths in a model.

To create a Color Map, first choose a parameter and units for which all pipes will be categorized. Then choose a color, select the operator (i.e. >, >=, =, <=, <) and enter a value, then press Add to Map. Values may be categorized on an absolute value basis by selecting this option.

The order of entry is not important. Impulse will sort the entries first by value and then by the operator, with the > being above the <. The pipes will be colored using the first match found in the map, starting from the top of the map. If no match is found, the pipe will be the Workspace pipe color. Be careful to avoid overlapping operators and values. If, for example, the map had the following four colors,

1) >=60

2) <60

3) >=20

4) <20

then there is no way colors 3 and 4 will be used because any pipe less than 20 is also less than 60. Since <60 is before <20 in the map, the pipe will be colored using 2.

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

1) >=60

2) >=20

3) <20

You can choose to categorize the pipes based on their absolute values by choosing the Display Absolute Value checkbox. To delete a map value, click the color you want to delete then click the Remove Setting button.

Instead of coloring the pipes, you could choose to color just the pipe numbers. This is done by choosing the Color Pipe Labels and clearing the Color Pipes checkboxes in the Color Options area. You could also apply the colorization to both. Choose the Print Black & White checkbox if you don't have a color printer. The pipes will be printed in black.

The color map values can be displayed in a legend by checking the Display in Legend checkbox. The color map will appear in the legend after the pipe and junction units, if that option was selected. The legend may be dragged and dropped to any location in the Visual Report window.

Auto Color Map

An automatic color map can be generated for you by selecting "Auto Color Map". This feature will generate a color map using greater than (>) conditions to divide the difference between your max and min value into the number of conditions. These conditions will have their own unique color that is automatically set on a rainbow spectrum. These color's can be manually changed by selecting the "..." button.

Saving Visual Report Layouts

The Visual Report window content is specified in the Visual Report Control window. To open the Visual Report Control window, choose it from the View menu or Toolbar. Through the Visual Report Control window, you can choose the parameters you want to see in the Visual Report window.

The parameters and text locations are automatically saved with the model. However, you may want to save the settings for use with a different model. To save your preferred layouts to an AFT Impulse visual layout file, click the Save Options button and assign a file name. The default file extension is .ivs. Later you can load your customized layouts for rapid preparation of the Visual Report window.

Related Topics

Related Blogs

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

User Options

AFT Impulse provides an array of User Options in a single window that can be found in the [Tools menu](#).

The General section of the User Options window allows you to adjust:

- Enabling the Startup Options Window

- Enabling the Tip of the Day feature

- Enabling orthogonal drawing tips

- Whether to confirm before clearing output results

- Enabling Highlighting Required Data on Property windows

- Options for opening models with scenarios

- Allowing messages for Force Set Calculation Time

- Visual Themes and Styles

- The default location of the Quick Access Panel

- Number of data points to show in input tables

- Maximum Number of Data Points per Graph

- Setting parameters for "button flashing" for curve fit buttons on Junction Property windows

User Option Defaults

There are five buttons at the bottom of the User Options window. Impulse has built-in default parameters, units and settings which you can choose by clicking the Reset to AFT Impulse Defaults button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of Impulse's defaults). To make your own default, first select the parameters, units and settings you would like to use then click the Save As New User Defaults button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the Load User Defaults button. The default settings are updated only when you click Set As New User Defaults.

If you have made changes which you don't want to keep, click the Cancel button. Click Save and Close to use the settings you have defined.

Related Topics

Unit System

You can set the units you prefer to use in AFT Impulse in the Unit System section. Once selected as the preferred units, these units appear as the default units in all instances in which empty fields exist in the model input. Any model input existing prior to changing the Preferred Units will maintain the units used when the data was input, regardless of changes made to unit preferences. After saving your preferred units, a window appears asking if you would like the output parameters to be displayed in your preferred units. Choose Yes if you do want the output parameters to be displayed in the preferred units, or No if you want the output parameters to be displayed in the units previously used.

Preferred Units

To customize the units used by AFT Impulse, first select Preferred Units under the Unit System section in the User Options window under the Tools menu.

AFT Impulse allows you to specify certain units from a large selection to be displayed in the main interface.

The primary selection is the Unit System:

- **US -** Only US units will be available
- **Metric -** Only Metric units will be available
- **Both with US Defaults -** All Units will be available, but US units will be chosen by default
- **Both with Metric Defaults -** All Units will be available, but Metric units will be chosen by default

Additionally, a filter can be applied to limit the selection:

- **All Units** - all units for the Unit System will be displayed
- **Common Only** - only "common" units will be displayed
- **Common Plus Selected Industries** - Units particular to certain industries can be included with the common units

If the selection has been limited to Common Only, there may units common for certain industries no longer present. The Additional Units for Industry section allows the user to append additional sets of units to the Common unit set. For example, barrels/day is a common unit in the petroleum industry but not many others. Checking the Petroleum box adds these units to the main interface.

Preferred Units allows fine grain control over what unit in the above selection is displayed as the default when defining model input. For example, it may be desired to have a particular parameter

such as pipe diameter default to inches, while keeping other parameters metric. This area allows the user to define such defaults.

Show/Hide Unit Items

For additional control of the available units, select Show/Hide Unit Items under the Unit System section. From here, you can select which units for each Unit Family Type you would like to be available for the model input and output by checking the box next to each unit you would like to be used. Removing units that you do not wish to use as input or view as output can simplify your modeling and reduce errors associated with incorrect units.

Edit Custom Unit Names

A custom unit name can be assigned to display different names in the interface for units. For example "feet/min2" could be changed to display as "ft/min2."

To assign a custom unit name choose the appropriate Unit Type from the drop-down and enter the Custom Unit Name in the table.

To speed up the process of entering custom unit names the units can be exported to csv and re-imported as follows:

1. Click "Save Units to CSV" and save the file where it will be easy to locate
2. Open the CSV file in a spreadsheet editing software like Excel, and fill in the desired custom unit names in the Custom Name column
3. Save the CSV file
4. In the User Options window click "Load Units from CSV" and select the edited CSV file

Note: To use units with superscripts, make sure the csv file is saved with UTF-8 encoding. This can be done in Excel by choosing save as and selecting "CSV UTF-8" as the file type.

Related Topics

Related Blogs

AFT Responds to 2020 Customer Survey Feedback

Workspace

In the Workspace section, you can change the symbols used, text content, and location of the text on the Workspace. Note that all display changes are made only for the user's convenience/preference and do not affect the model calculations. Additionally, many display options available in the User Options window are also available in individual pipes and junctions.

Grid

A traditional 2-D grid or an Isometric grid can be enabled on the workspace for modeling convenience by choosing the appropriate Pipe Drawing Mode. The number of lines in the grid can be adjusted to create a grid that is fine, medium or coarse. The grid can be hidden or shown by checking or unchecking the Show Grid box accordingly. If the grid is shown, the grid line pattern can be solid or dashed. The pipes and junctions can also be snapped to the grid, which moves junctions and pipe end points to the closest intersection of grid lines.

2D Orthogonal drawing mode can be enabled in the drawing area. This provides automatic Workspace drawing guides in which vertices are automatically inserted into pipes to enforce drawing only along the horizontal and vertical. When junctions are moved with connected pipes, then pipes may also have vertices inserted to maintain orthogonality. The various pipe drawing modes can also be toggled on the Arrange menu.

Pipes

The display of pipe lines, closed pipes, pipe endpoints, and pipe direction arrows can be adjusted according to your preferences here.

Pipe line thickness can be adjusted from 1 to 10 pixels by selecting the desired pixel value in the box next to Pipe Line Thickness (pixels). You may choose to show all pipes as having the same thickness, or you can vary the thickness with an increasing pipe diameter. If varying the thickness according to the pipe diameter, you can allow Impulse to vary the thickness in a gradual fashion with larger diameters shown as having increasingly larger thicknesses, or you can customize two diameters between which the thickness will vary. In the latter case, any pipes with diameters below this minimum value will have the pipe thickness of the minimum diameter pipe, and any pipes with diameters above this maximum value will have the pipe thickness of the maximum diameter pipe. To customize the pipe thickness variation, input the minimum and maximum diameter between which you wish the pipe thickness to vary, as well as the pipe thickness that will represent the minimum diameter pipe and the pipe thickness that will represent the maximum diameter pipe. The pipe thickness will vary in a linear fashion between the two diameters specified.

From the User Options window, you can select how you would like a closed pipe to be displayed in the workspace. The options include a solid line, dashed line, a dotted line, and dashed-dotted combinations.

When building a model, Impulse automatically adjusts pipe endpoints to the centers of junctions for user convenience. If you would like to disable this feature, uncheck the box next to Automatically Adjust Pipe Endpoints to Junction Centers in the Pipes window.

The display of flow direction arrows on the pipes can also be adjusted. Under the Pipe Direction Arrows section of the Pipes window, you can select the size of arrow you would like to be displayed as a percentage of the pipe thickness. Additionally, you can opt to not show a pipe direction arrow by checking the option Do Not Show Arrow.

Note: The pipe direction arrow is used as a reference direction for flow and does not change direction on the workspace if the calculated flow is in the opposite direction indicated by the arrow. Instead, the flow is reported as a negative flow in the output.

Junctions

The junction icon style displayed on the Workspace can be chosen in Junctions underneath the Workspace section in the User Options window. The two available icon styles include classic icons and transparent icon. Classic icons are the icons used in earlier versions of Impulse ending at Version 4 in which each junction is surrounded by a box, and transparent icons are updated icons not surrounded by boxes.

You can also select the icon size and label location for junctions. To adjust the size, click on the box below Icon Size in the Junctions window and choose the percentage of the default size that you would like the junction icons to be sized. For the junction label location, you can select from any of the eight locations shown in the window by clicking on the selection circle that corresponds to the desired location.

Note: Junction and pipe labels can be moved individually on the Workspace by using the mouse to drag and drop the label according to your preferences.

The final option in the Junctions window of the User Options is to enable or disable a dotted outline around a closed junction. If this option is enabled, a dotted box will appear around the junction. If this option is disabled, no dotted box will appear.

Labels

Display settings for pipe and junction labels on the Workspace can be customized under Labels in the Workspace section of the User Options window. When a pipe or junction is inserted onto the Workspace, the next whole number available at the increment specified in this window is automatically assigned to the new pipe and/or junction. Impulse's default is to increment pipe and junction numbers by one. Therefore, the first pipe you place on the Workspace will automatically be labeled as Pipe 1 (P1 for brevity), the second will automatically be labeled P2, etc. Junctions are labeled in a similar fashion with a "J" in front of the number to indicate it is a junction. However, if you choose a different increment in the User Options window for pipes and/or junctions, the automatic numbering will increment pipes and/or junctions according to the specified number. You can renumber any pipe or junction individually as long as the number is not already used in any scenario of the model. If a number at the increment is already in use, Impulse will skip to the next available whole number. Pipe and Junction label numbers are limited to integers between 0 and 99,999.

Note: A unique label must exist for each pipe and junction on the Workspace. Therefore, there cannot be more than one P1 in the model, but a P1 and J1 can exist simultaneously because the leading letter is different.

Additionally in this section in the User Options window, you can select if you would like the pipe and/or junction numbers only to be displayed, if you would like the names only to be displayed, or if you would like both the name and number to be displayed. To choose which display option you would like, select the radio button next to the desired display option under Pipe/Junction Default Display.

Note: If you would like a pipe and/or junction display label type to vary from the default option selected in Labels in the User Options window, you may choose the display label type for individual pipes and junctions in the Optional tab of the individual pipe or junction properties window under Display on Workspace.

You can also choose whether the pipe and junction labels should be moveable at all times, or if they should only be moveable when the user is holding down the shift key. Select which option you prefer by clicking on the radio button next to your preference. Restricting the movement of labels can be especially helpful to users with complicated or dense models, as it is possible to inadvertently grab a label when attempting to reposition or select an icon on the Workspace.

Colors and Fonts

The Colors and Fonts window located under the Workspace section in the User Options window allows you to change the colors of the following parameters seen on the Workspace:

- Pipe and Junction Symbols
- Junction Numbers
- Pipe Numbers
- Pipes
- Selected Items
- Undefined Numbers
- Workspace Background Base Scenario
- Workspace Background Children
- Workspace Background of Grandchildren
- Workspace Background 3rd Generation
- Workspace Background 4th Generation +
- Grid

The currently selected color for each parameter is shown to the right of the parameter. To change the color for any parameter, either select the box containing the parameter's color and then click on the Set Color box under the list, double click on the box containing the color you would like to change, or click the "..." button next to the color of the parameter you would like to change. A color window pops up that allows you to choose from 48 basic colors, or to create a

custom color by clicking anywhere in the color box and adjusting the slider at the right of the box to create the color of your choice. If you would like to save the custom color for use in other areas of the model, click on any empty box underneath Custom colors: and then click on the Add to Custom Colors box below the color box. When finished with selecting a color, click OK on the bottom left of the Color window.

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

You may also choose to change the font of all junction and pipe names and numbers by clicking on the Change Font button to the right of the Font for Junction and Pipe Numbers on the Colors and Fonts box in the Workspace section. A preview of the current font is given underneath the Font for Junction and Pipe Numbers title.

Other

Symbols representing pipe or junction special conditions and inflow or outflow are listed in the Other window under the Workspace section in the User Options window. These symbols can be changed by the user. To change a symbol, click on the box containing the symbol you want to change, and type in the new symbol.

If you choose to include a background picture in your model, you can adjust the picture size in this section, as well. To scale a background picture, enter a Scale Multiplier in the box underneath Background Picture Scaling.

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 option avoids you having to re-enter that data in the case that several pipes require this same input. Disabling this feature leaves the information 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 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, an elevation of your choice will automatically be input into each new junction 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 Setpoint box allows you to specify if all new control valves should be set to Always Control (Never Fail) by default in the Control Valve Properties window. This setting can be useful as a troubleshooting tool to see how far your control valves were from being able to control in the system.

The Default Design Factors box allows you to add a default Design Factor to each pipe and/or junction shown in the list on the Pipes and Junctions window Optional tab. Setting a default design factor for pipes and junctions can be especially useful when applying safety factors on given equipment.

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/or efficiency/power type you would like as the default option wherever flow rate, head/pressure, and/or efficiency/power 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 the efficiency/power type can be set to either efficiency or power.

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 Impulse to search for library files, click Add under the Library Search Locations box. Browse to the file location that contains the library, select the location, and click OK.You can add multiple folders to be searched. To add additional folders, repeat the process for each folder.

The User File consists of all saved settings and preferences. In order to backup these user settings in a *.ini file, click Add under User File Backup Locations. Browse to the file location in which you would like to save these settings. Once you have selected the desired file location, click OK. Note that the User File can be saved in multiple locations. To add additional locations, repeat the process for each folder that you would like to save the settings.

Backup Settings

A backup file is a copy of a previously saved version of your *.imp file that can function as a complete, standalone model. These files are used as a safety measure to provide you with one or more previous versions of the model in the case that the current model is deleted or damaged.

To customize the number of backup files (i.e. the number of previously saved versions of the model) and the frequency that the model is saved, open the Backup Settings window, which is located under the Files section of the User Options window.

From the Backup Settings window, choose the total number of backup files you want to save by clicking on the dropdown box next to Number of Backup Files Created and specify the total number of files. After the model has been saved initially, subsequent instances of saving the model will create a new backup file each instance up to the limit specified in Backup Settings. Older backup files will be deleted to make room for newer backups.

The most recent backup file will always contain the file extension *.bak001. The file extension of earlier versions is numbered in chronological order with earlier versions receiving larger

numbers in the file extension. Therefore, given any model file, the backup file with a file extension of *.bak002 will always contain the model file saved after (more recently than) the backup file containing the file extension of *.bak003. This extension naming convention applies to any number of backup files.

If you want Impulse to automatically save the model file periodically when the file is open, tick the box next to Automatically Save Model and select the frequency in the dropdown box.

If you need to use a backup file as a standalone model, change the extension to .imp for the backup file, and open the file in Impulse.

You can opt to save no backup files by choosing "0" in the Number of Backup Files Created box.

Encoding

The File Encoding option can be set to "Unicode" in order to use characters outside of the standard english keyboard such as diacritics, cyrillic, etc for text fields such as the pipe/junction name, notes, etc.

Related Topics

Waterhammer

The Waterhammer section of the User Options window allows you to:

- Require a check of the Waterhammer Assistant before running models
- Specify how AFT Impulse handles the situation where transient data for a junction does not go to the end of the transient simulation. Impulse can either use the last data point in the table or extrapolate using the last two data points. Using the last data point is the default option.
- Define if transient cavitation should be based on stagnation (recommended) or static pressure
- Enable or disable the Variable G feature for use in the Environmental Properties panel. By default this option is disabled.

Related Topics

File Import and Export

This section focuses on the various file types that can be imported and exported from AFT Impulse. Topics included discuss importing piping layouts from various sources, importing and exporting between AFT products, and importing and exporting data with Excel.

Contents

Related Blogs

Meet the Import Piping Layout Wizard

Now it's Super Easy! Changing Input Data in AFT Models using Excel Change Data

That was EASY! Quickly Change Pipe and Junction Input Data into AFT models using Excel Change Data

Importing Piping Component Files

AFT Impulse 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.
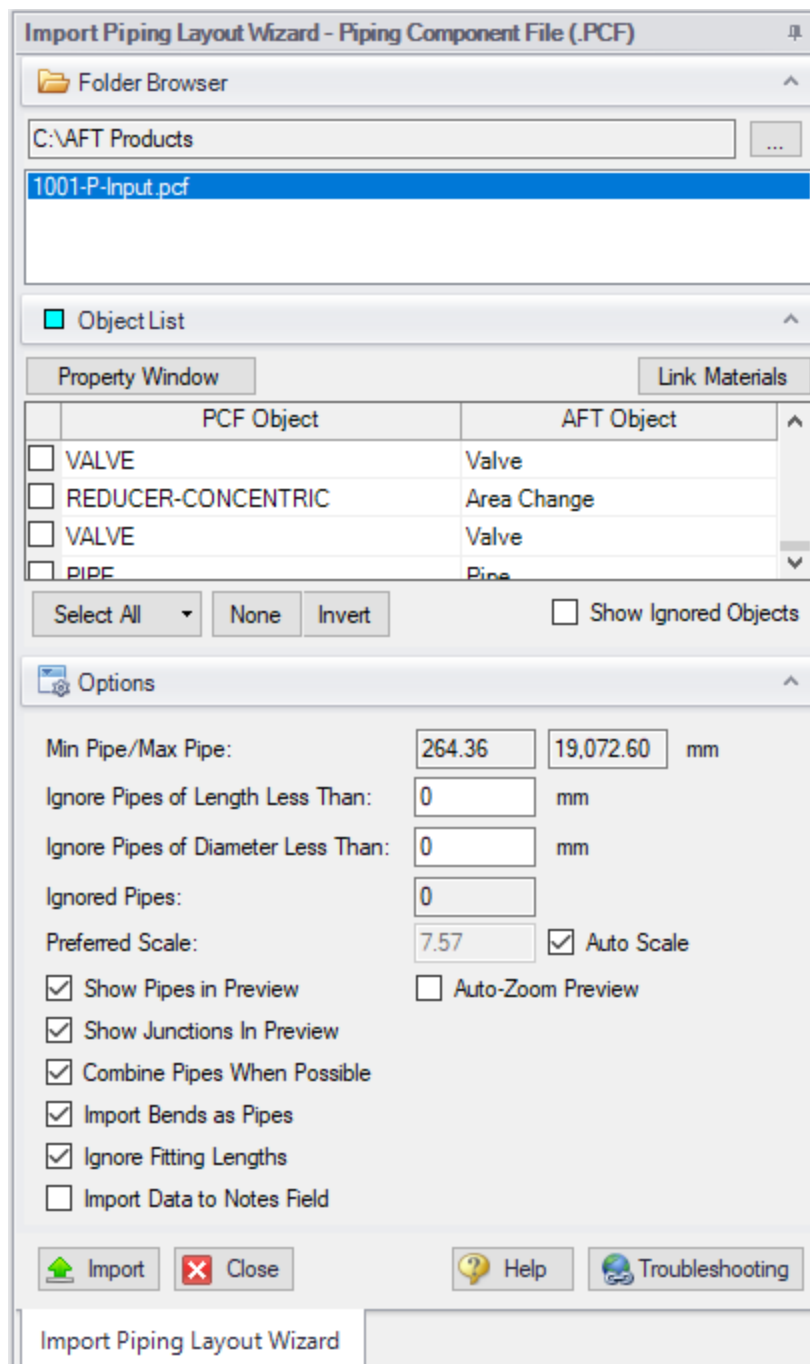
Figure 1: Piping Component File Import

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser -** After browsing to a folder, all .pcf files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the Workspace. The location of the preview can be moved by dragging the icon in the upper-left of the

selection area.

- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain Junction, if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a Relief Valve or a Control Valve. Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.

  - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.

  - **Link Materials** - Opens the File Object Link window, where the Materials Codes from the pcf file can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the pcf file. Selections made for a pcf material will be applied to all pipes that reference that pcf material. Pipes associated with each material can be seen by clicking the arrow on the left expand the group (Figure 2).

  - **Show Ignored Objects** - Certain types of objects are ignored on import - these can be objects such as flanges, or additional file components such as materials or bolt specifications.

- **Options**

  - **Min Pipe/Max Pipe -** Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.

  - **Ignore Pipes of Length Less Than -** Do not import pipes shorter than this amount.

  - **Ignore Pipes of Diameter Less Than -** Do not import pipes smaller than this diameter.

  - **Ignored Pipes -** How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.

  - **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.

  - **Show Pipes in Preview** - Display the pipes in the Workspace preview.

  - **Show Junctions in Preview** - Display the junctions in the Workspace preview.

  - **Combine Pipes When Possible -** It is required that all pipes in the Workspace be connected by two junctions. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be manually combined, before the model could be solved.

  - **Import Bends as Pipes** - Pipe bends can be considered as junctions or simply a visual change in direction of the pipe. This is identical to using segments to show piping layout. Note that these bends are visual only and have no losses associated with them - to account for losses, either junctions or fittings & losses must be used.

This option is helpful to reduce unnecessary model clutter.

- **Ignore Fitting Lengths** - Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.
- **Import Data to Pipe or Junction Notes Field** - Add the raw data from the originating file to the Notes tab of the corresponding object.

File Materials Code	File Materials Description	AFT Material	AFT Size	AFT Type
(None)		(User Specified)		
16HC01-1151	PIPE DIN 2448	Steel - DIN 2458	88.9 mm	1.4 mm w...
PIPE	36519.82 mm	Steel - DIN 2458	88.9 mm	1.6 mm w...
PIPE	238.90 mm	Steel - DIN 2458	88.9 mm	1.6 mm w...
PIPE	1095.35 mm	Steel - DIN 2458	88.9 mm	1.6 mm w...
16HC01-1153	PIPE DIN 2448	(User Specified)		
16HC01-1135	PIPE DIN 2448	(User Specified)		

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

Related Topics

Related Blogs

Meet the Import Piping Layout Wizard

Importing GIS Shapefiles

Import GIS Shapefile

GIS Shapefiles that adhere to the ESRI standard can be imported into Impulse 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.

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 Impulse 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 generate reservoir junctions.
- Valves can map:
  - **Labels** - Names
  - **Elevation**

Related Topics

Related Blogs

[Meet the Import Piping Layout Wizard](#)

Importing Caesar II Netral Files

AFT Impulse 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 File (.cii) brings up the Import Piping Layout Wizard in the Quick Access Panel, as shown below in Figure 1.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser -** After browsing to a folder, all .cii files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the Workspace. The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List -** When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain Junction, if the automatic assignment is

not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.

- **Property Window -** This button opens an additional panel that shows the raw properties for the currently selected object.

- **Link Materials -** Opens the File Object Link window, where the pipes can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the CII file, and will be applied to all of the pipes.

- **Show Ignored Objects -** Certain types of objects are ignored on import - these can be objects such as flanges, or additional file components such as materials or bolt specifications.

- **Options**

  - **Min Pipe/Max Pipe -** Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.

  - **Ignore Pipes of Length Less Than -** Do not import pipes shorter than this amount.

  - **Ignore Pipes of Diameter Less Than -** Do not import pipes smaller than this diameter.

  - **Ignored Pipes -** How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.

  - **Preferred Scale -** The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.

  - **Show Pipes in Preview -** Display the pipes in the Workspace preview.

  - **Show Junctions in Preview -** Display the junctions in the Workspace preview.

  - **Show Labels In Preview -** Show the corresponding label from the .cii file in the Workspace preview. This is only shown in the preview and will not be imported into the model itself.

  - **Combine Pipes When Possible -** It is required that all pipes in the Workspace be connected by two junctions. However, this is not a requirement in all software. Therefore, some .cii files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be manually combined, before the model could be solved.

  - **Import Bends as Pipes -** Pipe bends can be considered as junctions or simply a visual change in direction of the pipe. This is identical to using [segments](#) to show piping layout. Note that these bends are visual only and have no losses associated with them - to account for losses, either junctions or [fittings & losses](#) must be used. This option is helpful to reduce unnecessary model clutter.

- **Ignore Fitting Lengths -** Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.

- **Import Data to Pipe or Junction Notes Field -** Add the raw data from the originating file to the Notes tab of the corresponding object.

Figure 1: CAESAR II Neutral File Import

Related Topics

Related Blogs

[Meet the Import Piping Layout Wizard](#)

Importing and Exporting EPANET Files

AFT Impulse 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.

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.

Selecting File > Export Piping Layout To > EPANET File (.inp) brings up the EPANET Export window, as shown below in Figure 2.

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.
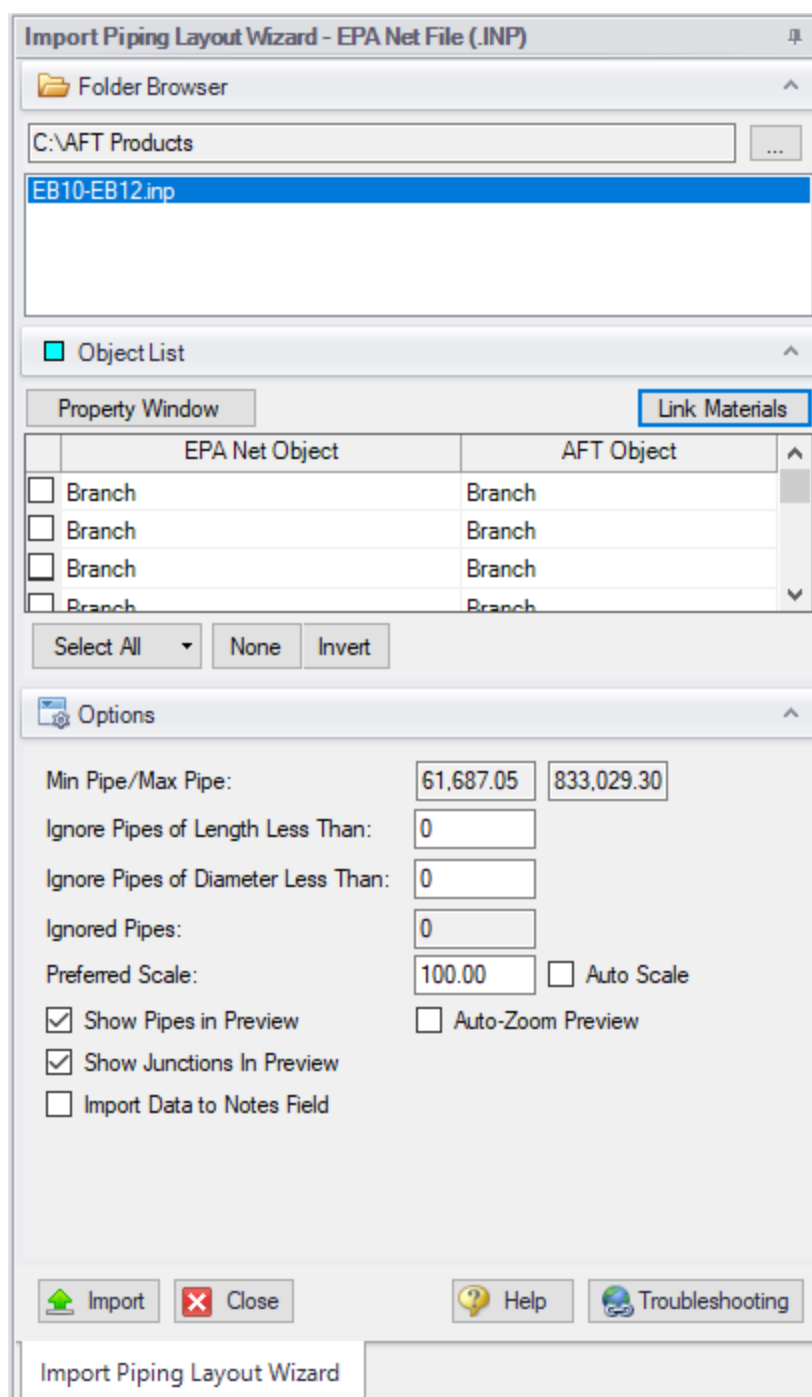
Figure 1: Import Piping Layout Wizard for EPANET files
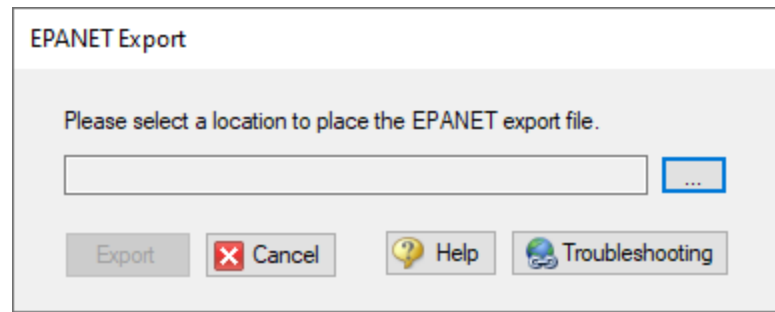
Figure 2: EPANET Export Dialog

Related Topics

Related Blogs

Meet the Import Piping Layout Wizard

Importing and Exporting from Other AFT Products

Models made in other AFT products such as AFT Fathom can be opened directly in AFT Impulse and vice versa. This can be a useful starting point when it is desired to perform additional analysis on a model that has already been built in another AFT product, such as if it is desired to perform transient analysis for a model originally built in Fathom.

To convert a model from one AFT product to another just browse to the model you would like to convert and open it in the software you are converting the model to. This can be done by changing the file extension selected, 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 Impulse 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 Impulse to either a General Component or Branch junction. In addition, Impulse will provide the option to merge junctions with static losses into the connected pipes using the Combining Pipes feature.

Combining pipes when a model is imported to Impulse typically results in a small decrease in accuracy for a large decrease in run time, as is discussed here.

Be aware that some features available for use in other AFT products are not available for use in AFT Impulse. If a junction is defined using a feature not available in AFT Impulse, 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.
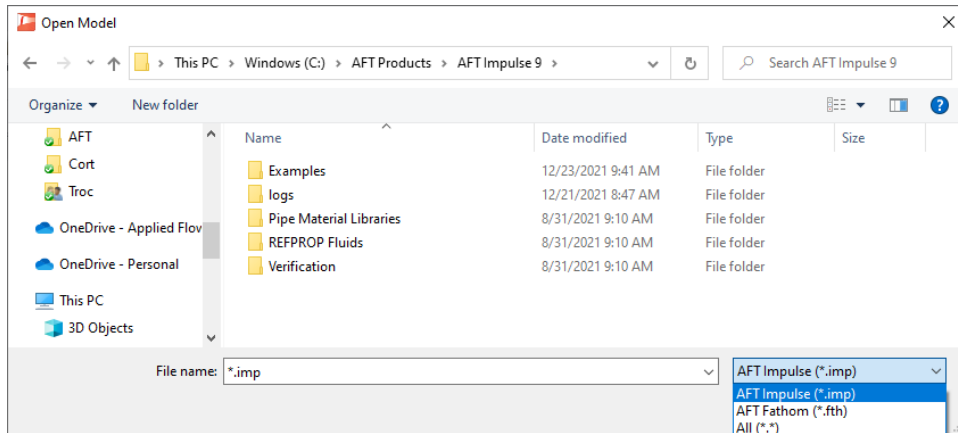
>*Figure 1: File extensions options for Open File window*

Related Topics

Exporting to Excel

AFT Impulse includes a powerful ability to export any Output data to any specific cell(s) in an Excel spreadsheet. Specific values from the model can be exported, as well as columns, rows, or entire output tables as defined in Output Control. These items are controlled with the Excel Export Manager, available from the File menu.
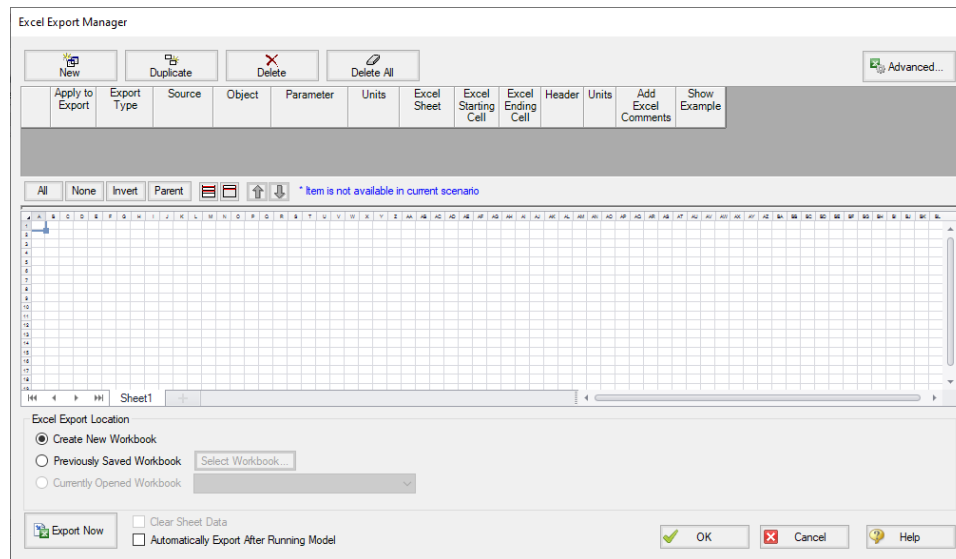
Figure 1: The Excel Export Manager

Adding Items to the Excel Export Manager

Data from the Output can be added to the Excel Export by clicking New, or by selecting an existing item and clicking Duplicate. Items can also be created directly from the Output Window by right-clicking on a cell or header.
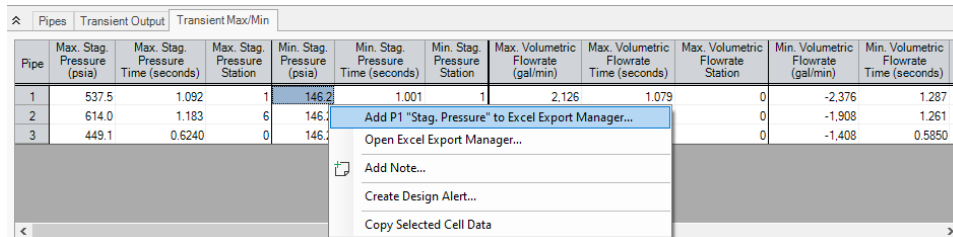
Figure 2: Adding items directly from the Output

Each row in the Excel Export Manager corresponds to a cell, column, row, or table of data to be added to the spreadsheet. Each column helps define what data will be transferred where. The grid below the entries shows a preview of where the data will be added to help prevent intersecting data by showing where there are overlapping regions. Note that graph items can vary in size and do not show a complete region here.

- **Apply to Export** - Items can be optionally included or excluded from the export.
- **Export Type** - What type of data is being exported. This affects other settings, and will affect how much room is required on the target Excel sheet.

- **Single Value** - The value of a single parameter for a single pipe or junction object.
- **Column** - Values of a single parameter for all of the displayed objects.
- **Row** - Values of all displayed parameters for a single object.
- **Table** - Values of all displayed parameters for all displayed objects.
- **Graph** - The graph points from a specified Graph List Item.
- **Other** - Additional information not available in a standard Output table, such as Fluid definition, Model Title, or Scenario Names.

- **Source** - Where the data is coming from.
  - **Pipe** - From the steady-state Pipe section of the output.
  - **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.
  - **Transient Max/Min** - From the Transient Max/Min tab in the Pipes section of the output.
  - **Forces** -
  - **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
- **For Existing Items** -
    - **Add New Item -** The item selected will be added to the Excel Export Manager even if it already exists.
    - **Select Existing -** If the item already exists, do not add a duplicate item and instead select the existing item.
    - **Ask** - If the item already exists, prompt the user on the action to take.
- **Create an Export Guide -** When exporting, an Export Guide can optionally be included in the export. The Export Guide contains information about the model and the generation of the export, to help keep the connection between the data and the model clear for future reference.

Exporting Data From Multiple Scenarios

The Excel Export Manager can be employed with Batch Runs with a couple of important distinctions.

First, if multiple sheets are specified in the Excel Export Manager, the only way to export multiple scenarios is to export **every scenario to its own Workbook**. To keep data from multiple scenarios in one Workbook, there can only be one sheet specified in the Excel Export Manager.

Second, **the Excel File will be saved after each scenario.** In normal single-scenario exports the file is never saved, but in batch mode it is required.

In the Batch Run tool, users are also able to use Excel Export without having to run the scenarios again. Note that if any of the scenarios in the Batch Run do not have output, the batch Export Only will not export any data for that scenario, as it will not run those scenarios to get output.
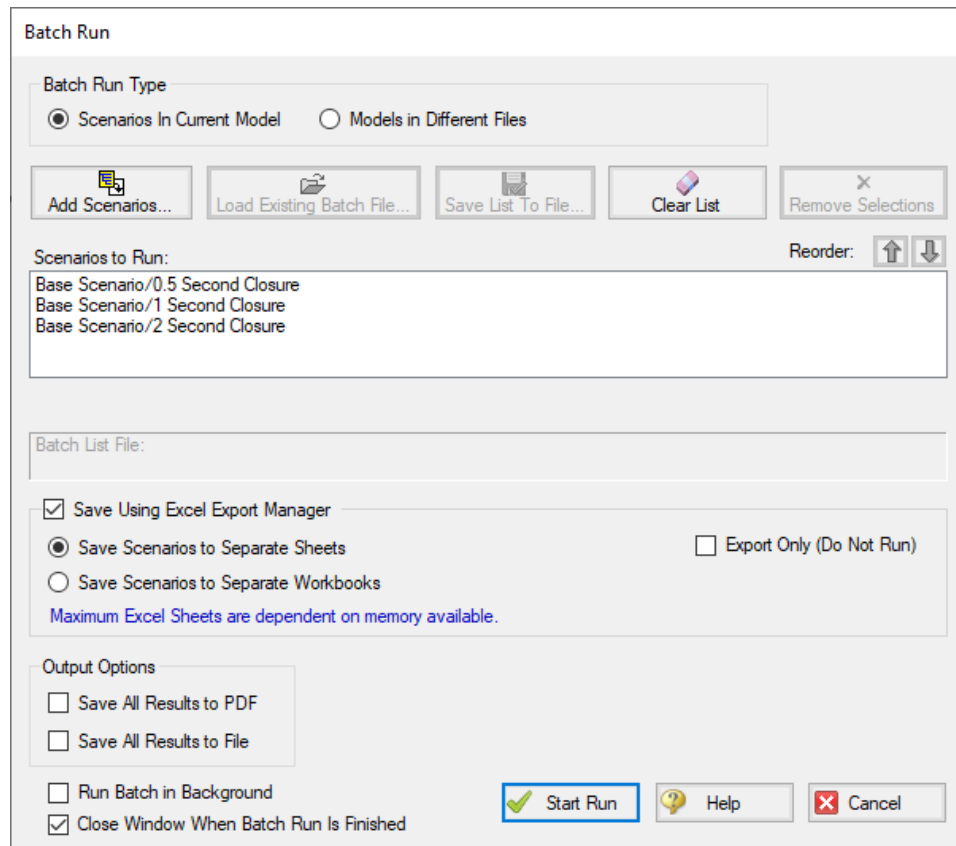
Figure 5: Saving Results From Multiple Scenarios

Related Topics

Related Blogs

Quick Guide to Custom Output

Paint a Clear Picture Faster Than Ever with AFT's New Excel Export Manager

Importing From Excel

A Microsoft Excel spreadsheet can be used to make changes to objects in an AFT Impulse 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 Impulse will search for a worksheet named **AFT Transfer**. There can be other worksheets in the workbook but Impulse will only look for a sheet with this name. The file structure looks like the following:

The changes are entered on a worksheet named "AFT Transfer". There can be other worksheets in the workbook but Impulse will only look for a sheet with this name. The first data is entered in cell B2. Column A and row 1 are not read and are available for user comments. The file structure looks like the following:

	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	FCV setting set	Yes	Control Valve	71	Control Setting	Set equal to value	300	High Demand
3	FCV setting reduced by 20 gpm	No	Control Valve	72	Control Setting	Change by value (+/-)	-20	Low Demand
4	FCV setting increase 10%	No	Control Valve	73	Control Setting	Change by percent (+/-)	10	Average
5	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	
6	Pipe Diameter increase	Yes	Pipe	58	Nominal Size Increment	Change by value (+/-)	1	

Figure 1: Sample AFT Transfer Excel Sheet

Row 1 and column A are not read by AFT Impulse - 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 Impulse installation, or can be downloaded here. By default the file path is: C:\AFT Products\AFT Impulse\AFT Impulse 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.

- 563 -

Object Number

This corresponds to the junction or pipe number of the object being changed.

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 Design Factor - Pipe Fittings & Losses	**S** Initial Guess Pressure	**S** Elevation	**S** Elevation	**S** Initial Guess Pressure
S Design Factor - Pipe Friction	**S** Inlet Elevation	**S** Flow Rate	**C** Junction Name	**S** Inlet Elevation
S ID Reduction (Scaling)	**C** Junction Name	**S** Flow Temperature	**S** Pressure	**C** Junction Name
S Initial Flow Guess	**S** Outlet Elevation	**S** Initial Guess Pressure		**S** Loss Model
S Inner Diameter (for unspecified, cylindirical pipes)	**S** User Specified K	**C** Junction Name		**S** Outlet Elevation
S Length		**I** Special Condition		**I** Special Condition
S Modulus of Elasticity (for unspecified, cylindrical pipes)				**S** User Specified K
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 Win-				

dow) **C** Pipe Name **S** Poisson Ratio **S** Roughness Value **I** Special Condition **S** User Specified Wavespeed **S** Wall Thickness (for unspecified, cylindrical pipes)				
Branch **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction Name **S** Outlet Elevation **S** Source/Sink Flow	**Check Valve** **S** Deceleration Calculation Interval (Estimated Fluid Velocity) **S** Delta Pressure/Head to Reopen (User Specified) **S** Design Factor **S** Disk/Plug Area (Force Balance) **S** Disk/Plug Submerged Weight (Force Balance) **S** Disk Distance to CG (Force Balance)	**Control Valve** **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction Name **S** Loss When Fully Open **S** Outlet Elevation **S** Setpoint **I** Special Condition	**Dead End** **S** Initial Guess Pressure **S** Elevation **C** Junction Name	**General Component** **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction Name **S** K Factor **S** Outlet Elevation **I** Special Condition

	S Disk Inertia (Force Balance)			
	S Forward Velocity to Close Valve (User Specified)			
	S Full Open Loss Value			
	S Initial Guess Pressure			
	S Inlet Elevation			
	C Junction Name			
	S Min Velocity Required to Fully Open (Vo) (Estimated Fluid Velocity)			
	S Outlet Elevation			
	S Plug Max Travel Length (Force Balance)			
	I Special Condition			
	S Spring Constant (Force Balance)			
	S Spring Force with Valve Closed (Force Balance, Translating)			
	S Spring Torque with			

	Valve Closed (Force Balance, Spring)			
Gas Accumulator	**Liquid Accumulator**	**Orifice**	**Pump**	**Relief Valve**
S Design Factor	**S** Elasticity	**S** Area/Diameter	**S** Control Setpoint	**S** Blowdown Pressure
S Initial Gas Volume	**S** Elevation	**S** Design Factor	**S** Delta Pressure to Re-Open Combined Check Valve	**S** Design Factor
S Initial Guess Pressure	**S** Initial Guess Pressure	**S** Exit Pressure		**S** Exit Pressure/Head
S Junction Elevation	**S** Initial Volume	**S** Initial Guess Pressure	**S** Design Factor	**S** Initial Guess Pressure
C Junction Name	**C** Junction Name	**S** Inlet Elevation	**S** Fixed Speed (%)	**S** Initial Elevation
S Maximum Gas Volume	**I** Special Condition	**C** Junction Name	**S** Forward Velocity to Close Combined Check Valve	**C** Junction Name
S Minimum Gas Volume		**S** Loss Value		**S** Loss Value
S Polytropic Constant in Transient		**S** Outlet Elevation	**S** Initial Guess Pressure	**S** Outlet Elevation
S Precharge Gas Pressure			**S** Inlet Elevation	**S** Remote High Pressure
S Precharge Gas Temperature			**C** Junction Name	**S** Remote Low Pressure
S Precharge Gas Volume			**I** Number of Heads (positive displacement)	**S** Set Pressure
I Special Condition			**S** Operating Speed (positive displacement)	**I** Special Condition
S Tank Diameter			**S** Outlet Elevation	
S Tank Height			**S** Rated Pump Speed (rpm)	
S Tank Length			**I** Special Condition	

S Tank Volume			**S** Steady/Average Flow Rate (positive displacement) **S** Submerged Suction Pressure **S** Trimmed Impeller Ratio as Percent	
Reservoir **S** Cross-sectional Area **S** Design Factor **C** Junction Name **S** Liquid Surface Elevation **S** Liquid Surface Pressure **I** Surface Elevation Type **S** Tank Bottom Elevation	**Screen** **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction Name **S** K Factor **S** Outlet Elevation	**Spray Discharge** **S** Design Factor **S** Discharge Flow Area **S** Elevation **S** Exit Pressure **S** Initial Guess Pressure **C** Junction Name **S** Loss Value **I** Number of Holes **I** Special Condition	**Surge Tank** **S** Cross-Sectional Area **S** Initial Guess Pressure **C** Junction Name **I** Special Condition **S** Surface Pressure	**Tee or Wye** **S** Design Factor **S** Initial Guess Pressure **S** Elevation **C** Junction Name
Turbine **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction	**Air Valve** **S** Inlet Cracking Pressure **S** Gas External Pressure **S** Initial Gas Mass In System	**Valve** **S** Design Factor **S** Exit Pressure **S** Initial Guess Pressure **S** Inlet Elev-	**Venturi** **S** Design Factor **S** Initial Guess Pressure **S** Inlet Elevation **C** Junction Name	**Volume Balance** **S** Initial Guess Pressure **S** Elevation **C** Junction Name

Name	**S** Inter-	ation	**S** Loss Value	
S Outlet Elev-ation	mediate Ori-fice Activation Value	**C** Junction Name	**S** Outlet Elev-ation	
I Special Condition	**S** Elevation	**S** Open Per-centage		
	S Initial Guess Pressure	**S** Outlet Elev-ation		
	C Junction Name	**I** Special Condition		
	S Outlet Elev-ation			
	S Full Open Orifice CdA Inflow			
	S Full Open Orifice CdA Outflow			
	S Full Open Orifice CdA Intermediate Outflow			
	I Special Condition			
Weir **S** Channel Width **S** Down-stream Bot-tom Elevation **S** Down-stream Cross-Sec-tional Area **S** Initial Guess Pres-sure **S** Inlet Elev-ation				

C Junction Name				
S Notch Width				
S Outlet Elevation				
I Special Condition				
S Surface Pressure				
S Upstream Bottom Elevation				
S Upstream Cross-Sectional Area				
S Height of Weir				
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	Val-ue
Assigne-d Flow	None	0
	Closed	1
Check Valve	None	0
	Closed	1
	Open	2
Control Valve	None	0
	Closed	1
	Fully Open - No Con-trol	2
Gas Accu-mulator	None	0
	Ignore Accu-mulator	5
General Com-ponent	None	0
	Closed	1
Liquid Accu-mulator	None	0
	Ignore Accu-mulator	5
Pump	None	0
	Pump Off No Flow	1
	Pump Off With Flow Through	2
Relief Valve	None	0

Junc-tion	Special Condi-tion	Val-ue
	Failed Open Ignore Relief Valve	2 5
Surge Tank	None Ignore Surge Tank	0 5
Spray Dis-charge	None Closed	0 1
Turbine	None Closed	0 1
Air Valve	None Ignore Air Valve	0 5
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 Impulse by right-clicking the scenario name and selecting "Copy Scenario Path Name."

- 574 -

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

Related Blogs

Now it's Super Easy! Changing Input Data in AFT Models using Excel Change Data

That was EASY! Quickly Change Pipe and Junction Input Data into AFT models using Excel Change Data

Design Alerts

Design Alerts are a powerful feature that can help speed up the modeling process and help verify that the model is behaving as expected. The creation of a Design Alert does not constrain or affect the operation of the model in any way. However, it automatically creates an output alert any time the design limit is violated. One common use case might be defining a maximum allowable pressure for the pipes in the model. While the output can be reviewed manually for these violations, this quickly becomes tedious as the model grows in size. Creating a Design Alert instead allows the engineer to spend less time reviewing output tables as any violation will be immediately identified.

The Design Alert Manager allows the user to create or modify Design Alerts. It can be accessed from the Tools Menu or in the Design Alerts tab in pipe and junction properties windows.

Design Alerts can also be created directly from the output, or can be imported/exported for use in other models.
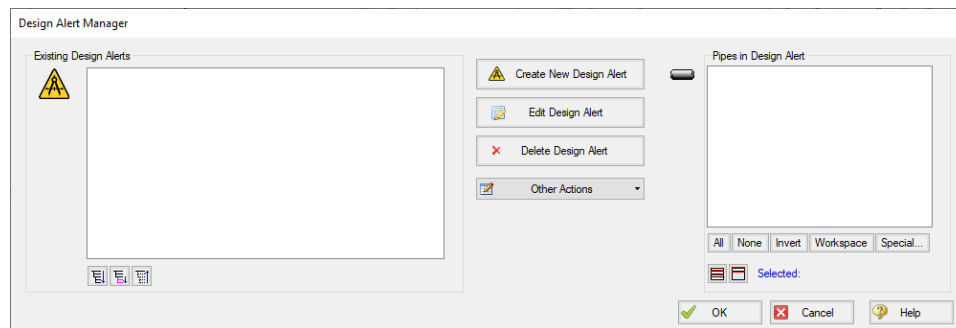
Figure 1: Design Alert Manager

Creating and Applying Design Alerts

Click Create New Design Alert to add a new Design Alert to the model. The Design Alert Manager expands to allow the creation of the new alert.

A Name, Object Type, and at least one Parameter are required. Depending on the Object Type, a different selection of Parameters is available, as shown below. To add a Parameter:

1. Select the desired parameter
2. Select whether this is the maximum or minimum allowable value
3. Enter the value with the appropriate units (if applicable)
4. Click Add to List.

Several Parameter limits can be added to one Design Alert. For example, an alert can be created that requires a pressure within a range by adding both the maximum and minimum pressure limits.

To update a Parameter, simply create it again and click Add to List - the old value will be replaced with the new one.
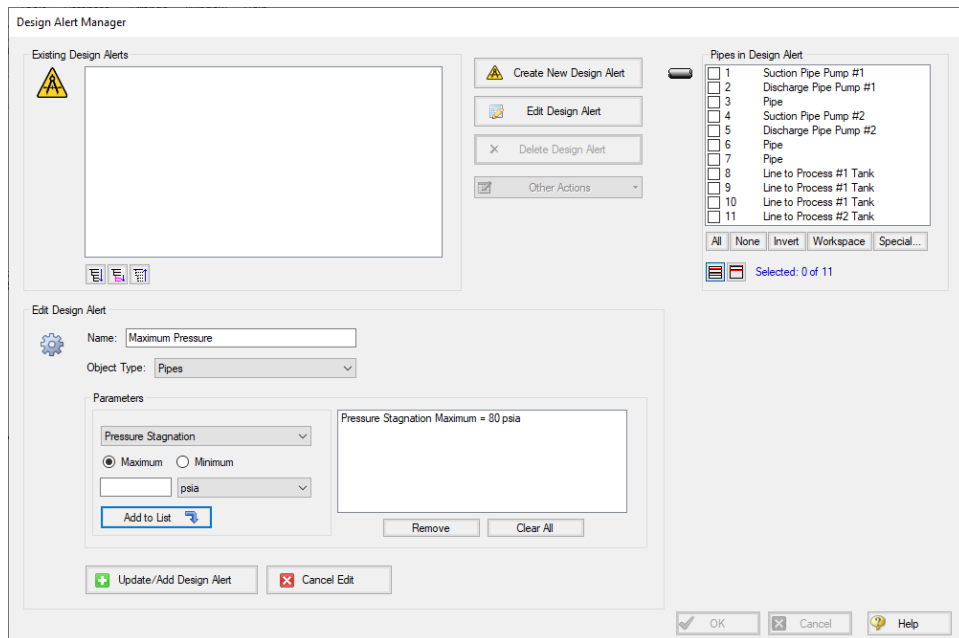
Figure 2: Creating a Design Alert

Pipes	General Junctions	Pumps	Valves	Forces
EGL	Gradeline Energy Inlet (EGL)	Head Rise	Cv	Maximum Absolute Force
HGL	Gradeline Energy Outlet (EGL)	Mass Flow Rate	Flow Area	Maximum Negative Force
Mass Flow Rate	Gradeline Hydraulic Inlet (HGL)	NPSHA	Head Loss	Maximum Positive Force
Pressure Gradient Stag. Total	Gradeline Hydraulic Outlet (HGL)	Percent of BEP	K	
Pressure Stagnation	Pressure Loss Stagnation Total	Power Overall	Kv	
Pressure Static	Pressure Loss Static Total	Pressure Discharge Stagnation	Mass Flow Rate	
Vapor Pressure Margin	Pressure Stagnation Inlet	Pressure Discharge Static	Open Percentage	
Velocity	Pressure Stagnation Outlet	Pressure Rise Stagnation	Pressure Drop Stagnation	
Volumetric Flow Rate	Pressure Static Inlet	Pressure Suction Stagnation	Pressure Inlet Static	
	Pressure Static Outlet	Pressure Suction Static	Pressure Outlet Static	
		Speed	Volumetric Flow Rate	
		Volumetric Flow Rate		

Figure 3: Available Design Alerts

At this point, the Design Alert still has not be created. To finalize it, click "Update/Add Design Alert."
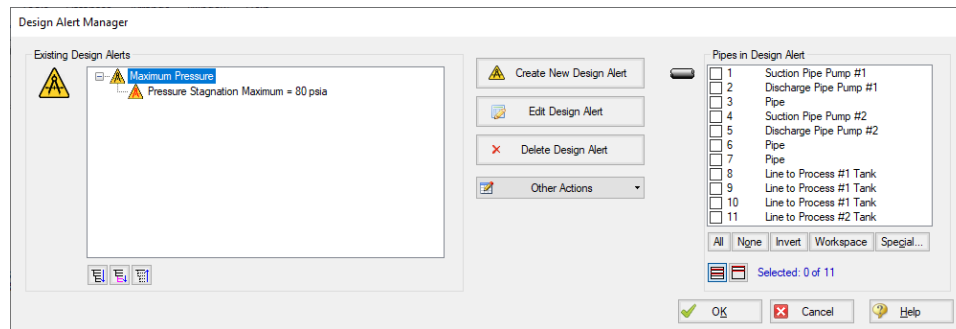
Design Alerts

Figure 4: An Existing Design Alert

The Design Alert now exists, but has not yet been applied to any objects in the model. To include objects in the model, select them in the pane on the right.
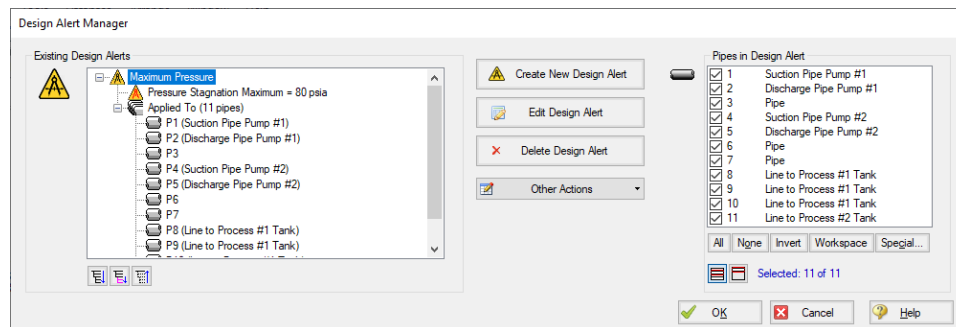
Figure 5: Fully Defined Design Alert

Note that the application of Design Alerts can also be managed for individual objects in their Properties Window's Design Alerts tab.

- 579 -

Figure 6: Apply Desired Design Alerts to this Object

When running the model, the selected object's output values will be checked against the limits that have been defined. If any of the Design Alerts are violated, the user will receive a message on the Design Alert tab and the violating parameters will be highlighted in the output tables.
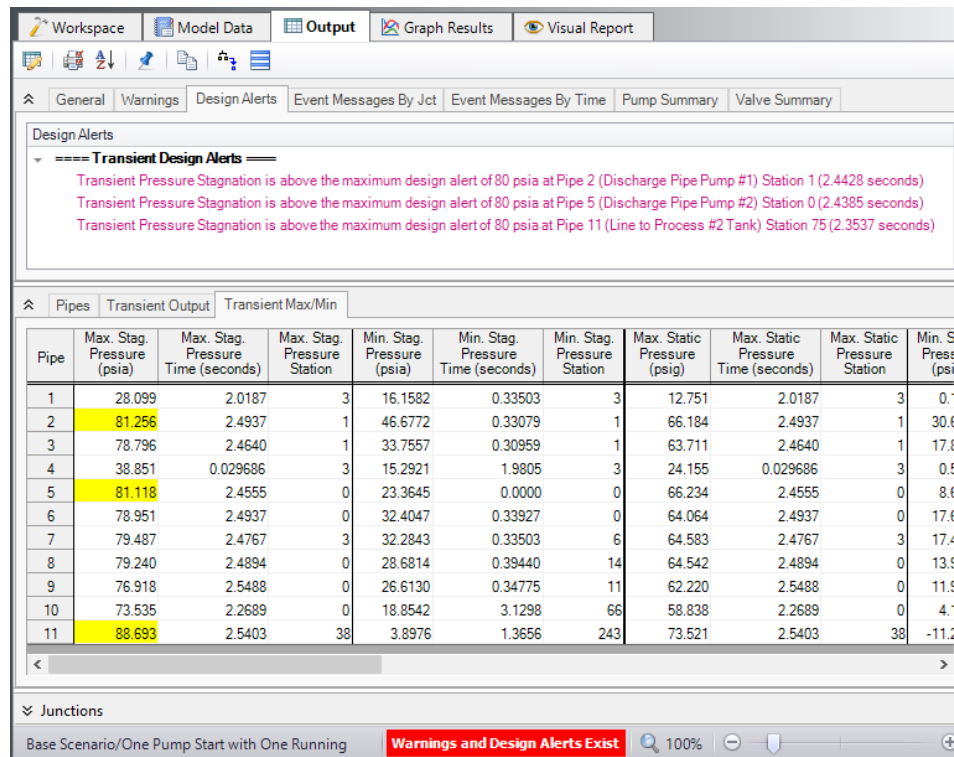
Figure 7: Design Alert Violations in Output

Design Alerts can be cross-plotted in the Graph Results window vs. the actual results. One good example where this is useful is for cross-plots of maximum and minimum allowed operating pressures.

Design alerts are visible in every scenario in a model, but can be turned off or on for specific scenarios. When creating a design alert for the first time it will be active in that scenario and all its child scenarios. For more information about scenarios, please read the Scenario Manager 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...
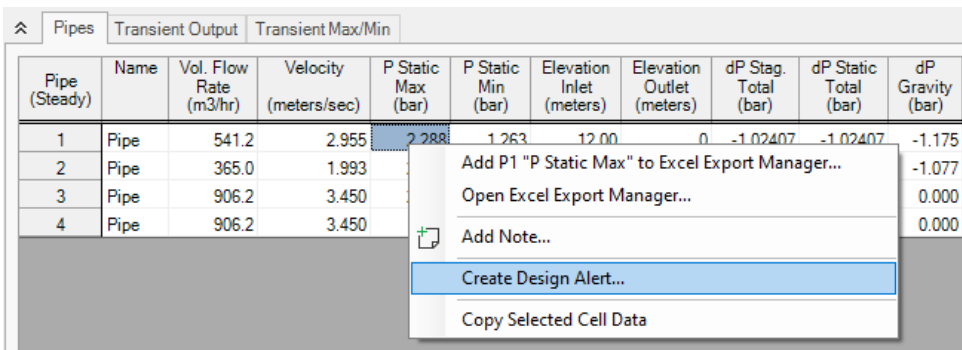
Figure 1: Output Context Menu

Create Design Alert automatically defines a Design Alert based on the selected value and quantity:
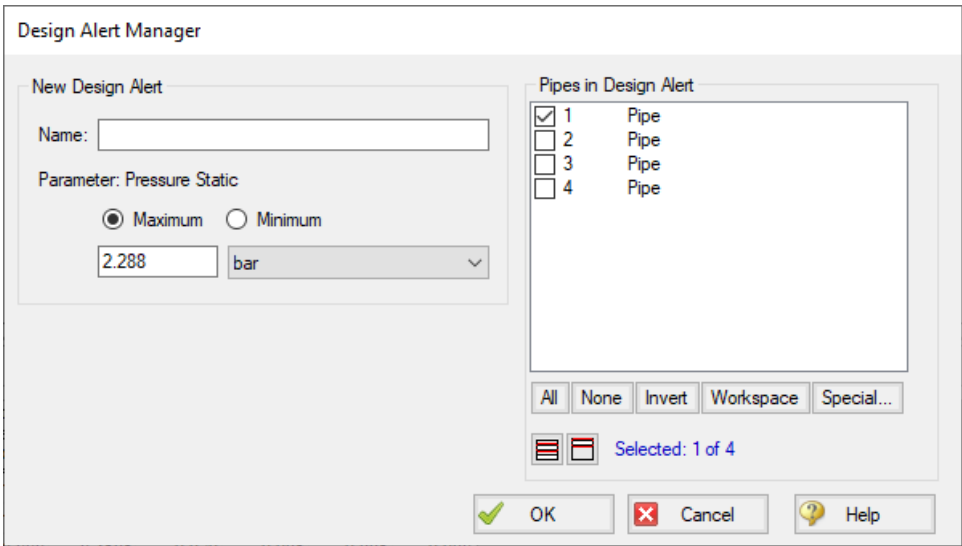
Figure 2: Automatically defined Design Alert

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 Alert Importing And 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.
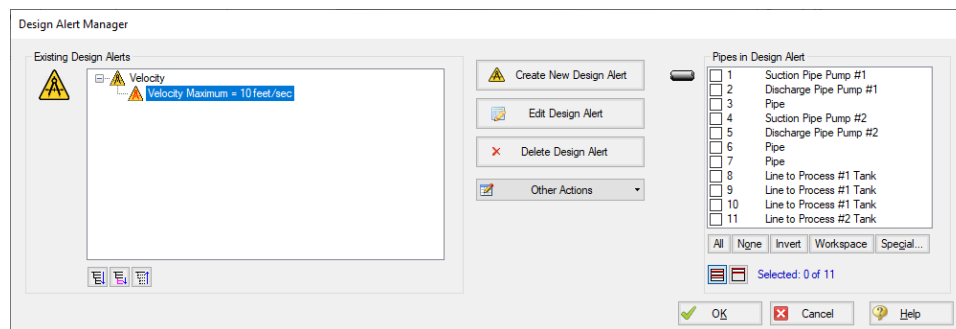
2. From the Other Actions menu, select Save, then save the file with a descriptive name.
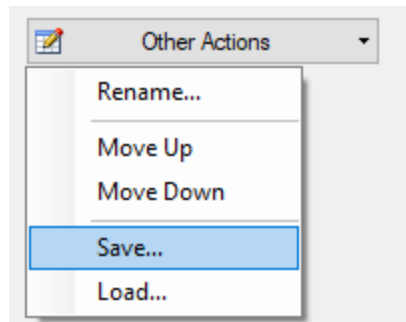
Import a Design Alert

1. Open the Design Alert Manager from the Tools menu.
2. From the Other Actions menu, select Load, then browse to the saved design alert file.
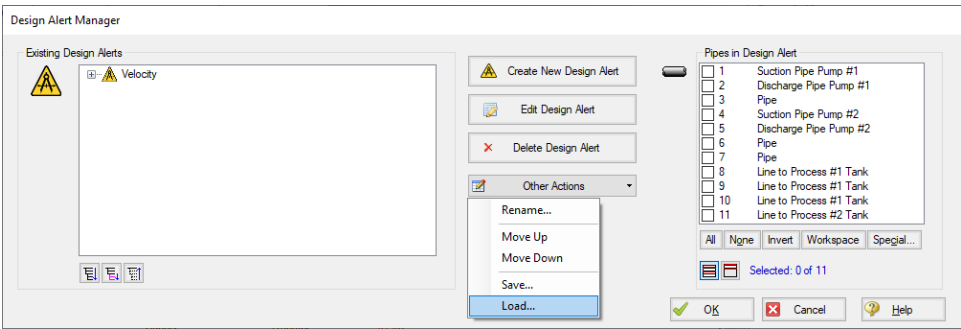
3. The Design Alert should now be added to the Existing Design Alerts list, and can be applied to objects in the model.

Related Topics

Scenario Manager Overview

Scenario Manager (in the Quick Access Panel) is a powerful tool for managing variations of a model, referred to as scenarios. It lets you to quickly test the same model with selected differences (such as pump size, pipe diameter, system location, units, etc.) to find the system which works best. The Scenario Manager allows you to:

- Create, name, and organize scenarios
- Select the scenario to appear in the [Workspace](the 'current' scenario)
- Delete, copy, and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario's properties
- Pass changes from a scenario to its variants
- See what scenarios have output by automatically changing the text color to blue if they have output

All scenarios are saved within the model file.

Scenario

A variant case of a model that is created in the Scenario Manager.

Base Scenario

The Base Scenario is the root scenario at the top of the scenario tree. As the highest level ancestor, It does not inherit from any other scenario. A new scenario can be inserted above the base scenario, at which point the new scenario would behave as the base scenario, and the original base scenario would behave as a child scenario. "Base Scenario" is the default name given in a new, single scenario model.

The Scenario Manager

Illustrated in Figure 1, the top of the Scenario Manager displays the scenario tree, showing the names of the currently defined scenarios, their relationship to other scenarios, and which is the current Workspace scenario (the current scenario is identified both as the scenario with a green check mark next to it and in AFT Impulse'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.
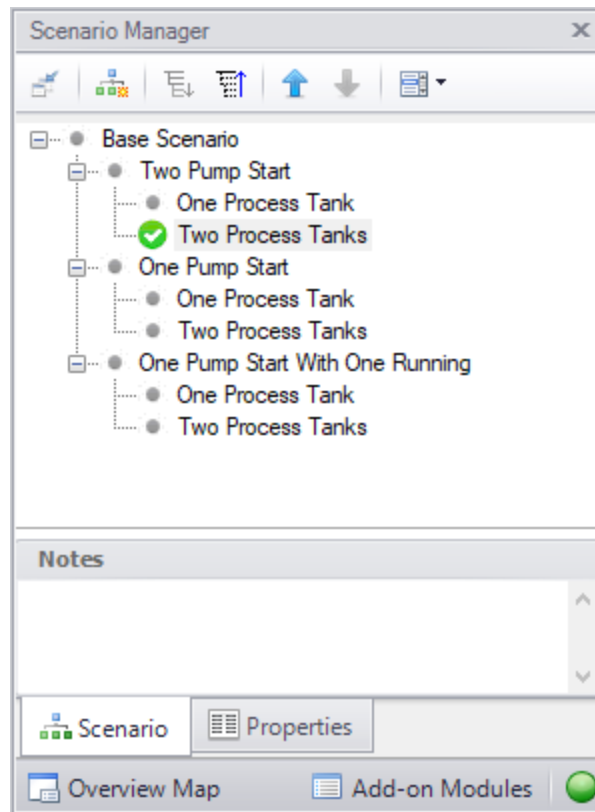
Figure 1: Scenario Manager on Quick Access Panel

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 three children: "Two Pump Start", "One Pump Start", and "One Pump Start with One Running". Each of these have two children: "One Process Tank" and "Two Process Tanks", representing the different operating cases where the process fluid could be fed to either one or two storage tanks.
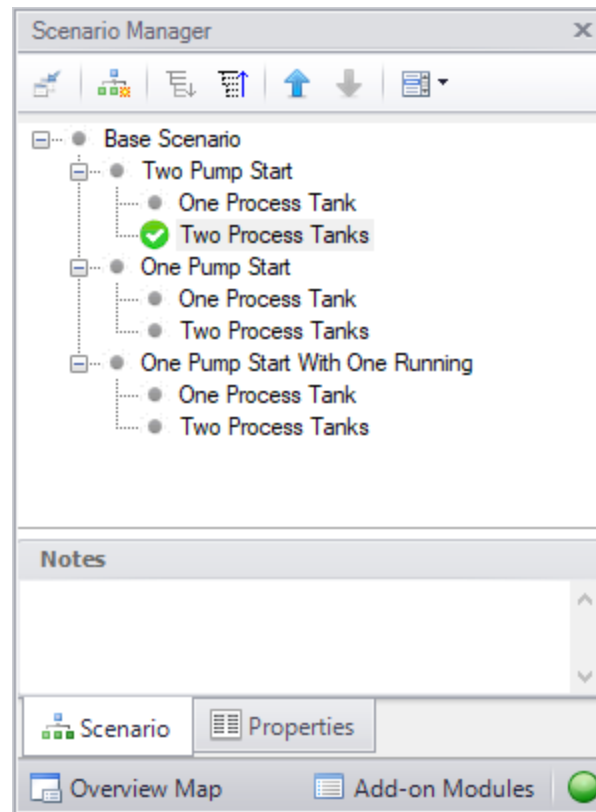
Figure 1: Scenario Manager Window

The "Two Pump Start", "One Pump Start", and "One Pump Start with One Running" scenarios were created by first selecting the Base Scenario and clicking 'Create Child'. Each of the "One Process Tank" and "Two Process Tanks" scenarios were created by choosing 'Create Child' for each of the three different pump configuration scenarios.

Note: Alternatively, one could have made the "Two Pump Start" scenario current and created the "One Process Tank" and "Two Process Tanks" scenarios below it. Then, with "Two Pump Start" still selected, select 'Clone With Children' twice, then rename the new clones "One Pump Start" and "One Pump Start with One Running".

The "One Process Tank" and "Two Process Tanks" scenarios may be thought of as grand children of the Base Scenario. This concept of scenario ancestry is helpful in keeping track of the differences and similarities between scenarios discussed later.

A box with a plus sign next to a scenario indicates it may be expanded to reveal children scenarios while a box with a minus sign indicates the level is fully expanded with all children scenarios shown, if any. No box indicates there are no children. Levels may be expanded by clicking on the plus sign and condensed by clicking on the minus sign.

Right-clicking a scenario provides the following choices:

- **Create Child with No Output -** create child scenarios without results saving duplication and load time
- **Rename -** renames the selected scenario
- **Delete Scenario -** deletes the selected scenario
- **Delete All Children -**deletes all children of the selected scenario
- **Clone Without Children -** creates a copy of the selected scenario at the same level (a sibling). Cloning a scenario prompts the user for a name for the new scenario
- **Clone With Children -** creates a copy of the selected along with its children. Cloning a scenario prompts the user for a name for the new scenario
- **Insert Scenario Above Selected... -** creates a new scenario where the selected scenario is and shifts the selected scenario and its children down a level. This can be used to create a new base scenario. If this is done the user will be prompted to rename the previous base scenario.
- **Promote -** raises a scenario one level, i.e. from a child to a sibling, active only for scenarios more than one level below the base scenario
- **Copy/Save Scenario Names**
    - **Copy Scenario Path Name -** copy the fully-qualified name of the scenario to the Windows clipboard ("Base Scenario/Case 1/Child 3")
    - **Copy Scenario Path Name With File Name -** copy the fully-qualified scenario name along with the model name ("Model.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 to scenario to a separate file without its children
- **Save Scenario to File With Children -** saves the selected scenario and its children to a separate file
- **Compare to Active Scenario -** compares all parameters for all objects between the selected scenario and active scenario and displays the Comparison Grid showing the results
- **Scenario Comparison Tool -** opens the Scenario Comparison tool to create a custom scenario comparison

Note: It is important to remember that AFT Impulse's Edit menu Undo does not apply to Scenario Manager operations. To undo a Rename scenario operation, you can rename the scenario again to the old name. To undo a Clone or Save Scenario operation, you can delete the clone or file created. Once a scenario is deleted or promoted, there is no way to un-delete or un-promote the scenario.

Viewing Scenario Differences

A scenario's data and that for all of its direct ancestors can be viewed in the Model Data Window. This functionality can be enabled in the Model Data Control window.

Fast Scenario Changes

AFT Impulse remembers the most recent scenario you had open, and allows you to quickly change back to that scenario by using the Last Scenario feature. Last Scenario is found on the View menu and Toolbar.

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 Impulse's Status Bar.) Once a scenario is the current scenario, changes may be made to differentiate it from its parent using any of AFT Impulse's editing tools and functions. When an attribute is changed within a scenario it breaks the link for this attribute. Subsequent changes in the scenario's ancestors will no longer affect this attribute.

Properties of a scenario that may differ from its ancestors include:

- Presence and location of objects - one may add or delete pipes and junctions within a scenario and the workspace locations may vary from scenario to scenario
- Pipe and junction properties - all values set in the Properties windows
- Fluid properties - fluid selected, temperature of fluid, viscosity model, system data
- Steady Solution Control - solution method, tolerances, relaxation
- Output Control - Output window content and formatting
- Transient Control - The transient settings can be varied between scenarios
- Library Manager - The connected library content can be varied between scenarios
- Visual Report Control - Visual Report content and color maps

Model Data Control and User Options apply to all scenarios.

Scenario Inheritance

Scenario Manager will pass changes made in a scenario to linked attributes of its descendants, allowing multiple scenarios to be modified in one operation and maintain similarity between scenarios where desired.

Any attribute whose value is the same as that of the scenario's parent is linked to the parent. In turn, if the parent's attribute value is the same as that of its parent, then the child's attribute is linked to its grandparent. This ancestral linking of attributes can exist across any number of generations. A change made to a scenario will be passed downward to its children, grandchildren, etc., as far as the link exists. Changes are NOT passed upward from child to parent.

Once an attribute is changed within a scenario, the link is broken to that individual attribute. Subsequent changes in the scenario's ancestors, parent, grandparent, etc., will no longer affect that attribute, but all other attributes within the scenario will remain linked. The Model Data window with 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 its Diameter would be 2, not 3, as would Grandchild #1.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--

If a scenario is changed, and its child has different data, then the change will not pass downwards (see Table 3).

Table 3: Changing the Base Scenario Diameter from 3 to 6 would not impact Child #1 or any descendants in that line. Changing the Length from 25 to 40 would also change the length in Child #1, Grandchild #1, and any descendants of Grandchild #1.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--

	Diameter	Length
Base	6	40
Child #1	2	--
Grandchild #1	--	--

If a child scenario has data that is different than the parent, its children cannot relink to the parent (see Table 4).

Table 4: Even if the Grandchild #1 has the same Diameter as the Base, it is not linked to the Base because it and its parent are not blank.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	3	--

If a child scenario's data, which was previously changed and is thus different from the parent, is changed back to the same 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 it's Diameter is also the same.

	Diameter	Length
Base	3	25
Child #1	3	--
Grandchild #1	3	--

	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

Table 6 shows data for two pipe properties across three scenarios. The data that would be used when each scenario is loaded is as follows:

- Base scenario
  - Diameter = 3
  - Length = 25
  - Changes to Diameter will not pass downwards
  - Changes to Length will affect only Child #1.
- Child #1 scenario
  - Diameter = 2
  - Length = 25
  - Changes to Base Diameter will not affect Diameter
  - Changes to Base Length will affect Length
- Grandchild #1 scenario
  - Diameter = 2
  - Length = 15
  - Changes to Base Diameter will not affect Diameter
  - Changes to Child #1 Diameter will affect Diameter
  - Changes to Base Length or Child #1 Length will not affect Length

Table 6: Input data for these properties explained in the text.

	Diameter	Length
Base	3	25
Child #1	2	–
Grandchild #1	--	15

Re-Establishing Broken Scenario Links

A link may be re-established by returning the attribute to the same value as that of its parent. This can be done manually by entering the value or selecting the Copy Data From Pipe list and selecting the Parent Pipe Data option. Within other data windows, it is done by selecting the Same As Parent option in Analysis Setup, Output Control, Library Manager or Visual Report Control.

Since links are identified by comparing attribute values of pipes or junctions with the same Workspace ID number, renumbering a scenario will break the links of all pipes and junctions renumbered. Since numbers must be unique, once a link has been broken by renumbering, it may not be re-established.

Related Topics

Running Models in Batch Mode

AFT Impulse scenarios or models can be run in batch mode, a feature that is especially useful for running a group of scenarios or models during lunch or overnight.

To run models in batch mode, follow these steps:

1. Click "Start Batch Run" on the File menu.
2. Select the Batch Run Type. This is where you select if you want to run scenarios from the current model or different model files. If the current model does not have any scenarios, then only the second option will be available.
3. If the Batch Run Type is "Scenarios in Current Model", then choose the Add Scenarios button and select the scenarios you wish to run.
4. If the Batch Run Type is "Models from Different Files", then the models can be selected individually using the "Add Model Files" button, or loaded from a Batch File (by Choosing the "Load Existing Batch File" button). A Batch File is a text file listing all models you would like to run. At any time, a selected model file can be removed by clicking the "Remove Selections". The current list of models can be saved to a Batch File for use in the future by clicking the "Save List to File" button.
5. Use the Output Options in the lower left to save the output data to a file and/or send it to a printer or Adobe PDF file. All reports will use the specified font.
6. Alternatively, the option can be chosen to Save Using Excel Export Manager by either saving to separate sheets, or saving the scenarios to separate workbooks. The Excel Export option can only be chosen if the batch run type is for Scenarios in Current Model. Additionally, the desired output must first be configured in the Excel Export Manager under the File menu for any data to be exported.
7. Check the box to Run Batch in Background if you wish for all selected scenarios to run without showing the Solution Progress window. AFT Impulse 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 Impulse then opens each scenario or model file in sequence, automatically runs each scenario/model that has a completed Analysis Setup window, and sends results to the specified destination. All pertinent information that is generated during the batch run (such as error messages) are automatically displayed when the final scenario/model is finished. Output files generated from each scenario or model run is automatically saved before moving to the next run.

The entire batch run can be canceled at any time by clicking the Cancel button in the Solution Progress window. Output from scenarios that have already been run is still retained when the Batch Run is canceled.

Note: You can plot output from multiple scenarios using Multi-Scenario Graphs.

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

Scenario Comparison Tool Overview

Scenario Comparison Tool (accessed via the Tools menu) is a powerful tool that allows users to compare model parameters between scenarios in a given model. It lets users quickly identify differences between two or more scenarios without needing to search through the model data for each scenario or inspect each element of the model. This tool can be especially powerful for users reviewing others' models, or for users returning to models after a period of time. It can also be powerful for troubleshooting purposes. The Scenario Comparison Tool lets you:

- Compare any two or more scenarios
- Compare selected Pipes and Junctions between selected scenarios
- Compare selected Pipe and Junction parameters between selected scenarios
- Compare selected General Properties between selected scenarios

Related Topics

Related Blogs

Five Fantastic New Features for Fast Fluid Transients: Impulse 8

New Feature Highlight - Scenario Comparison Tool

Accessing Scenario Comparison Tool

The Scenario Comparison tool can be accessed from the Scenario Manager, the Workspace, and the Tools menu.

From the Scenario Manager

To set up a comparison from the Scenario Manager, first select the pipes and junctions to compare in the workspace. Then, select the desired scenario to compare the current scenario to in the Scenario Manager. Note the current scenario is indicated by the green check-mark symbol. Right-click the desired scenario, and select Compare Selected Scenarios.... This will bring users to the Scenario Comparison window and display a grid of the differences between the selected objects in the two scenarios.

To modify the comparison use the Modify Comparison button at the bottom of the window.

From the Workspace

From the workspace, users can quickly select desired pipes and junctions in the scenario and compare them to those found in any other scenario.

First, select the desired pipe(s) and/or junction(s) in the workspace. Then, right-click the Workspace, choose Compare Scenarios, and select the desired comparison. This will bring users to the Scenario Comparison window and display a grid of the differences between the selected objects in the two scenarios.

To modify the comparison use the Modify Comparison button at the bottom of the window.

From the Tools Menu

From the Tools menu, there are multiple options to generate the Scenario Comparison Report.

A comparison grid can be directly generated by selecting the desired objects to compare in the Workspace, then selecting the scenarios to be compared from the Compare Scenarios menu option.

Alternatively, the Scenario Comparison Tool menu option can be selected to build a custom scenario comparison by directly selecting the scenarios, objects, and parameters to be included in the comparison. See Modify Comparison for more information on the available options.

Click the Show Comparison button at the bottom of the window to view the comparison. To make additional changes, use the Modify Comparison button at the bottom of the window.

Related Topics

Related Blogs

New Feature Highlight - Scenario Comparison Tool

Modify Comparison

Several Options are available to customize the Scenario Comparison Grid. These options are available from the Modify Comparison button at the bottom of the scenario comparison report, or by selecting Scenario Comparison Tool from the Tools menu. The Modify Comparison window is shown in Figure 1 below.
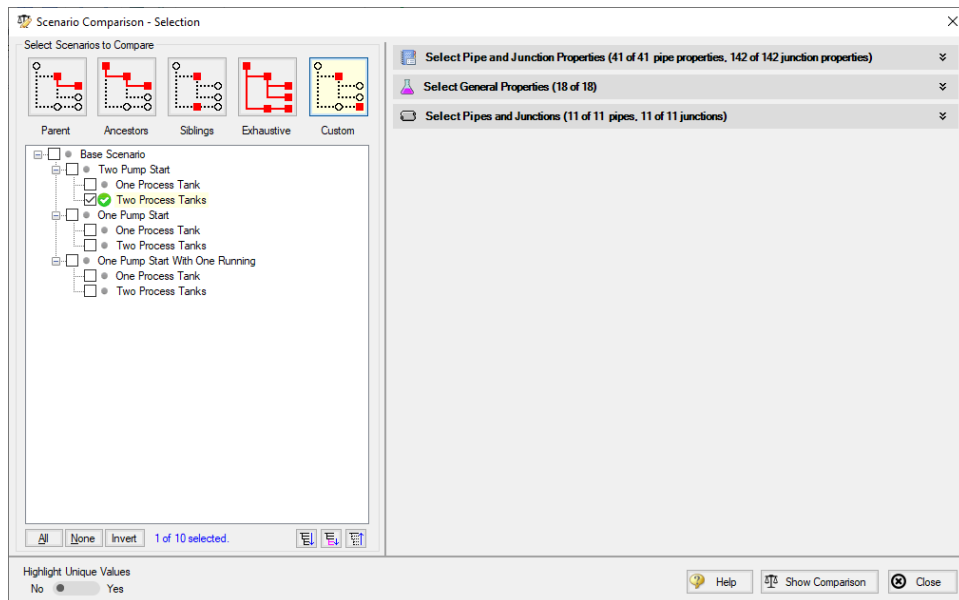
Figure 1: Scenario Comparison window options to Modify Comparison

Select Scenarios to Compare

The Scenario Comparison Tool has four pre-set templates for comparing scenarios, plus the ability to set up a custom comparison. These are:

- **Parent:** This template compares the current scenario with that scenario's parent.
- **Ancestors:** This template compares the current scenario with all of that scenario's direct ancestors. This setting would be similar to the comparison available in the Model Data window, though it does provide a more comprehensive comparison.
- **Siblings:** This template compares the current scenario with all of that scenario's siblings.
- **Exhaustive:** This template compares all of the scenarios in the model with each other.
- **Custom:** This option allows users to compare any two or more scenarios throughout the model.

Select Pipe and Junction Properties

This section sets what parameters will be compared such as length, diameter, roughness, etc.

Users can choose to compare a set of specified properties, or all pipe and junction properties by selecting the desired properties under Select Pipe and Junction Properties. Only properties selected in this section will be considered when the comparison is run.

A helpful option to filter the parameters to be displayed in the comparison is to display the parameters shown in the Model Data window. This can simplify the comparison, but might hide some differences between scenarios. Use the Model Data Control window to further define what parameters are shown with this option.

Select General Properties

This section sets what General Properties will be compared such as fluid properties, tolerances, cavitation models, pipe sectioning, etc.

Users can choose to compare all or a select set of general properties for the scenarios including options that are located in the Steady Solution Control, Fluid Properties, Pipe Sectioning, and Transient Control groups of 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

Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

<div style="background-color:#c0504d; color:black; padding:8px;">

Scenario Comparison Grid

</div>

Choosing the Compare Selected Scenarios menu item from the right-click or Tools menu will bring the user directly to the Scenario Comparison Grid based on any selected Workspace objects and scenarios. The settings for the scenario comparison can be seen/modified by clicking [Modify Comparison](#) at the bottom of the window.

The tabular data from the report can be copied to and pasted to a separate file or worksheet using the Copy button at the bottom of the window.

Figure 1: Default Scenario Comparison Report view

Filtering Options

When a large comparison is run, typically many of the objects/parameters in the model will be unchanged across scenarios, which may result in a large number of items with no differences. To help condense the Scenario Comparison Grid, several toggles are available at the top of the Item and Parameter columns which allow the user to show/hide rows in the report. Switching the toggle to All will show all items or parameters being compared. If the toggle is set to differences, then only Items/Parameters that have a different value in at least one scenario will be shown. Thus, if both toggles are set to All, then all parameters will be displayed for all items. If both toggles are set to Differences, then only items that have parameters with different values will be shown, and only the parameters that differ will be displayed.

Color Coding

The cells in the Item and Parameter columns will be colored either yellow or green to indicate whether that item/parameter has differences between the scenarios being compared. <mark>Yellow</mark> indicates differences, while <mark>green</mark> 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.
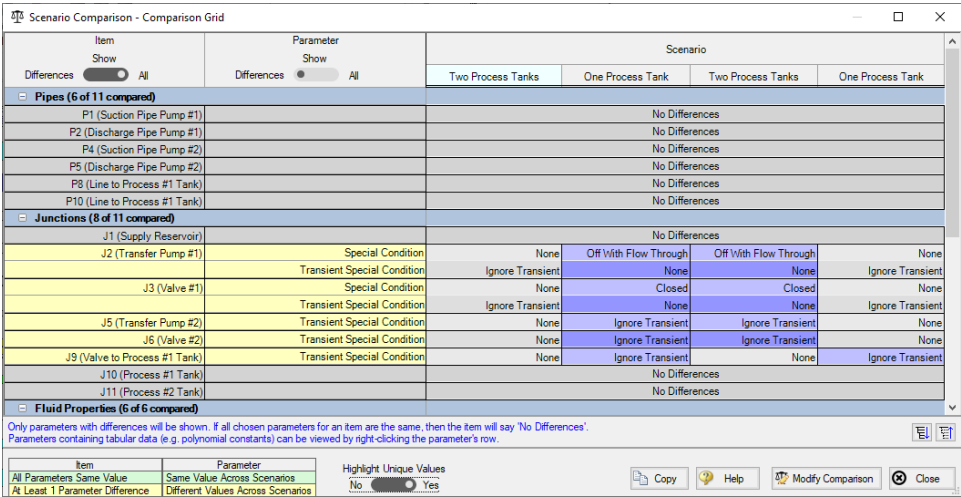
Figure 2: Scenario comparison showing differences only. Each unique value is highlighted with a different color to indicate which scenarios have differences from the currently active scenario, which is the first Two Process Tanks scenario for this example.

Related Topics

Related Blogs

[New Feature Highlight - Scenario Comparison Tool](#)

Add-on Modules

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

- PFA (Pulsation Frequency Analysis) - Helps identify and avoid resonant frequencies in systems, especially in those caused by positive displacement pumps.
- SSL (Settling Slurries) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.

Related Topics

PFA - Pulsation Frequency Analysis

What is the PFA Module?

(PFA Module Only) Steady state pressure and flow dynamics, henceforth referred to simply as "pulsation", is a common problem in PD pump systems, which can cause high vibration, fatigue failures, frequent maintenance outages, and flow uniformity or product quality problems.

The PFA module helps engineers identify and avoid resonant frequencies in systems, especially in those caused by positive displacement pumps. Problems can arise when the frequencies that are excited by pulsation are at or near the acoustic resonant frequencies of a fluid system. The module is used to calculate the acoustic resonance frequencies that could damage system equipment.

How Does the PFA Module Work?

The Pulsation Frequency Analysis (PFA) module is used to determine resonant frequencies that could damage system equipment by calculating the acoustic resonance.

The general steps of pulsation analysis involve the following:

1. The piping and process equipment system is first modeled.
2. A pulse is generated to determine excitation frequencies, then a Low Pass Filter is applied to the pulse in order to determine the forcing function. This forcing function is then placed in the transient data of the junction at which the pulse is applied.

3. The model is run with the forcing function to generate the Excitation Frequency Analysis graph, and the user selects frequencies for analysis from the graph. If the pulsation source is specified to be a PD pump, the PFA module calculates the PD pump speeds (RPM) that will excite the chosen frequencies, and a table of PD pump speeds that abide by the specifications in the Pulsation Setup group in Analysis Setup will be populated.

4. The pump speeds of interest to be analyzed are selected by the user, and the PFA module automatically generates read-only child scenarios that model the pump operating at the selected speeds.

5. The system can be modified accordingly and reanalyzed if desired.

The PFA module will also provide a warning in the Output if the maximum peak-to-peak pressure limit has been exceeded at any location in the system, according to the API-674 standard.

Using the PFA Module

The user has the option of activating or not activating the PFA module when AFT Impulse first loads. After AFT Impulse is loaded, the PFA module can be activated or deactivated for use from the Modules panel in the Analysis Setup window or the Tools menu. Whether or not PFA is activated impacts the Analysis Setup window, Analysis menu, Output window, and Graph Results window as discussed in the Modules panel topic.

If the PFA module is active, the user can still run models without pulsation. This is selected under Pulsation Frequency Analysis on the Analysis menu. Hence there are three possibilities for the PFA module.

- PFA is not active
- PFA is active and pulsation is disabled
- PFA is active and pulsation frequency analysis is performed

Related Topics

PFA Pump RPM Evaluation

The various pump speeds that excite frequencies of interest selected by the user are automatically calculated. These speeds are displayed in the Pump RPM Evaluation Panel labeled "Excitation Frequencies and Pump Speeds" that will appear in the Quick Access Panel when the Graph Results window is active and frequencies of interest have been specified. Instructions to evaluate excitation frequencies using AFT Impulse PFA are available when clicking on the question mark icon in the Pump RPM Evaluation Panel. The corresponding pump speeds that will

excite the frequencies being evaluated are displayed on this panel, and speeds of interest can be selected automatically to create child scenarios using those pump speeds. Figure 1 below shows the location of the Pump RPM Evaluation Panel that allows you to specify pump speeds you would like to evaluate after generating an Excitation Frequency Analysis graph.
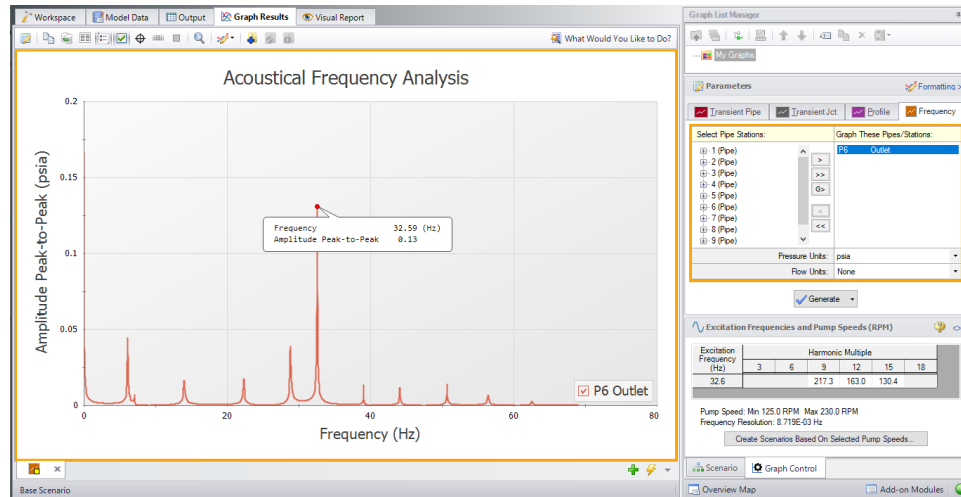
Figure 1: The Graph Results window with the Quick Access Panel and the Pump RPM Evaluation Panel

Note that the Frequency vs. Magnitude graph can also be displayed with the magnitude on a logarithmic scale. To change this graph to display the magnitude on a logarithmic axis, right-click on either axis and check the box under Logarithmic.
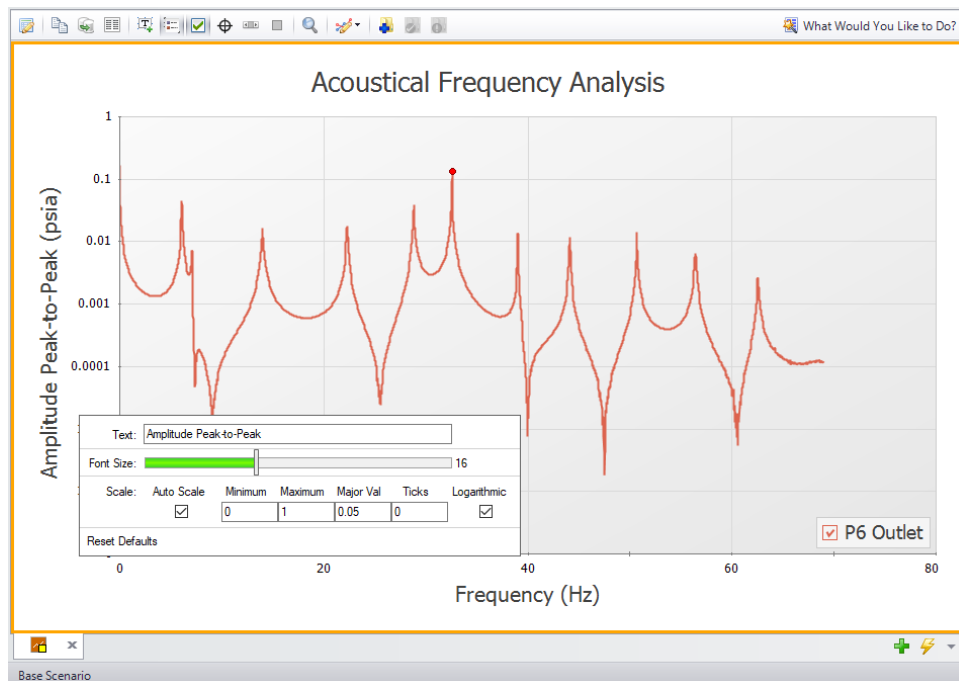
Figure 2: The Magnitude vs. Frequency Graph with the magnitude on the y-axis displayed on a log-arithmic scale

After the steps needed to generate an Excitation Frequency Analysis graph have been taken, you can analyze frequencies of interest. Large peaks on Excitation Frequency graphs correspond to natural acoustic frequencies of the system. The numerical "hammer" provided by the initial pulse has helped to identify the response of the system. By clicking and dragging from one side to the other over particularly high peaks and releasing the mouse button, the frequency in this area with the largest magnitude will be selected. Right-clicking on the label that appears enables you to add the frequency to the excitation frequencies being evaluated by selecting "Evaluate Excitation Frequency", as shown in Figure 3.
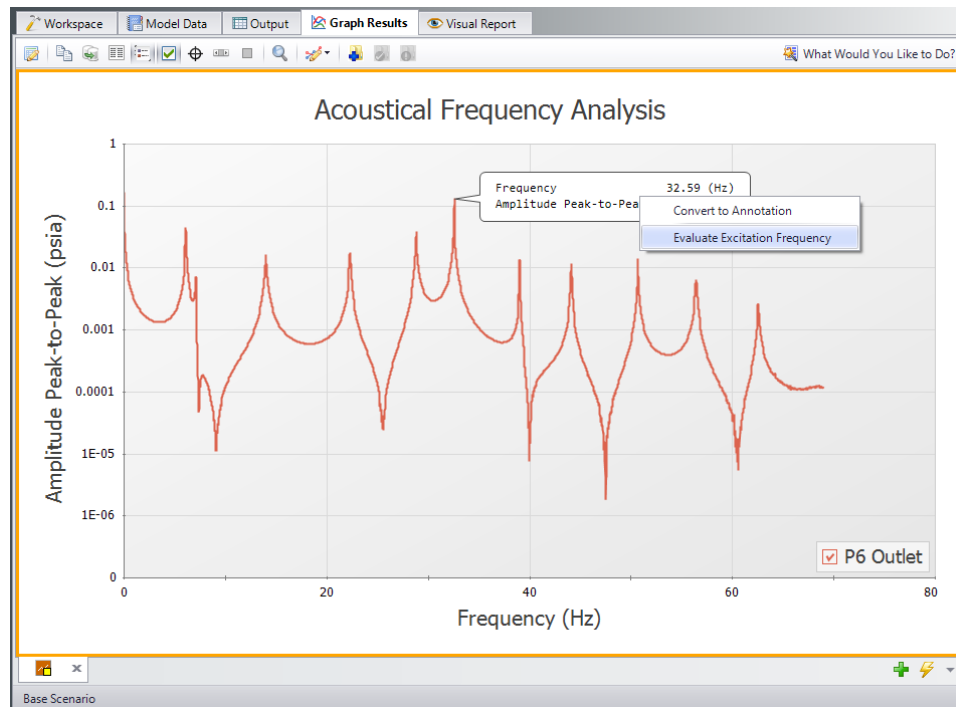
Figure 3: Right-click on the flag of the selected frequency to evaluate it

It is important to identify these natural frequencies to make informed decisions about operation. Operating at pump speeds that will excite these natural frequencies for extended periods of time can cause system fatigue and structural issues attributed to vibrations. These are commonly referred to as 'worst-case' pump speeds because they match up with the harmonic frequency of the system and cause the most vibrations when operating at that frequency. Figure 4 shows the instructions which can be accessed by clicking on the question mark icon on the top right of the Pump RPM Panel of the Quick Access Panel, which is indicated in Figure 1.
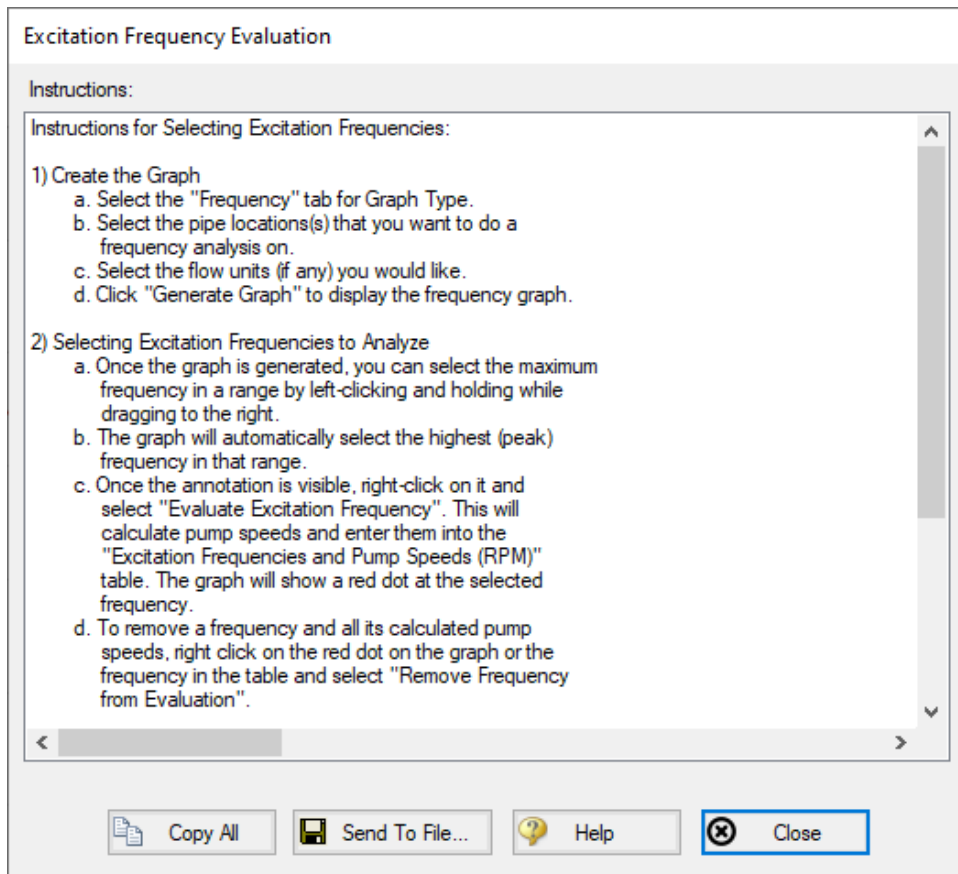
Figure 4: Detailed Instructions to evaluate excitation frequencies and determine problematic pump operating speeds

In order to determine what pump speeds will excite the natural frequencies that are selected, Equations 1 and 2 are used (Blanding & Walters, pp. 7-8).

$$N_{ij} = 60\frac{f_{ni}}{j}$$
(1)

Equation 1 identifies the various pump speeds for which integer multiples align with the natural frequencies.

- $N_{ij}\ [=]\ RPM$, corresponding pump speed that will excite the given natural frequency

- $f_{ni}\ [=]\ Hz$, selected natural frequency that is being evaluated

- $j\ [=]\dfrac{waves}{Rev}$, run-speed multiple, referred as the harmonic multiple

$$j = n\,s$$

Equation 2 is used to calculate the harmonic multiple where:

- $n\ [=]\ waves/Rev$, the number of plunges per revolution, equal to the number of PD pump heads

- $s =$ integer harmonic multiplier

The integer harmonic multiple corresponds to harmonic numbers of the selected natural frequency. Note, the N_{ij} decreases hyperbolically as s increases. Figure 5 shows the 'worst-case' pump speeds that correspond to the natural frequency (32.6 Hz) being evaluated.

Excitation Frequencies and Pump Speeds (RPM)

Excitation Frequency (Hz)	Harmonic Multiple					
	3	6	9	12	15	18
32.6			217.3	163.0	130.4	

Pump Speed: Min 125.0 RPM Max 230.0 RPM
Frequency Resolution: 8.719E-03 Hz

Create Scenarios Based On Selected Pump Speeds...
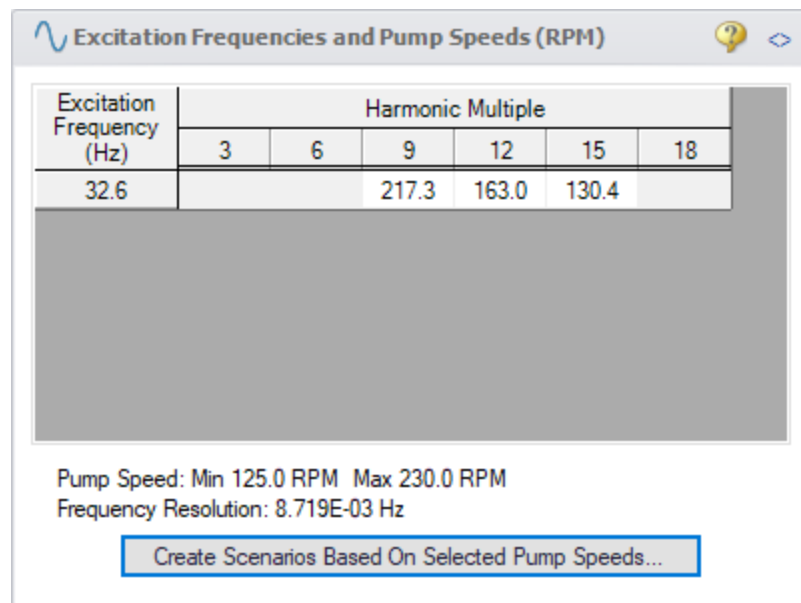
Figure 5: The Pump RPM Evaluation Panel with the pump speeds determined to excite evaluated frequencies

After the PD pump speeds are calculated, read-only scenarios are created by clicking the "Create Scenarios Based On Selected Pump Speeds" button. You can specify which pump speed(s) you want to make into a child scenario by clicking on any speeds you want to exclude. AFT Impulse PFA will then create and populate child scenarios to evaluate the selected pump speeds. For each child scenario Impulse PFA will calculate and define transients in the specified pulsation source junction that are based on the pump speeds displayed in Figure 5. Running the scenarios will determine the actual pressure response of the system at these frequencies, which can be evaluated with API-674.

References

1. Blanding, James M., Ph.D., and Trey Walters, PE. Pulsation Analysis in Positive Displacement Pump Systems Using Waterhammer, Modal, and Animation Software. Houston, TX: 45th Turbomachinery & 32nd Pump Symposia, Sept. 2016. http://www.aft.com/documents/TechnicalPapers/2016/TPS2016-PulsationAnalysis.pdf

Related Topics

API 674 Evaluation

After the Pump RPM Evaluation is finished in the base scenario, child scenarios are created to evaluate the identified pump operating speeds. In each pump operating speed scenario API-674 evaluations are performed as follows:

1. For each pipe station Impulse performs a Fast Fourier Transform (FFT) on the static pressure results at that pipe station to find the frequency response

2. The pressure limit is calculated for each harmonic using equation C.2 from API-674 as is shown here:

$$P = \frac{100}{\sqrt{d\,f}}$$

Where d is the inner diameter of the pipe, and f is the pulsation frequency which is defined as

$$f = N\,j\,/\,60$$

Where N is the pump speed in rpm and j is the harmonic multiple.

3. The FFT results are compared to the pressure limit results, and any pressures that exceed the pressure limits are reported in the Pulsation Summary in the Output window.

Related Topics

Steady-State Pulsation Tolerance

([PFA module only](#)) When the PFA module is enabled, an additional tolerance must be specified in the [Tolerance panel](#) in the [Analysis Setup window](#).

This is the Steady-State Pulsation Tolerance. This has a default value of 1.0E-05 which should be acceptable for the majority of pulsation studies.

"Steady-State Pulsation" means that the pulsation itself is repeating in a regular fashion. That is, the mean value, amplitude, and frequency of the pulsation are not changing over time, even though an individual property such as pressure might be rapidly changing. As the "start up" of a pulsation event is generally not of interest, this is accounted for by finding the steady state of pulsation.

The Relative Tolerance controls how tightly the Pulsation is considered "steady" - a loose tolerance may include some additional transient effects that are not desired.

Related Topics

PFA Module Glossary

Much of the terminology associated with pulsation analysis is specific to time-frequency analysis and/or to vibrational analysis. The following words have been defined for ease of comprehension when working with the PFA module. Note that most of the defined words include page numbers and, occasionally, equation numbers. All of these references can be found in the document referenced at the end of this page.

- **Block Size** - The number of samples in a set amount of time that are frequently calculated in powers of 2 (Page 14, Equation 26)

- **Low-Pass Filter (LPF)** - A mathematical filter that permits all frequencies at and below the specified frequency (termed the cutoff frequency) for analysis, and rejects all frequencies above the cutoff frequency (Page 7)

- **Sample Rate -** The inverse of the time step (Page 14)
- **Harmonic Frequencies -** Multiples of the base frequencies that excite the system
- **Forcing Function -** This is the function that will be used to "ring" the system. This function can be the pulse in a non-low-pass-filtered system, or can be determined by the cutoff frequency that is determined by using FFT and LPF functions (Page 7)
- **Pulse** - The near-instantaneous increase, then normalization, of flow
- **Fast Fourier Transform (FFT)** - A method used to transpose the time-domain (flow vs. time) into the frequency-domain (magnitude vs. frequency)
- dt_a - The initial time step, calculated by $1/2f_c$ (Page 4, Equation 7)
- dt_b - The selected time step, filling the criteria that dt_b is less than or equal to $dt_a/3$ (Page 5)
- **Cutoff frequency (f_c) -** This is the frequency that is the maximum frequency that will be analyzed in the system. Calculated by nN/10, where n is the maximum pump speed (with buffer) and N is the number of pump heads in the pump (Page 4, Equation 6)

References

1. Blanding, James M., Ph.D., and Trey Walters, PE. Pulsation Analysis in Positive Displacement Pump Systems Using Waterhammer, Modal, and Animation Software. Houston, TX: 45th Turbomachinery & 32nd Pump Symposia, Sept. 2016. http://www.aft.com/documents/TechnicalPapers/2016/TPS2016-PulsationAnalysis.pdf

Related Topics

SSL - Settling Slurry

What Is the SSL module?

An optional add-on module to AFT Impulse 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 behavior 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 Impulse 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 additional graphing parameters are available to assist the user with understanding the hydraulic behavior of the slurry.

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 Impulse first loads. After AFT Impulse is loaded, the SSL module can be activated or deactivated for use from the Tools menu. Whether or not SSL is activated impacts the Analysis menu, Library menu, and Output window.

If the SSL module is active, the user can still run models with normal pure liquids. 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

Enabling SSL Use

When the SSL module is active, two new menu items appear - one on the Analysis menu and the other on the Library menu. On the Analysis menu is the Slurry Modeling menu, from which the user can select "Disable" or "Settling".

The SSL module settling slurry calculation is enabled by selecting Slurry Modeling -> Settling from the Analysis menu (Figure 1). This can be selected before or after a model is built. If selected after, then additional input will be required for solids data. Pre-existing models built with standard AFT Impulse can be opened with SSL and settling slurry data added.

Settling slurry calculation can be turned off at any time by selecting Disable from the same menu. The Disable mode causes the SSL module to function like standard AFT Impulse. If the model is opened in standard AFT Impulse, this data will be lost. Table 1 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 having to deactivate the SSL module. This will cause the loss of the slurry input data for the active scenario.
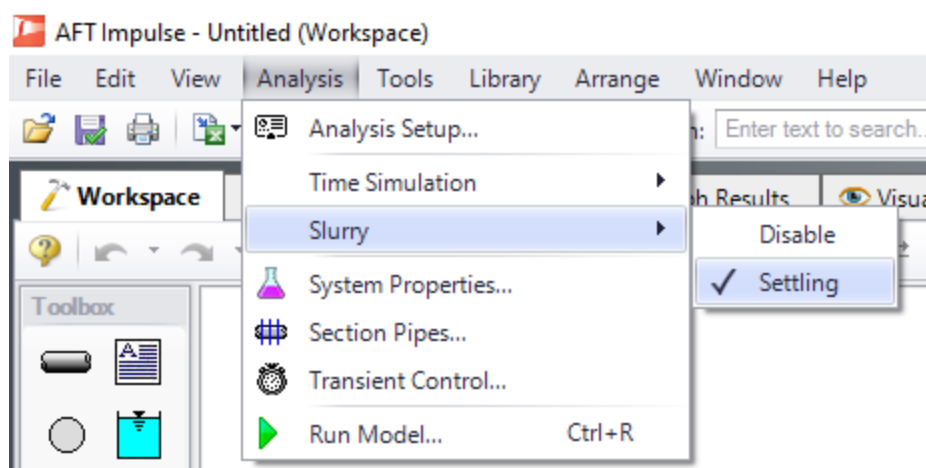
Figure 1: Select "Settling" from the Slurry Module menu item on the Analysis menu to instruct AFT Impulse to perform settling slurry calculations when it runs.

Related Topics

Slurry Calculation Method Overview

(SSL Module Only) The methodology for slurry pipe calculations is taken from Wilson 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

Related Topics

Minimal Slurry Calculation Method

(SSL Module Only) The following data are required for the Minimal Calculation Method

- Temperature
- Solids concentration
- Density
- Sliding friction coefficient, μ_s

Horizontal Pipes

The pressure drop in a given by the relationship given by Wilson 2006, on page 116. 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 in the Solids Definition panel and becomes the following:

$$\frac{i_m - i_f}{S_m - S_f} = \left(\frac{V_m}{0.55\,V_{sm}}\right)^{-0.25} \left(\frac{\mu_s}{0.40}\right) \tag{1}$$

Note that when the Minimal Input option is selected in 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

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by Wilson 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:

$$i_m = \frac{f_f}{2\,g\,D} V_m^2 + 0.22 \left(S_m - S_f\right) \left(\frac{V_m}{V_{50}}\right)^{-M} \tag{3}$$

Calculating the V50 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.

$$V_{50} \approx 3.93 \, d_{50}^{0.35} \left(\frac{S_s - 1}{1.65} \right)^{0.45} \nu_r^{-0.25} \qquad \text{(4)}$$

The relative viscosity v_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}

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by Wilson 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:

$$i_m = \frac{f_f}{2\,g\,D} V_m{}^2 + 0.22 \left(S_m - S_f\right) \left(\frac{V_m}{V_{50}}\right)^{-M} \tag{3}$$

Calculating the V_{50} Value

Equations 2 and 3 above needs 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.

$$V_{50} = w_{50} \sqrt{\frac{8}{f_f}} \cosh(60\,d_{50}/D) \tag{4}$$

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.

$$w_{50} = 0.9\,v_t + 2.7 \left[\frac{(\rho_s - \rho_f)\,g\,\mu}{\rho_f^2} \right]^{1/3} \tag{5}$$

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.

$$M = (0.25 + 13\,\sigma_s^2)^{-0.50} \tag{6}$$

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

Vertical Pipe Theory For Slurries

(SSL Module Only) Flow in vertical pipes offers a special-case flow regime. The basic equation for flow rising in a vertical pipe is given in Wilson 2006, on page 178.

$$\frac{dP}{dz} = \rho_w \, g \left[1 + (S_s - 1) \, C_{vi}\right] + \frac{\rho \, g \, f \, V_m^2}{2 \, g \, D} \qquad (1)$$

It is important to note that this is the total pressure drop per unit length whereas in the horizontal pipe equations above 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 we obtain the following basic relationship:

$$\frac{dP}{dz} = \rho_w \, g \, S_m + \frac{\rho \, g \, f \, V_m^2}{2 \, g \, D} \qquad (2)$$

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:

$$\frac{dP}{dz} = \rho_w \, g \, S_m \left(1 + \frac{f_w \, V_m^2}{2 \, g \, D}\right) \qquad (3)$$

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:

$$\frac{dP}{dz} = \rho_w\, g\left(S_m + \frac{f_w\, V_m^2}{2\, g\, D}\right)$$

(4)

Related Topics

Sloped Pipe Theory For Slurries

(SSL Module Only) Sloped pipe calculations are a combination of horizontal head loss and a vertical component. A widely used formula is given in Wilson 2006, on page 183.

$$\Delta i\left(\theta\right) = \Delta i\left(0\right)\cos\theta + (S_s - 1)C_{vd}\sin\theta$$

(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 the 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.

$$\Delta_D\sqrt{2\,g\,D(S_s - 1)}$$

(2)

Where Δ_D is determined from the graph in Wilson, et al, 2006, on page 181.

Related Topics

Wavespeed Calculations For Slurries

When performing transient slurry calculations, the wavespeed for fluids must be adjusted to account for solids suspended in the fluid.

The adjusted equation includes the addition of the bulk modulus and density of the solids, denoted with the 's' subscript, as well as the volumetric concentration of solids, C_v. Equation 21 from Jacobs 1991 can be rearranged as follows.

$$a = \sqrt{\dfrac{1}{\dfrac{\rho_f \left(1 - C_v\right) + \rho_s C_v}{\dfrac{1 - C_v}{K_f} + \dfrac{C_v}{K_s} + \dfrac{D}{E\,t}}}}$$

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	inner diameter of pipe
f	friction factor
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
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:

- Solid particles are moving with the fluid. Stationary beds cannot be modeled. Sliding beds can be modeled with caution.
- Base liquid behaves in a Newtonian viscosity manner
- Slurry viscosity is equivalent to the viscosity of the carrier fluid

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 flowrate definitions are of course interrelated. The slurry flowrate is the sum of the other two. As will be shown in the next section, all three flowrates can be displayed in the Output window.

The three flowrate definitions can be specified in three places in AFT Impulse. 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 flowrate of slurry.

Note that anywhere in AFT Impulse that specifies or displays flowrate information and does not offer the qualifier of slurry/solids/clear liquid, the flowrate of slurry is assumed.
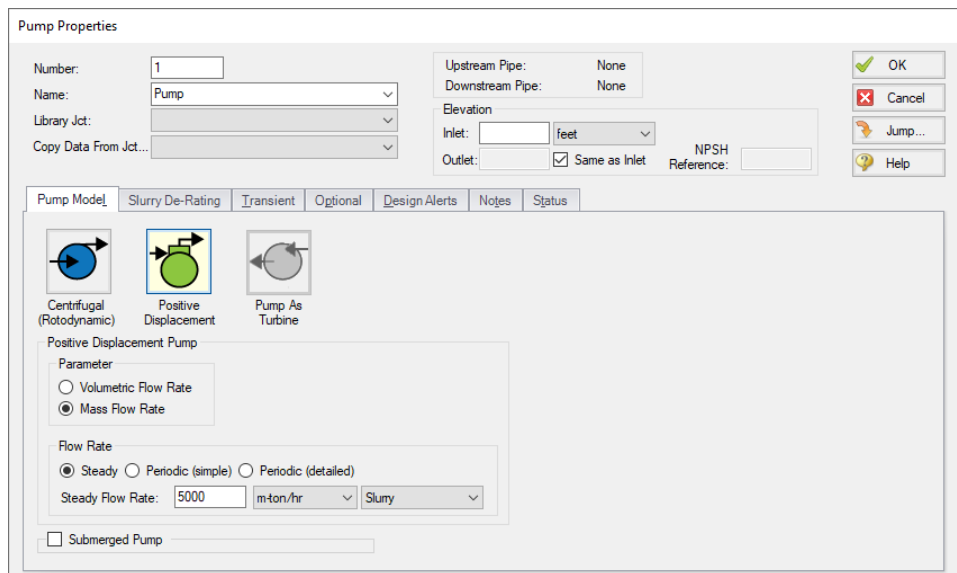
Figure 1: Pump Properties Window with fixed mass flowrate specified

Related Topics

Estimating M From Recorded Data

(SSL Module Only) There is an option in the Solids Definition panel in Analysis Setup and Solids Library to estimate *M* from recorded data. Choose the button for *M* Curve Fit. This is based on discussion in Wilson 2006, on page 135.

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 dis-

played 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.
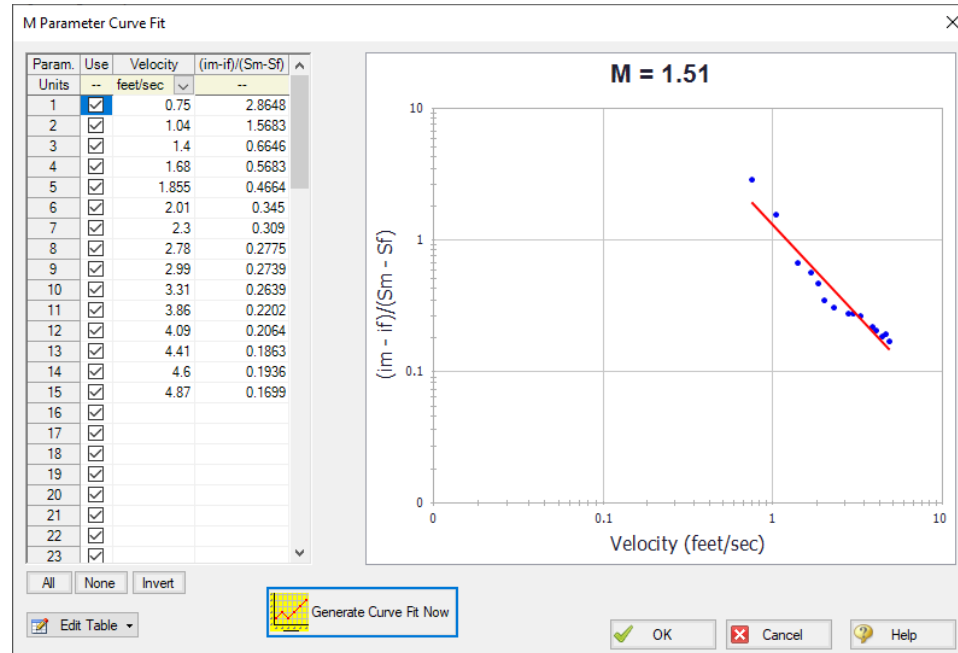
Figure 1: M curve fit window based on recorded data.

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 the Fluid Properties group Solids Definition panel or Slurry 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.

If one wants to have one or more properties vary with temperature, uncheck the Use Constant Properties box. The property values as a function of temperature are then 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.
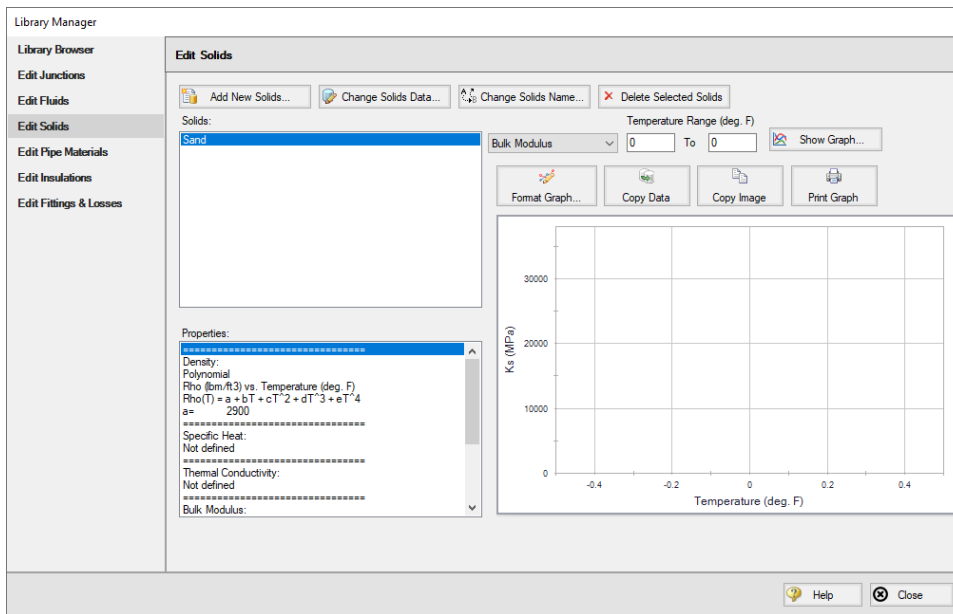
Figure 1: Solids Library showing example of user-entered solids data.

Figure 2: Change Solids Physical Properties window when defining constant physical properties.

Other Solids Parameters

The Terminal Velocity Parameter and the M exponent are entered on the Other Solids Parameters tab (Figure 2). The particle size distribution is also entered on this tab.

Similar to direct entry in the Solids Definition panel, the user can allow AFT Impulse 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 then 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 Impulse 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 the De-Rating Factors 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 2006, 2006 on page 243.

The Warman method is in use in industry but is not widely available. The method used in AFT Impulse is based on Walker, C.I page 2, and uses a correction factor for impeller size.

$$CH = 1 - \left[0.000385 \left(S_s - S_f \right) \left(1 + \frac{4}{S_s} \right) \right] C_m \ln \left(\frac{d_{50}}{0.0227} \right)$$

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 for the steady state (Figure 1). Slurry parameters can be selected for the transient under the Pipe Transient tab.
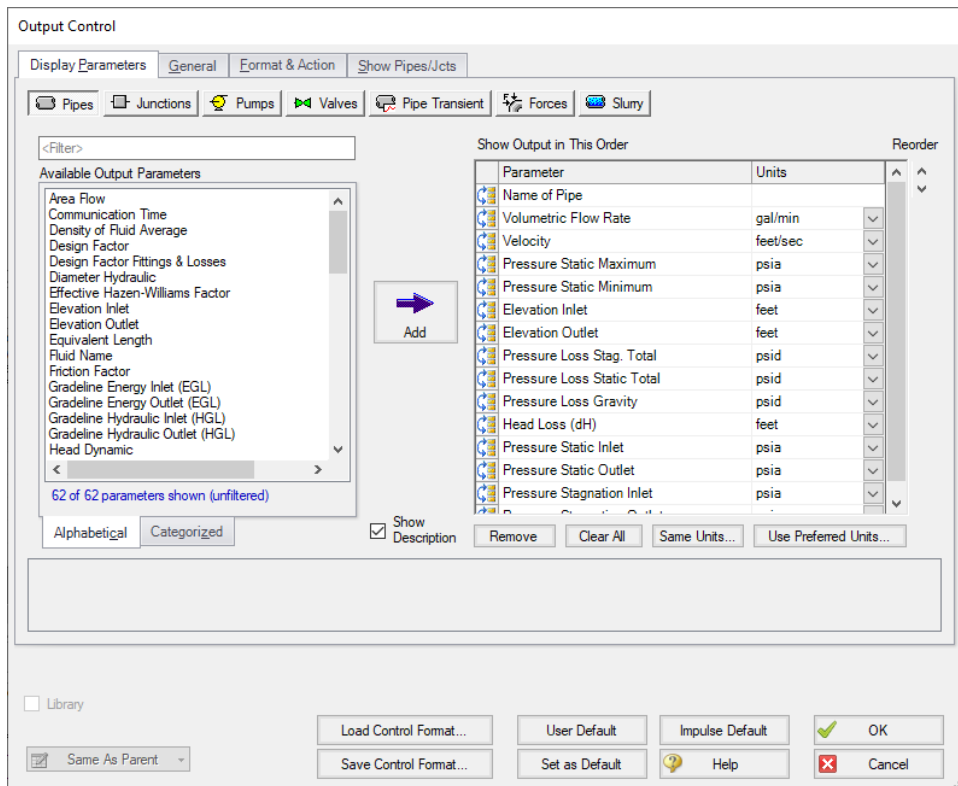
Figure 1: The Output Control window has a slurry button to allow choice of Output table parameters.

Viewing Slurry Output

The steady-state output slurry data is shown on a separate Slurry tab in the Pipes section of the Output window. The transient output data is shown in the pipe Transient Output and pipe Transient Max/Min tabs with the other transient data for the pipe (See Figure 2). The transient data can also be graphed.

Pipe (Steady)	Name	Im (ft/100 ft)	Jm (ft/100 ft)	Velocity (feet/sec)	Settling Velocity Maximum (feet/sec)	Vm/Vsm	dH Slurry (feet)	dP (psid)	Mass Flow Rate (lbm/sec)	Vol Flow Slurry (gal/min)	(Im - Iw) / (Sm - Sw)
1	Pipe	26.823	20.173	31.69	12.307	2.575	2.0173	3.4655	1,439.2	7,789	0.02737
2	Pipe	26.905	20.235	31.69	9.743	3.253	80.9409	46.6117	1,439.2	7,789	0.02986
3	Pipe	11.167	8.398	18.13	9.716	1.866	0.4199	0.2418	823.4	4,456	0.07578
4	Pipe	11.122	8.365	18.13	10.942	1.657	7.1101	13.3097	823.4	4,456	0.07442
5	Pipe	11.167	8.398	18.13	9.716	1.866	33.5938	19.3470	823.4	4,456	0.07578
13	Pipe	8.968	6.745	13.56	9.697	1.399	0.3372	0.1942	615.9	3,333	0.12253
14	Pipe	8.829	6.640	13.56	11.400	1.190	6.6401	18.7984	615.9	3,333	0.11832
15	Pipe	8.968	6.745	13.56	9.697	1.399	47.2128	27.1781	615.9	3,333	0.12253

Pipe	Max. Volumetric Flowrate (gal/min)	Max. Volumetric Flowrate Time (seconds)	Max. Volumetric Flowrate Station	Min. Volumetric Flowrate (gal/min)	Min. Volumetric Flowrate Time (seconds)	Min. Volumetric Flowrate Station	Max. Vm/Vsm	Max. Vm/Vsm Time (seconds)	Max. Vm/Vsm Station	Min. Vm/Vsm	Min. Vm/Vsm Time (seconds)	Min. Vm/Vsm Station
1	7,820	1.343	0	7,292	7.438	0	2.585	1.343	0	2.411	7.438	0
2	7,820	1.341	0	7,292	7.437	0	3.266	1.341	0	3.046	7.437	0
3	4,456	0.000	0	2,929	9.990	1	1.866	1.239E-03	1	1.230	9.990	1
4	4,456	0.01611	12	2,929	9.999	17	1.657	0.01982	15	1.092	9.999	17
5	4,456	0.1103	71	2,928	9.497	80	1.866	0.1078	69	1.230	9.497	80
13	4,370	9.999	0	3,333	1.239E-03	0	1.830	9.999	0	1.399	1.239E-03	0
14	4,370	9.999	5	3,333	0.02354	19	1.557	9.999	5	1.190	0.02478	20
15	4,370	9.998	0	3,333	0.1859	130	1.830	9.998	0	1.399	0.1859	130

Figure 2: Output window showing Pipes section of Output window with Slurry tab for steady state output and Transient Max/Min tab with Velocity Ratio for slurry.

Related Topics

SSL in Graph Results

(SSL Module Only) The transient slurry parameters are shown along side the regular pipe transient parameters in the Graph Control tab on the Quick Access panel (Figure 1). The resulting graph of the velocity ratio is shown in Figure 2.
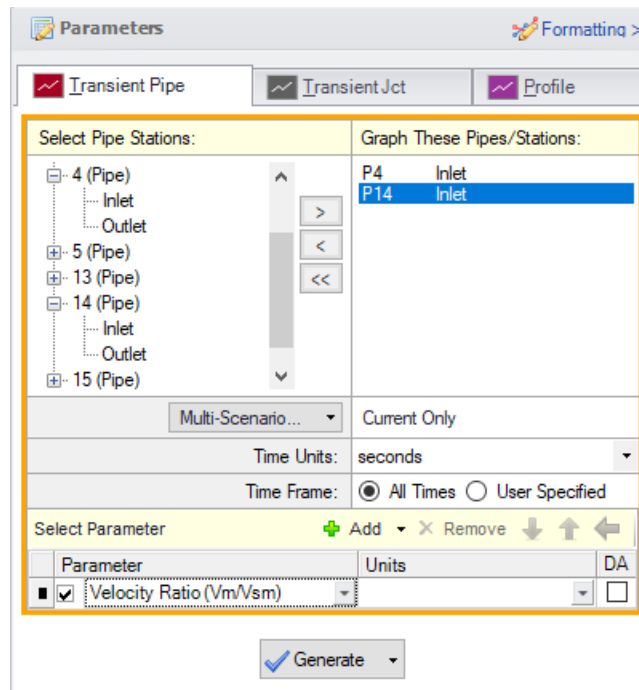
Figure 1: Transient slurry parameters are selected on the Quick Access panel
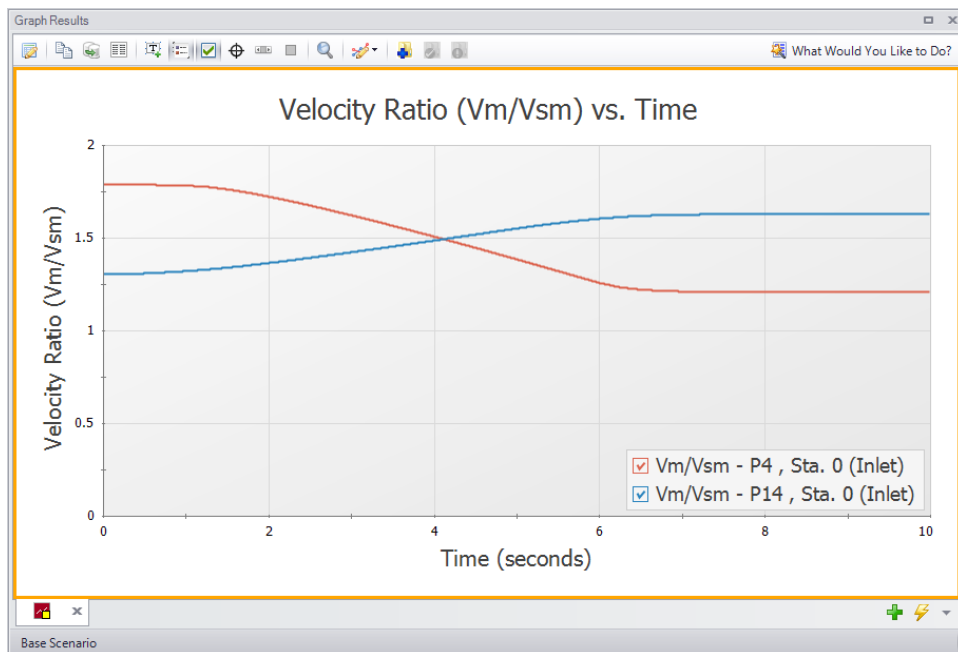
Figure 2: Graph Results window showing the Velocity Ratio over time for selected locations when using the SSL module.

Related Topics

SSL Intital Guesses for Pipe Velocity

(SSL Module Only) AFT Impulse 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 Impulse makes an initial guess itself.

Typical settling slurry applications operate at a higher velocity than non-slurry flows and AFT Impulse 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)

One of the distinct aspects of settling slurry behavior is the minimum point of the system curve. This means that for a given *jm* (or, in general, head difference) there are two potential flowrates, as is 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 im/jm point to achieve a desirable velocity ratio, and to avoid higher head losses that can occur to the left of the im/jm minimum. For example, in the below figure the minimum im 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.
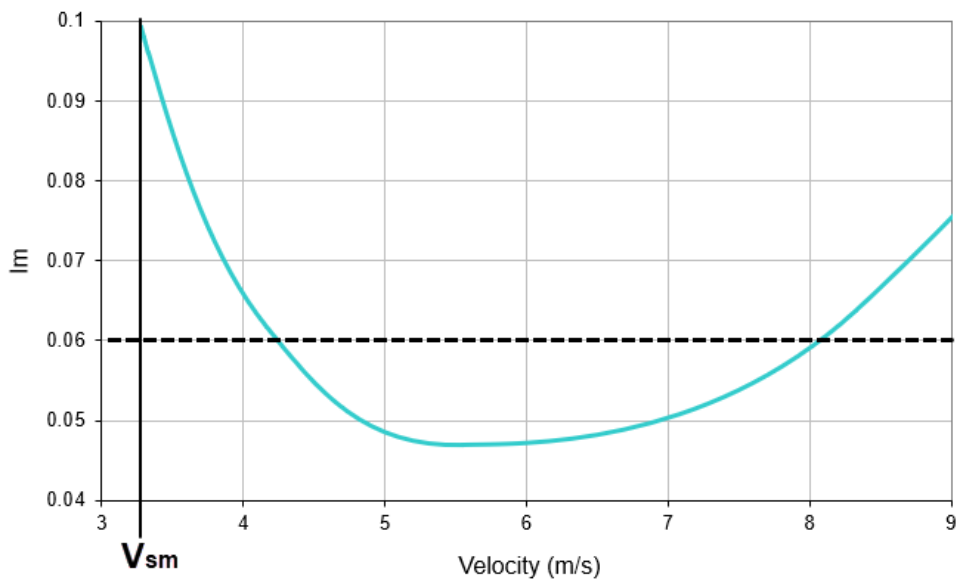
Figure 1: im vs mean velocity for a slurry system at a specific slurry concentration and pipe diameter.

Related Topics

Pipe Slope

(SSL Module Only) Pipes with different slopes will have different pressure drops for slurry flow because of the way solids are carried horizontally, vertically or in between. AFT Impulse determines the pipe slope based on the pipe length and connected junction elevations.

The pipe angle can be added to the Slurry output in the Output Control table.

Related Topics

Libraries

One of the more powerful features of AFT Impulse is that it provides extensive customization cap-abilities. AFT Impulse provides the tools to build custom libraries for fluid properties, pipe mater-ials, and all junction types except Volume Balance. AFT Impulse incorporates these libraries internally and offers them to you through list boxes.

There are three categories of libraries: Local User, External, and AFT Internal.

The **Local User Library** is located in the IMP_USER9.DAT file in the local user directory, and stores user-specified data for most component types for local use. It cannot be transferred to other users.

The **AFT Internal Library** is located in the Impulse9.DAT file in the local user directory and stores AFT-provided libraries, as well as user-specified fittings & losses. It cannot be transferred to other users.

External libraries are created by the user and can contain data for any of the library types. In AFT Impulse you can create multiple network or local custom libraries using the Library Manager on the Library menu. External libraries can be shared to multiple users from a network location as is discussed in Creating an Enterprise-wide Network Library System.

Related Topics

Related Blogs

Databases 101 - Get to Know Them

Databases 201 - Name That Fluids

Databases 301 - Those Are Some Nice Pipes!

Databases 401 - Managing Databases

Library Manager

Accessed from the Library Menu, the Library Manager is a new feature to AFT Impulse 9 and replaces the Database features used in previous versions. For a list of changes between Impulse 8 and Impulse 9 see Interface Improvements for Impulse 9. 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

Related Blogs

Databases 101 - Get to Know Them

Databases 201 - Name That Fluids

Databases 301 - Those Are Some Nice Pipes!

Databases 401 - Managing Databases

Library Browser

The first panel of the Library Manager is the Library Browser. Use the Library Browser to create a new library, add an existing library, view contents of an added library, and connect or disconnect libraries to the model.

Adding and Removing Existing Libraries

To work with a library you must first make it available to AFT Impulse. Referring to Figure 1, clicking on the Add Existing Library button will open a standard file dialog box with which you may browse to and select a library. A library may be removed from the list of available libraries by right-clicking on the library and selecting Remove Library from List. A summary of the library contents can be seen in the section on the right when a library, library section, or component is selected in the list of available libraries.

The AFT DEFAULT INTERNAL LIBRARY and the AFT IMPULSE LOCAL USER LIBRARY may not be removed and will always appear in the list of available libraries. External shared libraries will also always appear in the available library list.
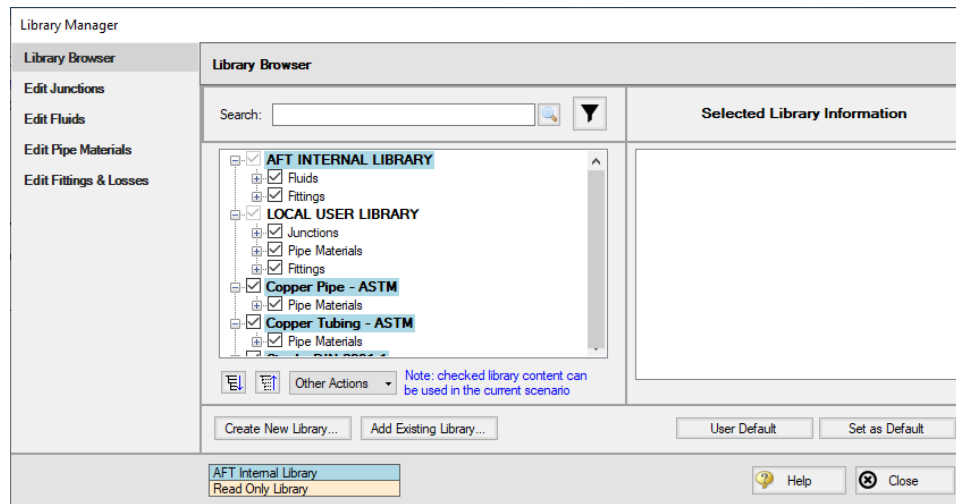
Figure 1: Library Browser in the Library Manager

Connecting and Disconnecting Libraries

In order to use the components from a library in the model the library must be **connected**. A library is connected to the model when it is shown with a check mark in the Library Browser. All libraries are automatically connected when they are first added to the Library Manager.

Sections or components in a library can also be connected/disconnected on an individual basis if the library as a whole is connected.

It may be desired to disconnect a library if you do not want the components from that library to be available for use in the current model, but you do want the library to be available for use in other models.

Clicking on Set As Default will establish the list of connected libraries as the default list, while clicking on User Default will set the connections to the default list.

AFT Internal Pipe Material libraries can be connected or disconnected as a group from the Other Actions button under the available libraries list.

Editing Library Contents

Existing content can be moved to/from, copied to/from, or deleted from available libraries in the Library Browser.

To edit an item right-click on the item in the Library Browser, then click the desired action from the right-click menu. Note that moving, copying, and deleting can be done to a library as a whole, or can be done for just a section or single component in a library. Individuals components can also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.
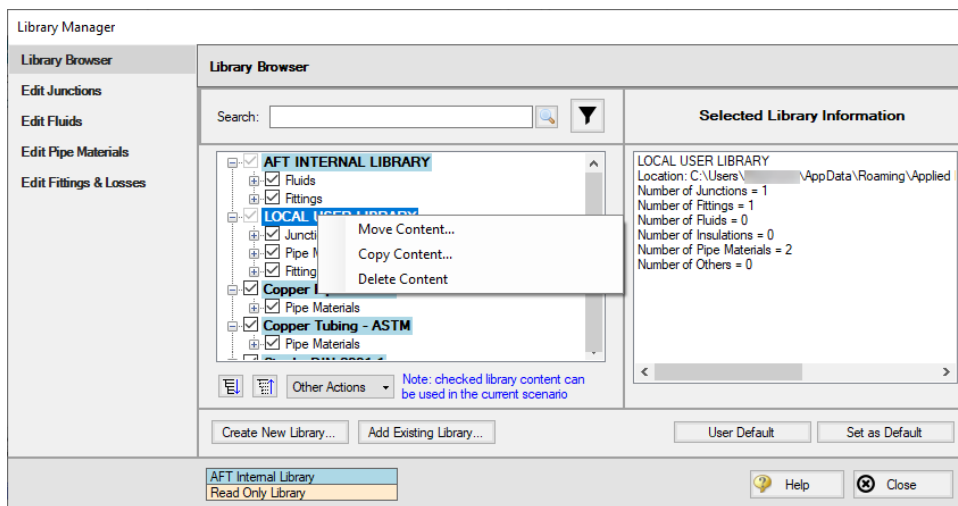
Figure 2: Library contents can be transferred or deleted from the right-click menu

Creating External Libraries

By default library items are stored in the AFT Default Internal Library or Local User Library, both of which cannot be shared with other users. Library items can be moved or copied to an external library which can be shared with multiple users.

To create an external library click the Create New Library button underneath the available libraries list. You will be prompted to provide a name for the file, and a library description (Figure 3). The provided description will be displayed as the library's name in the Library Manager.

To add content to an external library copy or move content from an existing library as described in the Editing Library Contents section. Junction components can also be directly added to external libraries as is described in the Junction Libraries topic.

For more details on creating and sharing external libraries see Creating an Enterprise-wide Network Library.
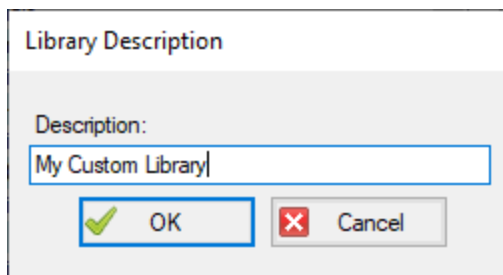
Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected

Related Topics

Related Blogs

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

Junction Library

Adding Custom Junction Data to the Component Library

To simplify modeling and eliminate repetitive data entry, AFT Impulse allows you to save frequently-used junction data to a custom library. Once saved, the custom equipment can be selected from the Library List in the [Properties window](#) at any later time for immediate recall of the data. Using this tool, you can easily build your own library of 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 component. Enter a descriptive name and close the window. If you open the junction again, the custom name will be selected in the Library List.

AFT Impulse accepts custom data for all junction types except 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

Related Blogs

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

Fluid Library

AFT Impulse provides a [fluid library](#) for numerous common fluids. If the working fluid you wish to model is not in this library, you may elect to enter the fluid properties in a custom library for later recall using the Edit Fluids panel in the [Library Manager](#).

You can open the Edit Fluids panel from the Library menu or from the Edit Fluid list button in the [Fluid panel](#) in [Analysis Setup](#). The Edit Fluids panel allows you to review currently defined AFT Standard and custom fluids, to change the data for custom fluids, and to enter new fluids. All AFT Standard fluid properties are assumed to be a function of temperature. Custom fluids can be a function of temperature or solids concentration.

Once you have entered a custom fluid, it will appear in the Fluids Available in Library list of the Fluid panel when AFT Standard is selected.

You can add new fluids in the Library Manager Edit Fluids panel by clicking the Add New Fluid button. Raw data can be provided for the variation of density, dynamic viscosity, vapor pressure, and bulk modulus 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.

Density, viscosity, and bulk modulus are required for all fluids. Vapor pressure is 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

Related Blogs

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

Pipe Material Library

Adding Custom Pipe Materials to the Pipe Materials Library

You can open the Edit Pipe Materials panel from the [Library Menu](#). Using this panel, you can easily add to the user library the geometric information for any pipe material in the length units of your choice. Custom pipe materials can be selected for use in the [Pipe Properties Window](#), minimizing the data entry required by the user.

Pipe materials typically have a range of nominal sizes and a specific type, class, or schedule that describes the actual dimension. AFT Impulse works with the inner diameter of the pipe, since that is the parameter that directly affects the fluid mechanics.

To add custom pipe materials to the user library, choose the entry location you desire in the list on the left (Pipe Materials, Sizes and Types), then click the New button below. For example, to add an entirely new material, click the Pipe Materials item at the top of the list then click the New Material button. To add a new size for an existing material (say Steel), click *Steel* in the list then click the New Size button. To add a new type to the Steel 1 inch size, click *1 inch* under the *Steel* material in the list then click the New Type button.

When adding a new material, you also will be asked to specify a geometry, enter a nominal size, type of material, diameter, wall thickness, friction data set, and friction value to define the first data. Optionally you can add in a wall thickness. From then on you can enter additional nominal sizes and types. The name descriptors you choose for the material, size, and type are at your discretion. They can contain text mixed with relevant numbers if you desire.

You can also add multiple Friction Data Sets for each pipe material. This allows you to keep, for example, different friction values that may apply to the same pipe under different conditions. These conditions could be related to the age of the pipe or the type of fluid being carried. See the Friction Data Sets section below for more information.

In addition to manual entry of the custom pipe materials, you also can import pipe material data from a file. When you click the Import From File button, a dialog box will appear explaining the necessary import file format. By choosing to import from a file, you will be able to have a separate copy of the data for later access.

Friction Data Sets

Friction Data Sets are pipe friction values for a particular pipe condition.

To create a new Friction Data Set select a Pipe Material, Size, or Schedule to associate the data set with, then click New Data Set from the Friction Data Set tab on the right (Figure 1). Modify the Friction Data Set description to provide a name, then enter the Friction Model and Value.

A Friction Data Set can be associated with the entire pipe material, or with specific sizes or types. For instance, a friction data set associated with Steel (i.e., the top level) might be called "25 year usage" and have a roughness value of 0.005 inches. This data set is then available to all sizes and types of steel. However, if "25 year usage" is associated only with the 1 inch of Steel, it will apply only to the pipe schedules that are 1 inch.

You can also associate friction data sets with a specific type (i.e., schedule). For instance, you can make a "25 year usage" and associate it only with 1 inch schedule 40 Steel pipe and assign its roughness value as 0.006 inches. Then all Steel pipe would use 0.005 inches for "25 year usage" friction while the different value for 1 inch schedule 40 of 0.006 inches would supersede the value of 0.005 inches associated with all Steel pipe.

Finally, among multiple friction data sets you can specify a default data set.

Multiple friction data sets can be created for each type of pipe material. For example, there may be separate friction data sets for Steel pipe such as: New Pipe, 10 Year Use, 20 Year Use and 30 Year Use. Typically, the older pipes will have higher friction.
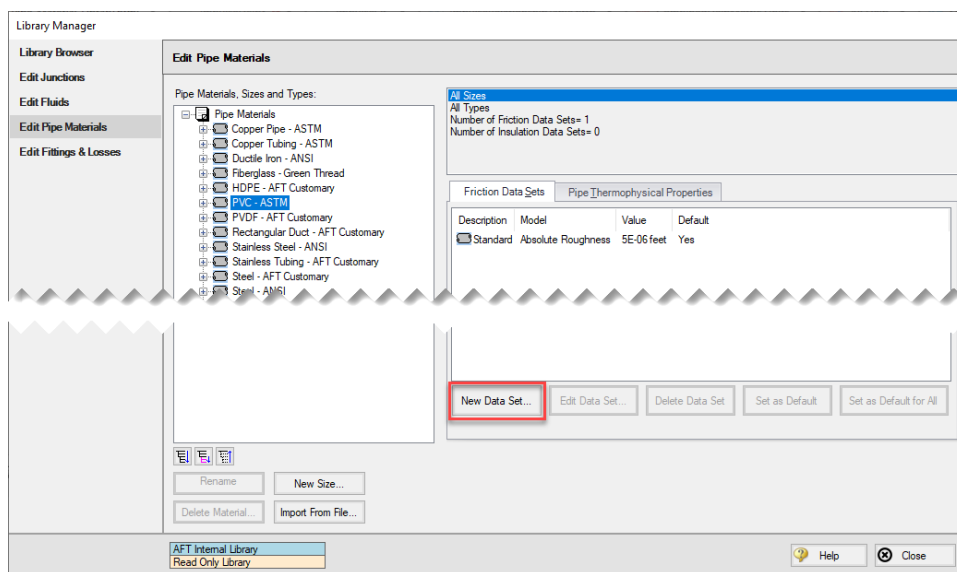
Figure 1: Friction Data Set tab on the Edit Pipe Materials panel

Pipe Physical Properties

The Pipe Material Library window allows you to assign physical properties of the pipe. The density and Poisson ratio of the pipe can be entered as constants, and the modulus of elasticity as a function of temperature.

Related Topics

Related Blogs

Databases 101 - Get to Know Them

Databases 201 - Name That Fluids

Databases 301 - Those Are Some Nice Pipes!

Databases 401 - Managing Databases

Fittings & Losses Library

The Edit Fittings & Losses panel can be accessed from the Library Menu and allows the user to review the Fittings & Losses data provided by default in Impulse, 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 spread-sheet-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 work-book. If a header row or a blank row is used, Impulse 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)	K (constant)
	Mitre	General	Pipe Size (nominal)	k / fT (Crane)
	Smooth Flanged	r/D=1	None	
		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)	K (constant)
		Type 2	Open Percentage	k / fT (Crane)
		Direct flow	L/D	
		General	Pipe Size (nominal)	
	Ball	None	None	
	Butterfly	Plane disk - AR=0.92		
		Plane disk		
		t/D=0.1		
		t/D=0.15		
		t/D=0.2		
		t/D=0.25		
		t/D=0.3		
		t/D=0.35		
		Type A		
		Type B		
		Type C		
	Cylindrical	General		
		Rect. Tube		
	Gate	Wedge\Plug		
		Wedge		
		Disk w/o recess		
		Disk with recess		
		Disk		
		Wedge - Full Open 10 deg.		
		Wedge - Full Open 20 deg.		
		Sluice		
		Disk with recess		
	Globe	90 degree wall		
		45 degree wall		
		Y Type		
		Standard 45 deg. wall		
		Standard 90 deg. wall		
		Standard		
		Y Type		
	Plug	Straight-way		
	Poppet	W/D=.035		
		W/D=.07		
		No Seat Width		
	Three-way	Through flow		
		Branch flow		
	3-K (Darby)	Angle		
		Globe		
		Plug		
		Gate		
		Ball		
		Diaphragm		
Check Valve	Swing	90 deg. seat	Angle (degrees)	K (constant)
		45 deg. seat	None	k / fT (Crane)
	Lift	90 deg. wall		
		45 deg. wall		
	Tilting disc	General		
	Stop-check	Globe 90 deg.		
		Globe 45 deg.		
		Globe thru flow		
	3-K (Darby)	Swing Check		
		Lift check		
Orifice	Sharp	None	Area Ratio	K (constant)
	Long	None	None	k / fT (Crane)
Entrance/Exit	Entrance	Sharp	R/D	
		Reentrant	None	
		Rounded Flush		
	Exit	None		
Other	Diff Flowmtr	Orifice Plate	Area Ratio	K (constant)
		Nozzle	None	k / fT (Crane)
		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.	None	N/A
		300 lb.		

Related Topics

Related Blogs

[Databases 101 - Get to Know Them](#)

[Databases 201 - Name That Fluids](#)

[Databases 301 - Those Are Some Nice Pipes!](#)

[Databases 401 - Managing Databases](#)

Network Library Overview

Custom libraries contain custom data generated by the user, including fluid properties, pipe materials, and data for junctions types such as pumps and valves.

All custom data created by the user is kept on the local computer in a file called IMP_ USER9.DAT in the local user directory. This data is not shared and is always available to the local user for access or for editing.

In addition to the local user library, you can create multiple network or local custom libraries. The Library Manager window on the Library menu simplifies this process. See Creating an Enterprise-wide Network Library System for details on how to do this.

Related Topics

Benefits of Shared Libraries

When the Impulse 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 Impulse itself, so it is guarded data.

Once the data is placed on a network, users can connect to any library listed in the Library Manager. Users can connect to as many libraries as they want.

If the user connects to a library with these files and selects to be connected to any of these control files, the model will inherit the control formatting specified in the control files. This means that special report layouts, special graphical representations and colors, and other special controls can be decided upon by the library constructor, and those control features can be loaded into

local models by end users. This is highly useful for creating customized reports for certain customers or project requirements.

By way of example, assume the Output Control content was specified as coming from the "Woodland Park Office Library" library.

If you uncheck the Library checkbox or change certain data in the Output Control window, the library will remain connected but change to an inactive state. When the library is inactive, none of its data flows through to the model. To bring the Output Control data back into sync with the connected library, recheck the Library check box in Output Control.

This entire custom library concept lends itself naturally to the idea of specific project libraries. Each project has differing requirements that depend on the nature of the project and the end customer. Data for each project can be assembled into separate libraries, allowing faster and more accurate transmission of the controlled data and desired reporting formats to engineers.

Maintaining a network library allows commonly-used data to be controlled, which may be helpful in instances where quality control is strict.

Related Topics

Creating an enterprise-wide, network library system

Overview

Users of AFT Impulse 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 Impulse by creating a file named DATABASE.LIB in the Impulse 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 Impulse uses to find these libraries. An External Library is any library file that is created by the user and listed in the IMP_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 Impulse 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 Impulse Internal Libraries which cannot be disconnected or made unavailable.

Table 1: A summary of the types of libraries used for the example below. Note the difference between External and External Shared libraries is just which *.LIB file their file path is listed in.

Library Groups & Filenames	Library Titles	Library Locations
Impulse Internal Libraries		
IMPULSE9.DAT IMP_USER9.DAT	AFT Default Internal Library AFT Impulse Local User Library	AFT Impulse 9 folder AppData folder
External Shared Libraries (DATABASE.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in DATABASE.LIB
External Libraries (IMP_DBUSER.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in IMP_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 (IMP_USER9.DAT) 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 Impulse 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 Impulse. 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 Impulse directory on the network. However, since Impulse 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 IMP_ 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 Impulse folder (e.g. AFT Products\AFT Impulse\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 Impulse. 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 Impulse 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 (IMP_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 Impulse will write the full path of the library to the External Library listing (IMP_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 Impulse 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 Impulse\Libraries\filename.dat). You can automatically have AFT Impulse 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 Impulse folder where the executable is located.

When the pipe materials library was added it became an External Library listed in IMP_ DBUSER.LIB. Now we need to move this listing from the local machine to the AFT Impulse 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 Impulse 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 Impulse, click Set As Default. To apply these settings to a model made previously, open the model, go to Library Manager, click User Default and, if you have set the default, the connections and setting will now be applied to the model.

Sharing Model Formats/Preferences

Note that the Library Manager used to provide the option to include formats/preference settings such as the Output Control or Visual Report Control layouts into shared libraries. These type of files can still be shared, but are no longer shared through the Library Manager. Instead these type of format files must be loaded directly through their control windows, such as loading the Output Control format being loaded in the Output Control window.

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

You may have a library item that is not found in any currently connected library. For example, a pump may be connected to a library, and derive its data from that library. However, when the model is loaded, the source library is not available. This could be for several reasons that will be discussed shortly.

The pump in the model does in fact have all of the required data needed to run the model. But when it is not found in the library, you will be offered several options:

- **Disconnect it from the source library** – this means that the model will forget the library the pump came from and make it a standalone pump not connected to any library.

- **Add it to a library** – this means that the pump can be added to a custom library

- **Leave the library connection** – this means that the pump will remember what library to which it was originally connected and connect to that library again when it is made available to AFT Impulse. If, however, you try to open the pump Properties window, AFT Impulse will disconnect the pump from the library and warn you.

The pump might have lost its library connection for one of these reasons:

- The library is no longer available because it was removed either through the Library Manager or because your libraries have been changed on the network.

- The model is being run on a different PC than where it was created, and the original libraries are no longer available. This could be, for example, because you have moved the model onto a laptop computer to take it on travel. Or it could happen because the model was developed by another company which had developed libraries of its own but which are not available to you.

Whatever choice you make, the model will retain all its engineering data and yield the same results as it did originally.

2. Item in Model is Different from Library

You may have a library item in the model that is connected to a external library, but for some reason the data is not the same. For example, you may have a pump curve for the Rocky Mountain Pump Company's Model 100A in your model, but it does not match the pump curve for this model in your library. Here you have three choices:

- **Update the items in the model to match the library** – this will actually change the model's input data, and it might yield different results than originally

- **Update the library (if it is local) to match the model's items** – this will change the library, but will leave the model's input data as it was. It will yield the same results as it did previously.

- **Disconnect it from the source library** – this will leave the model as it was, and it will yield the same results.

The pump in the model might have a different pump curve than the one in the library for one of these reasons:

- When the model was built, the pump was connected to the Rocky Mountain Pump Company (i.e., RMPC) Model 100A. But since that time, the RMPC 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 pumps to the library values.

- The library may have been changed by an unauthorized individual, and while it was accurate originally, now the data is no longer valid. In this case, it would be best to **not update** the pump in your model, and to inquire into the cause of the inaccurate data within your organization.

Related Topics

Library Locations in General Preferences

When performing an engineering analysis with or even without cost estimating, there may be many engineering libraries that underlie the model. These libraries need to be connected in the Library 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 Impulse tries to open a model that requires certain libraries, and those libraries have not been connected already, AFT Impulse looks for them in the locations specified in the User Options. In addition, AFT Impulse looks in the model's folder. If AFT Impulse 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.
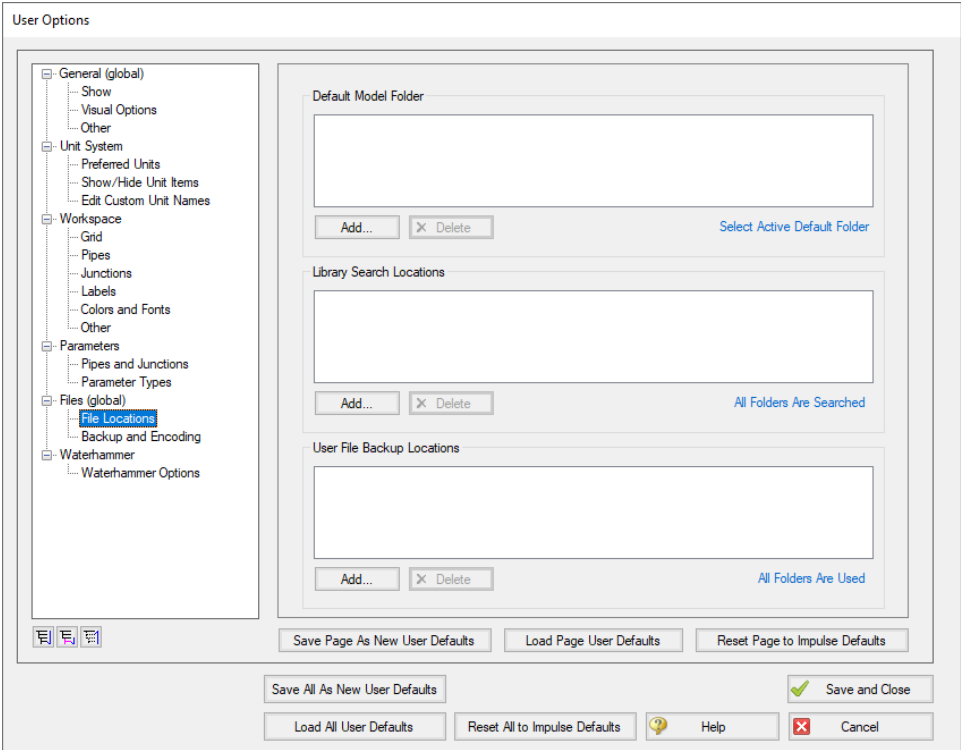
Figure 1: Library Search Locations can be defined in the User Options window

Related Topics

Theory and References

This section focuses on the fundamental theories that are used in AFT Impulse'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, waterhammer theory, and information on references used in the development of AFT Impulse.

Content

Nomenclature

a	wavespeed
A	cross-sectional flow area of a pipe
B_M	compatibility equations parameter for method of characteristics
B_P	compatibility equations parameter for method of characteristics
C_D	discharge coefficient
C_M	compatibility equations parameter for method of characteristics
C_P	compatibility equations parameter for method of characteristics
C_V	valve coefficient
C_{HW}	Hazen-Williams factor
D	diameter of a pipe
E	modulus of elasticity
e	pipe wall thickness
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
F_B	Dimensionless pump torque parameter (see Four Quadrant Model Overview)
F_H	Dimensionless pump head parameter (see Four Quadrant Model Overview)

g	gravitational constant
H	head
HGL	hydraulic gradeline (piezometric head)
I	moment of inertia of pump and entrained liquid
J	Jacobian matrix
K	loss factor
K	constant for Power Law fluid
K_B	liquid bulk modulus of elasticity
$K_{sprinkler}$	fire sprinkler loss factor
L	length of a pipe
$\dot{m}$	mass flow rate
n	general constant
n	constant for Power Law fluid
N	pump synchronous speed (rpm)
P	pressure, static
P_O	pressure, stagnation
P_w	wetted perimeter
$\underline{P}$	power
Q	volumetric flow rate
r	radius
r	relaxation
R, R'	resistance or gas constant
R_c	coefficient of rigidity
Re	Reynolds Number
s	pump speed
S_B	parameter as derived from conservation of mass
S_C	parameter as derived from conservation of mass
S_y	yield stress
t	time
T	torque on pump

v	pump flow ratio to reference
V	velocity
V	volume*x* distance along pipe centerline
z	elevation
α, μ, θ	angle
α	pump speed ratio
β	diameter ratio
β	torque ratio
λ	method of characteristics multiplier
η	efficiency
v	kinematic viscosity
ε	roughness
ω	pump speed
ρ	density
μ	dynamic viscosity

Subscripts

i	junction or pipe station at which solution is sought
j	junctions with pipes connecting to junction *i*
R	rated conditions for a pump, usually BEP
o	stagnation
∞	infinity, far away, ambient
new	current time for computation
old	most recent time for computation

Steady-State Network Solution Methodology

AFT Impulse makes use of standard matrix solution techniques when solving for a steady-state solution(Jeppson 1976). The method is known as the H-Equation method, where *H,* the piezo-metric 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 grade line (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:

$$HGL = \frac{P}{\rho g} + z \qquad (1)$$

where z is the elevation. Each concept offers certain advantages.

The solution technique makes use of the continuity and one dimensional momentum equations. AFT Impulse 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 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:

$$\sum_{j=1}^{n} \dot{m}_{ij} = 0 \qquad (2)$$

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:

$$\Delta P_f = f \frac{L}{D} \left(\frac{1}{2} \rho V^2 \right) \qquad (3)$$

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:

$$P_1 + \frac{1}{2}\rho V_1{}^2 + \rho g z_1 = P_2 + \frac{1}{2}\rho V_2{}^2 + \rho g z_2 + \Delta P_f \qquad (4)$$

The static and stagnation pressure are related as follows:

$$P_o = P + \frac{1}{2}\rho V^2 \qquad (5)$$

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:

$$P_{o,i} - P_{o,j} + \rho g (z_i - z_j) = \Delta P_f \qquad (6)$$

where *i* and *j* denote upstream and downstream junction values.

The definition of mass flow rate is:

$$\dot{m} = \rho A V \qquad (7)$$

Combining Equations 3 and 6 and substituting for velocity, *V*, using Equation 7 gives the mass flow for each pipe:

$$\left[\frac{P_{o,i} - P_{o,j} + \rho g (z_i - z_j)}{R'_{ij}} \right]^{1/2} = \dot{m}_{ij} \qquad (8)$$

where $R_{ij}{}'$ is the effective flow resistance in the pipe and the subscript *ij* refers to the pipe connecting junctions *i* and *j*.

$$R'_{ij} = \left(\frac{f_{ij} L_{ij}}{D_{ij}} + K_{ij} \right) \frac{1}{2 \rho A_{ij}^2} \qquad (9)$$

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*:

$$\sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho\, g\, (z_i - z_j)}{R'_{ij}} \right]^{1/2} = 0 \qquad (10)$$

where *n* is the number of pipes connected to junction *i*.

To be completely general, Equation 10 should be written for junction *i*:

$$\sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho\, g\, (z_i - z_j)}{R'_{ij}} \right]^{1/2} = \dot{m}_{i,Applied} \qquad (11)$$

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:

$$F_i = \sum_{j=1}^{n} \left[\frac{P_{o,i} - P_{o,j} + \rho\, g\, (z_i - z_j)}{R'_{ij}} \right]^{1/2} - \dot{m}_{i,Applied} \qquad (12)$$

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:

$$J_F = \begin{bmatrix} \dfrac{\partial F_1}{\partial P_{o,1}} & \dfrac{\partial F_1}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_1}{\partial P_{o,n}} \\[1.5em] \dfrac{\partial F_2}{\partial P_{o,1}} & \dfrac{\partial F_2}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_2}{\partial P_{o,n}} \\[1.5em] \vdots & \vdots & \ddots & \vdots \\[1.5em] \dfrac{\partial F_n}{\partial P_{o,1}} & \dfrac{\partial F_n}{\partial P_{o,2}} & \cdots & \dfrac{\partial F_n}{\partial P_{o,n}} \end{bmatrix}$$

The column matrix $\vec{P}_o$ contains the stagnation pressure at each junction, and column matrix $\bar{F}$ 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 Impulse 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 Impulse's first guess does not lead to a converged solution. In these cases, AFT Impulse provides you with the option of specifying your own initial guesses of flow and pressure in the Optional tab of the pipe/junction properties window.

Related Topics

Continuity Equation at a Branch

By applying the continuity equation at branch *i*, we loop over all connecting pipes and get

$$\sum_{j=1}^{n} \dot{m}_{ij} = 0 \qquad (1)$$
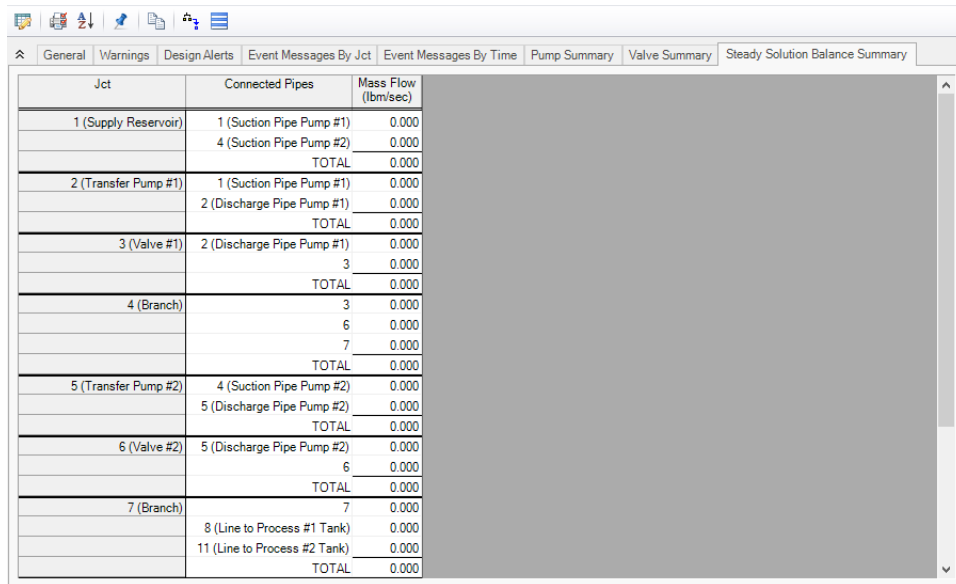
where $\dot{m}_{ij}$ is the flow rate in the pipe connecting junctions *i* and *j*.

Verifying Network Solutions

As discussed on the Network Solution Methodology page, AFT Impulse uses a Newton-Raphson matrix method to obtain a system level mass balance for the steady-state solution. When the Solver is finished, the continuity equation is satisfied at all branching sections. Once solved, it is a simple matter to go back and sum up all mass flows at each branch to verify that these equations are satisfied. In fact, AFT Impulse does just that.

After the Solver converges, AFT Impulse loops over each junction and adds up the inflow and outflow of mass 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 Impulse 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.

Jct	Connected Pipes	Mass Flow (lbm/sec)
1 (Supply Reservoir)	1 (Suction Pipe Pump #1)	0.000
	4 (Suction Pipe Pump #2)	0.000
	TOTAL	0.000
2 (Transfer Pump #1)	1 (Suction Pipe Pump #1)	0.000
	2 (Discharge Pipe Pump #1)	0.000
	TOTAL	0.000
3 (Valve #1)	2 (Discharge Pipe Pump #1)	0.000
	3	0.000
	TOTAL	0.000
4 (Branch)	3	0.000
	6	0.000
	7	0.000
	TOTAL	0.000
5 (Transfer Pump #2)	4 (Suction Pipe Pump #2)	0.000
	5 (Discharge Pipe Pump #2)	0.000
	TOTAL	0.000
6 (Valve #2)	5 (Discharge Pipe Pump #2)	0.000
	6	0.000
	TOTAL	0.000
7 (Branch)	7	0.000
	8 (Line to Process #1 Tank)	0.000
	11 (Line to Process #2 Tank)	0.000
	TOTAL	0.000

Figure 1: Solution Balance Summary table shown in the Output window offers a balance report of all junctions in the window.

Transient Theory

AFT Impulse simulates transient systems using fundamental equations based on conservation of mass and momentum. These fundamental equations are solved using the Method of Characteristics, which simplifies a set of two partial differential equations into four ordinary differential equations.

AFT Impulse also includes two cavitation models which are designed to capture limited transient cavitation in an otherwise liquid-full system.

The math and calculations needed to support AFT Impulse's transient simulations can be complex at times. The topics in this section attempt to clearly detail the theory behind AFT Impulse. Please reach out to AFT Support if you have further questions, or consider attending one of our AFT Impulse training seminars.

Wavespeed - Detailed Discussion

As described in Wavespeed and Communication Time, the speed at which a wave can travel through a fluid is limited by its bulk modulus of elasticity and density. When constrained to a pipe, the wavespeed is further reduced as some of the wave energy is lost to deforming the pipe. An in depth derivation of wavespeed is presented here, along with discussion of the meaning of the Pipe Support constant.

Derivation Including Pipe Effects

Consider again an instantaneous valve closure. When a valve at the end of a pipe closes, a number of things happen. As we have seen, a wave is propagated upstream as discussed in the Conceptual Example. However, what happens to the pipe itself?

Fluid continues to flow into the pipe, but not out of it. This causes both radial and axial strain - the pipe may be deformed in both directions, depending on how it is supported. This increase in fluid mass continues until the wave is reflected, or in other words, for a time equal to the time required for the wave to travel the pipe:

$$\Delta t_{end-to-end} = L/a$$

During the entirety of this period, the pipe is being deformed. If we consider the pipe to be supported only at the upstream end, it is free to expand lengthwise. The total axial increase in length occurs at the end of the period.

$$total\ increase\ in\ length = \Delta x$$

Further, this means that the closed valve has a certain velocity - it moves the distance Δx over the time period L/a.

$$V_{valve} = \frac{a \, \Delta x}{L}$$

The fluid velocity at the face of the valve decreases from the initial steady value V_0 to this V_{valve}, making the change in fluid velocity:

$$\Delta V_{fluid} = \frac{a \, \Delta x}{L} - V_0$$

The total fluid mass in the pipe increases during this period as well, by the steady rate.

$$\Delta m = \dot{m}_{steady} \Delta t = \frac{\rho \, V_0 \, A \, L}{a}$$

Combining the previous two equations:

$$\frac{\rho \, V_0 \, A \, L}{a} = \frac{\rho \, A \, L}{a} \left(\frac{a \, \Delta x}{L} - \Delta V_{fluid} \right)$$

There are three mechanisms that allow the mass to increase in the pipe - the pipe strains axially, strains radially, and the fluid compresses.

$$\frac{\rho \, A \, L}{a} \left(\frac{a \, \Delta x}{L} - \Delta V_{fluid} \right) = \rho \, L \, \Delta A + \rho \, A \, \Delta x + L \, A \, \Delta \rho$$

Simplifying and reorganizing:

$$-\frac{\Delta V_{fluid}}{a} = \frac{\Delta A}{A} + \frac{\Delta \rho}{\rho}$$

With the instantaneous waterhammer relationship,

$$\Delta P = -\rho \, a \, \Delta V$$

the velocity change term can be eliminated, and the result can be solved for a^2:

$$a^2 = \frac{\dfrac{\Delta P}{\rho}}{\dfrac{\Delta A}{A} + \dfrac{\Delta \rho}{\rho}}$$

The bulk modulus of a fluid relates the change in volume (or density) to an applied change in pressure:

$$K = \frac{\Delta P}{\Delta \rho / \rho}$$

Multiplying the a^2 equation through by $K/\Delta P$:

$$a^2 = \frac{K/\rho}{1 + K \dfrac{\Delta A}{A \, \Delta P}}$$

The area/pressure term is a measure of the elastic response of the pipe - its value will depend on how the pipe is constrained. For convenience, common terms that show up in this analysis are separated, and a new constant c_1 is defined.

$$a = \sqrt{\frac{K/\rho}{1 + c_1 \dfrac{K}{E} \dfrac{D}{e}}}$$

Pipe Support Constants

The value of c_1 varies based on how the pipe is constrained. As an example, a **thin-walled pipe fully anchored at the upstream end** is discussed here, with other values of c_1 listed below.

The change in internal area of the pipe A, is equal to the change in radius multiplied by the circumference. The change in diameter is equal to the radial strain multiplied by the diameter.

$$\Delta A = \frac{\Delta D}{2} \pi D = \frac{\xi_{radial} D}{2} \pi D$$

Dividing by area:

$$\frac{\Delta A}{A} = 2\xi_{radial}$$

The radial strain is affected both by axial and circumferential strain, which are related to one another via Poisson's Ratio v:

$$\xi_{radial} = \xi_{circumferential} - \nu \xi_{axial}$$

Further, strain is related to stress with Young's Modulus E:

$$\xi = \frac{\sigma}{E}$$

Combining the above and introducing the ΔP term:

$$\frac{\Delta A}{A \Delta P} = \frac{2}{\Delta P E}(\sigma_{circ} - \nu \sigma_{axial})$$

The two stresses are related to their respective forces and areas:

$$\sigma_{axial} = \frac{A \Delta P}{\pi D e} \qquad \sigma_{circ} = \frac{D \Delta P}{2 e}$$

Adding these to the previous equation and simplifying:

$$\frac{\Delta A}{A \Delta P} = \frac{D}{e E}\left(1 - \frac{\nu}{2}\right)$$

Which is able to give an expression for c_1:

$$K \frac{\Delta A}{A \, \Delta P} = K \frac{D}{e \, E} \left(1 - \frac{\nu}{2} \right) = c_1 \frac{K}{E} \frac{D}{e} \qquad\qquad c_1 = 1 - \frac{\nu}{2}$$

Note: The above value of c_1 is only valid for a pipe anchored only at the upstream end. See below for other values of c_1 for differing constraints.

- **Thin-Walled Anchored Upstream (detailed above)**

$$c_1 = 1 - \frac{\nu}{2}$$

- **Thin-Walled Anchored Throughout**

$$c_1 = 1 - \nu^2$$

- **Thin-Walled With Expansion Joints**

$$c_1 = 1$$

- **Thick-Walled Anchored Upstream**

$$c_1 = \frac{2\,e}{D}(1 + \nu) + \frac{D}{D + e}\left(1 - \frac{\nu}{2} \right)$$

- **Thick-Walled Anchored Throughout**

$$c_1 = \frac{2\,e}{D}(1 + \nu) + \frac{D}{D + e}\left(1 - \nu^2 \right)$$

- **Thick-Walled With Expansion Joints**

$$c_1 = \frac{2\,e}{D}(1 + \nu) + \frac{D}{D + e}$$

- **Circular Tunnel (unlined)**

$$c_1 = \frac{2\,e}{D}(1 + \nu)$$

- **Circular Tunnel (lined) / Buried Pipe**

$$c_1 = \frac{2\,E\,e}{E_R\,D + 2\,E\,e} \qquad E_R = Rock/Soil\ Modulus\ of\ Rigidity$$

Related Topics

Transient Flow Equations

To begin with any rigorous fluid dynamics solution, you must start with the most fundamental laws of physics. For isothermal flow of a liquid, there are two core equations - conservation of mass and Newton's second law of motion. We will consider the below control volume in a pipe. The control surfaces *1* and *2* are permitted to move relative to one another, and the unlabeled control surface on the pipe wall must remain fixed to the wall. For the purposes of analyzing a fluid transient, it is safe to consider the system one dimensional in *x*. Note that there is one radial effect that is important to consider - the change in cross-sectional area of the pipe.

The resulting equations nonlinear hyperbolic partial differential equations, requiring a numeric solution such as the Method of Characteristics.

For a complete development of the equations please see one of the many excellent references (Wylie, 1993, Chaudry, 1987, and Swaffield, 1993.
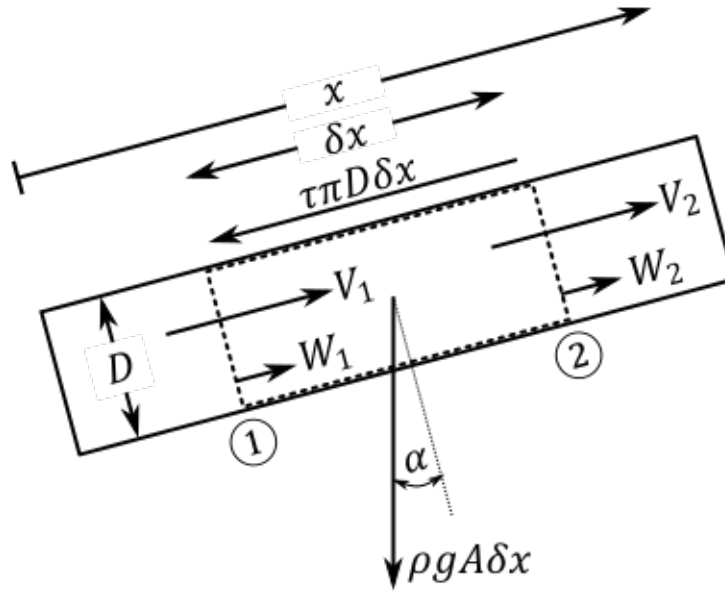
Figure 1: Control volume

Continuity Equation

First we apply conservation of mass to the control volume via Reynold's Transport Theorem:

$$\frac{d}{dt} \int_{C\mathbb{V}} \rho \, d\mathbb{V} - \oint_{CS} \rho \, (\vec{V} \cdot \hat{n}) \, dA = 0$$

Simplifying to one dimension:

$$\frac{d}{dt} \int_{x_1}^{x_2} \rho \, A \, dx - [\rho_1 \, A_1 \, (V_1 - W_1) - \rho_2 \, A_2 \, (V_2 - W_2)] = 0$$

According to the Leibniz integral rule:

$$\int_{x_1}^{x_2} \rho \, A \, dx = \int_{x_1}^{x_2} \frac{\partial}{\partial t} (\rho \, A) \, dx + \rho_2 \, A_2 \underbrace{\frac{dx_2}{dt}}_{W_2} - \rho_1 \, A_1 \underbrace{\frac{dx_1}{dt}}_{W_1}$$

Applying this relationship and canceling terms:

$$\int_{x_1}^{x_2} \frac{\partial}{\partial t}(\rho\, A)\, dx + \rho_2\, A_2\, V_2 - \rho_1\, A_1\, V_1 = 0$$

Considering a mean value of the integral:

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

Divide by δx

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

Expand and group terms:

$$\frac{1}{\rho}\underbrace{\left(\frac{\partial \rho}{\partial t} + V\frac{\partial \rho}{\partial x}\right)}_{\frac{d\rho}{dt}} + \frac{1}{A}\underbrace{\left(\frac{\partial A}{\partial t} + V\frac{\partial A}{\partial x}\right)}_{\frac{dA}{dt}} + \frac{\partial V}{\partial x} = 0$$

$$\frac{1}{\rho}\frac{d\rho}{dt} + \frac{1}{A}\frac{dA}{dt} + \frac{\partial V}{\partial x} = 0$$

This is a generally applicable equation, we have not made any significant assumptions on the nature of the fluid itself.

Next, we will apply the definition of bulk modulus of elasticity. This step carries with it additional assumptions - because we are effectively dropping thermodynamic effects with this substitution, the equation will only be able to be applied to "slightly compressible" fluids. This is not an issue in waterhammer applications - where the fluid is commonly referred to as "incompressible."

$$K = \rho\frac{dP}{d\rho} \qquad \Leftrightarrow \qquad \frac{1}{\rho}\frac{d\rho}{dt} = \frac{1}{K}\frac{dP}{dt}$$

$$\frac{1}{K}\frac{dP}{dt} + \frac{1}{A}\frac{dA}{dP}\frac{dP}{dt} + \frac{\partial V}{\partial x} = 0$$

Rearranging the above equation, we can find the definition of <u>wavespeed</u>.

$$\frac{dP}{dt} + \rho \underbrace{\left(\frac{K/\rho}{1 + \frac{K}{A}\frac{dA}{dP}}\right)}_{a^2}\frac{\partial V}{\partial x} = 0$$

Expanding the pressure term, we get a convective term that is typically ignored in waterhammer analysis due to its relatively very small magnitude.

$$\frac{\partial P}{\partial t} \quad + \quad \underbrace{V\frac{\partial P}{\partial x}}_{Convective\ term} \quad + \quad \rho\, a^2 \frac{\partial V}{\partial x} = 0$$

Finally, we arrive at what is termed the continuity equation.

$$\boxed{\frac{1}{\rho}\frac{\partial P}{\partial t} + a^2\frac{\partial V}{\partial x} = 0}$$

Momentum Equation

The application of Newton's second law follows from the same control volume above. Again applying Reynold's Transport Theorem we begin with the following relationship:

$$\sum F = \frac{d}{dt}\int_{C\mathbb{V}} \rho V\, d\mathbb{V} - \oint_{CS} \rho V\, (\vec{V}\cdot\hat{n})\, dA$$

Very similar to the continuity equation, we can apply a one dimensional simplification, the Leibniz integral rule, and consider the resulting integral as a mean over the interval. These steps are extremely similar to the above and are not shown for brevity.

$$\sum F = \frac{\partial}{\partial t}(\rho \, V \, A)\, \delta x + \Delta(\rho \, A \, V^2)$$

Summing the forces on the body:

$$\sum F = P_1 \, A_1 - P_2 \, A_2 - \tau \, \pi \, D \, \delta x - \rho \, g \, A \sin(\alpha) \, \delta x$$

To find the shear term, we consider a *steady* force balance with only two terms - the pressure across a section, and the shear term. We can find the pressure across the section with the Darcy-Weisbach relationship.

$$\sum F_{steady} = \frac{\pi D^2}{4} dP - \tau \, \pi \, D \, \delta x = 0$$

$$dP = \frac{f \, \rho \, V \, |V| \, \delta x}{2 \, D}$$

$$\tau = \frac{f \, \rho \, V \, |V|}{8}$$

Incorporating this, dividing through by *δx*, and assuming that the areas on the pressure terms can be considered constant, we arrive at the following combined equation:

$$\frac{\partial}{\partial t}(\rho \, V \, A) + \frac{\partial}{\partial x}(\rho \, A \, V^2) = -A\frac{\partial P}{\partial x} - \frac{f \, \rho \, V \, |V|}{8}\pi \, D - \rho \, g \, A \sin(\alpha)$$

Expanding the partial derivatives, and dividing through by *ρA*, we find the continuity equation and can drop those terms. We also again have a convective term which is normally ignored.

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

Gathering the remaining terms, we arrive at the momentum equation:

$$\frac{\partial V}{\partial t} + \frac{1}{\rho}\frac{\partial P}{\partial x} + \frac{f\,V\,|V|}{2\,D} + g\sin(\alpha) = 0$$

Related Topics

Method of Characteristics

The Method of Characteristics is a general approach for solving a partial differential equation by reducing it to a system of ordinary differential equations.

We see here how it is applied to the transient flow equations. The continuity equation and momentum conservation equation are both partial differential equations. Together, they have two unknowns P and V, and two independent variables x and t. Application of the Method of Characteristics will generate four ordinary differential equations.

Characteristic Equations

The equations are represented below as L_1 and L_2.

$$\frac{\partial V}{\partial t} + \frac{1}{\rho}\frac{\partial P}{\partial x} + \frac{f\,V\,|V|}{2D} + g\sin(\alpha) = L_1$$

$$\frac{1}{\rho}\frac{\partial P}{\partial t} + a^2\,\frac{\partial V}{\partial x} = L_2$$

Linearly combining the equations with the introduction of a constant factor λ

$$L_1 + \lambda\,L_2 = 0$$

$$\frac{\partial V}{\partial t} + \frac{1}{\rho}\frac{\partial P}{\partial x} + \frac{f\,V\,|V|}{2D} + g\,\sin(\alpha) + \lambda\left(\frac{1}{\rho}\frac{\partial P}{\partial t} + a^2\,\frac{\partial V}{\partial x}\right) = 0$$

Rearranging terms from the linear combination above

$$\frac{\lambda}{\rho}\left(\frac{\partial P}{\partial t} + \frac{1}{\lambda}\frac{\partial P}{\partial t}\right) + \left(\frac{\partial V}{\partial t} + \lambda\,a^2\,\frac{\partial V}{\partial x}\right) + \frac{f\,V\,|V|}{2D} + g\,\sin(\alpha) = 0$$

We define a relationship for λ:

$$\frac{dx}{dt} = \frac{1}{\lambda} = \lambda\,a^2$$

Where:

$$\lambda = \pm\frac{1}{a} \qquad and \qquad \frac{dx}{dt} = \pm\,a$$

This relationship between wavespeed, time step, and length step must be enforced for this method to work as expected. This is discussed in detail in <u>Pipe Sectioning - Introduction to Method of Characteristics</u>.

Substituting into the above equation:

$$\pm\frac{1}{a\,\rho}\left(\frac{\partial P}{\partial t} + \frac{dx}{dt}\frac{\partial P}{\partial x}\right) + \left(\frac{\partial V}{\partial t} + \frac{dx}{dt}\frac{\partial V}{\partial x}\right) + \frac{f\,V\,|V|}{2\,D} + g\,\sin(\alpha) = 0$$

The terms in the parantheses are simply the total derivatives for *P* and *V*.

$$\frac{dP}{dt} = \frac{\partial P}{\partial t} + \frac{dx}{dt}\frac{\partial P}{\partial x}$$

$$\frac{dV}{dt} = \frac{\partial V}{\partial t} + \frac{dx}{dt}\frac{\partial V}{\partial x}$$

Simplifying the expression to:

$$\pm \frac{1}{a\,\rho}\frac{dP}{dt} + \frac{dV}{dt} + \frac{f\,V\,|V|}{2\,D} + g\sin\left(\alpha\right) = 0$$

This represents the four *characteristic equations* that we are searching for, paired as positive (C+) and negative (C-) sets of equations:

$$C^{+} \begin{cases} +\dfrac{1}{a\,\rho}\dfrac{dP}{dt} + \dfrac{dV}{dt} + \dfrac{f\,V\,|V|}{2\,D} + g\sin\left(\alpha\right) = 0 \\[2em] \dfrac{dx}{dt} = a \end{cases}$$

$$C^{-} \begin{cases} -\dfrac{1}{a\,\rho}\dfrac{dP}{dt} + \dfrac{dV}{dt} + \dfrac{f\,V\,|V|}{2\,D} + g\sin\left(\alpha\right) = 0 \\[2em] \dfrac{dx}{dt} = -a \end{cases}$$

Note: There have not been any simplifying assumptions made in the development of these ordinary differential equations - they represent the original partial differential equations exactly.

Numerical Method

We can simplify the equations further into a form that is convenient for simulation.

For the **positive characteristic** multiply the top equation by *a dt*, which is equal to d*x*

$$\frac{1}{\rho}\,dP + a\,dV + \frac{f\,V\,|V|}{2\,D}\,dx + g\sin(\alpha)\,dx = 0$$

Recognizing that *sin(α)* is simply *dz/dx:*

$$\frac{1}{\rho}\,dP + a\,dV + \frac{f\,V\,|V|}{2\,D}\,dx + g\,dz = 0$$

It is more convenient to use mass flow instead of velocity, and multiplying through by ρ.

$$dP + \frac{a}{A}\,d\dot{m} + \frac{f\,\dot{m}\,|\dot{m}|}{2\,D\,\rho\,A^2}\,dx + \rho\,g\,dz = 0$$

This equation can be integrated *along the positive characteristic*.

It is useful at this stage to consider a graphical representation of the characteristic grid. Some point *P* is affected by the positive characteristic from point *A*, and by the negative characteristic from point *B*.
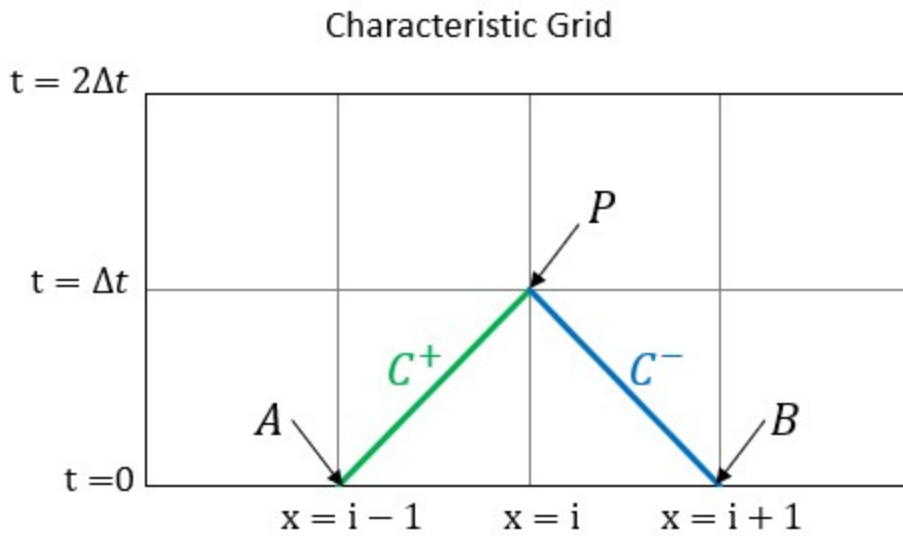
Figure 1: Characteristic grid. The point P sees effects from points A and B after some time dt, which has equal magnitude of dx/a

Setting up the integral, along the positive characteristic line:

$$\int_{P_A}^{P_P} dP + \frac{a}{A}\int_{\dot{m}_A}^{\dot{m}_P} d\dot{m} + \frac{f}{2\,D\,\rho\,A^2}\int_{x_A}^{x_P} \dot{m}\,|\dot{m}|\,dx + \rho\,g\int_{z_A}^{z_P} dz = 0$$

Most of the terms are straightforward. However, the third term contains mass flow which varies with respect to *x*. This term is approximated with the trapezoidal rule, which is second order accurate.

$$(P_P - P_A) + \frac{a}{A}(\dot{m}_P - \dot{m}_A) + \frac{f}{2\,\rho\,D\,A^2}[|\dot{m}_P\|\dot{m}_A|(x_P - x_A)] + \rho\,g\,(z_p - z_A) = 0$$

Defining *impedance*:

$$B = \frac{a}{A}$$

and *resistance*:

$$R = \frac{f \, \Delta x}{2 \, \rho \, D \, A^2}$$

and solving for pressure at the point of interest:

$$\boxed{P_P = P_A - B(\dot{m}_P - \dot{m}_A) - R \, \dot{m}_P \, |\dot{m}_A| - \rho \, g \, (z_p - z_A)}$$

we can repeat the above process for the *negative characteristic* and arrive at:

$$\boxed{P_P = P_B + B \, (\dot{m}_P - \dot{m}_B) + R \, \dot{m}_P \, |\dot{m}_B| + \rho \, g \, (z_p - z_B)}$$

We have finally arrived a system of two equations with only two unknowns - mass flow and pressure at the point *P*. These equations can be further arranged into a convenient form known as the compatibility equations.

Related Topics

Compatibility Equations

From the characteristic equations (repeated below), we can construct two convenient equations known as the *compatibility equations*.

$$P_P = P_A - B \, (\dot{m}_P - \dot{m}_A) - R \, \dot{m}_P |\dot{m}_A| - \rho \, g \, (z_P - z_A) \qquad (C^+)$$

$$P_P = P_B + B \, (\dot{m}_P - \dot{m}_B) + R \, \dot{m}_P |\dot{m}_B| + \rho \, g \, (z_P - z_B) \qquad (C^-)$$

The only unknown terms in these equations are the pressure and mass flow at point P. The other terms are known either from a previous time step, or from the initial conditions as determined from a steady state simulation.

Rearranging terms:

$$P_P = P_A + B\dot{m}_A - \rho g (z_P - z_A) - \dot{m}_P (B + R|\dot{m}_A|) \qquad (C^+)$$

$$P_P = P_B - B\dot{m}_B - \rho g (z_P - z_B) - \dot{m}_P (B + R|\dot{m}_B|) \qquad (C^-)$$

and defining four new constants (constant for *this time step*):

$$C_P = P_A + B\dot{m}_A - \rho g (z_P - z_A)$$

$$C_M = P_B - B\dot{m}_B + \rho g (z_P - z_B)$$

$$B_P = B + R|\dot{m}_A|$$

$$B_M = B + R|\dot{m}_B|$$

We have our simplest equations yet:

$$\boxed{P_P = C_P - \dot{m}_P B_P} \qquad (1)$$

$$\boxed{P_P = C_M + \dot{m}_P B_M} \qquad (2)$$

Which can be solved directly for pressure or mass flow:

$$P_P = \frac{C_P B_M + C_M B_P}{B_P + B_M} \qquad (3)$$

$$\dot{m}_P = \frac{C_P - C_M}{B_P + B_M} \qquad (4)$$

Note: The compatibility equations allow a solution of all interior points in a pipe. However, the pipe endpoints are solved by applying specific boundary conditions. The boundary condition relationships build on the previous methods, and offer the final needed information to generate a complete solution.

Related Topics

Transient Cavitation Models

AFT Impulse has two cavitation models available to capture transient cavitation in pipes: the Discrete Vapor Cavity Model (DVCM) and the Discrete Gas Cavity Model (DGCM).

Any cavitation in a system inherently violates AFT Impulse's fundamental assumption that all pipes are 100% liquid-full. The cavitation models attempt to work around that assumption by modeling cavitation only at pipe computation stations, with the intervening pipe remaining liquid-full.

In a real system, vapor pockets can form continuously in the liquid. However, Impulse lumps all vapor formed into discrete pockets fixed at the grid points determined by the Method of Characteristics (MOC). Vapor volume is tracked at each station, and the MOC equations are applied to the pipe between each station following the liquid-full assumption.

Both models, DVCM and DGCM, are based on the methodology described in Wylie 1993, pg 66-69 (DVCM) and pg 184-187 (DGCM), and supplemented by other literature sources. These models are considered industry standard, and are considered good models for transient cavitation. However, their accuracy is inherently limited by the liquid-full assumption behind AFT Impulse and the MOC. See the Interpreting Transient Cavitation Predictions page.

Discrete Vapor Cavity Model (DVCM)

DVCM evaluates transient cavitation by assuming one of two conditions exists. Either the pressure at the pipe computation station is greater than P_{vapor} and the MOC equations can be applied as normal, or the pressure at the pipe computation station is equal to P_{vapor} and cavitation is present.

When the MOC equations calculate that the pressure at a station reaches (or falls below) P_{vapor}, the pressure at that station is fixed exactly at P_{vapor}. The MOC equations are then evaluated treating the station as a boundary condition, simplifying the compatibility equations for the flowrate upstream and downstream of the station to:

$$\dot{m}_{new,up} = \frac{C_P - P_{vap}}{B_P} \; ; \dot{m}_{new,down} = \frac{P_{vap} - C_M}{B_M}$$

Asserting a fixed pressure boundary of P_{vapor} creates a mass flow discontinuity across the station. Impulse assumes this mass flow discontinuity is caused by vapor pockets forming and tracks it as the total vapor volume. The actual mass flowrate as a function of time is unknown, so an average of the mass flowrate at the old time step and new time step is used:

$$V_{vap,new} = V_{vap,old} + \frac{\Delta t}{\rho} \left[\frac{\dot{m}_{old,down} + \dot{m}_{new,down}}{2} - \frac{\dot{m}_{old,up} + \dot{m}_{new,up}}{2} \right]$$

When the vapor volume tracked at a station reaches (or falls below) zero, the station returns to the normal MOC equations.

Discrete Gas Caivty Model (DGCM)

DGCM builds on DVCM by introducing a small amount of free gas into the system. Most liquid systems contain some small amount of gas flowing with the liquid or dissolved in the liquid. This gas expands as system pressure approaches vapor pressure and can be considered a precursor to the cavitation which exists when the liquid is at vapor pressure.

Similar to DVCM, DGCM evaluates transient cavitation by assuming one of two conditions exists: either the station stagnation pressure is above P_{vapor}, or the system pressure is equal to P_{vapor}.

Calculations when $P_{stag} > P_{vapor}$:

When the station stagnation pressure is above P_{vapor}, DGCM uses the Ideal Gas Law to determine the free gas volume:

$$PV = nRT$$

Specifically, *nRT* are lumped together as a constant value determined by the Initial Void Fraction and Initial Void Fraction Reference Pressure parameters on the Transient Cavitation Panel such that:

$$V_{gas} = \frac{nRT}{P_{stag} - P_{vap}} = \frac{constant}{P_{stag} - P_{vap}}$$

Impulse then couples the above equation with the MOC compatibility equations to simultaneously solve for the pressure, upstream mass flow rate, and downstream mass flowrate at each station for each time step. These calculations are always performed regardless of how

close the system is to Pvap, adding additional computational complexity. As a result, a simulation using DGCM will take longer to run than a simulation using DVCM.

The free gas volume calculated at each station allows DGCM to improve on DVCM. Specifically, it allows DGCM to capture the effect that gas or vapor in a liquid system has on wavespeed. Wavespeed depends on how compressible a fluid is, with gas being more compressible than a liquid. At low pressures, gas volume will increase relative to the liquid, lowering the wavespeed in the fluid. This effect can become more pronounced as the fluid approaches the vapor pressure of the liquid, and is an important effect to capture.

The Method of Characteristics solution requires wavespeed to be fixed prior to the simulation, meaning wavespeed can't be directly modified. However, DGCM forces a pressure wave to expend some energy compressing the gas volume at a station as the wave travels along the pipe, effectively slowing wave motion as gas volume increases at low pressures.

The small gas volume at each station also helps avoid some of the numerical noise associated with DVCM, effectively dampening high-frequency waves as they move through the system.

Note that the free gas in the system can be set using the Initial Void Fraction and Reference Pressure settings in the Transient Cavitation Panel. Default values are 1E-7 and 1 atm, respectively. Literature discussing DGCM, and AFT's internal tests show that these values are meant to be constants rather than measured system values. AFT does not recommend changing these values from the default.

Calculations when $P_{stag} = P_{vapor}$:

When the station stagnation pressure is equal to P_{vapor}, DGCM simplifies to DVCM. The pressure at the station is fixed exactly at P_{vapor}, the MOC equations are evaluated treating that station as a boundary condition, and vapor volume is tracked as the mass flow discontinuity across the station.

DGCM contains one further difference from DVCM, seen in the equation used to integrate the mass flow discontinuity. DVCM integrates the total change in vapor volume from one time step to the next by averaging the mass flow evenly between the old time step and the new time step. DGCM instead uses rectangular integration and a weighting factor, ψ, with a value of 0.95, as shown in the equation below.

Using a value of ψ=0.95 weights the equation towards the new time step, adding numerical stability to the results, and providing a better match between Impulse results and available reference data. AFT does not recommend changing this value from the default.

$$V_{vap,new} = V_{vap,old} + \frac{\Delta t}{\rho} [\psi(\dot{m}_{new,down} + \dot{m}_{new,up}) + (1 - \psi)(\dot{m}_{old,down} + \dot{m}_{old,up})]$$

Related Topics

Related Blogs

Infinite Pipe Transient Theory

An infinite pipe is a representation of an interior pipe station - this is not the end of the pipe, but is identical to the same interior station in an equivalent system were the pipe is explicitly modeled. Therefore, all of the Method of Characteristics solution will apply to the infinite pipe boundary as well. However, by not explicitly modeling the entire pipe, there is some information we lose.

To determine the properties at the infinite boundary, we must know the properties of the **adjacent stations from the previous time step**. This is the case for any interior station. But this poses an immediate problem - there is only one "adjacent station" - we have removed the other by not modeling the pipe.
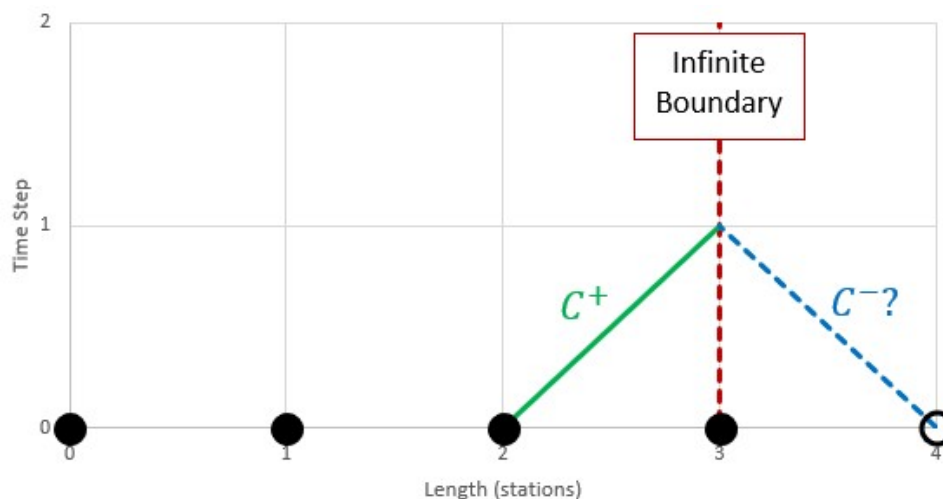
Figure 1: Downstream properties are required to calculate the negative characteristic

We can overcome this restriction by creating a "virtual" station one section past the end of the modeled pipe. Again, note that this virtual station represents a real interior pipe point, it is only "virtual" in the sense that no pipe is included in the **model**. With the virtual station, we then have stations adjacent to the boundary, and can proceed with the compatibility equations.
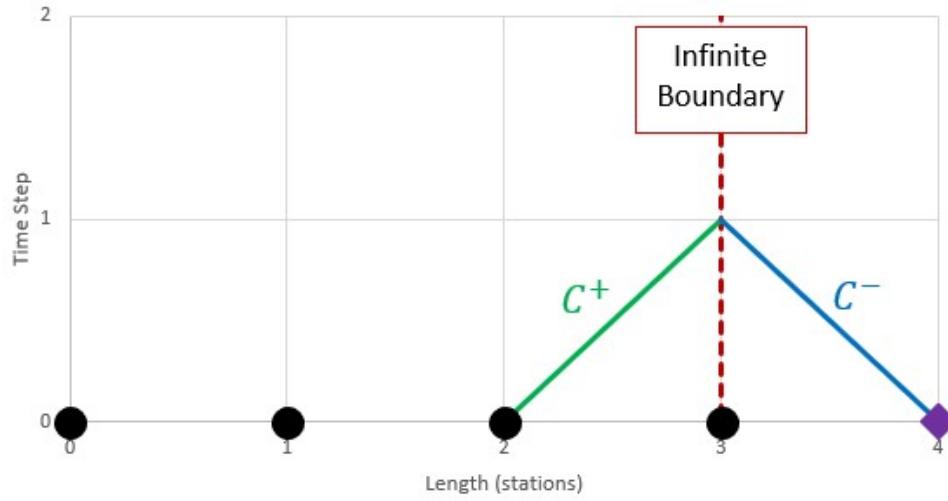
Figure 2: Create a "virtual" station to allow calculation of the negative characteristic

However, we have introduced a new problem. We need to know the pressure and flow at the virtual station to implement the compatibility equations for the subsequent time step. To proceed, we must estimate these values. This can be accomplished with the data we do know - adjoining stations and previous time steps. Without transient effects, we can make a very good approximation with a second order extrapolation.

$$P_{virtual,new} = P_{i+1,t} = 3P_{i,t-\Delta t} - 3P_{i-1,t-2\Delta t} + P_{i-2,t-3\Delta t}$$

$$\dot{m}_{virtual,new} = \dot{m}_{i+1,t} = 3\dot{m}_{i,t-\Delta t} - 3\dot{m}_{i-1,t-2\Delta t} + \dot{m}_{i-2,t-3\Delta t}$$

This estimation requires data from the 3 adjoining stations, and 3 time steps previous.
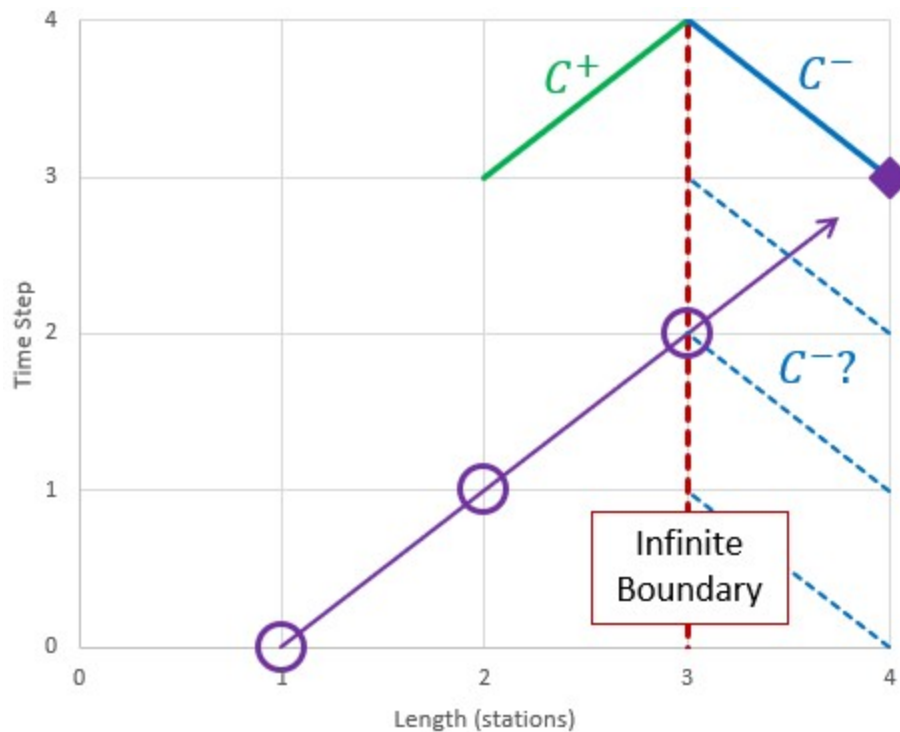
Figure 3: Estimate the properties at the virtual station with the 3 previous time steps and stations

This introduces yet another obstacle to overcome - if we require 3 stations and time steps for estimating the virtual station in the time step 3 (3 time steps after 0), we are not able to utilize the compatibility equations to calculate the properties at the boundary until time step 4. Fortunately, there is an easy solution.

The properties at the boundary can be considered constant until a transient from the system affects it. Any transient change requires 1 time step to travel 1 section. We can state that the properties at the boundary remain unchanged for the first 3 time steps if we keep the boundary at least 3 stations away from any potential transient perturbation. In other words, we need to require a minimum number of sections in the pipe. This minimum value is only **two** sections. This is because the initial transient time step (time step 1 in the diagram) can be considered the second fixed data point available to us - the first (time step 0) being the steady state initial conditions. The properties at station 1 cannot change until time step 2.

With a minimum of two sections, we can state that the pressure and flow at the boundary do in fact remain fixed for the first 3 time steps (4 including steady state).
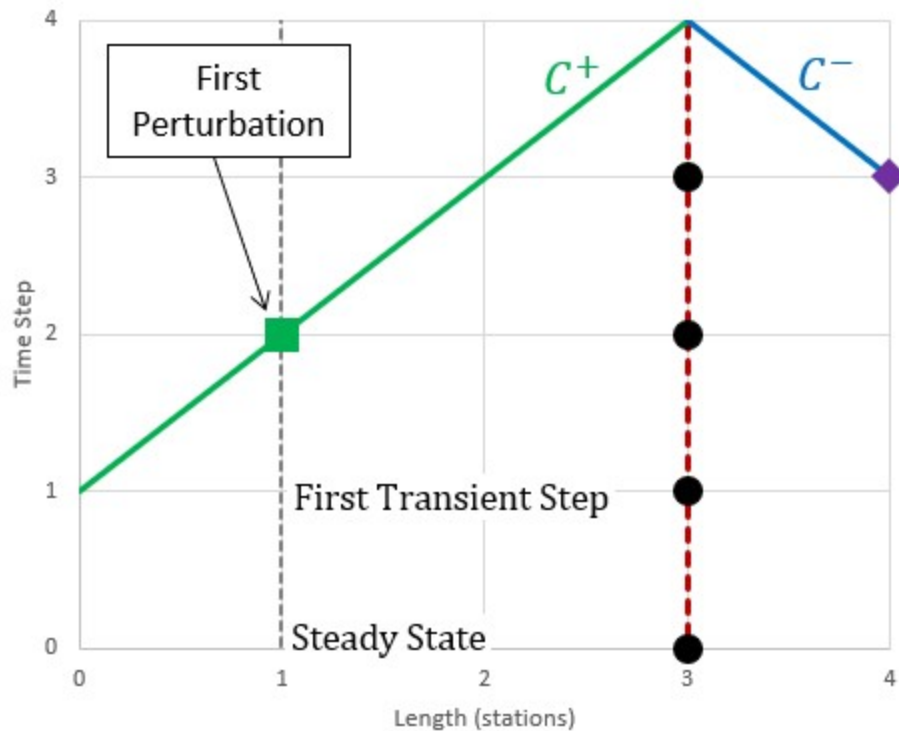
Figure 4: The estimation forces a certain minimum number of sections. Note that the properties cannot change at station 1 until time step 2 - one time step is required for any defined transient to impact the local solution.

See Wylie, et al, 1993, pp. 121-122 for more information.

Infinite Pipe Vapor Cavitation Theory

The Infinite Pipe junction is designed to only pass pressure waves out through the junction, and never to allow any reflections. If the junction drops to vapor pressure, a cavity would occur, which would cause reflections. Therefore, vapor cavities at these junctions are ignored and the pressure is allowed to fall below the vapor pressure. This maintains the function of the junction. If this occurs the model should be updated.

Related Topics

References

1. Adamkowski, A. and Lewandowski M., "Experimental Examination of Unsteady Friction Models for Transient Pipe Flow Simulation", *Journal of Fluids Engineering*, Vol. 128, pp. 1351-1363, November 2006.

2. American Petroleum Institute. 2010. API Standard 674 with Errata (May 2014) 3rd Edition. Positive Displacement Pumps-Reciprocating. Annex C, sections A.1 and A.2, pages 67-68

3. 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

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 12.1-12.6-2016*, Rotodynamic (Centrifugal) Slurry Pumps, 2016, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA

7. *ANSI/HI Standard 14.3-2019*, American National Standard for Rotodynamic Pumps - for Design and Application, 2019, published by Hydraulic Institute, New Jersey, USA

8. ANSI/API 526 7th Edition - Flanged Steel Pressure-relief Valves, 2017, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA

9. 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

10. Ballun, John V., "A Methodology for Predicting Check Valve Slam", *Journal AWWA, Vol.99, No. 3, pp. 60-65, March 2007*.

11. Blanding, James M., Ph.D., and Trey Walters, PE. Pulsation Analysis in Positive Displacement Pump Systems Using Waterhammer, Modal, and Animation Software. Houston, TX: 45th Turbomachinery & 32nd Pump Symposia, Sept. 2016.

12. Brown R.J. , and D.C. Rogers, Development of Pump Characteristics from Field Tests, Journal of Mechanical Design, ASME, vol. 102, pp. 807-817, Oct., 1980.

13. Carthew, G.A., Goehring, C.A., van Teylingen, J.E., Development of dynamic head loss criteria for raw sludge pumping, Journal Water Pollution Control Federation (WPCF), Volume 55, Number 5, (May 1983).

14. Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988.

15. Darby, R., Chemical Engineering Fluid Mechanics. 2nd edition. New York, NY, 2001

16. Donsky, B., Complete Pump Characteristics and the Effects of Specific Speeds on Hydraulic Transients, ASME Journal of Basic Engineering, pp. 685-699, Dec. 1961.

17. Fox, R. W. and A.T. McDonald, Introduction to Fluid Mechanics, 3rd edition, Wiley & Sons, New York, NY, 1985.

18. 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

19. Idelchik, I. E., Handbook of Hydraulic Resistance, 4th edition, CRC Press, Boca Raton, FL, 2007.

20. Ingersoll-Dresser Pumps, Cameron Hydraulic Data, Liberty Corner, NJ, 1995.

21. Jacobs, B.E.A., Design of Slurry Transport Systems, ELSEVIER SCIENCE PUBLISHERS LTD, New York, NY, 1991.

22. Jeppson, R. W., Analysis of Flow in Pipe Networks, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.

23. Karassik, I.J., J.P. Messina, P. Cooper and C.C. Heald, Pump Handbook, 3rd Edition, McGraw Hill, New York, NY, 2001.

24. Karney, B.W., and D. McInnis, Efficient Calculation of Transient Flow in Simple Pipe Networks, ASCE Journal of Hydraulic Engineering, Vol. 118, No. 7, July, 1992.

25. Kittredge, C.P., Hydraulic Transients in Centrifugal Pump Systems, ASME Journal of Basic Engineering, Vol. 78, No. 6, pp. 1307-1322, Aug. 1956.

26. Lyons, J. L., Lyons' Valve Designers' Handbook, Van Nostrand Reinhold, New York, NY, 1982.

27. Miller, D. S., Internal Flow Systems, 2nd edition, Gulf Publishing Company, Houston, TX, 1990.

28. Suter, P., Representation of Pump Characteristics for Calculation of Waterhammer, *Sultzer Technical Review Research Issue*, pp. 45-48, 1966.

29. TAPPI, Generalized method for determining the pipe friction loss of flowing pulp suspensions, TAPPI, TIS 0410-14, Atlanta, GA, 1988.

30. Tilton, J. N., Perry's Chemical Engineer's Handbook, Seventh Edition, Chapter 6, McGraw-Hill, New York, NY, 1997.

31. Thorley, A.R.D., Fluid Transients in Pipeline Systems, 2nd Ed., D&L George, Ltd. 2004

32. Thorley, A.R.D., and A. Chaudry, Pump Characteristics for Transient Flow, Pressure Surges and Fluid Transients, BHR Group, 1996.

33. Val-Matic Valve and Manufacturing Corp., "Surge Control in Pumping Systems," Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.

34. Vazquez, J.A., C.H. Cloud, and R.J Eizember, 2012, "Simplified Modal Analysis for the Plant Machinery Engineer," Proceedings of the 41st Turbomachinery Symposium.

35. Walters, T.W., Analytical Approaches to Modeling Transient Vaporous Cavitation in Multi-Pipe Fluid Systems, ASME FED-Vol. 116, New York, NY, 1991.

36. Walters, T., Dahl, T. and Rogers, D.C., "Pump Specific Speed and Four Quadrant Data in Waterhammer Simulation - Taking Another Look", *Proceedings of the ASME 2020 Pressure Vessels and Piping Conference*, PVP 2020, Minneapolis, MN, USA, July 19-24 2020.

37. Walters, T. W., Lang S. A., and Miller D. O., "Unappreciated Challenges in Applying Four Quadrant Pump Data to Waterhammer Simulation, Part 1: Fundamentals", Proc. 13th International Conference on Pressure Surges, BHR Group, Bordeaux, France, November 2018.

38. Watters, G.Z, Modern Analysis and Control of Unsteady Flow in Pipelines, Ann Arbor Science, Ann Arbor, MI, 1979.

39. White, F. M., Fluid Mechanics, McGraw Hill, New York, NY, 1979.

40. Wilson, K.C., "Effect of Solids Concentration on Deposit Velocity", Journal of Pipelines, 5, 1986, pp. 251-257.

41. Wilson, K.C., G.R. Addie, A. Sellgren and R. Clift, *Slurry Transport Using Centrifugal Pumps*, 3rd Ed., published by Springer, 2006.

42. 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

43. Wylie, E.B., V.L. Streeter & L. Suo, Fluid Transients in Systems, Prentice Hall, Englewood Hills, New Jersey, 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 substances, estimates of the critical volume were derived from the Joback method.

8.2.4 Normal Boiling Points

All values are believed to be experimental. In some cases, the values were slightly adjusted for vapor pressure.

8.2.5 Freezing Points

Where possible, quoted freezing points are experimental. No accurate method of estimation of compound freezing point is available. In the absence of experimental data, a rough estimate was derived from the Joback method.

8.2.6 Acentric Factors

The acentric factor is defined as

$$w = -\log10(Pvr \text{ at } Tr = 0.7) - 1$$

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.

$$Tc = Tb/(0.584 + 0.965 \text{ Sum}(Dt) - \text{Sum}(Dt)2)$$

$$Pc = 1/(0.113 + 0.0032 \text{ na} - \text{Sum}(Dp))2$$

$$Vc = 17.5 + \text{Sum}(Dv)$$

$$Tf = 122 + \text{Sum}(Df)$$

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)}   0.25 < Tr < 0.95

Vr(1) = Sum{bnTrn/(Tr - 1.00001)}   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

$$Zra = (PcVref / RTc)n$$

$$n = 1/(1 + (1 - Tref / Tc)m)$$

$$m = 2/7 \text{ or other empirical constant}$$

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,

$$VL = VLs - VLs,es + VL,es$$

$$VL = \text{specific volume at T and P}$$

$$VLs = VL \text{ at T and Ps from methods of this section}$$

$$VLs,es = VL \text{ at T and Ps from the equation of state}$$

$$VL,es = VL \text{ at T and P from the equation of state}$$

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

$$(CpL - Cpo)/R = 1.45 + 0.45/X + 0.25w(17.11 + 25.2X0.333/Tr + 1.742/X)$$

$$X = 1 - Tr$$

$$w = \text{acentric factor}$$

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

$$\ln(v/v^*) = 2.3026 \, J(1/Tr - 1)$$

$$J = (0.577 + Sum(D_i))0.5$$

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 < Tr < 1. The method also predicts the viscosity at the critical point (Tr = 1). The accuracy is normally better than 5% up to Tr = 0.92 with higher errors encountered as the critical point is approached. Overall this is an excellent estimation method for high-temperature liquid viscosity. The only serious limitation is the restricted range of applicability.

The form of the relation is

$$v = (f_0 + w.f_1)/A$$

with

$$w = \text{acentric factor}$$

$$f_0 = a_0 + b_0 Tr + c_0 Tr2$$

$$f_1 = a_1 + b_1 Tr + c_1 Tr2$$

$$A = 0.176x106 \; Tc0.1667/M0.5Pc0.667$$

$$a_0 = 2.648 \qquad a_1 = 7.425$$

$$b_0 = -3.725 \qquad b_1 = -13.39$$

$$c_0 = 1.309 \qquad c_1 = 5.933$$

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.

$$v = V_0/E(V - V_0) \text{ centipoise}$$

$$V = \text{liquid molar volume in cc/mol}$$

$$E = -1.12 + Vc/D$$

$$D = 12.94 + 0.1 \, M - 0.23 \, Pc + 0.0424 \, Tf - 11.58 \, Tf/Tc$$

$$V_0 = 0.0085 \, wTc - 2.02 + Vf / \{0.342(Tf / Tc) + 0.894\}$$

with

$$Tc = \text{critical temperature, K}$$

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:

$$k/k_s = 0.98 + 0.0079\ P_r T_r^{1.4} + 0.63\ T_r^{1.2} P_r/(30 + P_r)$$

k = conductivity at P

k_s = conductivity at Ps

8.8 Vapor Pressure

The vapor pressure is expressed in its reduced form

$$P_{vr} = P_v/P_c$$

Reduced vapor pressure varies from very low values at freezing point to unity at the critical point.

8.8.1 Published Correlations

The experimental correlations are commonly given in the following formats:

Wagner Equation

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

with

$$X = 1 - T_r$$

FKT Equation

$$\ln(P_v) = a + b/T + c\ln(T) + dP_v/T^2$$

Antoine Equation

$$\ln(P_v) = a + b/(T + c)$$

8.8.2 Gomez-Thodos Vapor Pressure Equation

Gomez-Nieto and Thodos give the following equation:

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

$$G = aH + bB$$

$$a = (1 - 1/T_{br})/(T_{br}^7 - 1)$$

$$b = (1 - 1/T_{br}^m)/(T_{br}^7 - 1)$$

$$H = T_{br}\ln(P_c/P_b)/(1 - T_{br})$$

For non-polar compounds,

$$B = -4.267 - 221.79/(H^{2.5}\exp(0.038\ H^{2.5})) + 3.8126/\exp(2272.33/H^3) + D$$

$$m = 0.78425\ \exp(0.089315\ H) - 8.5217/\exp(0.74826\ H)$$

$$D = 0$$

except D = 0.41815 for He, 0.19904 for H2, 0.02319 for Ne

For polar non-hydrogen-bonding compounds (e.g. ammonia and acetic acid),

$$m = 0.466\ T_c^{0.1667}$$

$$G = 0.08594\ \exp(0.0007462\ T_c)$$

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

8.8.2 Gomez-Thodos Vapor Pressure Equations

For polar hydrogen-bonding compounds (water, alcohols),

$$m = 0.0052\ M^{0.29} Tc^{0.72}$$

$$G = (2.464.M)\ \exp(0.0000098\ MTc)$$

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

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:

$$\ln(Pvr) = f(0) + wf(1)$$

$$w = \text{acentric factor}$$

$$f(0) = 5.92714 - 6.09648/Tr - 1.28862\ \ln(Tr) + 0.169347\ Tr^6$$

$$f(1) = 15.2518 - 15.6875/Tr - 13.4721\ \ln(Tr) + 0.43577\ Tr^6$$

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 (Long)

In modeling a generalized pipe network, it is possible to construct models that do not have a unique solution. A common occurrence of this is when a model contains one or more sections which are completely bounded by known flow rates. A series of examples with accompanying explanation will clarify situations where this occurs. The length of this topic is proportional to the frequency with which engineers get confused over the issue. It became apparent that an exhaustive, lengthy discussion was necessary for some engineers. Do not dismiss this topic too quickly because of its length.

Before beginning it will be helpful to consider an aspect of the philosophy of computer modeling. At the risk of stating the obvious, it must be recognized by the user that a computer model cannot calculate anything that cannot in principle be calculated by hand. All the computer does is accelerate the calculation process. Some users expect the model to generate independent information and become frustrated when the software requests additional information from them. But if the user was doing the calculation by hand that same information would still be required. The difference is that in the case of the hand calculation, the engineer would be forced to think through why the information is needed.

However, when the engineer interacts with AFT Impulse, they do not need to think at the same depth as in the hand calculation. So when AFT Impulse 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 Impulse terms, the system has two assigned flow junctions.

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

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

The model in Figure 1 can be built with AFT Impulse and if you try and run this model, AFT Impulse will inform you that it cannot run it because of the lack of a reference pressure. In AFT Impulse there are nine junctions that can act as a reference pressure: the reservoir, assigned pressure, spray discharge, pump with submerged option, and valve and relief valve after cracking (with the exit valve option) behave as reference pressures because that is their nature. Three other junction types, surge tank, gas accumulator, and assigned flow (when the steady-state flow is zero) have a special feature that allows them to act as reference pressures. This feature is found on each junction's Optional tab.

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 Impulse 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. Clearly they must have the same flow, so an inconsistency occurs. The basic reason the inconsistency is possible is because Figure 1 does not have a unique solution.

Figure 2 a-c shows three other model configuration possibilities. It is not possible to specify inconsistent conditions for any of the Figure 2 models, and they always have a unique solution no matter what input is specified by the user.
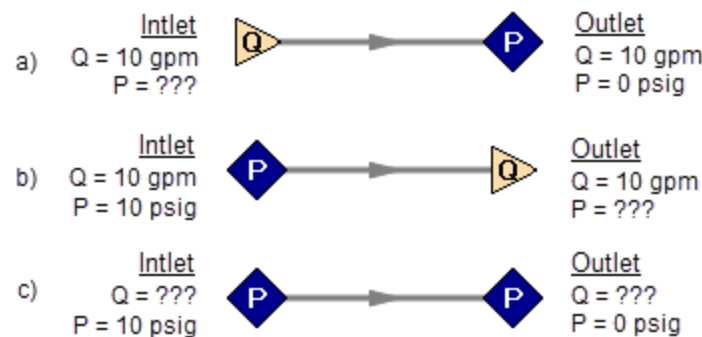
Figure 2: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.

The four models in Figures 1 and 2 a-c contain all logical possibilities. In all four cases there are four things we want to know. The pressure and flow at the inlet, and the pressure and flow in the outlet. In each case we know two of these four. The problem is that in the Figure 1 model the lack of a reference pressure makes it impossible to determine the inlet and outlet pressures, even though we know the flow. The three models in Figure 2 a-c all have at least one pressure and thus all four of the desired parameters can be determined. Table 1 summarizes this. Also note in Table 1 that in the Figure 2c case the flow can be determined, but it is by iteration.

Table 1: Four logical possibilities of boundary conditions for single pipe systems

Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 1		X		X	NO	No reference pressure, cannot calculate pressures
Figure 2a		X	X		YES	1 known flow, can calculate everything directly
Figure 2b	X			X	YES	1 known flow, can calculate everything directly
Figure 2c	X		X		YES	2 pressures, can iterate for flow

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

The pipe lengths, diameters and friction factors are not included in the remaining models because they do not influence the main point of this topic. It is assumed that these parameters can be obtained for each pipe and the resulting pressure drop calculated with standard relationships.

Consider the model in Figure 3. With three boundaries having a known flow, this clearly has the same problem as the model in Figure 1. No unique pressures at any location can be calculated.

A pressure at any one of the three boundaries in Figure 3 would be sufficient to allow a unique solution of the system. It is also possible that there could be two pressures and one flow, or three pressures. As long as there is one pressure a unique solution exists.

Outlet
Q = 7 gpm
P = ???

Intlet
Q = 10 gpm
P = ???

Outlet
Q = 3 gpm
P = ???
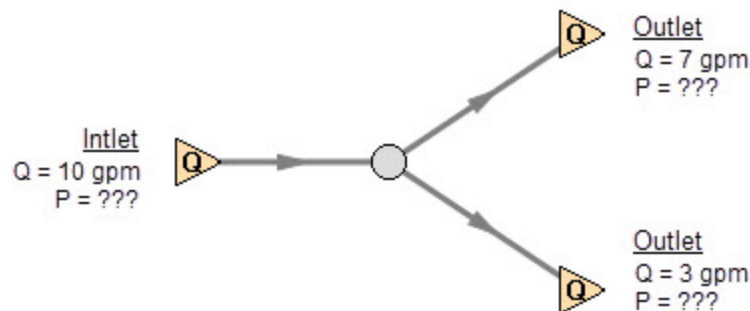
Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.

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 Impulse, you will get the message shown in Figure 5. AFT Impulse identifies the part(s) of the model where a known pressure is needed.
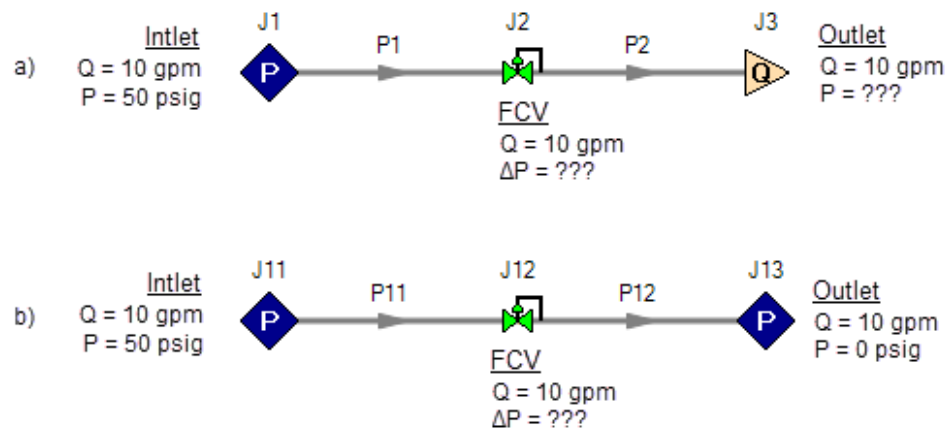
Figure 4a-b.: The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists.
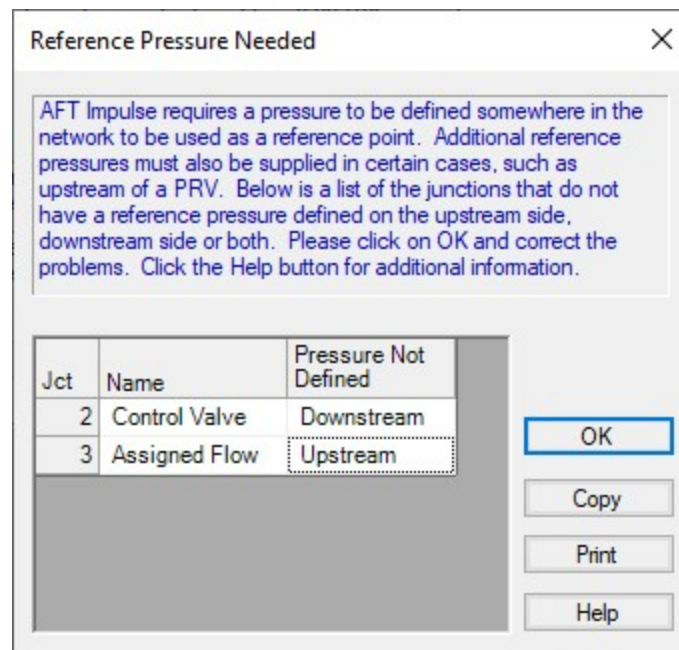
Figure 5: AFT Impulse message when you try to run the model shown in Figure 4a.

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 an assigned flow. The Pump with an assigned 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.

Intlet
Q = 10 gpm
P = 50 psig

Outlet
Q = 10 gpm
P = ???

Pump with Known (fixed) Flow
Q = 10 gpm
ΔH = ???
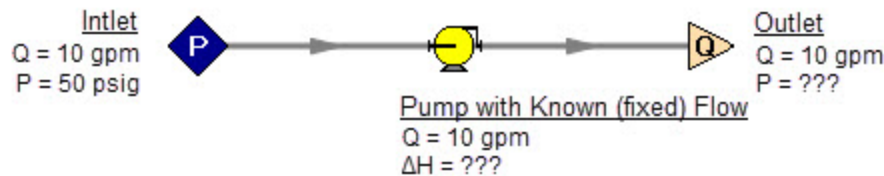
Figure 6: A pump modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists

Increasing in model complexity, Figure 7 shows some additional examples of models without a unique solution. In particular, Figure 7b has a pump with an assigned 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 FCVs 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 an assigned 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.
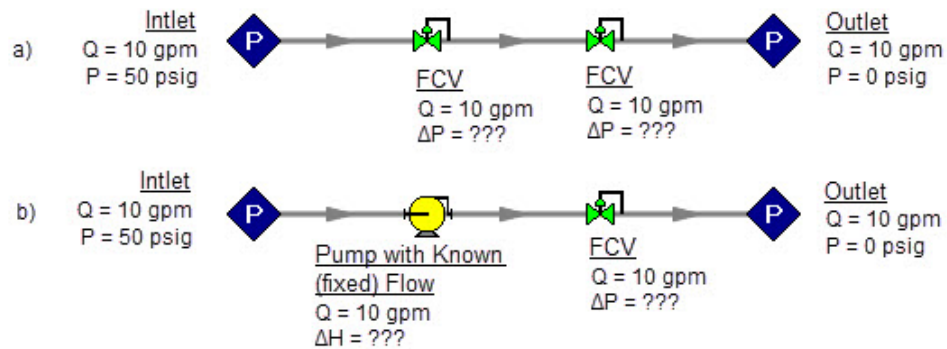
Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a pump modeled as an assigned flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions must be changed from a flow controlling device.

To avoid these kinds of problems in installed systems, a minimum 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.
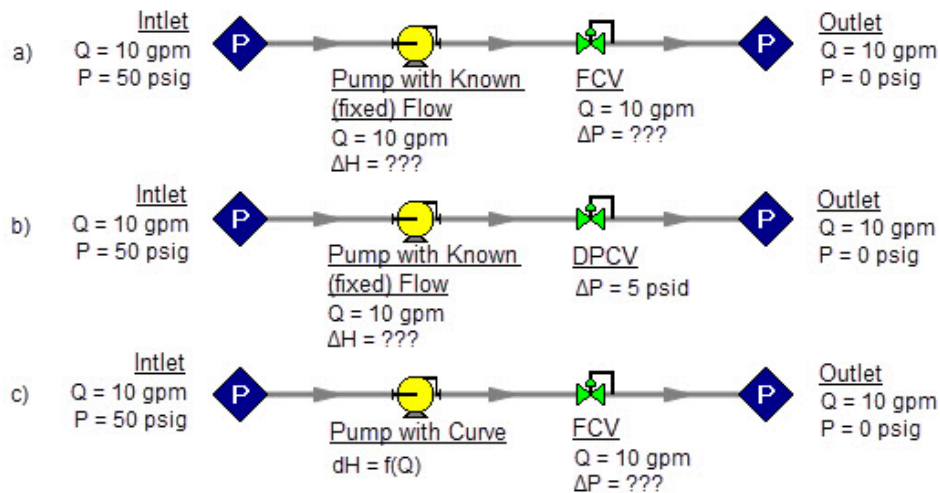
Figure 8a-c: The model at the top is the same as Figure 7b and does not have a unique solution. To size the pump, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the pump is sized, a pump curve exists, and 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 (see Figure 9). To size this pump, we apply the process described in Figure 8. We choose one of the FCVs, 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.
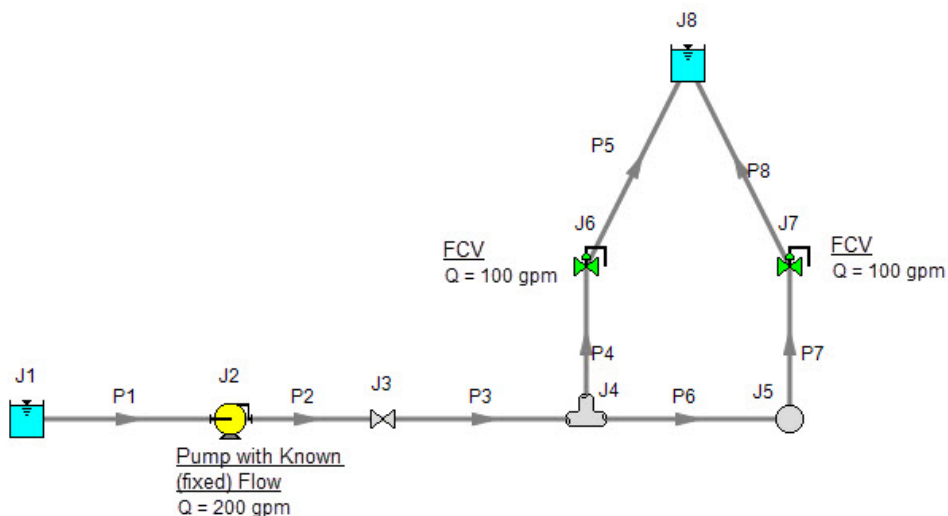
Figure 9: A pump with an assigned flow in series with two parallel FCVs. The model in its current form does not have a unique solution. To size the pump, first change the FCV at J7 to a PDCV.

This brings up the question, which FCV should one choose to turn into a PDCV? When FCVs are put in parallel, frequently the pipe design has one of the FCVs further away from the pump that the others. Because of the additional piping leading to this FCV, it will be the weakest link in

the chain of parallel FCVs 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 FCVs in the parallel system will have greater than the minimum pressure drop - thus satisfying the pressure drop requirement for those FCVs 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 FCVs 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 FCVs have identical pressure drops, any of the FCVs will serve as the PDCV.

The preceding process can be extended to cases where there are three or more FCVs in parallel, and also cases with more than one pump in parallel supplying the FCVs.

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 pgm 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 (113.5 ft.).

General	Warnings	Pump Summary	Valve Summary	Reservoir Summary						
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
3	Valve	REGULAR	200.0	27.76	0.000	0.00	66.50	N/A	0.000	Open
6	Control Valve	FCV	100.0	13.88	6.837	15.80	56.73	38.22	11.123	Open
7	Control Valve	PDCV	100.0	13.88	5.000	11.56	54.90	44.70	8.134	Open

General	Warnings	Pump Summary	Valve Summary	Reservoir Summary									
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	54.94	127.0	100.0	N/A	6.409	N/A	N/A	48.04	N/A

Figure 10a-b: Results from changing J7 in Figure 9 to pressure drop control valve

Pressure Control Valves

The problem of non-unique solutions can also occur when a Pressure Reducing Valve (PRV) or Pressure Sustaining Valve (PSV) is used. A PRV is used to control pressure downstream of the valve, while a PSV controls pressure at the valve inlet. One thing that PRVs and PSVs have in common with FCVs is that the pressure drop cannot be known ahead of time. The pressure drop depends on the balance of the pipe system.

Consider the simple system shown in Figure 11. With the pressure controlled downstream of the PRV, the pressure upstream is not known. Without knowledge of the PRV's pressure drop, the pressure upstream of the PRV cannot be determined and thus no unique solution exists.

Figure 11: If a PRV is downstream of a known flow the PRV pressure drop cannot be calculated. This model does not have a unique solution.

Figure 12 shows the possible cases with PRVs and PSVs, while Table 2 comments on the six cases. In summary, at least one known pressure is always needed on the side of a pressure control valve opposite of the controlled side.
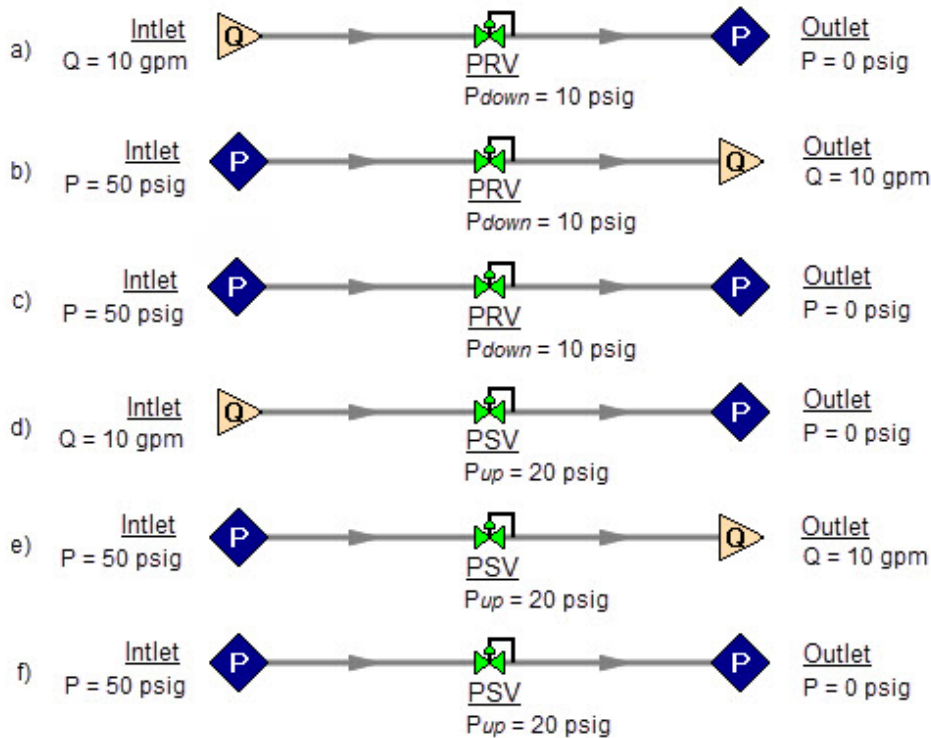
Figure 12a-f.: Six possibilities for PRV and PSV configurations. Cases A and E do not have unique solutions. See Table 2 for comments.

Table 2: Summary of six possibilities for PRV and PSV configurations.

| Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 12a - PRV		X	X		NO	No reference pressure upstream of PRV, cannot calculate
Figure 12b - PRV	X			X	YES	1 known flow, can calculate everything directly
Figure 12c- PRV	X		X		YES	2 pressures, can iterate for flow
Figure 12d - PSV	X	X	X		YES	1 known flow, can calculate everything directly
Figure 12e - PSV	X			X	NO	No reference pressure downstream of PSV, cannot calculate
Figure 12f - PSV	X		X		YES	2 pressures, can iterate for flow

If a pump is being sized in series with a PRV or PSV, modeling the pump as an assigned 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 FCVs.

In combination with the previous examples, it should be apparent how the non-unique solution problem can occur in more complicated systems. In all such cases, the reasoning will boil down to the same problems already discussed.

Closing Parts of a System

A model may be built which has sufficient information to obtain a unique solution. Then a user closes a junction and all of a sudden the problem of no unique solution occurs. This is shown in Figure 13. The top model has a unique solution and will run fine. In the bottom model the valve is turned off and the section of the system downstream of the valve has only known flows. No reference pressure exists in this section because it is isolated from the known pressures in the section on the left.
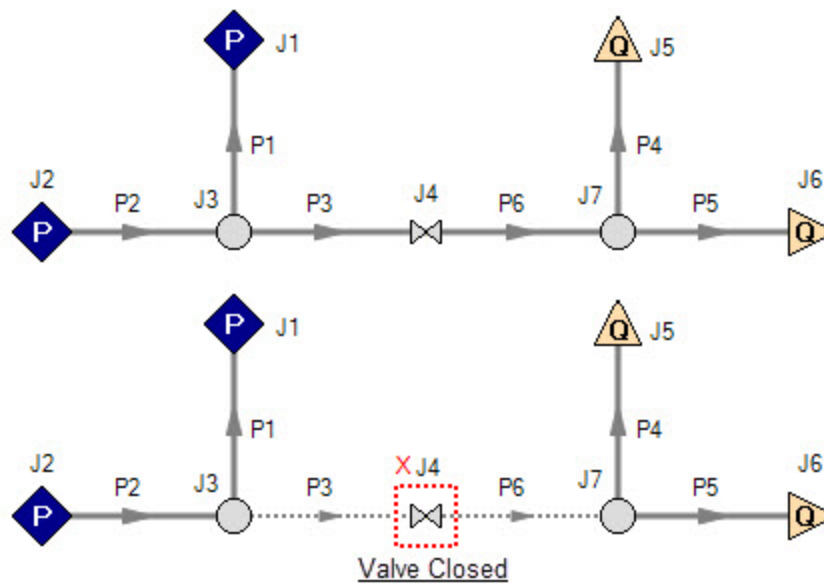
Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.

Steady State Considerations

There may be some situations - in particular for closed loop systems - where system components can act as a pressurizer and fix the pressure in the system during steady state when there is no flow. In this situation, it may not make sense to provide a reference pressure for the system using a traditional pressure junction. An example for this behavior would be a gas accumulator that pressurizes the system while there is no flow in steady state and then allow space to accommodate thermal expansion in the system.

In order to allow users to model that behavior, three different junctions have the ability to act as pressure reference points in steady state. These are the Assigned Flow junction, the Gas Accumulator junction, and the Surge Tank junction. The Properties Window pages for each of these junctions give a better description on how to properly define and use this feature.

For the Gas Accumulator and Surge Tank junctions, the ability to fix a zero-flow pressure is intended to allow users to better model a pressurizer component. During the transient simulation these junctions work to mitigate surge events by compressing or expanding the gas inside them as pressure waves travel through the system. During steady state, the initial conditions are fixed such that the pressure is a known, flow is zero, and the gas volume inside the junction is defined by the user as normal.

For the Assigned Flow junction, the ability to fix a zero-flow pressure is intended to allow users to model a situation where no flow is present in a pipeline and as a valve opens flow is introduced to the system. It is possible that during steady state, the reference pressure is known downstream of the closed valve, isolating the rest of the pipeline from a reference pressure. Defining this

zero-flow reference pressure would allow the user to model a system that might not otherwise have a valid reference pressure.

It should be noted that while normal pressure junctions are typically capable of acting as an infinite fluid source, these three zero-flow reference pressure options will not. As the option implies, these features can only be used in a situation where zero flow is present. During the transient simulation, these junctions no longer act as reference pressures, do not have the potential to act as infinite fluid sources at constant pressure, and the pressure in the system is determined solely through the Method of Characteristics equations.

Summary

AFT Impulse models always require one reference pressure, and may require more if the engineer uses control valves, pumps with assigned flows, or closes pipes or junctions. The actual number of reference pressures needed depends on the model, but AFT Impulse 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

Related Blogs

Reference Pressure Needed: Where It Comes from and How to Address it

Pipe Material Library References

Prior to AFT Impulse 6, pipe material properties were compiled from a variety of unofficial sources. Many of the original sources are now obsolete. Beginning in AFT Impulse 6, pipe material properties are taken directly from national/international standards such as: ASTM, ANSI, AWWA, and ASME. Several new material standards are being added. For example: DIN and JIS. With the addition of a significant number of new pipe material standards, the increased volume and scope necessitate splitting the pipe material library into several smaller independent libraries. All of the pipe material found in AFT Impulse 5 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 Manager can be used to "Connect" or Disconnect" each of the pipe material libraries. A list of available pipe libraries can be found below.

Note that while AFT attempts to provide this standard information to make defining pipe information easier for the user, the prudent engineer will always verify this information against manufacturer specifications. ASTM and ANSI standards exist for the thickness and diameter

specifications for some of these materials. Furthermore, AFT attempts to source reference information from reputable and authoritative sources. However, data for pipe roughness and thermophysical properties including density, thermal conductivity, modulus of elasticity, and Poisson's ratio have no single standard to follow. Thus, the user should always check that the pipe material inputs match the actual pipe material data for their system.

- **Copper Pipe - ASTM -**
  - Source = ASTM B42-10
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-33
- **Copper Tubing - ASTM -**
  - Source = ASTM B88-09
  - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-31; www.cranecopper.com.au/sizes/astmb88.asp
- **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

Critical Warnings, Warnings, and Cautions

A number of warnings may be given in the output when you run an AFT Impulse model. These warnings are intended to alert you about potential problems with the model or solution. There are different warning levels depending on the severity, but all warning types should be reviewed when they are displayed in the output.

1. CRITICAL WARNING – Very important and reflects a high possibility of incorrect results in the output. Output with critical warnings should be used only for error resolution purposes and never for engineering decisions.
2. WARNING – Offers important information about the converged solution or the model itself.
3. CAUTION – A cautionary warning is of less importance than a normal warning or critical warning. The included information may or may not be of interest to the user.
4. INFORMATION – An Information message is only intended to make the user aware of an advanced feature being used in the model. This advanced feature could cause incorrect results if not applied properly. If the user understands the feature then no action is required.

Any of the warning message types above can be double-clicked in the Output window to take you directly to the pipe/junction referenced in the message. Whenever any warnings occur that are of the WARNING or CRITICAL WARNING level, the General Results section of the Output Window will be displayed with the text in red.

The above links will take you to a list that includes most of the warnings you may receive, with a brief explanation given for each.

Related Topics

Critical Warnings

A Critical Warning message in AFT Impulse alerts users to serious issues in the model results. These messages should always be reviewed by the user, and results containing these messages should never be used for anything other than error resolution.

CRITICAL WARNINGS

- Artificial Transient - Initial pressure conditions at Pipe X Station Y are not steady (Error = Z%)
- FCV/PRV/PSV Could Not Control Flow/Pressure - Pressure Was Added...
- Junction X cavitates. Infinite pipe junctions do not support cavitation
- Junction X Had Reverse Flow - Pump Head Could Not Be Predicted
- Pipe Sequences of zero friction exist, flow rate cannot be calculated

- Setting relaxation lower than relative pressure/flow tolerance gives unreliable results
- The air volume entering through Junction X exceeds the volume of a connected pipe
- The following pipes have vapor volumes in computing stations greater than 100% of the liquid computing volume.
- These results do not represent a converged solution.
- Weir liquid level convergence problem
- Zero-Length Connector experienced cavitation

Artificial Transient Initial pressure conditions at Pipe X Station Y are not steady Error Z

A Critical Warning message in AFT Impulse.

AFT Impulse detected an artificial transient. If the magnitude is small you may be able to ignore it. If you want to ignore it you can change artificial transient criteria in the Artificial Transient Detection Panel.

In general, artificial transients will affect your results, and you should be careful to understand what is happening.

Related Topics

Related Blogs

Are you for real? Understanding and Troubleshooting Artificial Transients in AFT Impulse

FCV PRV PSV could not control

A Critical Warning message in AFT Impulse.

Full message is:

FCV/PRV/PSV could not control pressure/flow at Junction X. Pressure was added to achieve control flow. See Valve Summary for more information.

A flow or pressure control valve could not control pressure/flow and the setting is to Always Control (Never Fail). In order to achieve the control flow or pressure, the valve must add pressure and act like a pump.

This warning is useful for resolving operational problems in your system.

Junction X cavitates Infinite pipe junctions do not support cavitation

A **Critical Warning message** in AFT Impulse.

An infinite pipe boundary on an Assigned Pressure or Assigned Flow junction is designed to have zero reflections. Transient cavitation that occurs at the junction is a sure sign of reflections. The infinite pipe model is not appropriate.

Furthermore, any cavitation that would occur in the real pipe past the infinite pipe boundary but does not occur because that part of pipe is being neglected also means that the infinite pipe model is not appropriate.

Related Topics

Junction X Had Reverse Flow Pump Head Could Not Be Predicted

A **Critical Warning message** in AFT Impulse.

This message displays for centrifugal pumps when the flowrate goes backwards through the pump and the Manufacturer's Curve is being used for calculations, rather than defining a Four Quadrant Data Set. 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 Impulse

This message is displayed if all of the pipes in a flow path are defined as frictionless by using the zero-length connector option, or the "frictionless" friction model.

At least one pipe in every flow path needs to include frictional losses in order to perform flow calculations.

Setting relaxation lower than relative pressure flow tolerance gives unreliable results

A **Critical Warning message** in AFT Impulse.

Setting the relaxation to a value close to or lower than relative tolerance can lead to false convergence and invalid results. AFT Impulse alerts the user to this when inputting data in the Iterations Panel, and also alerts the user once the solution has been obtained.

The air volume entering through Junction X exceeds the volume of a connected pipe

A **Critical Warning message** in **AFT Impulse**.

The air volume entering through Junction X exceeds the volume of a connected pipe. The air valve can only model air in a connected pipe. Results with more than 100% air volume are unreliable.

If there are two connected pipes this warning will always apply to the pipe with the smaller volume.

Impulse's solution method assumes that the pipes are liquid full. If the volume of air in the system has not exceeded the volume of a single pipe station, then the liquid full assumption only has a minor impact on accuracy. However, if the volume of air entering the system exceeds the volume of a computational section then the liquid full assumption will cause a larger decrease in accuracy. Impulse uses an adjusted solution method to account for the air entering at the air valve, but does not account for air traveling past the pipes connected to the air valve. Any calculations past the point that the air volume exceeds the connected pipe volume will be invalid.

It is recommended to review the model inputs. Depending on how the air valve and physical system is configured it may be useful to consider changes such as modeling the air valve inline (with two pipes connected), resizing the valve, or moving the valve to a different location.

Modeling the air valve as inline helps to eliminate the potentially short connecting pipe, which will improve model run times and allow the air valve to be attached to longer pipes. Resizing or relocating the air valve can help to reduce the amount of air let into the system and can help reduce operational issues due to entrapped air.

Related Topics

The Following Pipes Have Vapor Volumes in Computing Stations Greather Than 100% of the Liquid Computing Volume

A **Critical Warning message** in **AFT Impulse**.

This Critical Warning is followed by a list of all pipe stations with Vapor Volumes greater than 100% of the liquid volume, shown in the format: Pipe *X*, Stations: *Y* (*Z*%).

Vapor volumes larger than the total volume in the pipe section are physically unrealistic and indicate that full column separation and extensive transient cavitation are likely occurring in the system at that point. In this situation, the cavitation models are no longer physically meaningful, and the conditions in the model may be outside of the single-phase assumption that Impulse is based on.

Users should explore the various cavitation topics - and in particular the Interpreting Cavitation Predictions topic - listed below in the Related Topics section to better understand what is causing these results, why they are possible, and how to interpret results in scenarios with this warning. If all else fails, users should treat systems with these results with extreme caution.

Related Topics

These results do not represent a converged solution

A **Critical Warning message** in **AFT Impulse**.

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.

Weir Liquid Level Convergence Problem

A **Critical Warning message** in **AFT Impulse**.

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.

Zero-Length Connector Experienced Cavitation

A **Critical Warning message** in **AFT Impulse**.

Full message text is:

The following Zero-Length Connectors experienced cavitation during the simulation, results are not valid

A Zero-Length Connector is a pipe model option which allows the user to define the pipe with no length associated with it. This tool is meant to help simplify a model with many short connections between system components and is not meant to capture some complex transient events including cavitation. Results shown with this message should not be used to make engineering decisions.

The user should assign lengths to the listed pipes and re-run the model to avoid cavitation in the Zero-Length Connector. Note that Zero-Length Connectors are commonly used to avoid run time concerns associated with having short pipes in a model, and replacing the Zero-Length Connector with a pipe of negligible length may cause unnecessarily long run times due to the sectioning process.

Warnings

A Warning message in AFT Impulse alerts users to issues in the model that will affect the model results. These warnings often offer important information about the converged solution or the model itself, and should always be reviewed before considering the results to be meaningful for

the system.

WARNINGS

- Air Valve Junction X steady-state pressure Y is below cracking pressure Z
- Chempak temperature and/or pressure is out of range for the following
- FCV could not control flow at Junction X
- Flow may not be balanced at Junction X
- Flow is into system at Reservoir Junction X...
- HGL, EGL and head loss results may not be meaningful for variable density systems
- Junction X cavitates. Cavitation at air valve junctions can cause anomalous results which may be incorrect.
- Junction X Exceeded Runout Flow Rate...
- Junction X Has Insufficient Pump NPSH
- Maximum peak to peak pressure limit has been exceeded (PFA)
- Negative added pressure by pump at Junction X
- Pipe force calculations are not reliable when cavitation occurs during the simulation
- Pipe X completely drained at Y seconds. Predicted results are not reliable after this time.
- Pipe X - Station Y pressure is below 10% vapor pressure margin. Pressure = Z. Note: There are also A other locations (PFA Only)
- PRV could not control pressure at Junction X
- PSV could not control pressure at Junction X
- Pump X had efficiency greater than 100%
- Pump X is using a Standard Pump Curve and... (Family of warnings)
- Static pressure lower than vapor pressure at inlet/outlet of pipe/junction X
- Surge Tank junction X completely drained at Y seconds. Predicted results are not reliable after this time.
- Tank Junction X completely drained at Y seconds. Predicted results are not reliable after this time.
- Temperature at inlet of pipe X is above/below max/min fluid temperature range
- The following pipes have vapor volumes in computing stations greater than 10% but less than 100% of the liquid computing volume
- Transient data was extrapolated at Junction X
- Transient static pressure lower than vapor pressure at Pipe X Station Y
- Unique solution not possible - Relief valve cycles

Air Valve Junction X steady state pressure Y is below cracking pressure Z

A **Warning message** in AFT Impulse.

Air Valves must be closed during steady-state, but if the steady-state pressure (shown as Y) is below the cracking pressure (shown as Z) they will open when the transient begins. This will cause an artificial transient in the system.

Related Topics

Chempak temperature and or pressure is out of range for the following

A **Warning message** in AFT Impulse.

The final solution had temperature and/or pressure conditions in the specified pipes obtained with Chempak physical property data which were outside Chempak's valid range. 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.

FCV at Junction X could not control flow

A **Warning message** in AFT Impulse.

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 at Junction X may not be balanced total Y

A **Warning message** in AFT Impulse.

This warning alerts you to the fact that the converged results may be incorrect. Once a converged solution is obtained, AFT Impulse checks all junctions to ensure a flow balance exists. It compares the imbalance to the flow tolerance you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to reduce your tolerances further and rerun the model.

If the magnitude of the flow out of balance is large, then you have a converged solution that is incorrect. This can occur in rare circumstances. When this happens, then it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and perform hand calculations on the flow balance at the affected junctions to verify whether an acceptable solution exists.

Flow is into system at Reservoir Junction X

A [Warning message](#) in AFT Impulse.

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 instead supplies whatever vapor is above the reservoir liquid. This warning alerts you to the fact that the reservoir had a pipe above liquid level. AFT Impulse assumes the intake fluid is the same as the reservoir fluid, but that is likely not the case. Results are probably invalid.

HGL EGL and head loss results may not be meaningful for variable density systems

A [Warning message](#) in AFT Impulse.

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.

Junction X cavitates. Cavitation at air valve junctions can cause anomalous results which may be incorrect

A [Warning message](#) in AFT Impulse.

Full message is:

Junction X cavitates. Cavitation at Air Valve junctions can cause anomalous results which may be incorrect.

An [Air Valve](#) junction is designed to allow air into the pipe but cannot keep track of vapor forming at the junction due to low pressure. If cavitation is predicted at the junction, the valve is likely undersized and is not preventing vacuum conditions.

Related Topics

Junction X Exceeded Runout Flow Rate

A [Warning message](#) in AFT Impulse.

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, and the system is operating past the runout flow rate. Thus the results are likely invalid, as the real pump may not be able to operate at that point.

Junction X Has Insufficient Pump NPSH

A Warning message in AFT Impulse.

The available pump NPSH (or NPSP) is less than that required. Thus the real pump will likely cavitate. Note that AFT Impulse does not model cavitation events inside junctions, including NPSH issues.

Maximum peak to peak pressure limit has been exceeded PFA

A Warning message in AFT Impulse. (PFA Module Only)

Full message is:

The maximum peak-to-peak pressure limit has been exceeded at X location(s).

The maximum peak-to-peak pressure occurred at Pipe Y – Station Z (dP=A, maximum dP=B)

In the PFA module, resonant frequencies are calculated for the modeled system. If these frequencies are caused by a positive displacement pump, the PFA module will automatically generate scenarios that evaluate problematic pump speeds. As part of the calculations when modeling these pump speeds, the PFA module will compare the peak-to-peak pressures generated with that pump speed, and it will produce a warning if these pressures violate the standards described in American Petroleum Institute 2010 Anex C sections A.1 and A.2, pages 67-68.

Note that the peak-to-peak pressure is defined as the difference between the maximum and minimum pressures

Negative added pressure by pump at Junction X

A Warning message in AFT Impulse.

Full message text is:

Positive Displacement Pump J2 Added Negative Pressure

A pump junction with an assigned flow rate had a pressure loss across the pump junction either in the steady state or transient system. This warning is meant to alert the user that the pump is not adding pressure as they may be expecting.

Pipe force calculations are not reliable when cavitation occurs during the simulation

A Warning message in AFT Impulse.

Full message reads:

Pipe force calculations are not reliable when cavitation occurs during the simulation.

A key component of AFT Impulse's transient hydraulic force calculations is the pressure differential across the force set. Specifically, the pressure differential caused by transient pressure waves in the system will cause a transient force to act on the system. Thus, the transient forces reported by AFT Impulse are only as accurate as the pressure waves predicted in the system.

Cavitation events including vapor volume formation, growth, and collapse all generate transient waves in the system, meaning they all cause transient forces to act on the system. Furthermore, cavitation inherently violates AFT Impulse's assumption that all pipes are 100% liquid full. Thus, the transient forces reported during and after cavitation events are only as reliable as the predicted cavitation events themselves.

If limited cavitation is predicted and determined to be accurate, the associated transient force results are likely also accurate. However, if cavitation is predicted to extend throughout large portions of the system or cause large vapor volumes to form, those results and associated transient forces may not be accurate, and should be used with caution.

Pipe X completely drained at Y seconds

A Warning message in AFT Impulse.

If pipes use the partially full feature, it is possible they will completely drain. AFT Impulse cannot properly model the pipes after this point, and subsequent results may not be valid. This warning alerts you that the specified pipes drained during the transient.

Pipe X pressure is below 10% vapor pressure PFA

A Warning message in AFT Impulse. (PFA Module Only)

Full message is:

Pipe X - Station Y pressure is below 10% vapor pressure margin. Pressure = Z. Note: There are also N other locations.

In the PFA module, resonant frequencies are calculated for the modeled system. If these frequencies are caused by a positive displacement pump, the PFA module will automatically generate scenarios that evaluate problematic pump speeds. As part of the calculations when modeling these pump speeds, the PFA module will calculate the lowest pressure margin above vapor pressure. If the relationship between the lowest pressure and vapor pressure shown in Equation 1 below (from American Petroleum Institute, 2010 Equation A.3) is not satisfied, this warning is displayed.

$$P_{min} > 1.1\, P_V \qquad\qquad (1)$$

PRV at Junction X could not control pressure

A **Warning message** in AFT Impulse.

This warning alerts you that the Pressure Reducing Valve at junction X was not able to control pressure, and backflow may have occurred.

PSV at Junction X could not control pressure

A **Warning message** in AFT Impulse.

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 Impulse.

Full message is:

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. For non-inertial models, only the output values for efficiency and power are affected. For inertial models, all pump results will be impacted.

This warning message could be caused by incorrect input data, or could be a result of using Four Quadrant pump curves:

1. 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.
2. For a pump defined using a Trip with Inertia or Startup with Inertia transient, check that the Total Rotating Inertia is entered correctly. The power data and reference density should still be checked as discussed above.
3. For a pump startup, verify that the provided torque data is the motor torque, and not the pump torque data, as is discussed in the Startup with Inertia topic.
4. If a Four Quadrant pump curve model is being used, check if the Four Quadrant efficiency curve is at or above 100% for any of the flowrates experienced in the transient.

Pump X is using a Standard Pump Curve and...

A series of Warning messages in AFT Impulse.

There are several different versions of this warning message that may occur when using a Standard Pump Curve (SPC) rather than a Four Quadrant Pump Curve (4QC). These include:

- **Pump *X* is using a Standard Pump Curve and had reverse flow. Head rise and power for negative flowrates were estimated using zero flow values. Predictions after reverse flow may not be reliable. A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection" for details.** This warning indicates that the pump junction is using a SPC and saw reverse flow. In that case the pump performance was estimated using the values entered for zero flowrate. This estimate may be acceptable when small amounts of reverse flow are seen through the pump. When reverse flow is present for longer durations or with larger magnitudes, the user should strongly consider using a 4QC to model their pump.

- **Pump *X* is using a Standard Pump Curve and head rise was negative. A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection" for details.** This warning indicates that the head rise across the pump junction was negative, either in the Steady-State solution or during the Transient Simulation. This behavior might have happened due either to the upstream and downstream pressures being set incorrectly, or to the pump operating far enough to the right on its curve that the head rise was negative. For centrifugal pumps, this behavior is only allowed if the "Allow Negative dP" Advanced Parameter in the Advanced Parameters Panel has been set to "Yes".

- **Pump *X* is using a Standard Pump Curve and head rise was predicted as negative and set to zero. A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection" for details.** This warning indicates that the head rise across the pump junction was predicted to be negative, but set to zero either in the Steady-State solution or during the Transient Simulation. This behavior could have happened due either to the upstream and downstream pressures being set incorrectly, or to the pump operating far enough to the right on its curve that the head rise was negative. Users should examine their pump input including pump curve data, pump curve fit, and pump curve reference density to ensure correct pump performance calculations. Fixing the head rise to zero in this situation is the default behavior of AFT Impulse. Users can allow the head rise to go negative by setting the "Allow Negative dP" Advanced Parameter in the Advanced Parameters Panel to "Yes". Positive Displacement Pumps will never see a negative head rise set to zero.

- **Pump *X* is using a Standard Pump Curve and power was negative. A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection" for details.** This warning indicates that the power requirements for the pump were estimated as negative. Users should examine their pump input including pump curve data, pump curve fit, and pump curve reference density to ensure correct pump performance calculations, and consider using a Four Quadrant Pump Curve if possible.

- **Pump *X* is using a Standard Pump Curve and flow exceeded End of Curve. If no End of Curve was specified, this is the largest entered flow. (*Y* gal/min, *Z* seconds) The head or power curve does not close the quadrant. Check the curve data at the calculated flows. A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection" for details.** This warning indicates that

not only is the flow through the pump past either the End of Curve setpoint or the last data point for the curve that the user has entered, but that the curves increase unbounded as flow increases rather than crossing the zero head axis (x-axis) and staying there. In the transient simulation, the time listed is the first time that this End of Curve value is exceeded. The pump may operate past this point later in the simulation as well. Note that if the pump is operating at a speed other than 100%, Impulse will apply the Affinity Laws to the End of Curve or last data point entered for the pump at 100% speed. Thus, this warning can be generated at flowrates lower than the specified End of Curve if the pump is running at low speeds. Users should check the pump performance both during steady state and the transient simulation to ensure that the pump is only operating on the portion of the curve prior to the flowrate where head and power begin to increase again. Alternatively, users could return to the Pump Configuration window and update their entered data or curve fit such that the head or power curves close the quadrant.

Static pressure lower than vapor pressure at inlet outlet of pipe/junction X

A Warning message in AFT Impulse.

This warning alerts the users that the steady-state static pressure is lower than the vapor pressure in the specified pipe or junction and that cavitation may occur. This warning is only shown if cavitation is not being modeled. If cavitation modeling is enabled (which is true by default), and cavitation exists in the steady state, the transient simulation will not proceed.

Related Topics

Surge Tank junction X completely drained at Y seconds

A Warning message in AFT Impulse.

If a surge tank completely drains, AFT Impulse cannot properly model the surge tank after this point. This warning alerts you that this happened during the transient.

Tank junction X completely drained at Y seconds

A Warning message in AFT Impulse.

If a Finite Tank completely drains, AFT Impulse cannot properly model the tank after this point, and results are unreliable. This warning alerts you that this happened during the transient.

AFT Impulse will continue to supply liquid to the system from the drained finite tank at the surface pressure specified for the junction.

Temperature at inlet of pipe X is above below max min fluid temperature range

A **Warning message** in AFT Impulse.

The temperature specified for the pipe is outside the range of valid temperatures. See the Fluid panel.

The following pipes have vapor volumes in computing stations greater than 10% but less than 100% of the liquid computing volume

A **Warning message** in AFT Impulse.

This Warning is followed by a list of all pipe stations with Vapor Volumes between 10 and 100% of the liquid volume, shown in the format: Pipe *X*, Stations: *Y* (*Z*%).

Vapor volumes between 10 and 100% of the liquid volume indicate that substantial transient cavitation is occurring at that point in the system. Impulse's cavitation models are meant to capture limited transient cavitation in a system, and have limited accuracy as vapor volume increases. In many systems, this amount of cavitation can be cause for concern, as it can lead to substantial waterhammer effects.

Users should explore the various cavitation topics - and in particular the Interpreting Cavitation Predictions topic - listed below in the Related Topics section to better understand what is causing these results, why they are possible, and how to interpret results in scenarios seeing this warning. If all else fails, users should treat systems with these results with caution, and incorporate large safety factors into their designs.

Related Topics

Transient data was extrapolated at Junction X

A **Warning message** in AFT Impulse.

The transient data at Junction X did not have time data that covered the entire time frame of the simulation. Any transient data after the final time data point was extrapolated using the final two values. You can enter additional time data points at Junction X to avoid extrapolation.

This extrapolation behavior can be changed in the User Options window on the Tools menu, under the Waterhammer Options.

Transient static pressure lower than vapor pressure at Pipe X Station Y

A **Warning message** in AFT Impulse.

This warning alerts the users that the transient static pressure is lower than the vapor pressure in the specified pipe and that transient cavitation may occur. This warning is only generated when cavitation is not being modeled.

Related Topics

Unique Solution Not Possible Relief Valve Cycles

A Warning message in AFT Impulse.

The relief valve specified toggles between its open and closed state, preventing Impulse from determining its actual 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.

Cautions

A Caution message in AFT Impulse is of less importance than a full warning. While some Caution messages may be left unresolved without affecting model results, they should never be ignored by the user.

CAUTIONS

- Assigned Flow at Junction X has a constant flow rate during the transient...
- Check Valve closed at Junction X
- Constant friction factors were used during the transient analysis
- Flow is negative through Junction X. Loss factors may be in error
- Head output does not have meaning for zero g systems
- Head rise across fixed flow pumps is referenced to system g level
- Junction X exceeded maximum event messages
- Junction X has a density change across it
- Junction X uses the Dimensional Check Valve Type Y which has a diameter of Z
- Pipe X length is shorter than the elevation change (between Junction Y and Z)
- Pipe X segment length Y is shorter than the elevation change.
- Pump X is using a Standard Pump Curve and flow exceeded End of Curve
- Pump with Check Valve closed at Junction X
- Some four quadrant data was based on estimated, not measured, data

- The air volume entering through Junction X exceeds 100% of a pipe station's computing volume
- The performance of Pump Junction X may be affected by fluid viscosity
- Transient data did not extend to Stop Time at Junction X

Assigned Flow at Junction X has a constant flow rate during the transient

A Caution message in AFT Impulse.

Full message is:

Assigned Flow at Junction X has a constant flow rate during the transient. During a transient the flow usually changes. This may not be an appropriate use of the junction.

An Assigned Flow junction fixes the flow at a boundary. When a transient occurs in the system, it is common to see all flows affected by the transient. This is a reminder that this junction may not be the appropriate junction to model a transient boundary condition. In general it is better to use Assigned Pressure or Reservoir junctions at boundaries. If there is a specific reason for using a fixed flowrate then this junction is the best way to do this.

Check Valve closed at Junction X

A Caution message in AFT Impulse.

This is a cautionary notice that a check valve closed.

Constant Friction Factors Were Used for the Transient Analysis

A Caution message in AFT Impulse.

Full message is:

Constant friction factors from the steady-state calculations were used during the transient analysis. The pipes listed below appear to have a wide range of actual friction factors and the predicted results may not be reliable. If a pipe is listed as closed, then the friction factor was estimated using a velocity of 5 feet/sec (1.52 meters/sec). It is recommended to use a Time Varying friction model which is defined in the Transient Friction panel.

This caution message is intended to alert users that the model results may not be reliable, as the transient analysis is using the initial steady-state friction factors. In some cases, this estimate may be appropriate, while in others it may not. Below the text of the caution, the steady-state friction factor being used and the potential maximum transient friction factor are listed. This can help give users an idea of how well this transient friction model fits their system. In the case that a pipe is closed or has no flow through it in the steady state, the friction factor is estimated using a velocity of 5 feet/sec (1.52 meters/sec).

When this message is shown, it is recommended that a Time Varying friction model is used for the transient simulation. This option can be specified in the Transient Friction Panel.

Flow through Junction X is negative Loss factors may be in error

A Caution message in AFT Impulse.

This warning alerts you to flow in your junctions which is in the opposite direction from the connecting pipes reference directions. Any loss information you specified for junctions may be in error because it was defined assuming flow in the other direction.

Head output does not have meaning for zero g systems

A **Caution message** in AFT Impulse.

Full message is:

Head loss, head rise, hydraulic gradeline, and energy gradeline output does not have meaning for zero g systems.

If the user sets the gravity to zero presumably for a space application then the concept of head and gradeline lose meaning. This can be set in the Environmental Properties Panel. For example see Equation 1 in Network Solution Methodology and see that a zero g will leads to a meaningless HGL (hydraulic gradeline).

Users should ignore all head loss, head rise and gradelines displayed in the output and focus on pressure results.

Head rise across assigned flow pumps is referenced to system g level

A **Caution message** in AFT Impulse.

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 the Environmental Properties Panel then the head rise at a pump is referenced to that g level on not 1 g.

Junction X exceeded maximum event messages

A **Caution message** in AFT Impulse.

To limit how much data is displayed in the output there is a maximum number of event messages are displayed for a junction. This limit was reached. This means that there are some events that happened but no event message is actually displayed. The maximum event messages setting can be changed in the Output Control window on the Format & Action tab.

Junction X has a density change across it

A **Caution message** in AFT Impulse.

Full message is:

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.

This warning is shown when the user has selected variable fluid properties in the Variable Fluids panel, and the density of the fluid is changing between pipes. AFT Impulse 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.

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 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 a pipeline carrying fluids of different properties such as oil of different density. 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.

Junction X uses the Dimensional Check Valve Type Y which has a diameter of Z

A **Caution message** in AFT Impulse.

Full message is:

Junction X uses the Dimensional Check Valve Type of Y which has a diameter of Z. The downstream Pipe A has a diameter of B. A Dimensional Valve Type with a diameter of B or a Non-Dimensional Valve Type is recommended.

The Dimensional Check Valve options are based on actual valves for which empirical data is available. These dimensional options should only be used when they match the actual size of the valve in the real system. If the dimensional option does not match the actual valve size, the user should choose a more appropriate dimensional option or use the non-dimensional options.

Pipe X length is shorter than the elevation change between Junction Y and Z

A **Caution message** in AFT Impulse.

This caution alerts you to a possible inconsistency in your input. When this warning occurs, you have defined a pipe with a length that is shorter than the elevation change between its connecting junctions. 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 Impulse.

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.

Related Topics

Pump X is using a Standard Pump Curve and flow exceeded End of Curve

A **Caution message** in AFT Impulse.

Full message is:

Pump X is using a Standard Pump Curve and flow exceeded End of Curve. If no End of Curve was specified, this is the largest entered flow. (Y gal/min at Z% Pump Speed = α gal/min, β seconds) A four quadrant model may be appropriate - see the Help Topic "Four Quadrant Data Set Selection for details.

This caution indicates that the flow through the pump found to be past either the End of Curve setpoint or the last data point for the curve that the user has entered. In the transient simulation, the time listed is the first time that this End of Curve value is exceeded. The pump may operate past this point later in the simulation as well.

Users should ensure the pump curve and curve fit being used are still valid at this flowrate, and past this flowrate if the pump operates at higher flowrates later in the transient. Note that if the pump is operating at a speed other than 100%, Impulse will apply the Affinity Laws to the End of Curve or last data point entered for the pump at 100% speed.

There are several similar Warning messages that may occur when using a Standard Pump Curve (SPC). Users can read more about those warning messages Here.

Pump with Check Valve closed at Junction X

A **Caution message** in AFT Impulse.

Pump junction X was specified with an internal check valve which closed during the simulation.

Some four quadrant data was based on estimated not measured data

A [Caution message](#) in AFT Impulse.

Some of the source data for [four quadrant curves](#) does not complete the four quadrants. In such cases data was estimated/interpolated in the missing quadrants. If this message is encountered the user may want to run with another four quadrant source to see if results change.

The air volume entering through Junction X exceeds 100% of a pipe station's computing volume

A [Caution message](#) in AFT Impulse.

The air volume entering through Junction X exceeds 100% of a connected pipe station's computing volume.

The pipe station's computing volume is determined by the section length calculated in the [Sectioning panel](#) and the cross-sectional area of the connected pipe. If two pipes are connected the average of the pipe station volumes of each of the connected pipes is used.

Impulse's solution method assumes that the pipes are liquid full. If the volume of air in the system has not exceeded the volume of a single pipe station, then the liquid full assumption only has a minor impact on accuracy. However, if the volume of air entering the system exceeds the volume of a computational section then the liquid full assumption will cause a larger decrease in accuracy. Impulse uses an adjusted solution method to account for the air entering at the air valve, but caution should be used when making decisions based on these results.

Related Topics

The performance of Pump Junction X may be affected by the fluid viscosity

A [Caution message](#) in AFT Impulse

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 Impulse.

The transient data at Junction X did not have time data that covered the entire time frame of the simulation. Any transient data after the final time data point was assumed to be at the final data value. You may want to enter additional time data points at Junction X.

This extrapolation behavior can be changed in the User Options window on the Tools menu.

Information

An Information message in AFT Impulse provides suggestions related to a feature that is used in the model. It is recommended to follow the suggestions provided to achieve higher accuracy results.

INFORMATION

The following pumps use a Four Quadrant Curve and...

A series of Information messages in AFT Impulse

There are several versions of this information message that may appear when using a Four Quadrant Pump Curve. Each message is intended to remind the user that using a Four Quadrant Curve introduces inherent uncertainty to the model even if it allows Impulse to better capture centrifugal pump behavior outside of standard pump operating zones. The user should always check to ensure that the Four Quadrant Curve used in the model is a good choice for their pump.

These information messages include:

- **The following pumps are using an estimated Four Quadrant data set. Sensitivity studies are recommended for a thorough understanding of possible transient behavior. See Help topic "Four Quadrant Data Set Selection" for more information.**

  This information message is generated when the four quadrant data set is selected by the user. Each data set will give slightly different results, and it is important the user evaluate multiple data sets to understand the range of possible pump behavior.

- **The following pumps are using an Automatically Selected Four Quadrant data set. The automatically selected data set should be confirmed as reasonable for this simulation. Sensitivity studies are recommended for a thorough understanding of possible transient behavior. See Help topic "Four Quadrant Data Set Selection" for more information.**

  This information message is generated when the four quadrant data set is selected automatically by Impulse. The selection is based on which data set has the closest specific

speed to the original pump curve defined by the user. This selection process is generally regarded as the best way to select a four quadrant data set, but the user should evaluate multiple data sets to understand the range of possible pump behavior.

- **The following pumps are defined with a Four Quadrant Curve based on the Best Efficiency Point. While this is the recommended setting for most simulations, the steady conditions will likely vary from the Standard Pump Curve. See Help topic "Four Quadrant Dimensional Reference Point" for more information.**

  This information message is generated when the four quadrant curve is dimensionalized using the pump's Best Efficiency Point (BEP). This dimensionalization means the Standard Pump Curve and the Four Quadrant Curve will intersect exactly at BEP. Impulse uses the Four Quadrant Curve for the steady-state and transient solutions, so the steady-state results will likely not fall exactly on the original Standard Pump Curve. This behavior is expected, and using BEP as the dimensional reference point is recommended, but the user should be aware of how it impacts the steady-state results.

- **The following pumps are defined with a Four Quadrant Curve based on the Steady-State Operating Point. If the initial operating flow is far from BEP, significant distortion in the resulting pump curves is possible, potentially impacting transient results. Check the fit of the Four Quadrant curves. See Help topic "Four Quadrant Dimensional Reference Point" for more information.**

  This information message is generated when the four quadrant curve is dimensionalized using the operating point of the system during steady-state. The Standard Pump Curve is used for the steady-state simulation, and the Four Quadrant Curve is dimensionalized based on those results. This approach allows the pump to operate on the original Standard Pump Curve in steady-state. However, if the Four Quadrant Curve is dimensionalized at the extreme ends of the Standard Pump Curve, away from BEP, the Four Quadrant Curve may not match well with the Standard Pump Curve, The user should check to ensure a good Four Quadrant Curve is used.

- **The following pumps are defined with a Four Quadrant Curve based on the Steady-State Operating Point, but the initial speed or flow was zero. The Dimensional Reference Point was automatically changed to Best Efficiency Point. See Help topic "Four Quadrant Dimensional Reference Point" for more information.**

  This information message is generated when the four quadrant curve is dimensionalized using the operating point of the system during steady-state, but the pump is at zero flow or zero speed. In this situation, the steady-state operating point cannot be used to generate a valid for quadrant curve, so the Best Efficiency Point was used instead.

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

Related Blogs

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

Reducing Run Time

The below topics discuss several common model configurations that can cause Impulse model run times to be longer than expected. Select a topic below for information on why each configuration increases the run time, and suggestions on how each configuration can be avoided.

If one of the below configurations is detected by Impulse and the run time is relatively long, Impulse will present a message to the user showing which of the below configurations may be applicable.

Note that the configurations listed here as causing long run times may not be inaccurate for the system being modeled. However, they are all likely inefficient modeling choices for transient analysis - especially for preliminary iterations.

The configurations discussed below are not meant to capture every possible model configuration which can cause long model run times in Impulse. Many other factors can also cause long run times. Following the suggestions listed below may help reduce run time in any model, though it may not be practical to implement the listed suggestions for all models.

Related Topics

Related Blogs

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

[You Should Combine Your Pipes](#)

The model has both very long and very short pipes

A model with both long and short pipes requires a larger number of pipe sections in order to perform calculations. The more pipe sections in a model, the more computations the solver needs to perform, which results in a longer run time.

How are short and long pipe lengths contributing to long run times?

To run a transient simulation, AFT Impulse sections pipes to apply the [Method of Characteristics](#) solution to the waterhammer equations. In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - generally one. AFT Impulse uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

When a model has both very long and very short pipes, the short pipes determine the controlling pipe length, and the long pipes must be divided into many sections to properly apply the Method of Characteristics. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times.

Consider a model that consists of two pipes: a very short pipe, such as 1 meter, and a very long pipe, such as 10,000 meters. The 1 meter pipe will be the controlling pipe, and the 10,000 meter pipe must be broken into 10,000 sections. The model thus requires 10,001 sections in order to perform the calculations, which results in a long run time. Increasing the short pipe's length artificially from 1 to 10 meters decreases the number of sections required by a factor of 10, reducing run time by a factor of 100. While the length increase may also increase the error in the short pipe, experience shows that the error is usually negligible in a liquid transient context.

What can the user do?

Several different approaches are available to reduce run time. These approaches can work in parallel, and it is up to the engineer to make appropriate judgments as to which is best for the model.

Consider using the [zero-length](#) connector for very short pipes. Oftentimes two fittings in a system are connected with a very short pipe or are welded together directly. In AFT Impulse, it is common for users to attempt to model these connections using pipes with very short defined lengths. These short pipes offer little accuracy improvement and result in a large number of sections for the solver to calculate. This also causes significant time for the model to run. The zero-length connector option allows AFT Impulse to exclude the pipe from the sectioning process since it represents a lossless connection between two junctions. Zero-length connectors can be used in up to two sequential pipes, but intermediate branching flow is not allowed between two zero-length connector pipes.

Consider modeling [fittings and losses](#) in pipes and combining multiple short pipes. In the Pipe Properties window, use the Fittings & Losses tab to add in fittings to the pipe instead of using junctions on the workspace. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings. For example, a user can add valves and bends to a pipe, and the solver will still apply the losses directly to the pipe during the calculations. By removing these junctions and combining pipes in the workspace, many short pipes between these fittings are eliminated. Reducing and combing short pipes reduces the number of sections in the model and the overall run time.

Consider adding artificial length to short pipes. Adding length to short pipes increases the length of the controlling pipe, which results in fewer sections for the other pipes in the model. Reducing the overall number of sections in the model improves run time. Using pipe lengths that are common multiples, such as 5 feet, 10 feet, 15 feet, etc. provides improved sectioning since short pipes can be perfectly divided into longer pipes.

Do not include extremely long pipes unless you are directly analyzing the transient behavior in those pipes. Consider using the [Infinite Pipe](#) feature in the Assigned Pressure and Assigned Flow junctions. The infinite pipe feature prevents the junction from reflecting pressure waves, which can be useful in modeling the part of a pipe system which is so long that its communication time prevents wave reflection interactions with the rest of the system during the simulation.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

The model has many pipes that can be combined

Users often model losses in AFT Impulse using junctions with no transient behavior, such as a bend, static valve, or orifice, between two pipes of identical diameter. While this may help show where these losses are occurring in the overall pipe system, it can result in extra pipes which can have a detrimental effect on pipe sectioning and run time. Combining pipes and modeling the losses within the pipe itself can improve sectioning and run time while maintaining accuracy.

How are pipes that can be combined contributing to long run times?

To run a transient simulation, AFT Impulse sections pipes to apply the Method of Characteristics solution to the waterhammer equations. In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - generally one. AFT Impulse uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

Each pipe in a model must be sectioned individually, meaning section count must increase with pipe count. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times. Reducing the number of pipes in a model is often an effective way to reduce run time.

Consider a model that includes a pipeline that is 100 meters long, and has 9 bends in it. If those 9 bends are modeled as unique junctions, the pipeline must be modeled as 10 individual pipes. Each pipe must have at least 1 section, meaning the pipeline requires at least 10 sections. If the pipeline is modeled as a single pipe, and the 9 bends are modeled as additional pipe friction using the Fittings & Losses tab in the Pipe Properties window, the number of sections in the pipeline could be reduced from 10 to 1, in turn reducing the run time by a factor of 100.

What can the user do?

The Fittings & Losses tab in the Pipe Properties window can be used to add losses directly to pipes to replace individual junctions on the workspace. This process can be done manually, or automatically using tools provided by AFT Impulse.

Select the pipes to be combined and the junctions between those pipes on the Workspace, and choose the Join Selected Pipes/Jcts from the Edit menu. This option will combine the selected pipes into one pipe, add the losses from the junctions into the Fittings & Losses tab, and use the

junction elevation and previous pipe lengths to fill out the Intermediate Pipe Elevations table on the Optional tab of the Pipe Properties window.

A similar tool can be found in the Sectioning panel in Analysis Setup to the right of the Select Sectioning interface. This tool shows the number of pipes that could be combined and by clicking the Combine Pipes button, you can select which pipes to include or exclude in the combine process. After clicking OK, the pipes are combined and you can immediately click the Update Results button on the Sectioning panel to resection the pipes in the model.

Related Topics

Related Blogs

[Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning](#)

[You Should Combine Your Pipes](#)

The model has many unique pipe lengths

A model with many unique pipe lengths often require a large number of sections in order to correctly use the [Method of Characteristics](#). A large number of pipe sections in a model results in long run times and typically offers little improvement in accuracy.

How are unique pipe lengths contributing to long run times?

To run a transient simulation, AFT Impulse sections pipes to apply the [Method of Characteristics](#) solution to the waterhammer equations. In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - generally one. AFT Impulse uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

A model with many unique pipe lengths often requires the controlling pipe to be broken into multiple sections in order to divide all pipes into uniform sections and uniform time steps. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times.

What can the user do?

Consider adjusting the lengths of the pipes in the model such that short pipes share a common pipe length and that longer pipes can be perfectly divided by the controlling pipe. Using exact pipe lengths can enhance calculation accuracy in many applications. However, transient simulations have many inherent uncertainties, and precise modeling of pipe lengths may not offer a noticeable improvement. Rounding short pipe lengths such that the lengths are perfectly divisible by the controlling pipe length, introduces minimal inaccuracy while significantly reducing the

number of sections in the model and resulting run time. Be sure to perform sensitivity studies and confirm the steady-state results with these modifications.

For instance, consider a model with five pipes of lengths 1.01, 2.31, 3.75, 6.00, and 1000 meters. All pipes have unique lengths and none are perfectly divisible into another. To section these pipes within the acceptable variance limits, the controlling pipes needs to be split into two sections, and over 2100 total sections are required to perform the calculations. Rounding the four shortest pipes to 5 meters increases the total pipe length of the model by only 0.7%. In return, the number of pipe sections is reduced by a factor of 10 and the run time is reduced by a factor of 100! This 0.7% change in total pipe length can be considered negligible for any reasonable transient simulation - there are greater uncertainties than this in almost all studies.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

The controlling pipe has a large number of sections

The controlling pipe is the shortest pipe in the model and is used to determine the number of computational sections needed in all other pipes in the model. If the controlling pipe has a large number of sections the other pipes will also require more sections, which results in more calculations needed and a longer run time.

How is controlling pipe sectioning contributing to long run times?

To run a transient simulation, AFT Impulse sections pipes to apply the Method of Characteristics solution to the waterhammer equations. In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - generally one. AFT Impulse uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

When the controlling pipe is broken into more than one section, the section length in all pipes is shortened, and all pipes must contain more sections as well. The run time is proportional to the square of the number of sections, meaning many sections in a model will often cause extremely long run times.

Consider a model that takes 10 minutes to run with one section in the controlling pipe. With 2 sections in the controlling pipe the model will take about 40 minutes, and with 3 sections in the controlling pipe the model will take about 90 minutes.

What can the user do?

There are multiple ways to decrease the number of sections in the controlling pipe.

Consider using the minimum number of sections within the suggested maximum variance. The maximum wavespeed variance for a given number of sections in the controlling pipe is shown in the tables on the Sectioning panel and is just referred to as "Variance". AFT Impulse will automatically recommend the smallest number of sections which limit the wavespeed variance to 10% or less. The user should generally choose Impulse's automatic sectioning selection since the Method of Characteristics used in AFT Impulse has little increase in accuracy when more sections are used. Some exceptions include when the wavespeed variance is greater than 10% for a given sectioning scheme, when transient cavitation occurs during the transient simulation, or when the partially full pipe feature is used. Engineering judgement should be used to determine if more sections are needed in the controlling pipe.

Consider reducing the number of unique pipe lengths. Models with numerous unique pipe lengths will often require the controlling pipe to be broken into multiple sections to satisfy the sectioning requirements for the Method of Characteristics. Artificially adjusting those pipe lengths such that all are divisible by the controlling pipe length can reduce the number of sections in the controlling pipe, and thus reduce the total number of sections in the model along with the model run time. For more information, see the topic discussing Models with many unique pipe lengths.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

The model has an unusually short pipe

A model with a very short pipe requires a larger number of pipe sections in order to perform calculations. The more pipe sections in a model, the more computations there are that need to be performed, which results in a longer run time.

How are pipe lengths contributing to long run times?

To run a transient simulation, AFT Impulse sections pipes to apply the Method of Characteristics solution to the waterhammer equations. In any system there will be a single controlling pipe. The controlling pipe is the pipe with the fewest sections - generally one. AFT Impulse uses the length of a section in the controlling pipe to divide all pipes in the model into sections and determine a uniform time step for the transient simulation.

When the controlling pipe is unusually short, the other pipes in the model must be divided into many sections to apply the Method of Characteristics. The run time is proportional to the square

of the number of sections, meaning many sections in a model will often cause extremely long run times.

Consider a model that has one 0.05 ft long pipe and 100 pipes that are each 10 ft long. Each 10 ft pipe would require 200 sections, requiring 20,000 sections overall. Modeling the very short pipe as 0.1 ft would reduce the number of sections by a factor of 2, which would potentially decrease the run time by a factor of 4. Alternatively if the very short pipe is not considered in the sectioning using one of the methods discussed below then the controlling pipe length would become 10 ft. With one section in each pipe this would decrease the number of sections from 20,000 to 100, decreasing the run time by a factor of 40,000.

What can the user do?

There are several different approaches to reduce run time. These approaches can work in parallel, and it is up to the engineer to make appropriate judgments as to which approach is best for the model.

Consider using the [zero-length](#) connector for very short pipes. Oftentimes two fittings in a system are connected with a very short pipe or are welded together directly. In AFT Impulse, it is common for users to attempt to model these connections using pipes with very short defined lengths. These short pipes offer little accuracy improvement and result in a large number of sections for the solver to calculate. This also causes significant time for the model to run. The zero-length connector option allows AFT Impulse to exclude the pipe from the sectioning process since it represents a lossless connection between two junctions. Zero-length connectors can be used in up to two sequential pipes, but intermediate branching flow is not allowed between two zero-length connector pipes.

Consider modeling [fittings and losses](#) in pipes and combining multiple short pipes. In the Pipe Properties window, use the Fittings & Losses tab to add in fittings to the pipe instead of using junctions on the workspace. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings. For example, a user can add valves and bends to a pipe, and the solver will still apply the losses directly to the pipe during the calculations. By removing these junctions and combining pipes in the workspace, many short pipes between these fittings are eliminated. Reducing and combing short pipes reduces the number of sections in the model and the overall run time.

Consider adding artificial length to short pipes. Adding length to short pipes increases the length of the controlling pipe, which results in fewer sections for the other pipes in the model. Reducing the overall number of sections in the model improves run time. Using pipe lengths that are common multiples, such as 5 feet, 10 feet, 15 feet, etc. provides improved sectioning since short pipes can be perfectly divided into longer pipes.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

The simulation has a large number of time steps

A model with a large number of time steps will require a large number of calculations. The more calculations that need to be performed, the longer the run time.

How are the time steps contributing to long run times?

Impulse sections pipe to apply the Method of Characteristics solution method. The Method of Characteristics requires a uniform time step to be used for all pipes in the model, which is determined during the sectioning process using the section length and adjusted wavespeed. On average the wavespeed is typically on the magnitude of 10^3, producing a time step on the magnitude of 10^{-3}.

A time step of 0.001 seconds means that a transient simulation of 1 minute will have 60,000 total time steps. Similarly, a transient simulation of 1 hour will have 3.6 million time steps. Run time is roughly proportional to the number of time steps, and models with large numbers of time steps will generally have long run times. In addition, models with higher wavespeed may require shorter time steps, increasing the number of time steps required.

What can the user do?

There are several different approaches to reduce run time when the number of time steps is larger than expected.

Consider reducing the simulation duration. The simulation duration is defined on the Simulation Mode/Duration panel. For a simple system, the full effect of the waterhammer cycle occurs over twice the communication time, where the communication time is the time it takes a pressure wave to travel the length of the system and back (length of system / wavespeed). Determining the communication time can become more complex for systems with branching paths or major reflection points in the path. However, limiting the simulation duration to 2 times the communication time may be a good starting point. If multiple transient events occur the simulation duration should be extended to end after the waterhammer cycle has completed for the last transient event.

In addition, for a piping network attached to a very long pipe it may not be necessary to include the length of the very long pipe in the communication time calculation. Use your engineering judgement on whether the reflections from the pipeline will be important to consider.

Consider reducing the number of unique pipe lengths. Models with numerous unique pipe lengths will often require the controlling pipe to be broken into multiple sections to satisfy the sectioning requirements for the Method of Characteristics. Artificially adjusting those pipe lengths such that all are divisible by the controlling pipe length can reduce the number of sections in the controlling pipe, increase the length of each time step, and reduce the total number of time steps

required for the simulation. For more information, see the topic discussing Models with many unique pipe lengths.

Consider lengthening the controlling pipe. The controlling pipe length is used to determine the time step for the simulation, setting the time step equal to the section length divided by the wavespeed. Artificially lengthening the controlling pipe means the time step for the simulation can be longer, and fewer time steps are required for the simulation.

Related Topics

Related Blogs

Analyzing Transients Faster - a Hacker's Guide to Pipe Sectioning

More time-consuming calculations are being used

Many transient simulations model systems which are relatively simple at the beginning of the simulation. Pipes are liquid full, pumps are running at full speed, and relief and check valves are in their steady-state conditions. However, as transient events occur in the system, transient cavitation, changes in pump speed, and relief valve or check valve chattering can occur. These and other similar phenomenon can require additional calculations which slow down the solver and increase the run time. Events such as cavitation cannot be viewed while the model is running, but can be observed in the output.

In many cases, these events are indicative of the behavior that would occur in the actual system. It is therefore important to run the model with some potentially time intensive calculation methods to gain an understanding of the real system. If you determine that the previously mentioned phenomenon are unrealistic to the physical system, then steps should be taken to review the model inputs to prevent the phenomenon from occurring during the transient, which will in turn reduce the run time of the AFT Impulse scenario.

Related Topics

Transient Solver Errors

A transient solver error will be shown during the transient solution if Impulse encounters a calculation error which would invalidate the calculation results for the rest of the transient. Transient solver errors halt the transient solution and must be resolved for the full simulation to be run.

The below topics offer descriptions of the more common solver errors and tips for common resolutions. Reach out to AFT Support for further assistance.

TRANSIENT SOLVER ERRORS

Fluid deceleration is outside the valid range

A **Transient Solver Error** in AFT Impulse

Full Message is:

Error occurred while running Transient Solver. Fluid deceleration is outside the valid range for the check valve type selected

This transient solver error message can occur if the model has a check valve defined using the Estimate from Fluid Deceleration valve model. The error message indicates that the fluid deceleration calculated for the check valve exceeds the maximum deceleration for the current valve type. If the deceleration exceeds the known deceleration vs maximum reverse velocity data the reverse velocity to close the check valve can't be calculated and the transient calculations can't proceed. See the Check Valve topic for more information on how the Estimate from Fluid Deceleration option is modeled.

The user will need to consider using the User Specified option to directly define the reverse velocity to close for the check valve, or should consider using a different check valve type if appropriate for the system.

Diagnostics Window

The Model Diagnostics Window is a feature introduced to AFT Impulse with AFT Impulse 8. It is intended to help users check on a number of settings that may have been changed during the model-building process and that may affect the model results.

The Model Diagnostics window can be accessed from the Tools menu.

When the Model Diagnostics Window is opened 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 pipes P1, P2, and P3 have unusual pipe dimensions, and should be reviewed.

Figure 1: Example of the Model Diagnostics window. For this example, potential issues have been flagged in the System Data window and the dimensions for pipes P1, P2, and P3.

Related Topics

Related Blogs

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

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 meters rather than feet.
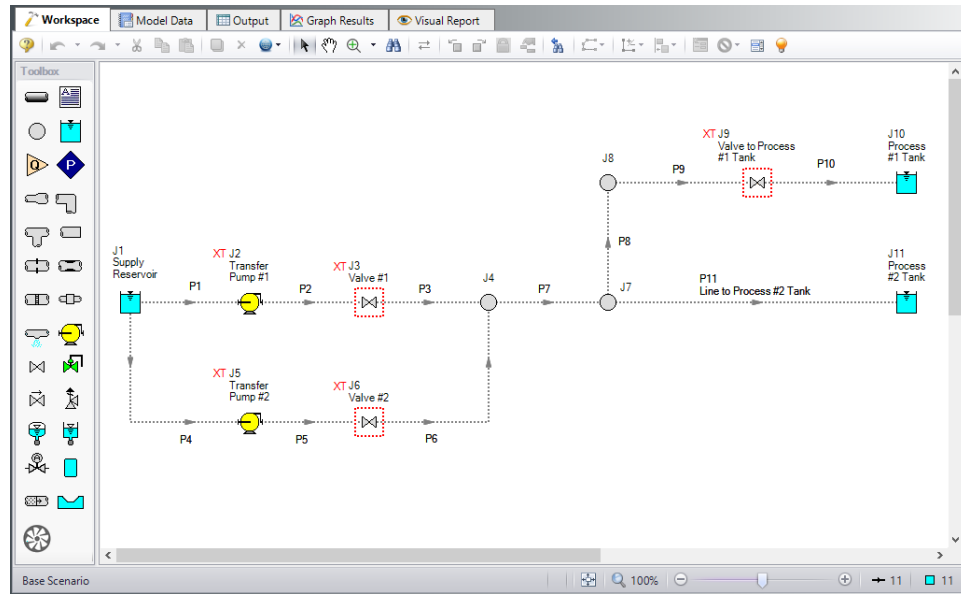
Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown).

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 (#2) has length data entered in meters rather than feet. This is not automatically an input error, as AFT Impulse 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

Related Blogs

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

Transfer Results to Initial Guesses

AFT Impulse's iterative [steady-state solver](#) takes initial guesses (also called initial conditions) and applies the principles of mass and momentum balance to improve the guesses. This is called iteration. It continues to refine the solutions until agreement with the governing equations is obtained.

After a converged steady-state answer is obtained, you can choose to take these results and use them as the initial guesses for a subsequent run. This causes much faster convergence in sub-

sequent steady-state runs because you are starting the solver at or very close to the actual solution.

Note: This will not increase the speed of the transient solution progress.

This feature is called Transfer Results to Initial Guess, and it is on the Output window, Edit menu and toolbar.

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 similar to the orange curve in Figure 1 below.

AFT Impulse 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 Impulse 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.
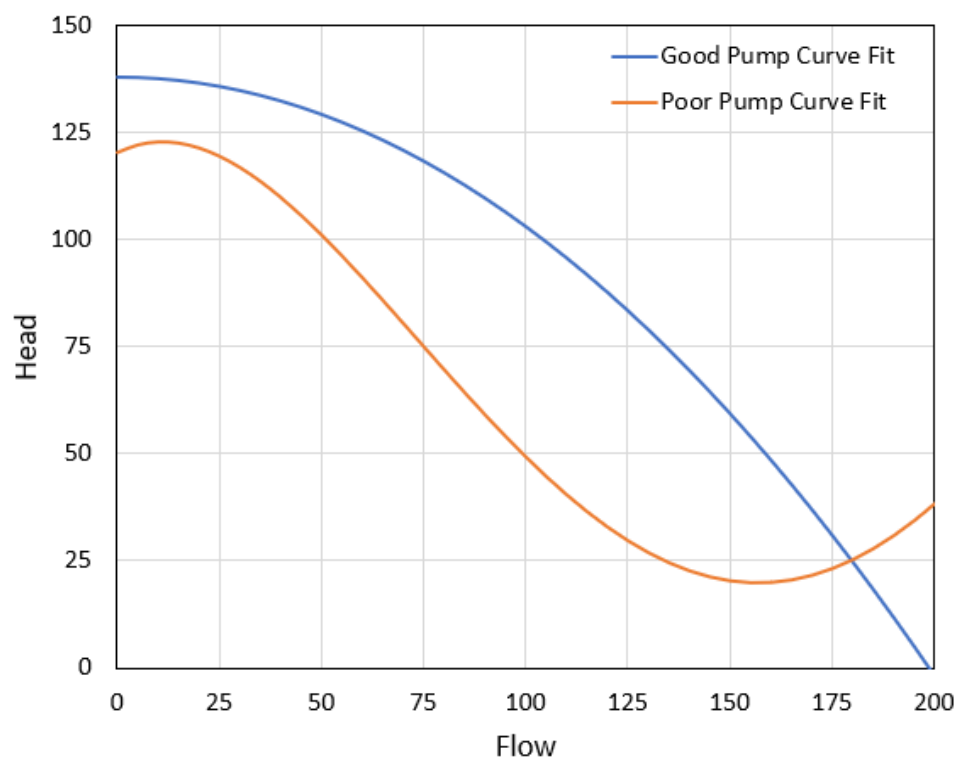
Figure 1: A good pump performance curve fit in blue and a poor pump performance curve fit in orange. All pump curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.

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.
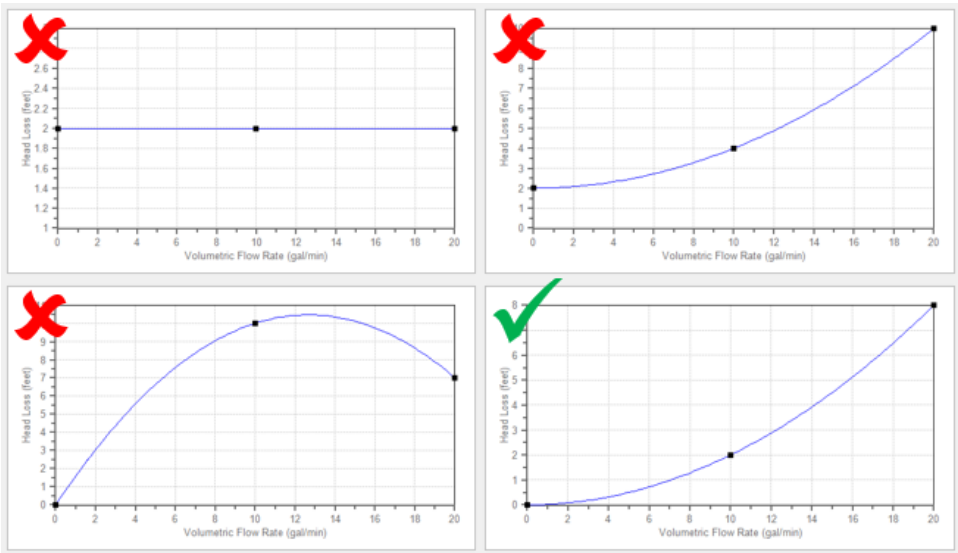
Figure 1: Example resistance curves showing correct and incorrect curve types. The incorrect curves either have non-zero loss at zero flow, or have losses which do not increase with flow. The correct curve has zero loss with zero flow and has losses which increase with flow.

These common issues create two potential problems in a model.

First, if the system sees a constant pressure drop even at low flowrates, AFT Impulse's iterative solver can encounter difficulty converging on a solution since the system flow and pressure drop are no longer dependent on each other.

Second, real equipment in a system will typically cause a higher pressure drop at larger flowrates, and will not provide any loss to the system at zero flow. Users should capture a single known pressure loss data point by defining a curve which passes through two points: the known pressure drop at the tested flowrate and zero pressure drop at zero flow. The topic on Resistance Curves gives more information on defining this type of curve.

Related Topics

Use the Output Window Sort Feature

If a model runs but the results look incorrect, use the Sort feature on the toolbar in the Output Window to look for extremes in velocity, pressure drop or other parameters. An input error (like incorrect diameter) may be easier to see by looking at its effect on the results.

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. The complexity can lead to convergence problems in models with numerous tees. Here are a couple of recommendations.

Tee/Wye junctions have two loss options: Simple and Detailed. The Simple loss model ignores losses at the tee. The Detailed model uses the correlation summarized in the Tee/Wye Loss topic. The potential convergence problems occur when using the Detailed loss model.

The potential problem is not with the Detailed tee model itself. Rather, it is due to multiple tees coupling together in such a way that AFT Impulse cannot converge on all of the flow splits. 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 tees. Using the Global Junction Edit window, you can change all tees to the Simple loss model and rerun the model. If it now converges 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 in the Tolerance panel rather than relative tolerance if the issue only involves the steady-state convergence.

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.

Additional configurations for calculation of the Detailed model can be set on the Optional tab to help with convergence using the detailed tee model in the transient. See Tee/Wye Waterhammer Theory for more information.

Related Topics

Related Blogs

Having model convergence problems? Start here!

Give me some details about Detailed Tees!

Lower the Flow Rate Relaxation

The Iterations Panel allows you to change the Numerical Relaxation parameter for both flow rate and pressure. It is recommended that you use only 0.5 or 1.0 for pressure relaxation. But lower values of flow rate relaxation can be very helpful.

A good first step is to lower the flow rate relaxation to 0.05 or 0.1. By lowering the flow rate relaxation you improve AFT Impulse's ability to obtain convergence. The cost is slower steady-state 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

Related Blogs

Having model convergence problems? Start here!

Try Absolute Tolerance

Most models converge more reliably with relative tolerances, which is the default in AFT Impulse's Tolerance Panel. But some models converge much better with absolute tolerance. This usually means that there is at least one element of the model that is close to the real answer in absolute terms but does not lock in on a relative (percentage) basis.

If a model will not converge with relative tolerance, you can try changing to absolute tolerance. Even better, try the option to converge on either absolute or relative, which will offer the most flexibility.

Related Topics

Related Blogs

Having model convergence problems? Start here!

Make Initial Flow Rate Guesses for Pipes

If you do not provide any initial guesses, AFT Impulse will make its own guesses. On occasion the AFT Impulse 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 for the steady state.

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, if convergence can only be achieved within a narrow range of fixed flowrates, there may be 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.

Related Topics

Break a Large Model Into Submodels

AFT Impulse 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

Zero-Length Connector Configurations

Pipes defined as zero-length connectors are only allowed to be used in limited cases. If the zero-length connector is used for a restricted case Impulse will show the following message: "Zero-Length Connector configurations that are not allowed for transient simulation exist."

To resolve the above error navigate to all pipes/junctions indicated by Impulse and disable the zero-length connector feature so that the restricted configuration no longer exists at those pipes/junctions. For a full list of restricted configurations see the discussion of zero-length connectors in the Pipe Model topic.

When changing the zero-length connector pipes to short pipes be careful to choose a reasonable pipe length to help avoid significantly long run times. See the topic on the affects of using unusually short pipes for suggestions on alternate ways to configure the model.

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 Impulse.

Contents

Related Blogs

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

Technical Support Options

Support, Upgrade and Maintenance (SUM)

This primary level of support is for assistance with specific features within the software, or resolving issues with a model such as difficult to converge models. We are limited in our ability to interpret model results, and cannot recommend specific changes in a system.

One full year of SUM is provided with the purchase of a full license. Leases have SUM coverage during the period of the lease. Further support after the first year requires SUM renewal.

For technical support questions falling in this category, [contact AFT](#).

Flow Expert Package (FEP)

The FEP is a higher level of consulting assistance for a particular project. This service will assign a qualified AFT engineer to assist in not only modeling your system, but also with providing operational suggestions and interpretation of results.

The intention of this service is to provide assistance to active users of AFT software and address questions on the appropriate use of the software for specific system requirements.

Please visit our website at: https://www.aft.com/products/aft-flow-expert-package for more information on our FEP services.

Flow Modeling Consulting Services

AFT's sister company, Purple Mountain Technology Group (PMTG), provides comprehensive consulting services utilizing experienced consulting engineers and AFT tools. PMTG offers a solution to a wide variety of system analysis by building and investigating the models and providing direction on system changes and appropriate operation. Visit PMTG on the web: https://www.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 Impulse does not include the Chempak Fluid Library by default. Chempak Data is a separate product that must be downloaded and installed as an addition to AFT Impulse 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 Impulse and Chempak Data are installed to their default locations, no further configuration is necessary. However, if AFT Impulse or the Chempak Data are not installed in a default location, AFT Impulse will need to know where to look for the Chempak Data. This is specified in the control.aft file, located in the same folder as the Impulse.exe. If the files cannot be found the following message is displayed:
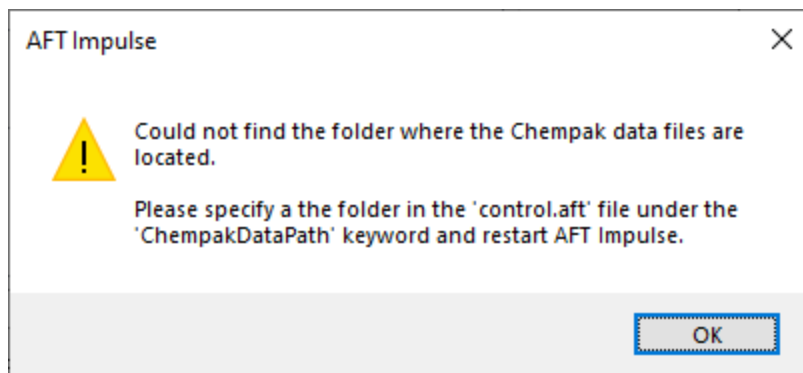
Figure 1: Error message when Chempak data cannot be found in default location

Control.aft can be edited with a text editor, and by default looks like:
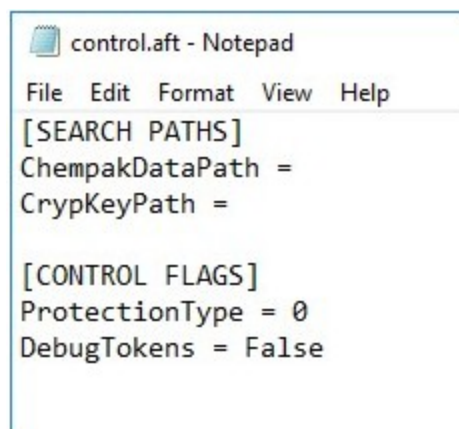
Figure 2: Default control.aft file

If the Chempak Data 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 Data Version

When upgrading to a new version of AFT Impulse, you may receive a message that Chempak is out of date.
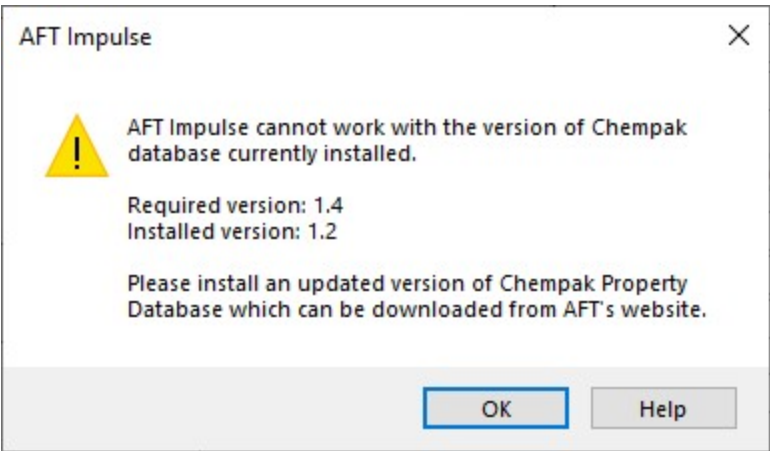
Figure 3: Outdated Chempak data

To fix this issue, download the newest version of the Chempak Data from the AFT website. Be sure to install to the same location as the existing Chempak Data, and double check the control.aft file if necessary.

Related Topics

Updated Fluids Library

If the data in a shared fluid library is changed outside of the model you are opening, the following informational message will appear when opening the file.
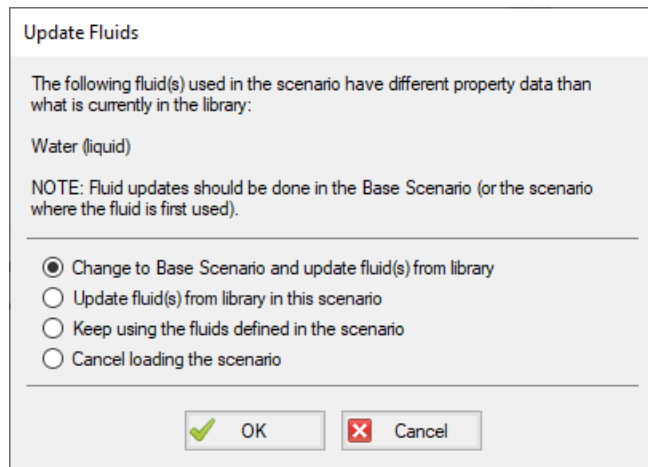
Figure 1: Update Fluids dialog window which appears if the fluid properties for the model have changed since it was last opened

There are a few reasons that the fluid data may have been changed:

1. The model is defined in a previous version of Impulse using a fluid from the AFT Standard fluid library. The AFT Standard fluid library was updated in AFT Impulse 9 in order to standardize the reference for the AFT Standard fluid library using data from the NIST REFPROP Version 10 fluid database. 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.

   Most models will only experience a minor change in results, but some models may experience more significant changes. Contact support for further information on specific fluid changes, and to get a comparison of fluid data between Impulse 8 and Impulse 9.

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 option the "Keep using the fluids defined in the scenario" or cancel loading the model and change the fluid. If the changes were intended then update the fluid data used in the model using the "Change to Base Scenario and update..." option.

Special File Control Features

What Is the Control.aft File?

Conrol.aft is read as AFT Impulse begins to load and instructs AFT Impulse 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 Impulse 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 Impulse will not find the important data.

Specifying Where Chempak is Located

Before AFT Impulse can use the Chempak Fluid Library it needs to know where both the data is located. These paths are specified in the control.aft file.

When Chempak is installed the setup procedure automatically updates any control.aft file specified with the path to the data.

You can manually edit the file by opening it in Notepad (or any text editor) and adding the following line to the [SEARCH PATHS] section (replace C:\AFT Products\Chempak Data with the path information based on your system):

ChempakDataPath = C:\AFT Products\Chempak Data

This will instruct AFT Impulse 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 version of AFT Impulse can be used with this version. In most cases, the model file can be opened in the newer version and automatically converted to the updated format. It is considered good practice to do a complete check on the model to ensure the conversion process was complete and accurate. In some cases, especially with older models, the automatic conversion is not successful. If this occurs, please contact our support team with a copy of the model file.

Other files such as libraries (previously databases) or preference files can also be converted.

Related Topics

Index

H

I

J

L

M

N

W