User Guide

PipeFlow is a trading name of Daxesoft Ltd.
www.pipeflow.com www.pipeflow.co.uk

Printed in the United Kingdom - November 2016

Table of Contents

Table of Figures

Introduction

Pipe Flow Expert is a software application that runs on the Microsoft Windows operating system.

It is used by engineers in over 100 countries worldwide, to model pipe systems where the flow rates, pressure losses, and pumping requirements of the system need to be calculated.

The Pipe Flow Expert software has an intuitive interface that makes it easy for users to start working on their pipe designs, which can be drawn out on a 2D or 3D isometric grid.

The software comes with an unrivalled support service that provides help to users when they need it.

Pipe Flow Expert Software

Pipe Flow Expert is designed to help today's engineers analyze and solve a wide range of hydraulic problems where the flow rates, pressure losses and pumping requirements throughout a pipe network must be determined.

The Pipe Flow Expert software will allow you to easily draw out a pipeline system and analyze the performance of the system when flow is occurring. Pipe Flow Expert calculates the balanced steady flow and pressure conditions of the system.

The software will allow you to perform analysis of alternate systems under various operating conditions.

The reported results include:

- flow rates for each pipe
- fluid velocities for each pipe
- Reynolds numbers
- friction factors
- friction pressure losses
- fitting pressures losses
- component pressure losses
- pressures at each node
- pump head
- pump operating point
- NPSHa at pump inlet

The input and display of system information on the Pipe Flow Expert drawing and in the results tables can be shown in metric or imperial units to suit your preference and specific units for each item (such as flow rate) can also be configured and set on an individual basis as required.

The Pipe Flow Expert software has been designed for the professional engineer who needs a powerful tool that has a class leading, easy to use and robust interface that makes it simple to design and analyze pipe networks.

Pipe Flow Expert Software Overview

Pipeline systems range from very simple ones with a single pipe to very large and complex networks with hundreds of interconnecting pipes. They may be as simple as a single pipe carrying water from one reservoir to another reservoir, or they may be more complex with many pipelines interconnecting to distribute fluid over a large area, or they could fall somewhere in-between such as a system that transfers a chemical from a supply container to various process points.

The pipelines may vary in size and nature and will usually involve changes in elevation from one point to another. These pipeline systems may include reservoirs, pressurized tanks, pumps, valves, flow control devices, heat exchangers and other components that affect flow in the pipelines.

The pipeline system is modeled by drawing the join points and the connecting pipes on a drawing pane. Horizontal, vertical or sloping lines can be used to connect one node to another node. The physical data describing the system is entered by the user and typically includes:

- The internal size, internal roughness and length of each connecting pipe
- The elevation of each join point (node)
- The In-flow and the Out-flow at each join point (if applicable)
- The elevation, liquid level and surface pressure data for each tank
- The performance data for each pump

Data input boxes are located at the left hand side of the drawing pane. These input boxes will display the data for the currently selected node or pipe and may be used to amend the current data. The data for a node, pipe, pump, etc. can be amended at any point during the design process.

Once the design has been completed, the system can be analyzed and the flow and pressure results can be calculated. For liquid systems, the pressure losses within the system are calculated using friction factors obtained from the Colebrook-White equation, and the pressure loss due to friction in each pipe is obtained from the Darcy-Weisbach equation. For gas systems, the pressure losses are calculated using a compressible isothermal flow equations such as the General Flow Equation.

An initial approximate solution is obtained using Linear Theory methods and an iterative approach that adjusts the flow rates until an approximate pressure balance is achieved. The solution is then converged to an accurate solution using sophisticated matrix techniques and other iterative algorithms.

Pipe Flow Expert defines the elements of the pipeline system in a series of mathematical equations. Pipe systems can produce a highly non-linear set of equations that are difficult to solve. The Pipe Flow Expert software uses the Newton method and other proprietary algorithms to solve the equations, to determine the flow rate and pressure loss in each pipe that provides a balanced solution.

The results of the flow rates for each pipe, the fluid velocities for each pipe, Reynolds numbers, friction factors, friction pressure losses for each pipe, fittings pressure losses, pressure at join points (nodes), head pressure at nodes, pump operating points and more, can be viewed on the results drawing and on the results grid.

Minimum Operating System Requirements

Pipe Flow Expert has been designed to work on the following operating systems:

- Microsoft® Windows 10 (All Versions)
- Microsoft® Windows 8 (all versions except RT [used on ARM based tablets])
- Microsoft® Windows 7 (all versions)

We recommended using one of the above operating systems, however the software will also run on:

- Microsoft® Windows Vista (all versions)
- Microsoft® Windows XP (all versions)

It is recommended that your computer system has at least the following minimum specification:

- Processor: 1.00 GHz or faster CPU
- Memory (RAM): 2Gb or higher
- Screen Display: 1024 x 768 pixels (minimum resolution)
1920 x 1080 pixels (or higher recommended)
- Graphics (RAM): 128 Mb or higher
- Hard Disk: 75 Mb install (250 Mb free space recommended)

Microsoft® Excel is required if the user wishes to export the results tables to a spreadsheet format.

Adobe® Acrobat Reader is required if the user wishes to generate and view customized PDF Reports based on the calculated results data.

Note:

If your computer will run Windows 7 (or later) then Pipe Flow Expert should work without any issues.

Registration and Licensing Information

Pipe Flow Expert is a software package that is provided for use on personal computer systems.

Installation

When the software is installed, it generates a unique product number that is shown on the Software License screen when you run the program. A matching license code must be entered to fully enable and license the software.

Figure 1 Pipe Flow Expert License Software

Purchasing a License

Each installation of the Pipe Flow Expert software requires a license code to be entered in order to fully activate the program. Until the license code is entered the software can only be used in trial mode.

License codes can be purchased from www.PipeFlow.com

Standard licensing and pricing options are shown on the web site at www.pipeflow.com.

You do not require an internet connection when licensing and enabling the software. The software does not need an internet connection to operate.

Moving a License

We allow a user to move the license to another machine. The old license is de-activated which generates a License Removal Confirmation Number. We then provide a new license code for a new installation of the software. This service is provided at no further cost as part of the Annual License fee which also includes support.

Network Licensing

The software can be operated over a network and used by a number of concurrent users. Network installation is simple and only requires that the software is installed to a shared network drive. Each user then runs the program as normal from the shared drive. When licensing the program you will need to purchase a software license that supports the maximum number of concurrent users that you require.

Please email info@pipeflow.com with your network licensing requirements to obtain a competitive quote for your specific needs.

Software Distribution Note

All of our software is provided via download from our web sites at www.pipeflow.com and www.pipeflow.co.uk

We do not provide the software on CD, since it would be exactly the same as the program that can be downloaded from our web site. This helps to reduce our costs, which allows us to offer lower prices for our customers and also guarantees that the user is always installing the latest version of the software. Users are free to make their own CD copy of the software, however every installation of the program will generate a new product code and a new license code will still be required.

New Customers

Those of you who have previously purchased software from us will already be assured of our reputation and will not be concerned about receiving the software via download. However we understand that as a new customer you may be concerned that you do not get a CD with the software on.

If we provided the software on CD it would be the exact same program that you can download from our web site (in fact the web site will always carry the very latest version of the software). Having a CD would not allow you to install and enable the software on multiple computers, since each installation would still generate a unique product number and would require a matching license code.

When you purchase a program from our web site, you will automatically be emailed with a unique purchase token and an invoice/receipt that confirms your purchase. This is all you need for proof of purchase and we will always be able to confirm your purchase in our database.

We have sold software via download for over 15 years. We are not one of those companies who you will not be able to contact or get a response from. As our existing customers know, our service and assistance with any issues you may have is worth far more than just having the software on CD.

We do sell a license to use a copy of the Pipe Flow Expert software on a USB drive. When this option is purchased, we send you a ‘Pipe Flow’ branded USB drive and allow you to license this. This provides a completely portable copy of the software. There is nothing to install. The USB drive can be plugged in to any Windows based computer and the Pipe Flow Expert software can then be run directly from the USB drive, up into the memory of the local computer.

Summary

- The Pipe Flow Expert software can be downloaded and installed for a free trial.
- A license code can be purchased from our web site at www.pipeflow.com that will enable use of the software for an initial period of time (normally 12 months).
- A user can be up and running with a licensed version of the software within a few minutes.
- An Annual License fee must be paid to extend the license to use the software for a further 12 months, once the initial 12 month license period ends. The Annual License fee can be paid at www.pipeflow.com and is usually about 15% of the cost of the initial 12 month license fee.
- Software support and maintenance, including technical assistance, help with modeling issues and free upgrades to new releases, are provided for free while you are running a licensed copy of the Pipe Flow Expert software. This ensures that you will always be able to use the latest version of the software.

Contacting Pipe Flow Software

Email: info@pipeflow.com

Internet: http://www.pipeflow.com

UK Telephone: +44 (0)1625 509142

USA Telephone: +1 650-276-3569

+1 650-276-FLOW

PipeFlow.com and PipeFlow.co.uk are trading names of Daxesoft Ltd, a U.K. Registered Company)

Additional Pipe Flow Software Programs

Pipe Flow Wizard - "What if?" Calculations on a Single Pipe for Liquids and Gases

Pipe Flow Wizard is able to perform four different calculations depending on the known information. It can calculate:

- Pressure Drops
- Flow Rates
- Size of Internal Diameters
- Pipe Lengths

Pipe Flow Wizard will perform calculations for individual pipes.
Pipe Flow Wizard will calculate results for **LIQUIDS or COMPRESSED GASES**

A **Fluid Database is included** with viscosity and density of common fluids.

Flow Advisor - for Channels and Tanks

Flow Advisor may be used to estimate water flow rate from various shaped channels and tanks. It can calculate:

- Open Channel Flow
- Water Flow Rates
- Time taken to empty tanks
- Volume, Capacity, Weight and Expansion

A **Materials Database is included** with density and coefficient of expansion of common materials.

All of our software can be downloaded for a free trial by visiting www.pipeflow.com

Once you have installed a trial version of one of our software products it can be fully enabled and licensed by buying a license code from our web site. You can be up and running with a fully licensed program within just a couple of minutes

Interface and Menus

This section details the different features of the Pipe Flow Expert interface. For each feature, there is an explanation, a screen shot and a table providing descriptions for each element of the feature. The sections following this section provide instructions for using the Pipe Flow Expert application.

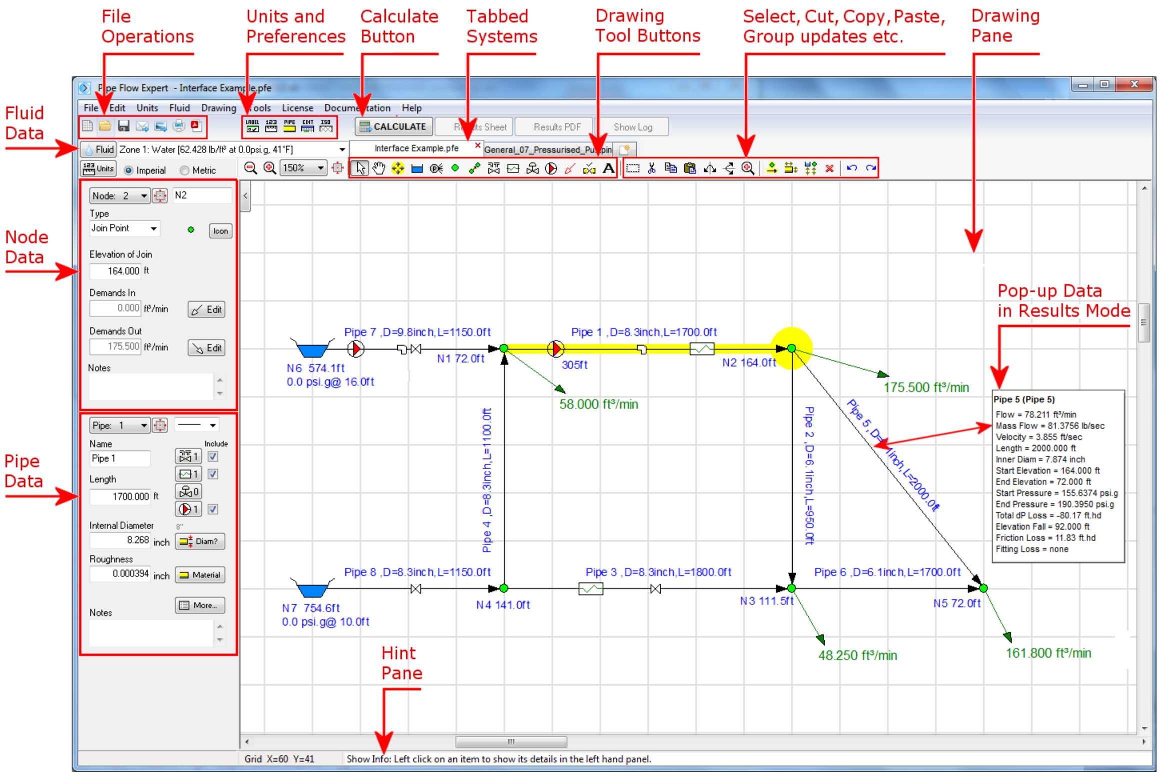


Figure 2 Pipe Flow Expert interface

Menu Bar

The menu bar has nine different menu selections to help you navigate and work in Pipe Flow Expert.

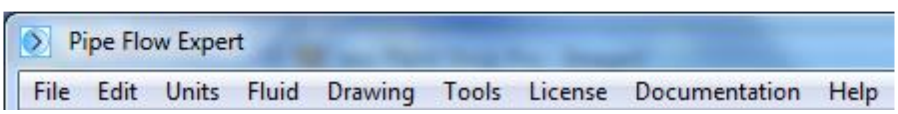


Figure 3 Menu Bar

File Menu

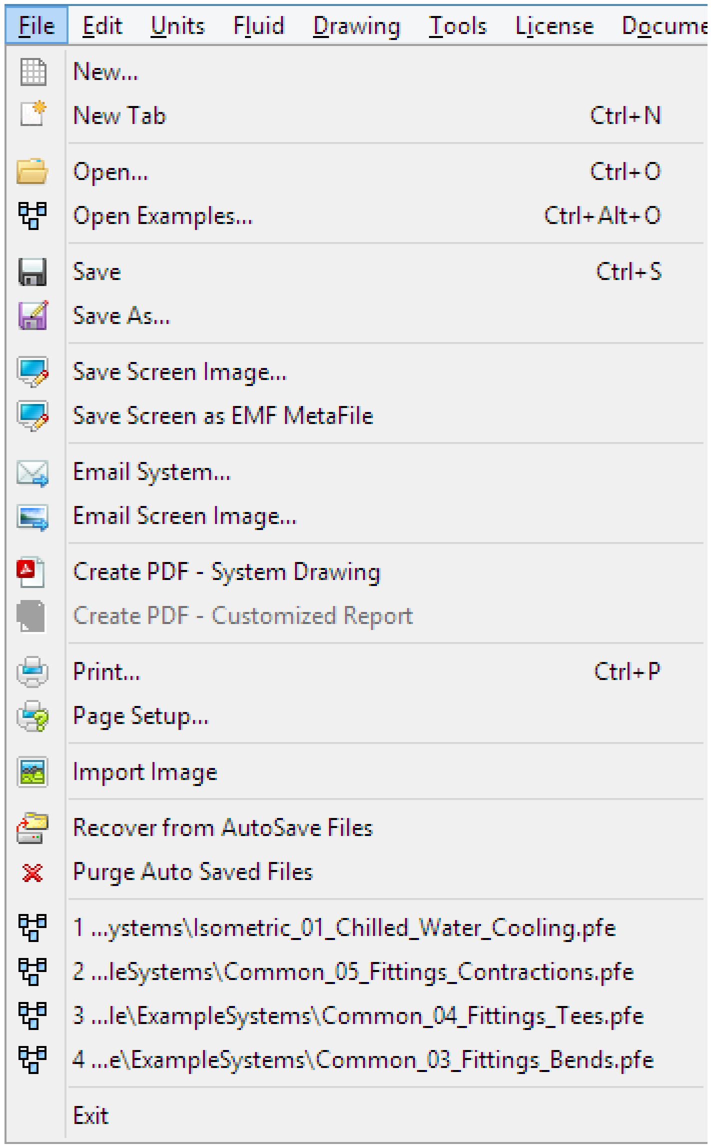


Figure 4 File Menu

Menu Items	Description
New	Opens a new drawing grid in the Drawing pane.
New Tab	Opens a new drawing grid in a new tab. Tabbed Design Sheets let you work on multiple systems simultaneously. Up to 4 different systems may be worked on concurrently using the 'Tabbed' system feature.
Open	Opens the **Open** dialog. Select a .pfe file and click the **Open** button to open a pipe system drawing in Pipe Flow Expert.
Open Examples	Opens the Example Systems screen. Select an example and click the **Load This Example System** button to open an example system drawing in Pipe Flow Expert.
Save	Saves your changes to the pipe system drawing.

Save As	Opens the **Save As** dialog. Browse to where you want to save the pipe system, name the .pfe file, and click **Save**.
Save Screen Image	Save an image of the visible drawing area as a jpeg file.
Save Screen as EMF Metafile	Save the drawing as an Enhanced Metafile.
Email System	Opens a new email message with a .pfe file of your current design attached to an email.
Email Screen Image	Opens a new e-mail message with a jpeg of the screen image attached to the e-mail.
Create PDF – System Drawing	Create a PDF document of the system drawing.
Create PDF – Customized Report	Create a PDF customized report, including cover sheet with logo image, system drawing and results tables. Including a high resolution pump graph with performance curves.
Print	Opens the **Printing Information** dialog. Type the page title and click the **Print** button to print the image currently visible in the Drawing pane.
Page Setup	Opens the **Print Setup** dialog. Define your printing preferences and click **OK**
Import Image	Opens the Import Image dialog. Allows browse of folders and files, to select import of image file on to the system drawing.
Recover from AutoSave Files	Pipe Flow Expert will automatically save a backup of your current design at approximately 5 minute intervals. In the event of a problem you can try to load the most recent backup file to try and recover your system.
Purge Auto Saved Files	Delete the files created by the auto save feature.
Exit	Closes the application.

Edit Menu

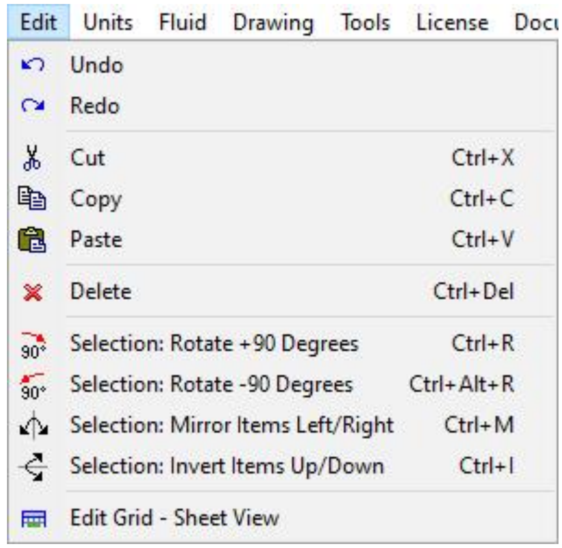


Figure 5 Edit Menu

Menu Items	Description
Undo	The last node or pipe action performed in the Drawing pane is undone.
Redo	The last undo action is redone.
Cut	Selected items are cut to the clipboard.
Copy	Selected items are copied to the clipboard.
Paste	Items on the clipboard are pasted in the Drawing pane.
Delete	Selected item(s) are deleted.
Selection: Rotate +90 Degrees	Selected items are rotated 90 degrees clockwise
Selection: Rotate -90 Degrees	Selected items are rotated 90 degrees anticlockwise
Selection: Mirror Items Left/Right	Selected items are mirrored left to right
Selection: Invert Items Up/Down	Selected items are inverted up to down
Edit Grid – Sheet View	Opens an edit grid allowing details of pipes, nodes and valves etc. to be amended.

Units Menu

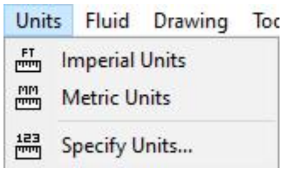


Figure 6 Units Menu

Menu Items	Description
Imperial Units	Converts all values to imperial units.
Metric Units	Converts all values to metric units.
Specify Units	Opens the **Units** tab of the **Configurations Options** dialog. Use the **Units** tab to define the units for each element of the pipe system drawing.

Fluid Menu

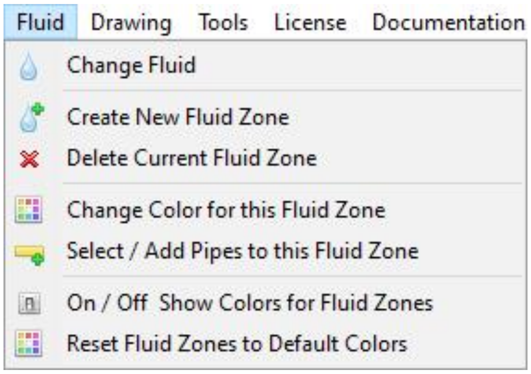


Figure 7 Fluid Menu

Menu Items	Description
Change Fluid	Opens the **Fluid data** dialog. Use the **Fluid data** dialog to define the fluid for the selected fluid zone.
Create New Fluid Zone	Opens the Fluid Manager to select a fluid for a new Fluid Zone
Delete Current Fluid Zone	Delete the current fluid zone from the system. All pipes that previously contained this fluid will now contain the fluid from zone 1.
Change Color for this Fluid Zone	Select a background highlight color for this fluid zone.
Select / Add Pipes to this Fluid Zone	Display the fluid zone selection rectangle. Click and drag the rectangle to enclose the pipes to be set to the current fluid zone
On / Off Show Colors for Fluid Zones	Toggle the fluid zone background highlight colors on or off.
Reset Fluid Zones to Default Colors	Reset all fluid zone colors to the default color for each fluid zone.

Drawing Menu

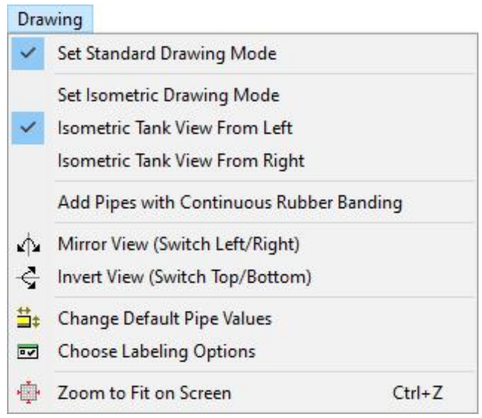


Figure 8 Drawing Menu

Menu Items	Description
Set Standard Drawing Mode	Set rectangular grid drawing mode
Set Isometric Drawing Mode	Set isometric grid drawing mode
Isometric Tank View From Left	Show isometric tanks viewed from left
Isometric Tank View From Right	Show isometric tanks viewed from right
Add Pipes with Continuous Rubber Banding	Enable / disable drawing pipes using continuous rubber banding.
Mirror View (Switch Left/Right)	Mirror the complete drawing
Invert View (Switch Top/Bottom)	Invert the complete drawing
Change Default Pipe Values	Opens the Default Values tab of the Configurations Options dialog. Use the Default Values tab to define default values for the next pipe to be drawn in the pipe system.
Choose Labelling Options	Opens the Labelling tab of the Configurations Options dialog. Use the Labelling tab to define how the pipe system drawing is labelled.
Zoom to Fit on Screen	Resize the drawing and center in the visible drawing area

Tools Menu

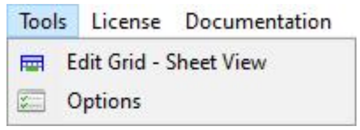


Figure 9 Tools Menu

Menu Items	Description
Edit Grid – Sheet View	Amend data for Pipes, Tanks, End Pressures, Nodes, FCVs, PRVs, BPVs, Cv Values, Kv Values, Fixed Losses, Pumps (Fixed Flow), Pumps (Fixed Head) and Pumps (Curve). The data in the grid may be over typed with new data – changes will be made immediately as each item of data is amended.
Options	Opens the **Configuration Options** screen. Use the **Configuration Options** to configure the pipe system.

License Menu

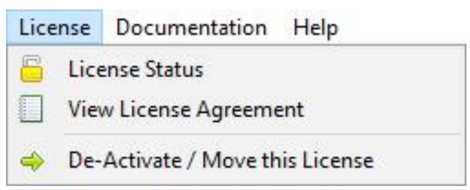


Figure 10 License Menu

Menu Items	Description
License Status	Displays the current license status and registration information.
View License Agreement	View agreement between Licensor and Licensee
Move this License	Displays instructions that explain how to deactivate the current license and obtain a license removal confirmation code. How to create a new installation (which will require a new license code).

Documentation Menu

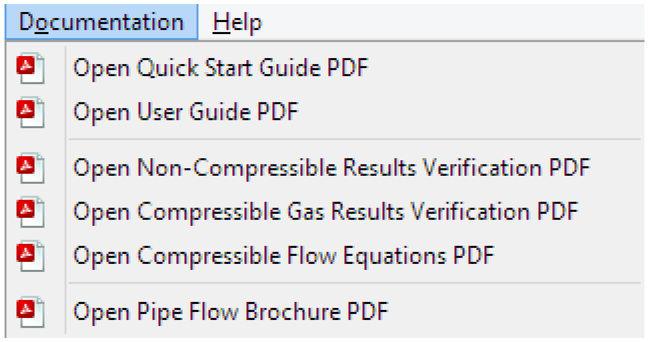


Figure 11 Documentation Menu

Menu Items	Description
Open Quick Start Guide PDF	Open the Quick Start Guide PDF document.
Open User Guide PDF	Open the full User Guide PDF document.
Open Non-Compressible Results Verification PDF	Open the Non-Compressible (Liquids) Calculation Results Verification PDF document.
Open Compressible Results Verification PDF	Open the Compressible (Gases) Calculations Results Verification PDF document.
Open Compressible Flow Equations PDF	Open the Compressible Flow Equations & formula descriptions PDF.
Open Pipe Flow Brochure PDF	Open the Pipe Flow Expert PDF Brochure.

Help Menu

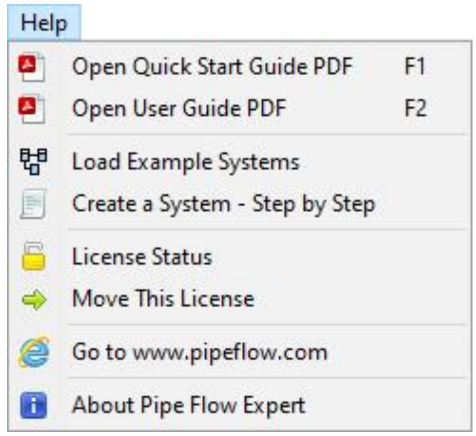


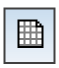
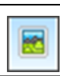
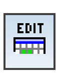
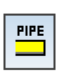
Figure 12 Help Menu

Menu Items	Description
Open Quick Start Guide PDF	Open the Quick Start Guide PDF document.
Open User Guide PDF	Open the full User Guide PDF document.
Load Example Systems	Lists the examples systems that can be loaded, viewed, and the results calculated.
Create a System – Step by Step	Displays the Help information starting at the page that describes how to create a system using a step by step walkthrough of each operation.
License Status	Displays the current license status and registration information.
Move This License	Displays instructions that explain how to deactivate the current license and obtain a license removal confirmation code. How to create a new installation (which will require a new license code).
Go to www.pipeflow.com	Launches an internet browser window to view the Pipe Flow web site.
About Pipe Flow Expert	Displays a pop-up window that shows details about the software.

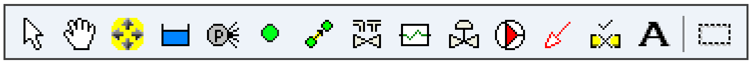
Tool Bar

Most of the Pipe Flow Expert functions can be performed by using a button on the tool bar. The Re-Design, Results Sheet, Results PDF and Show Log buttons are only available on the tool bar after you click the Calculate button on the tool bar.
The tool bar also contains a field for determining whether the unit values displayed in Pipe Flow Expert and the pipe system are imperial or metric. The first step in creating a pipe system is to select the Imperial or Metric option. You can define whether imperial or metric values are used from the tool bar, the Units menu, or from the Units tab in the Configuration Options dialog. Inches, feet, and gallons are examples of imperial units. Centimeters, meters, and liters are examples of metric units.

Tool Bar Buttons

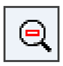
Button	Name and Description
	**New Grid** – Opens a new drawing grid.
	**Open File** – Opens the Open dialog. Select a .pfe file and click the Open button to open an existing pipe system drawing in Pipe Flow Expert.
	**Save File** – Saves your changes to the pipe system drawing.
	**Email System Information** – Opens a new e-mail message with the pipe system .pfe file attached to the e-mail.
	**Email Screen Image** – Opens a new e-mail message with a jpeg of the screen image attached to the e-mail.
	**Print** – Opens the **Printing Information** dialog. Type the page title and click the Print button to print the image currently visible in the Drawing pane.
	**Create PDF of System Drawing** – Produces a high resolution PDF of the system drawing including items which may be currently off-screen.
	**Import Image** – Opens the Open dialog. Select an image file to import on to the pipe system drawing. Images can be resized and repositioned as required.
	**Toggle Isometric Grid** – Switch to Isometric Drawing Grid, or back to Standard Grid.
	**Network Grid View** – Opens a grid to display details of pipes, tanks, end pressures and nodes. The data displayed in the grid may be edited to change the network data. Any changes are applied immediately to the drawing.
	**Set Pipe Default Drawing Values** – Opens the **Pipe Settings tab** of the **Configurations Options** dialog. Use the **Pipe Settings tab** to define the pipe default values for drawing or to copy some of the pipe attributes to selected pipes in the system.
	**Choose Labelling** – Opens the **Labelling tab** of the **Configurations Options** dialog. Use the **Labelling tab** to define how the pipe system is labelled in the Drawing pane.
	**Choose Units** – Opens the Units tab of the **Configurations Options** dialog. Use the **Units tab** to define the units for each element of the pipe system.
	**Calculation Settings** – Opens the Calculations tab of the **Configurations Options** dialog. Use the **Calculations tab** to select method of solution and method of calculation, and set calculation parameters.

The **Hint** pane displays tips for using the button currently selected on the tool bar. The **Hint** pane displays tips for the following buttons:



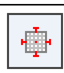
In **Results** mode the **Hint** pane is used to display information about the flow rates in pipes, velocities in pipes, pressure drop in pipes, pressures at join points, head pressure (grade line), elevations of tanks and nodes, and fluid heads in the pipeline system. Items are highlighted by clicking on an item in the drawing pane. Data for the highlighted item is displayed in the hint pane.

Fluid – Opens the **Fluid menu,** select change fluid to open the **Fluid data** dialog. Use the **Fluid data** dialog to define the fluid for the currently selected fluid zone. Select other fluid menu options to change the **fluid zone color** or to **create a new fluid zone** or to **delete a fluid zone.**

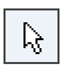


Zoom Out – Moves the focus of the pipe system out to see more of the drawing at a reduced size in the Drawing pane.

Zoom In – Moves the focus of the pipe system in to get a close-up view in the Drawing pane.



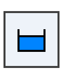
Move to Centre of Grid – Centers the pipe system on the drawing grid.



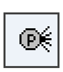
Show Item Info – Select the show information cursor. Click an item in the Drawing pane to display its details in the Node pane, Pipe pane, or dialog associated with the selected component.

Pan the Drawing – Select the move grid cursor. Click in the Drawing pane, and while holding down the left mouse button, move the mouse to pan the drawing.

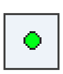
Drag and Move Items – Highlights all items on the drawing. Select a highlighted object, and while holding down the left mouse button, drag the object to its new position.



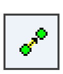
Add Tank – Select the tank cursor. Click where you want to add a tank in the Drawing pane. Use the Node pane to enter the Elevation, Liquid level and Fluid surface pressure for the tank.



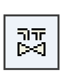
Add Demand (Pressure) – Select the end pressure cursor. Click where you want to add a fixed pressure demand in the Drawing pane. Use the **Node pane** to enter the pressure and elevation for this fixed pressure demand.



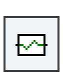
Add Join Point – Select the join point (node) cursor. Click where you want to add a pipe join point in the Drawing pane.



Add Pipes – Select the draw pipes cursor. Click on a node in the Drawing pane where you want to add a pipe, and then click where you want the pipe to end. Right-click to turn off the rubber banding.



Add Fittings – Select the valve and fittings cursor. Click on the pipe on which you want to add valves or fittings. Use the **Pipe fittings friction coefficients dialog** to select the valves or fitting to be added.



Add Component Pressure Loss – Select the component cursor. Click on the pipe on which you want to add a component. Use the **Component pressure loss dialog** to enter the pressure loss characteristics of the component.

Add Control Valve – Select the control valve cursor. Click on the pipe on which you want to add a control valve. Use the **Control data dialog** to choose a flow valve type. Choose a Flow Control Valve (FCV) to set the flow rate required in the pipe, or a Pressure Reducing Valve (PRV) to set the pressure required at the end of the pipe, or a Back Pressure Valve (BPV) to set the pressure required at the start of the pipe.

Add Pump – Select the pump cursor. Click on the pipe on which you want to add a pump. Use **the Pump data dialog** to enter the pump performance characteristics.

Add Demand (Flow) – Select the add demand flow cursor. Click on a node in the Drawing pane where you want to set a demand flow. Use **the Flow demands dialog** to set the **In-flows** or **Out-flows** at the selected node.

Toggle to Open/Close a Pipe – Select the open/close pipe cursor. Click on a pipe in the Drawing pane where you want to close a pipe or to re-open a pipe that has been closed previously.

Add Text – Add Free text labels to the drawing.

Selection Tool – Select the selection rectangle cursor. Click in the Drawing pane, and while holding down the left mouse button, drag the mouse to create a rectangle around the item(s) you want to select for cutting, copying, pasting, deleting or moving. You can select multiple items to cut, copy, paste, delete or move as a group. Click the right mouse button to exit from selection mode.

Cut – Selected items are cut to the clipboard.

Copy – Selected items are copied to the clipboard.

Paste – Items on the clipboard are pasted in the Drawing pane.

Mirror – Selected items are mirrored left / right.

Invert – Selected items are inverted up / down.

Zoom Selected Area – Displays the selection area in a close-up view in the centre of the Drawing pane.

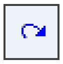
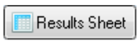
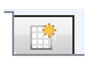
Select Individual Pipes and Nodes – Use the Adjust attributes of Selected Pipes tool button to display the Pipe Settings tab to make changes to all selected pipes.
Use the Adjust attributes of Selected Nodes tool button to display the Node Settings tab to make changes to all selected nodes.

Adjust Attributes of Selected Pipes – Opens the **Pipe Settings** tab of the **Configurations Options** dialog. Use the **Pipe Settings** tab to copy various attributes of the default pipe to each of the pipes selected. The **Selection Tool** may be used to Select a group of pipes prior to opening the **Pipe Settings** tab. Individual pipes can be added or removed from the selected list by checking or un-checking the box adjacent to the pipe description.

Adjust Attributes of Selected Nodes – Opens the **Node Updates** tab of the **Configurations Options** dialog. Use the **Node Updates** tab to copy node elevation and images to each of the nodes selected. The **Selection Tool** may be used to Select a group of nodes prior to opening the **Node Updates** tab. Individual nodes can be added or removed from the selected list by checking or un-checking the box adjacent to the node description.

Delete – Selected item(s) are deleted. If a selected item is a node then all pipes which connect to the node will also be deleted.

Undo last action – The last add pipe or node action performed in the Drawing pane is undone.

	**Redo last undo action –** The last undo action is redone.
	**Calculate –** Calculates the flow and pressure at each node of the pipe system drawing. The calculation results are displayed on the **Results** window.
	**Re Design** – Changes the view back to the design mode. The pipe system is displayed in drawing mode again. This button is only visible on the tool bar after clicking the **Calculate** button.
	**Results** – Opens the **Results** window. This button is only visible on the tool bar after clicking the **Calculate** button.
	**Results PDF** – Opens the **Results PDF** dialog. Add a logo, chose the options required and produce a PDF of the results, including a system image and pump graphs. This button is only visible on the tool bar after clicking the **Calculate** button.
	**Show Log** – Opens the **Results Log** dialog. The **Results Log** dialog indicates whether the pipe system was solved. This button is only visible on the tool bar after clicking the **Calculate** button.
	**Example Systems** – Opens the **Example Systems** dialog. Over 40 examples systems are provided to illustrate usage of the Pipe Flow Expert software. This button is only visible on the tool bar if design mode is selected.
	**Next Example** – Loads the **Next Example System** from the examples systems dialog. This button is only visible on the tool bar if design mode is selected.
	**New Tab** – Opens a new design sheet allowing users to work on multiple systems simultaneously. Tabbed Sheets work in both Design View and Results Mode. Switch between different system models with a single click

Keyboard Shortcuts

Various keyboard shortcuts are provided which duplicate the actions of some Menu system options and some of the Tool Bar buttons.

- CTRL+N, for New Tab
- CTRL+X, for Cut
- CTRL-C, for Copy
- CTRL-V for Paste
- CTRL-DEL key for delete operations.
- CTRL+O for Open.
- CTRL+S for Save.
- CTRL+P for Print.
- CTRL+ALT+O for Open Examples.
- CTRL+R for Rotate +90 Degrees
- CTRL+ALT+R for Rotate -90 Degrees
- CTRL+M for Mirror Items Left/Right.
- CTRL+I for Invert Items Up/Down.
- CTRL+Z for Zoom to fit on screen

Node Pane

The Node pane displays the properties of the node selected on the pipe system in the Drawing pane. The type of node selected in the Type field determines which features are available in the Node pane.

Node Types

A node can be a join point, tank, or demand pressure–referred to as End Pressure in the Node pane. The type of node selected is shown in the Type field. The type of node can be changed by selecting from the options shown in drop down type list.

Nodes are located at the beginning and end of a pipe and at the junction of pipes in a pipe system. You can use the drop down Node Id list next to the Node name field to select an individual node or to scroll through each of the nodes in the pipe system.

Click the zoom button next to the drop down Node Id list to redisplay the drawing with the selected node at the center of the visible drawing area.

The Node pane is also used to enter and edit the specific details for the selected node. The values and units displayed in the node pane are determined by the units selected on the Units tab in the Configuration Options dialog. The selected units can also be switched on mass between a predefined set of imperial and metric units.

Tank Node Data

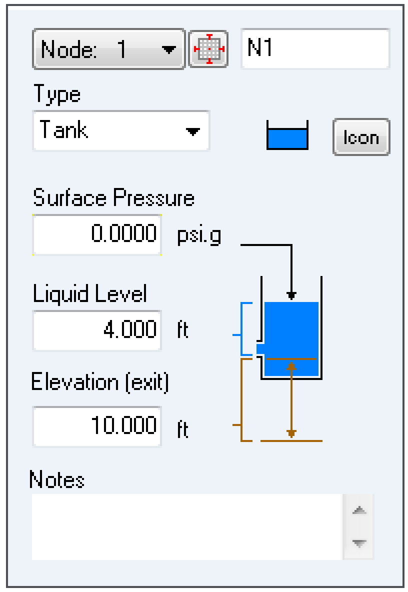


Figure 13 Tank Node Pane

Feature	Description
Node	The name of the node currently selected in the Drawing pane. Use the **Node** field to edit the node name.
Node Id List	Use the **Node Drop Down List** to select a node or to scroll through each of the nodes in the pipe system.
Type (Tank)	The type of node currently selected in the Drawing pane. Use the **Type** field to modify the node type. There are three types of nodes – join point, tank, or fixed pressure demand (End Pressure).
Type (Tank Icon)	Displays the image representing the selected tank in the Drawing pane. Use the **Icon** button to select the tank image you want displayed on the pipe system drawing. The **icon size** can be selected from a range of scales. The image you select does not affect any of the tank's properties or values. 
Surface Pressure	The surface pressure of the fluid in the tank expressed in the units displayed.
Liquid Level	The amount of fluid above the tank exit point expressed in the units displayed.
Elevation (Exit from tank)	The elevation at the exit point of the tank expressed in the units displayed.
Notes	Notes about the node currently selected in the Drawing pane. Notes can be up to 200 characters.

End Pressure Data

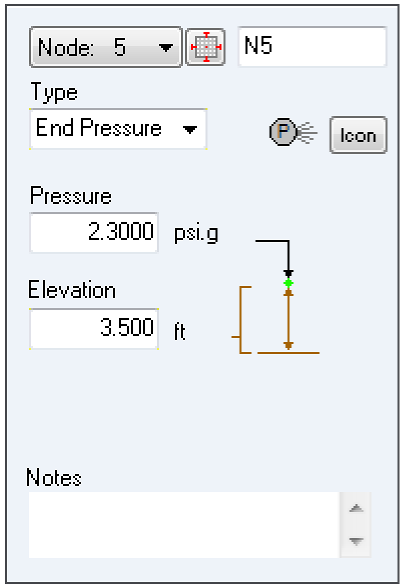


Figure 14 End Pressure Node Pane

Feature	Description
Node	The name of the node currently selected in the Drawing pane. Use the **Node** field to edit the node name.
Node Id List	Use the **Node Drop Down List** to select a node or to scroll through each of the nodes in the pipe system.
Type (End Pressure)	The type of node currently selected in the Drawing pane. Use the **Type** field to modify the node type. There are three types of nodes – join point, tank, or demand pressure (End Pressure).
Type (End Pressure Icon)	Displays the image representing the selected demand pressure in the Drawing pane. Use the **Icon** button to select the end pressure image you want displayed on the pipe system drawing. The **icon size** can be selected from a range of scales. The image you select does not affect any of the demand pressure's properties or values. 
Pressure	The required pressure for the end pressure expressed in the units displayed.
Elevation (of Pressure Point)	The elevation of the end pressure expressed in the units displayed.
Notes	Notes about the node currently selected in the Drawing pane. Notes can be up to 200 characters.

Join Point Data

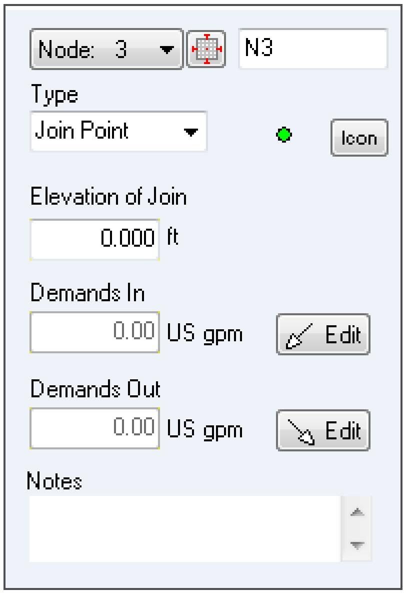
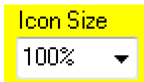


Figure 15 Join Point Node Pane

Feature	Description
Node Identification Number	Use the **Node Drop Down List** to select a node or to scroll through each of the nodes in the pipe system.
Node	The name of the node currently selected in the Drawing pane. Use the **Node** field to edit the node name.
Type (Join Point)	The type of node currently selected in the Drawing pane. Use the **Type** field to modify the node type. There are three types of nodes – join point, tank, or demand pressure (End Pressure).
Type (Join Point) Icon	Displays the image representing the join point in the Drawing pane. Use the **Icon** button to select the join point image you want displayed on the pipe system drawing. The **icon size** can be selected from a range of scales. The image you select does not affect any of the joint point properties or values. 
Elevation of Join Point	The elevation of the join point expressed in the units displayed.
Demands In (at Join Point)	View the **In Flow** to the system at this node. Click edit to change.
Demands Out (at Join Point)	View the **Out Flow** from the system at this node. Click edit to change.
Edit buttons (Join Point)	Opens the **Set Flow Demands** dialog. This dialog is used to set a demand flow on the join point.
Notes	Notes about the node currently selected in the Drawing pane. Notes can be up to 200 characters.

Flow Demands

Flow demands can only be added at a node if the type selected is a Join Point.

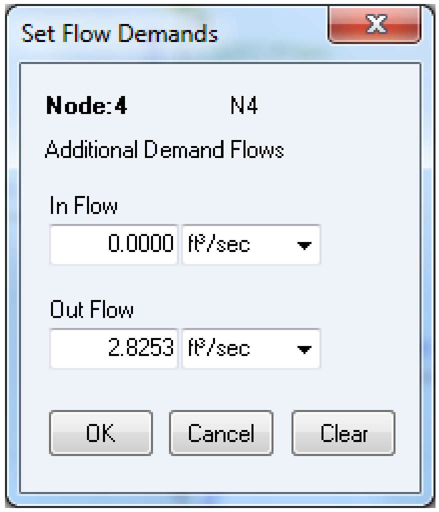


Figure 16 Flow Demands

Feature	Description
In Flow Rate Entering The System	The flow rate entering the system at this node expressed in the units displayed.
Out Flow Rate Leaving The System	The flow rate leaving the system at this node expressed in the units displayed.
OK	Confirm any changes that have been made.
Cancel	Close the dialog without making any changes.

Pipe Pane

A pipe can have many features, such as name, length, internal diameter, roughness, nominal size, material, schedule or class reference, internal volume and surface area.

The data for some of these features, such as name, length, internal diameter and roughness can be entered directly into the pipe pane input boxes (other features would be left unchanged and may have to be amended at a later time).

To set all the features of a pipe use the Diameter and Material buttons to access the selection dialogs.

Pipe Flow Expert allows fittings and valves, components, flow control valves and pumps to be added to a pipe. The selection dialogs to add or change the data for these items can be accessed by clicking on the appropriate button in the pipe pane.

Pipe Features

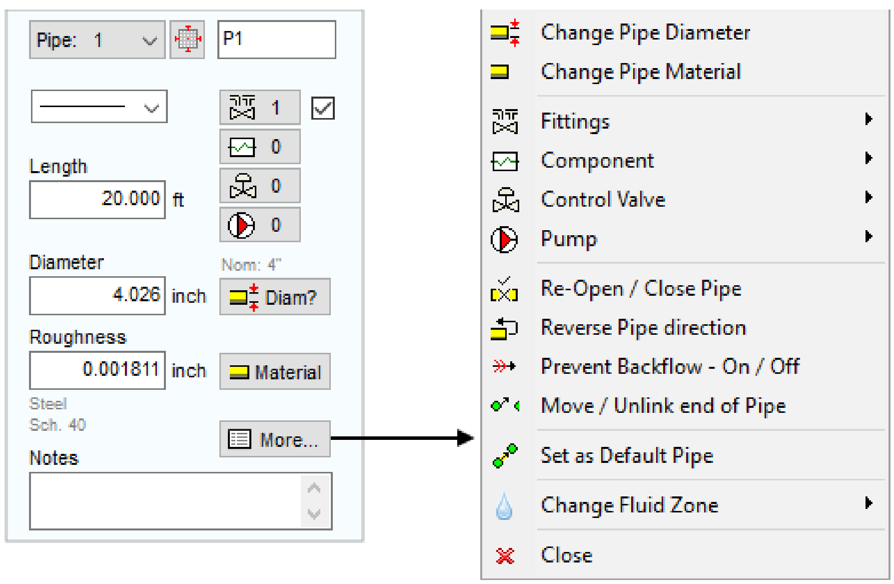


Figure 17 Pipe Pane and Pipe Pop-up menu

Feature	Description
Pipe Identification Number	Use the **Pipe Drop Down List** to select a pipe or to scroll through each of the pipes in the pipe system.
Name	The name of the pipe currently selected in the Drawing pane. Use the Name field to edit the pipe name.
Length	The length of the pipe currently selected in the Drawing pane. Use the length field to change the length of the pipe.
Internal Diameter	The internal diameter of the pipe currently selected in the Drawing pane. Use the internal diameter field to change the internal size of the pipe. Click the Diam? button to display the Pipe Diameter Database sizes.

Roughness	The internal roughness of the pipe currently selected in the Drawing pane. Use the roughness field to change the internal roughness of the pipe. Click the Material button to display the Pipe Material Database.
Notes	Notes about the pipe currently selected in the Drawing pane. Notes can be up to 200 characters.
Pipe Color and Line Width	Set the color and line width of the pipe currently selected in the Drawing pane.
Add/Change Fittings button	Opens the **Pipe fitting friction coefficients** database. Use the database to add and maintain fittings on a pipe. The number of fittings on the pipe is displayed next to the fitting image on the **Add/Change Fitting button**.
Add/Change Component Pressure Loss button	Add a component to the pipe currently selected in the drawing pane. Change the Pressure Loss characteristics for the component. A number is displayed next to the Component image to indicate if the component is active.
Add/Change Control Valve button	Add a control valve to the pipe currently selected in the drawing pane. Set the maximum flow rate in the pipe by using a flow Control Valve (FCV), or set the pressure at the end of the pipe by using a Pressure Reducing Valve (PRV), or set the pressure at the start of the pipe by using a Back Pressure Valve (BPV). A number is displayed next to the Control Valve image to indicate if the Control Valve is active.
Add/Change Pump button	Add a pump to the pipe currently selected in the Drawing pane. Enter the flow rate and head characteristics for various points on the pump performance curve. A number is displayed next to the Pump image to indicate if the Pump is active.
Include check boxes	Include/Exclude status for various items added to the pipe currently selected in the Drawing pane. **i.e. Fittings & Valves, Components, Control Valves, Pumps.**
Diam?	Display the Pipe Diameter Size Database. A **Double Click** on a pipe will also display the Pipe Diameter Size Database.
Material	Display the Pipe Material Database.
More….	Opens the pipe sub menu which provides further options to: • Change Pipe Diameter • Change Pipe Material • Edit, Copy, Paste & Delete Fittings, Components, Control Valves, and Pumps • Re-Open / Close Pipe • Reverse Pipe Direction • Prevent Backflow – On / Off • Move / Unlink end of Pipe • Use Pipe Values for as default for the next pipe to be drawn • Change Fluid Zone (for pipe) • Close sub menu A **Right Click** on a pipe will also display the pipe sub menu.

Pipe Material Data

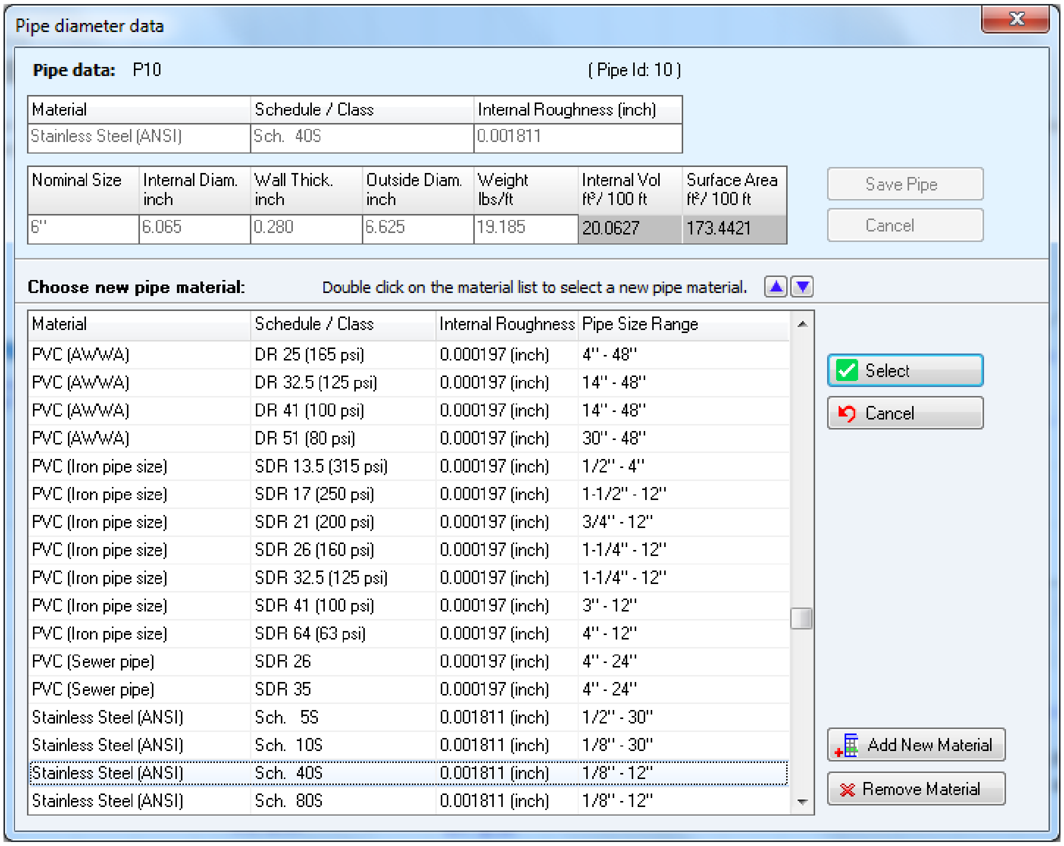


Figure 18 Pipe Material Data

Feature	Description
Material	The material description of the pipe currently selected in the Drawing pane.
Schedule/Class	The Schedule/Class description of the pipe currently selected in the Drawing pane.
Internal Roughness	The internal roughness of the pipe currently selected in the Drawing pane.
Nominal Size	The Nominal Size description of the pipe currently selected in the Drawing pane.
Internal Diam.	The internal diameter of the pipe currently selected in the Drawing pane.
Wall Thick	The wall thickness of the pipe currently selected in the Drawing pane.
Outside Diam	The outside diameter of the pipe currently selected in the Drawing pane.
Weight	The weight per unit length of the pipe currently selected in the Drawing pane.
Internal Vol.	The internal volume (ft^3/100 ft or m^3/100m) of the pipe currently selected in the Drawing pane.
Surface Area	The external surface area (ft^2/100 ft or m^2/100m) of the pipe currently selected in the Drawing pane.
Save Pipe	This button is not available during pipe material selection.
Cancel (Changes)	This button is not available during pipe material selection.
Change Material	Show Listing of available pipe data materials.

Select	Show size data for selected material.
Cancel	Revert to previously selected size listing.
Add New Material	Create a new material category to which new pipe sizes can be added.
Remove Material	Remove a material category.

Pipe Diameter Data

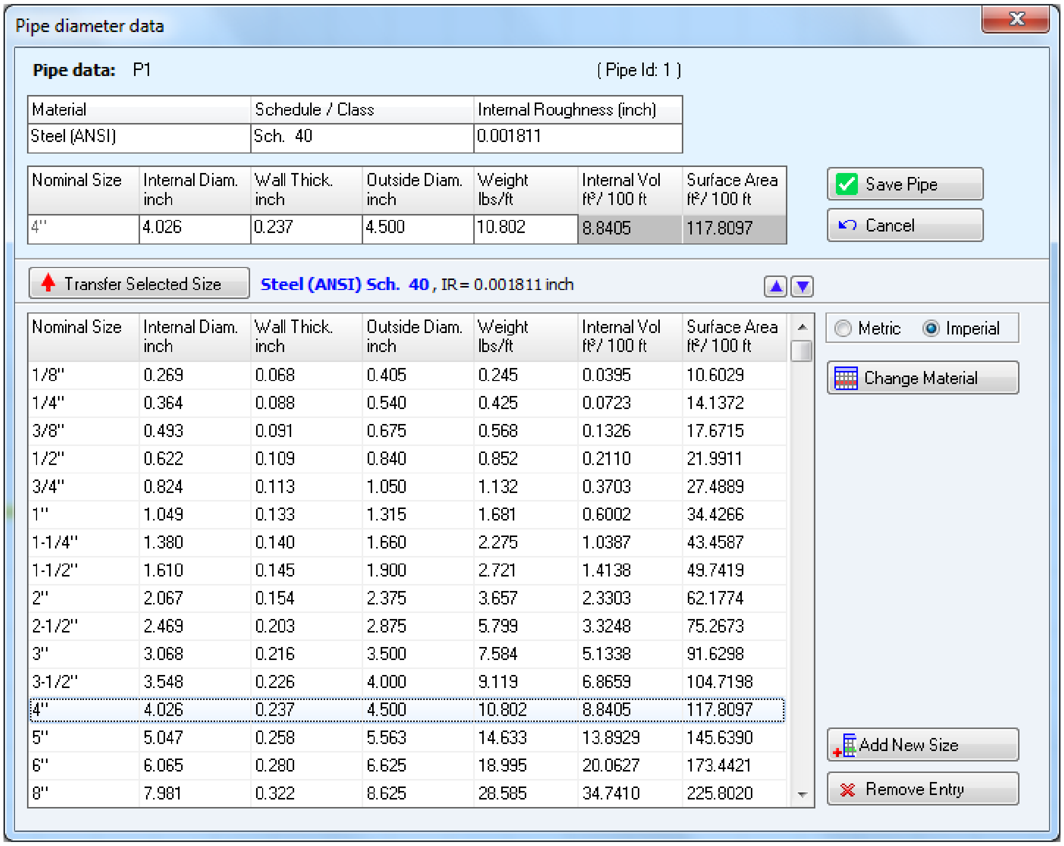


Figure 19 Pipe Diameter Data

Feature	Description
Material	The material description of the pipe currently selected in the drawing pane.
Schedule/Class	The Schedule/Class description of the pipe currently selected in the drawing pane.
Internal Roughness	The internal roughness of the pipe currently selected in the drawing pane.
Nominal Size	The Nominal Size description of the pipe currently selected in the drawing pane.
Internal Diam.	The internal diameter of the pipe currently selected in the drawing pane.
Wall Thick	The wall thickness of the pipe currently selected in the drawing pane.
Outside Diam	The outside diameter of the pipe currently selected in the drawing pane.
Weight	The weight per unit length of the pipe currently selected in the drawing pane
Internal Vol.	The internal volume (ft^3/100 ft or m^3/100m) of the pipe currently selected in the drawing pane.
Surface Area	The external surface area (ft^2/100 ft or m^2/100m) of the pipe currently selected in the drawing pane.
Save Pipe	Save the selected data to the pipe currently selected in the drawing pane.

Cancel	Close the Pipe Diameter database without making any changes.
Transfer Selected Size	Copy size data from data listing to entry boxes (at top of the screen).
Size data listing	Listing of pipe sizes.
Metric or Imperial	Display size data in metric or imperial units.
Change Material	Display data dialog for pipe material.
Add new size	Display input boxes for new pipe size data.
Remove entry	Remove a pipe size.

Pipe Fittings Database

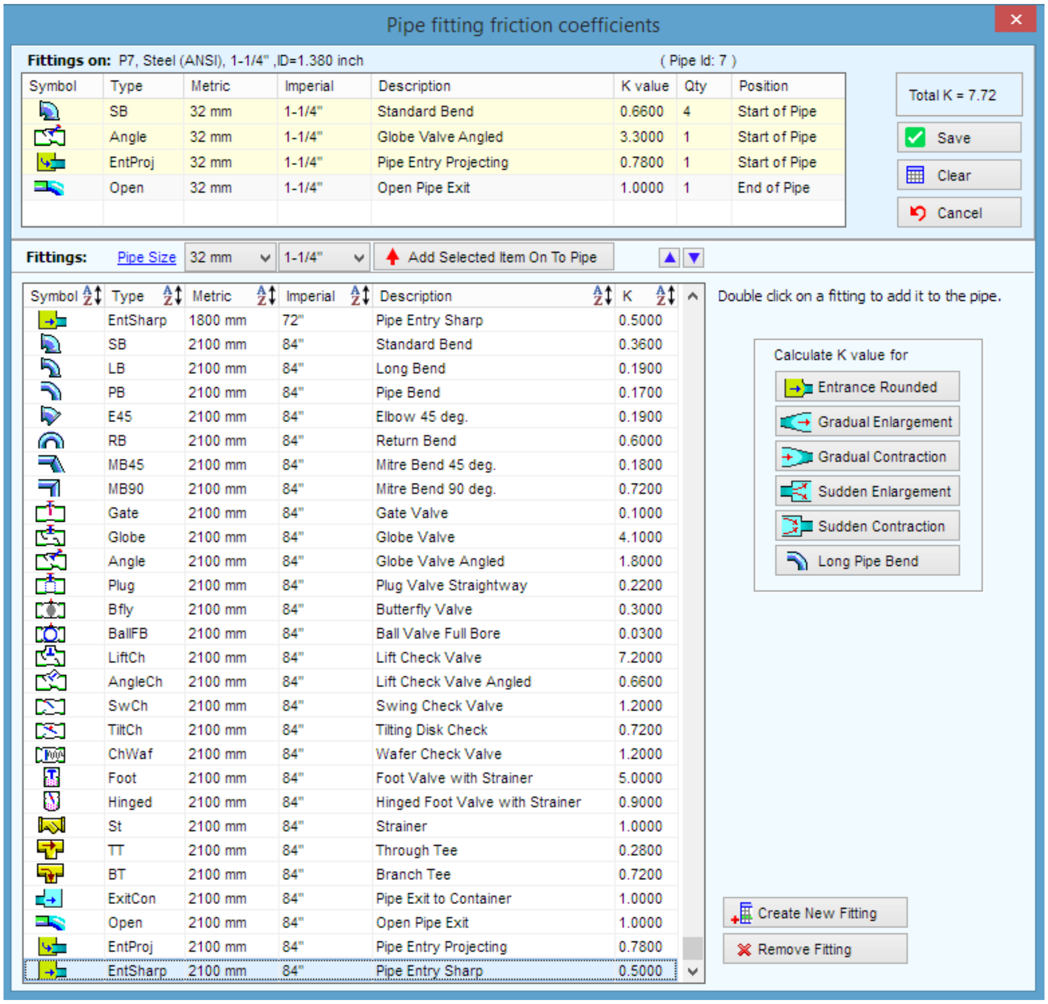


Figure 20 Pipe Fittings Database

Feature	Description
Symbol	Fitting type symbol.
Type	Fitting type short code.
Metric	Nominal metric fitting size.
Imperial	Nominal imperial fitting size.
Description	Fitting description.
K value	Local loss coefficient of fitting.
Qty	Quantity of fittings.
Position	Fitting position. Define to be at Start or End of pipe.
Save	Save the fitting selections to the pipe currently selected in the drawing pane.
Clear	Clear the fitting selections.
Cancel	Close the Pipe Fittings Database without making any changes.
Pipe Size	Click to display fittings to match the current pipe size.

Metric Pipe Sizes (mm)	Drop down list of metric pipe sizes, click to display matching fitting sizes.
Imperial Pipe Sizes (inch)	Drop down list of imperial pipe sizes, click to display matching fitting sizes.
Fitting Database	Listing of fittings and friction coefficients.
Add Selected Item On To Pipe	Copy fitting data from listing to the list on the currently selected pipe.
Calculate K value	Calculate local loss coefficient for: entrance rounded, gradual enlargement, gradual contraction, sudden enlargement, sudden contraction or long pipe bend.
Create new fitting	Create data for non standard fittings.
Remove entry	Remove data from fittings database.

Component Pressure Loss

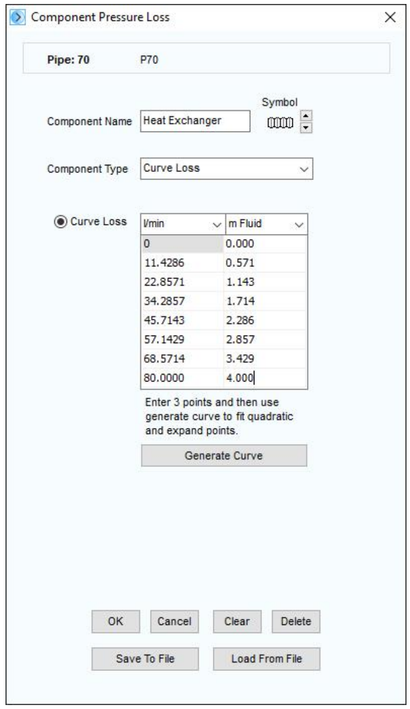


Figure 21 Component Pressure Loss

Feature	Description
Component Name	Name of the component.
Symbol Scroll buttons	Choose symbol for the component.
Component Type	Choose the type of pressure loss component (Fixed Loss, Curve Loss, Cv Value, Kv Value, Sprinkler K (metric), Sprinkler K (imperial), Orifice). Inputs applicable for defining the chosen component type are displayed. See later section about **COMPONENTS** for further details.
OK	Add the component to the pipe currently selected in the Drawing pane.
Cancel	Close the dialog without making any changes.
Clear	Clear the pressure loss data.
Delete	Remove the component from the pipe currently selected in the drawing pane.
Save To File	Save the component details to a file.
Load From File	Load the component details from a file.

Control Valve Data

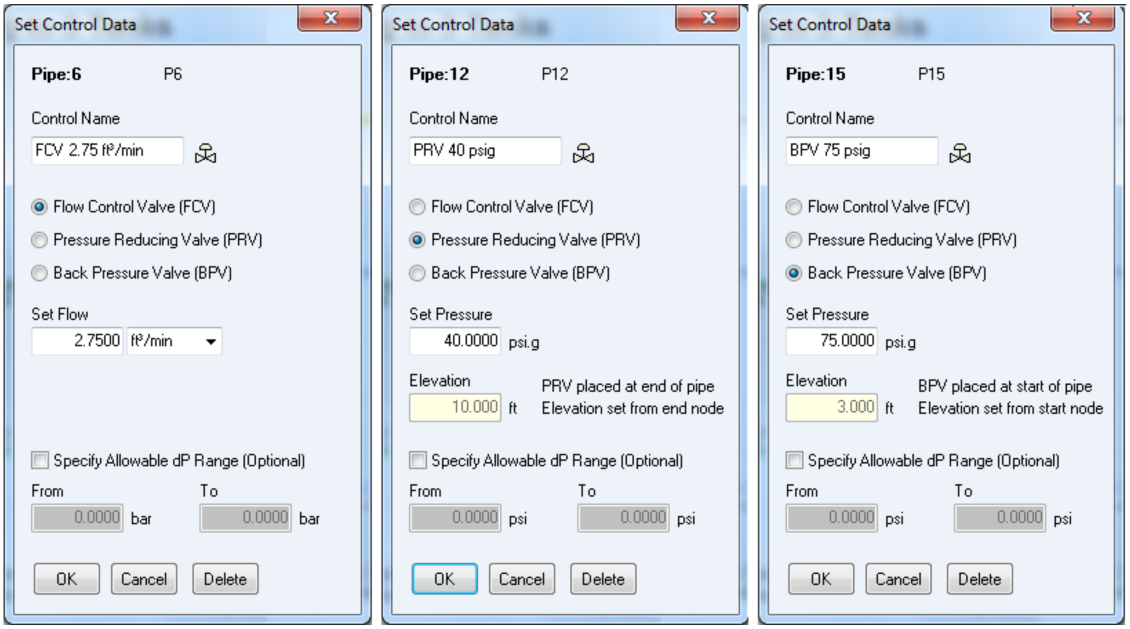


Figure 22 Control Valve Data

Feature	Description
Control Name	Name of the control valve or P&ID number.
Flow Control Valve	Set control flow rate in the units displayed.
Pressure Reducing Valve	Set pressure required at the end of the pipe.
Back Pressure Valve	Set back pressure required at the start of the pipe.
Specify Allowable dP Range	Set allowable dP Range, a warning will be issued if the actual dP is outside the lower and upper values.
OK	Add the control valve type to the pipe currently selected in the Drawing pane.
Cancel	Close the dialog without making any changes.
Delete	Remove the control valve from the pipe currently selected in the Drawing pane.

Pump Data

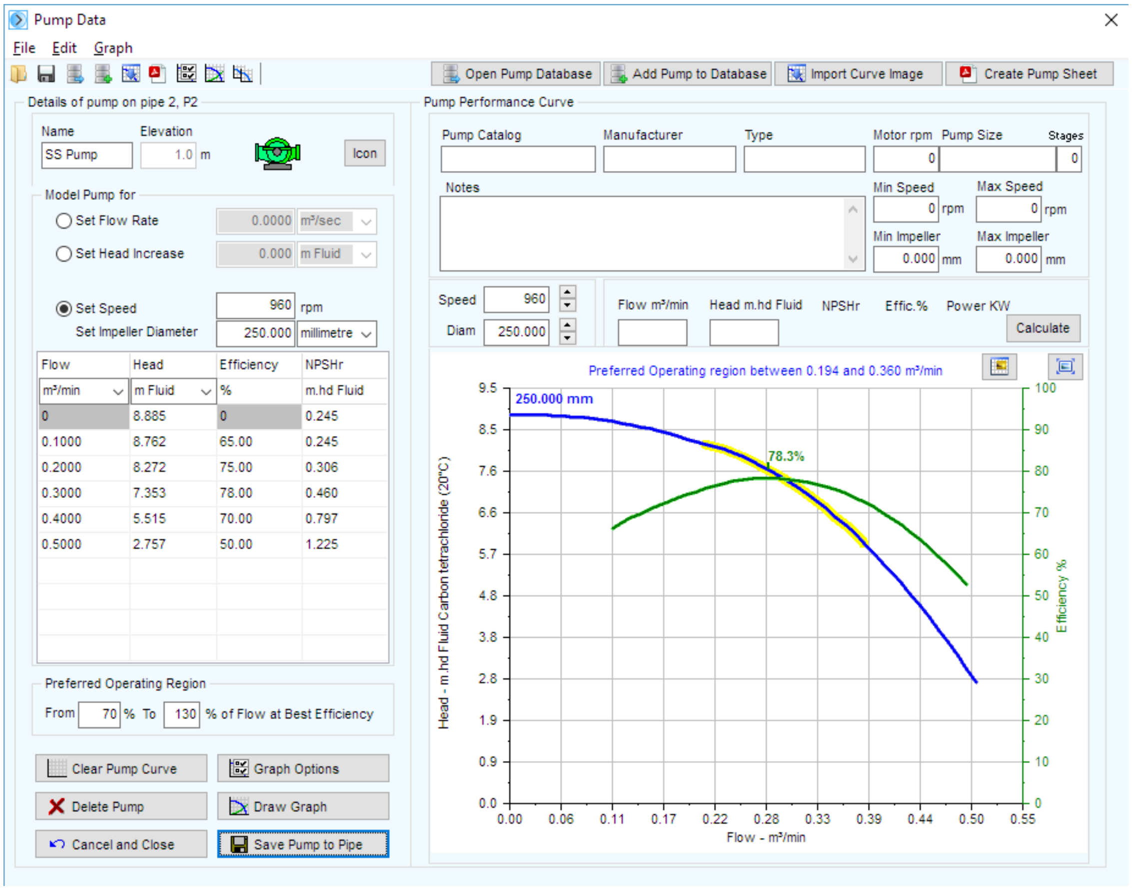


Figure 23 Pump Data

The main features of the pump data dialog are listed in the table below. The pump section later in this document provides further details of the Graph Options, the Pump Data Menu options and the options for opening the Pump Database, adding a pump to the Pump Database, Import of a Curve Image and creating a Pump Data Sheet.

Feature	Description
Name	Name of the pump.
at Elevation	Elevation of the pump.
Pump Scroll buttons	Choose symbol for the pump.
Set Flow Rate	Enter flow rate to model a pump with a fixed flow rate
Set Head Increase	Enter a head or pressure to model the additional motive force from a pump
Set Speed (rpm)	Operating speed of a pump with a performance curve.
Set Impeller Diameter	Set impeller diameter size & units
Flow	Flow rate values for operating range of pump. Displayed in units selected from drop down list.
Head	Head generated for each of the flow rate entries. Displayed in units selected from drop down list.

Efficiency	Efficiency of the pump at each of the operating data points.
NPSHr	The Net Positive Suction Head requirement at the inlet of the pump for the flow rate value.
From (% of BEP)	Start of preferred operating region.
To (% of BEP)	End of preferred operating region.
Clear Pump Curve	Clear the pump data.
Delete Pump	Remove the pump from the pipe currently selected in the drawing pane.
Cancel and Close	Close the dialog without saving any changes
Graph Options	Opens additional graph options customization dialog. Allows creation of additional data curves and adjustment of Iso efficiency lines. See later section about **Pumps** for further details.
Draw Graph	Draw a pump graph based on entered data points.
Save Pump to Pipe	Add the pump to the pipe currently selected in the drawing pane.
Pump Catalog	The name of the pump catalog.
Manufacturer	The name of the pump manufacturer.
Type	The type of pump.
Motor rpm	The motor speed.
Pump Size	The pump size reference.
Stages	The number of pump stages.
Notes	Notes about the pump. Notes can be up to 200 characters.
Min Speed	The minimum operating speed of the pump.
Max Speed	The maximum operating speed of the pump.
Min Impeller	The minimum impeller diameter for the pump.
Max Impeller	The maximum impeller diameter for the pump.
Speed (rpm)	Operating speed of the pump impellor diameter for the duty curve.
Diam	The diameter of the pump impeller, in the units shown.
Flow	The flow rate of the pump duty point, in the units shown.
Head	The head produced by the pump at the duty point, in the units shown.
NPSHr	The Net Positive Suction Head requirement at the inlet of the pump for the flow rate at the duty point.
Effic. %	The pump efficiency at the duty point.
Power	The shaft power required at the duty point.
Calculate (Button)	Click to calculate the duty point details for an entered flow rate or head requirement.
Logo (Button)	Add a logo to the pump graph image
Expand (Button)	Expand graph to show in full window

Drawing Pane

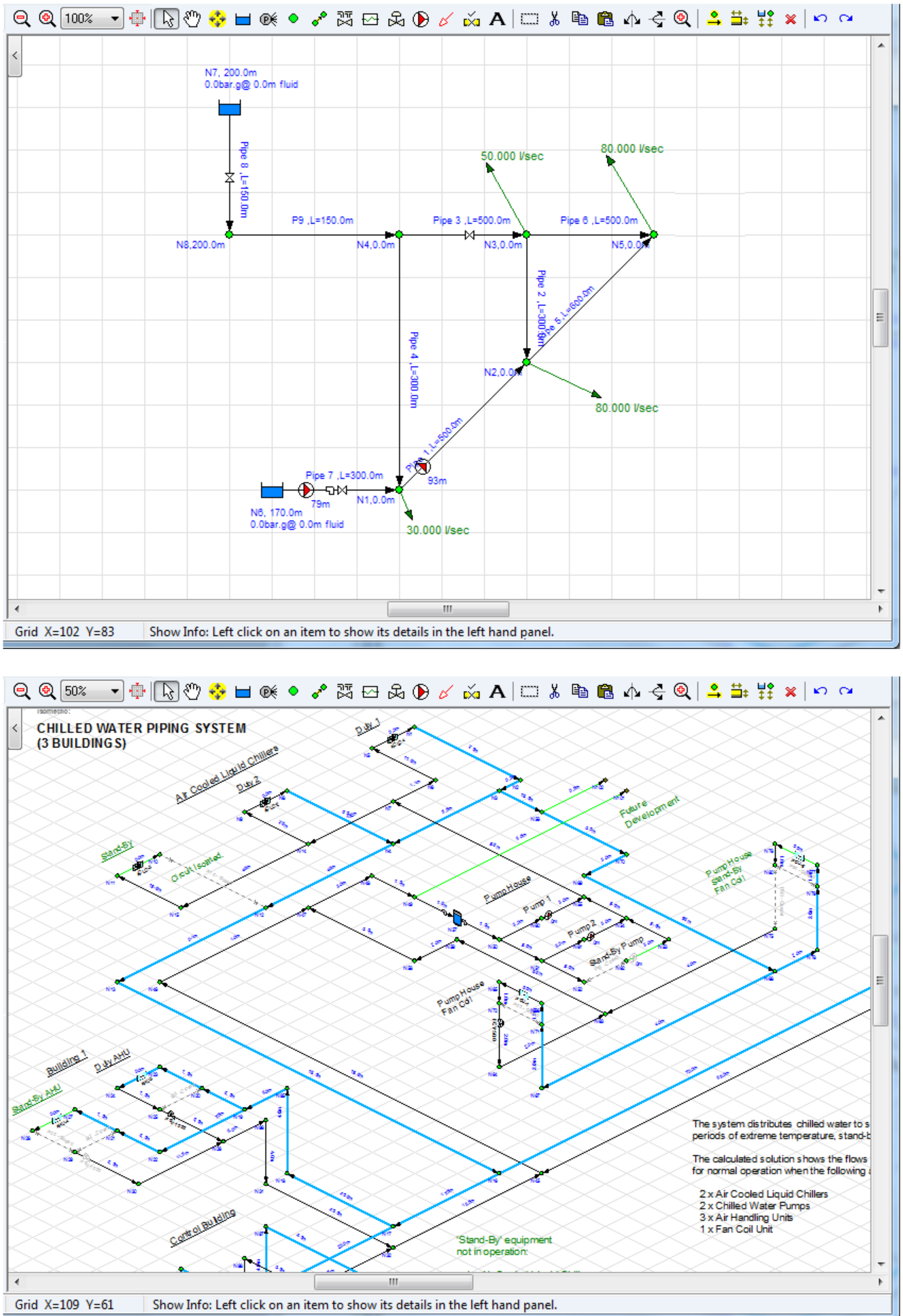


Figure 24 Drawing Panes Standard or Isometric

Feature	Description
Drawing Grid	The drawing grid shows details of the pipeline system. The information to be displayed, such as size, length, elevation, flow rate etc. can be selected from **the configuration options labelling tab**.
Hint Feature	The **Hint** pane, located above the tool bar, displays tips for using the button currently selected on the tool bar.
Zoom Feature	If your mouse is suitable the drawing size can be zoomed larger or smaller using the mouse scroll wheel.
Drawing Size Display	The drawing size can be selected from the drop down listing between the **Zoom Out** and the **Zoom In** buttons.
Zoom Out Button	Click to make the drawing size smaller by one step.
Zoom In Button	Click to make the drawing size larger by one step.
Zoom drawing to fit on screen	Click to fit all of the drawing on the screen.

Pipe Flow Expert offers a choice of a drawing grid with square increments for standard designs or an isometric grid for use where it is desirable to create an isometric view.

It is possible to toggle between standard / isometric design grids.

Moving from a standard grid to an isometric grid transposes the standard drawing on to a flat isometric plane. A user can then adjust/move the nodes as necessary to produce a drawing that looks 3D.

Moving from an isometric grid to a standard grid transposes the isometric drawing to a flattened view that produces a near plan view (except that PipeFlow Expert does not allow nodes to appear at the same point on the grid and therefore nodes directly above or below each other on the isometric grid are offset at a diagonal angle when transposed to the flat view).

The system design can be amended in either drawing mode.

Configuration Options Screen

Pipe Flow Expert can be configured to suit your requirements.

- Labels: The labels that are shown on the drawing can be chosen.
- Units: The units in which to display lengths, diameters, head loss, velocity etc. can be chosen.
- Pipe Data: The default pipe data can be chosen. These details are used to draw any new pipes.
- Pipe Data: Various default pipe attributes may be copied to a range of selected pipes.
- Node Data: An elevation value and a node image may be copied to a range of selected nodes. Demand In / Out Flow values can be set or adjusted.
- Results Colors: The results screen drawing can be displayed using a color gradient, to indicate the range of flow rates, fluid velocities, friction loss, hydraulic gradient line and elevations etc.
- Calculations: The calculation parameters and tolerances can be changed (if necessary).
- Editor: The editor preferences can be configured.

Labels Tab

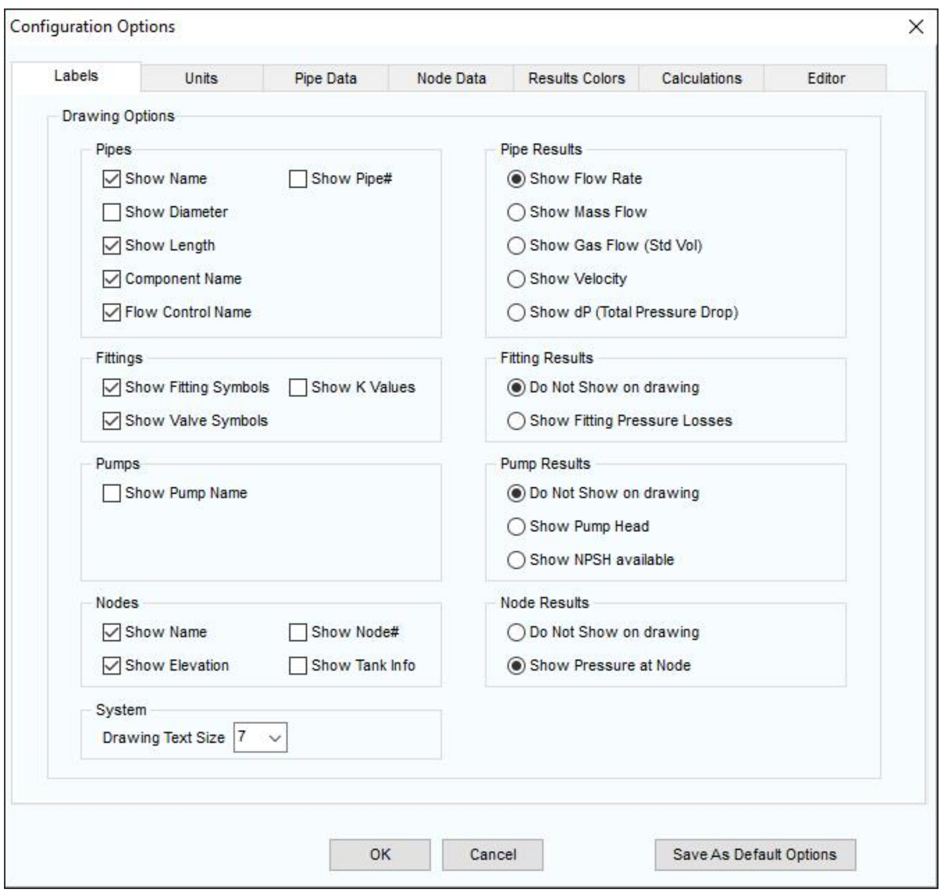


Figure 25 Configuration Options - Labels Tab

Feature	Description
Show Name (Pipes)	Show name of the pipe on the drawing.
Show Diameter	Show diameter of the pipe on the drawing.
Show Length	Show length of the pipe on the drawing.
Show Pipe #	Show the pipe number on the drawing.
Component Name	Show component name on the drawing.
Flow Control Name	Show flow control name on the drawing.
Show Values	Show valve symbol on the drawing, if valve is present on pipe.
Show Total K Value	Show total K value on the drawing, for all fittings on the pipe.
Show Bends	Show bend symbol on the drawing, if bend is present on pipe.
Show Pump Name	Show pump name on the drawing.
Show Name (Node)	Show the name of node on the drawing.
Show Elevation (Node)	Show the node elevation on the drawing.
Show Tank Info	Show the Tank Information on the drawing.
Show Node #	Show the pipe number on the drawing.
Pipe Results **Show Flow Rate**	Show the volume flow rate on drawing.
Show Mass Flow	Show the mass flow rate on drawing.
Show Velocity	Show the fluid velocity on the drawing.
Show Total Pressure Drop	Show the total pressure drop of the pipe, the fittings and the component on the drawing.
Fitting Results **Do Not Show on drawing**	Do not show the total pressure drop of the fittings on the drawing.
Show Pressure Loss through fittings	Show the pressure drop of the fittings on the drawing.
Pump Results **Do Not Show on drawing**	Do not show the pump results data on the drawing.
Show Pump Head	Show the pump head on the drawing.
Show NPSH available	Show the net positive suction head available at the pump inlet on the drawing.
Node Results **Do Not Show on drawing**	Do not show the pressure at the nodes.
Show Pressure at Node	Show the pressure at the nodes on the drawing.
Save As Default Options	Save the current Labels, Units, Pipe Data, Results Colors & Editor options as defaults for a new system. **See System Options – Default Options for further information.**
OK	Confirm changes to configuration options.
Cancel	Close the labelling tab without making any changes.

Units Tab

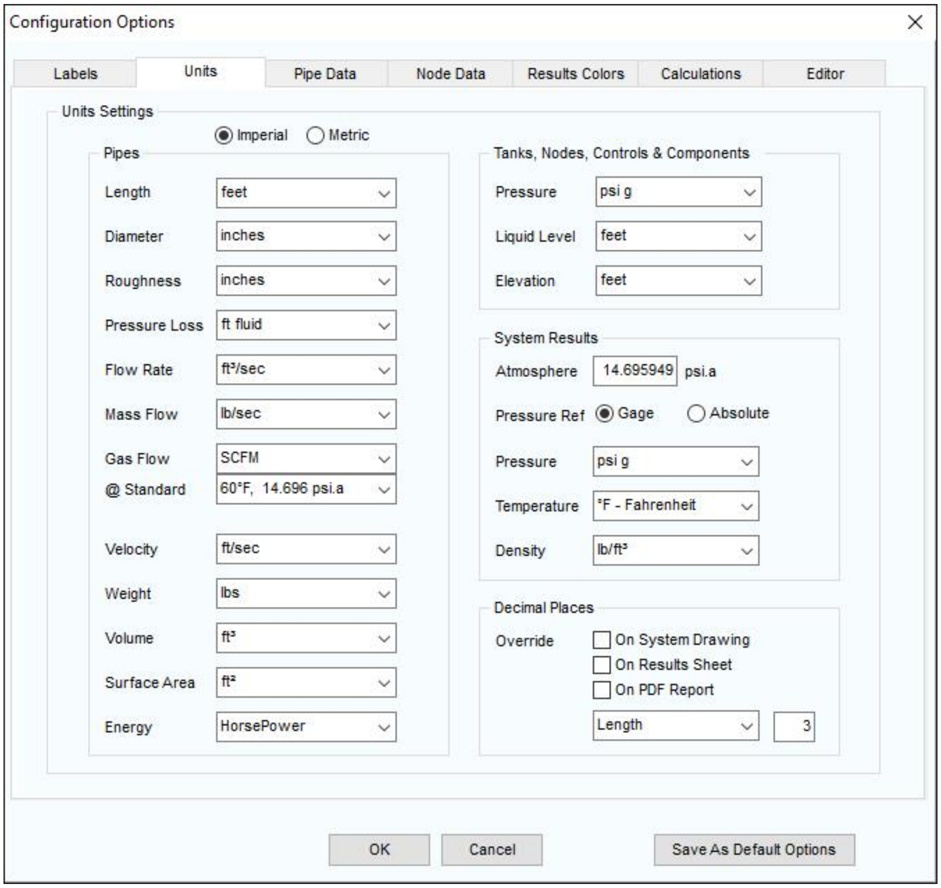


Figure 26 Configuration Options - Units Tab

Feature	Description
Length	Select the units for display of the pipe lengths.
Diameter	Select the units for display of the pipe internal diameters.
Roughness	Select the units for display of the pipe internal roughness.
Head Losses	Select the units for display of the head losses
Flow Rate	Select the units for display of the flow rates.
Mass Flow	Select the units for display of the mass flow rates
Gas Flow	Select the units for display of the gas flow rates.
@Standard	Select the standard conditions to calculate standard gas flow at.
Velocity	Select the units for display of the fluid velocities.
Weight	Select the units for display of the pipe weights.
Volume	Select the units for display of the internal pipe volumes.
Surface Area	Select the units for display of the surface areas of the pipes.
Energy	Select the units for display of the energy usage
Tanks, Nodes, Control & Components: **Pressure**	Select the units for display of pressure.

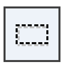
Liquid Level	Select the units for display of the fluid liquid level.
System Results: Elevation	Select the units for display of the elevations of the nodes.
Atmosphere	Value of standard atmosphere in absolute pressure units.
Pressure Ref.	Select Gage or Absolute pressure units.
Pressure	Select the units for display of the pressures at join points (nodes).
Temperature	Select option for display of the fluid temperatures.
Density	Select the units for display of the fluid density
Imperial or Metric	Select general option for Imperial or Metric displays.
Decimal Places	Define the number of post decimal places used to display values on the system drawing, results sheet and PDF report.
Save As Default Options	Save the current Labels, Units, Pipe Data, Results Colors & Editor options as defaults for a new system. **See System Options – Save As Default Values for further information.**
OK	Confirm changes to configuration options.
Cancel	Close the dialog without making any changes.

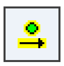
Pipe Settings Tab

If you will be using the same type of pipe throughout the pipe system, you can define the default pipe type using the **Pipe Settings** tab in the **Configuration Options** dialog. When the default pipe values are set, each time you add a pipe to the system, the pipe type and values are automatically defined for the pipe. You can customize the values for an individual pipe in the system by editing its values in the Pipe pane or in the **Pipe diameter data** dialog. The **Pipe diameter data** dialog contains a list of the pipe materials and sizes available in the pipe database and additional properties specific to a pipe in the system.

NOTE: Be sure to enter the value for the pipe's properties in the units displayed next to the field. For example, if you are using metric units, and *m* for meter is displayed next to the Length field, enter the pipe length value in meters.

Various attributes of the default pipe may be copied to the list of selected pipes.





The **Selection Tool** or the **Individual Pipe Selection Tool** may be used to select which pipes are to selected prior to opening the **Pipe Settings** tab. Individual pipes can be added or removed from the selected list by checking or un-checking the box adjacent to the pipe description.

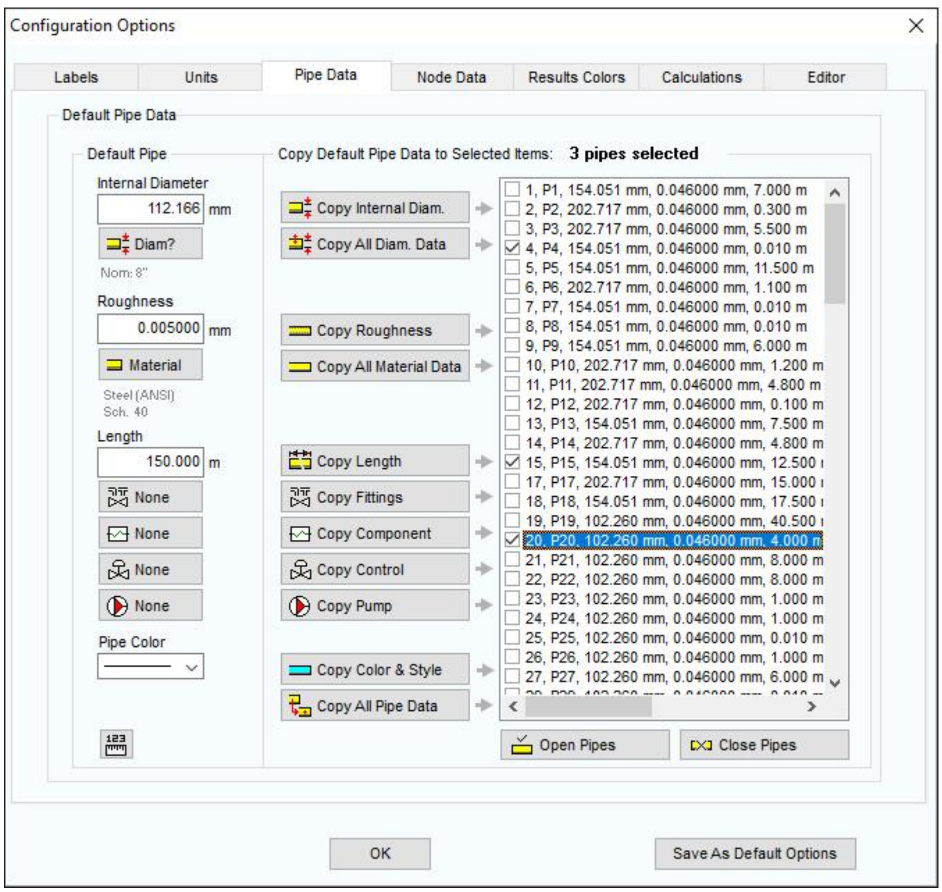


Figure 27 Configuration Options - Pipe Settings Tab

Feature	Description
Internal Diameter	Set the default internal diameter for the next pipe to be added to the drawing.
Diam?	Display the Pipe Diameter Database.
Roughness	Set the default internal roughness for the next pipe to be added to the drawing.
Material	Display the Pipe Material Database.
Length	Set the default pipe length for the next pipe to be added to the drawing.
Add/Change Fittings	Select the default fittings for the next pipe to be added to the drawing.
Add/Edit Component	Set the default data for a component to be added to the drawing.
Pipe Color	Select the pipe color and line width for the default pipe.
Copy Internal Diameter Only	Copy the internal diameter data to the pipes selected in the listing.
Copy All Diameter Data	Copy the internal and outer diameter data to pipes selected in the listing.
Copy Roughness Only	Copy the internal roughness to the pipes selected in the listing.
Copy All Material Data	Copy the pipe material, schedule/class and internal roughness to the pipes selected in the listing.
Copy Length Only	Copy the default pipe length to the pipes selected in the listing.
Copy Fittings Only	Copy the default pipe fittings to the pipes selected in the listing.
Copy Components Only	Copy the default component to the pipes selected in the listing.
Copy Controls Only	Copy the default flow control to the pipes selected in the listing.
Copy Pumps Only	Copy the default pump to the pipes selected in the listing.
Copy Color and Style	Copy the line color and line width to the pipes selected in the listing.
Copy All Pipe Data	Copy the default pipe values, including fittings and valves, component and pumps to the pipes selected in the listing.
Open Pipes	Open the pipes selected in the listing. Pipes which are currently closed will be shown at the end of the listing.
Close Pipes	Close the pipes selected in the listing.
Save As Default Options	Save the current Labels, Units, Pipe Data, Results Colors & Editor options as defaults for a new system. **See System Options – Default Options for further information.**
OK	Close the configuration options pipe updates tab.

Node Updates Tab

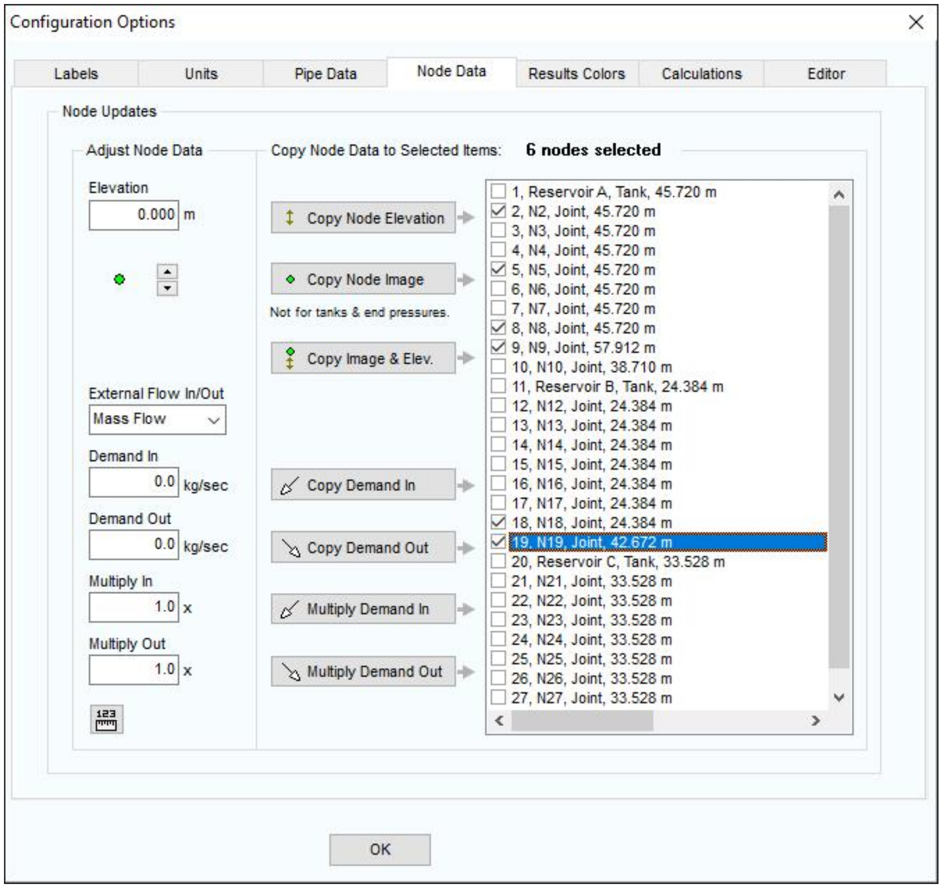


Figure 28 Configuration Options - Node Updates Tab

Feature	Description
Copy Node Elevation	Copy the elevation to the nodes selected in the listing.
Copy Node Image	Copy the node image to the nodes selected in the listing.
Copy Image and Elevation	Copy the node image and the elevation to the nodes selected in the listing.
Copy Demand – Flow In	Copy the additional flow in values entering the system at the node points selected in the listing.
Copy Demand – Flow Out	Copy the additional flow out values leaving the system at the node points selected in the listing.
Multiply Demand In	Multiply the in flow demand values by the specified amount for the nodes selected in the listing.
Multiply Demand Out	Multiply the out flow demand values by the specified amount for the nodes selected in the listing.
OK	Close the configuration options node updates tab.

Results Colors Tab

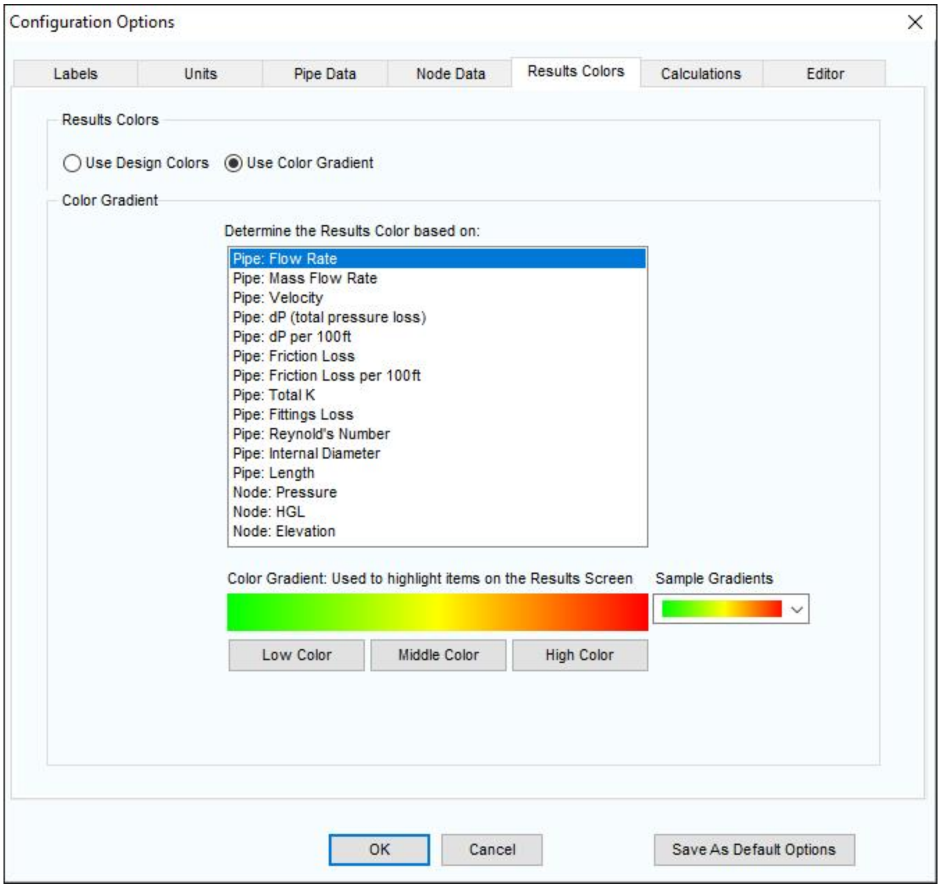


Figure 29 Configuration Options - Results Colors tab

Feature	Description
Use Design Colors / Use Color Gradient	Select option to display results according to the chosen color gradient.
Determine Results Colors based on a Chosen Category.	Select a results category to be displayed using the chosen color gradient.
Sample Gradients	Select a color gradient to be used on the results screen.
Low Color / Middle Color / High Color	Select customized colors to generate a user defined color gradient.
Save As Default Options	Save the current Labels, Units, Pipe Data, Results Colors & Editor options as defaults for a new system. **See System Options – Default Options for further information.**
OK	Close the results colors tab.
Cancel	Close the results colors tab and revert to the previous selection

Calculations Tab

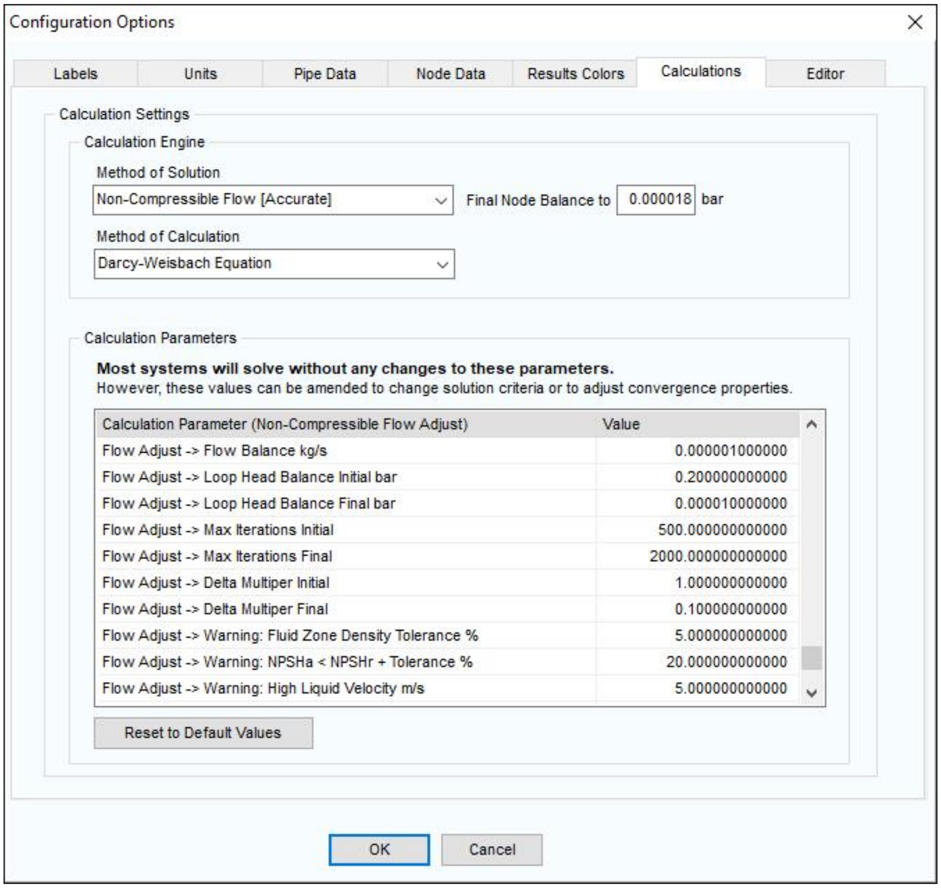


Figure 30 Configuration Options - Calculations Tab

Feature	Description
Method of Solution	Non-Compressible or Compressible flow calculation engine
Method of Calculation	Equation used to calculate friction loss.
Final Node Balance	Balance pressure at all nodes to this accuracy
Calculation Parameters	A set of values that are used during convergence to determine a solution. The actual parameters vary depending on the Method of Solution selected and the Method of Calculation equation that is selected.
Reset All entries to the Default Values	Restore the standard settings for the calculation parameters.
OK	Confirm changes to system calculation options.
Cancel	Close the Calculations Tab without making any changes.

Note: We recommend that you do not change these values, unless it becomes necessary to solve a very finely balanced system.

Editor Tab

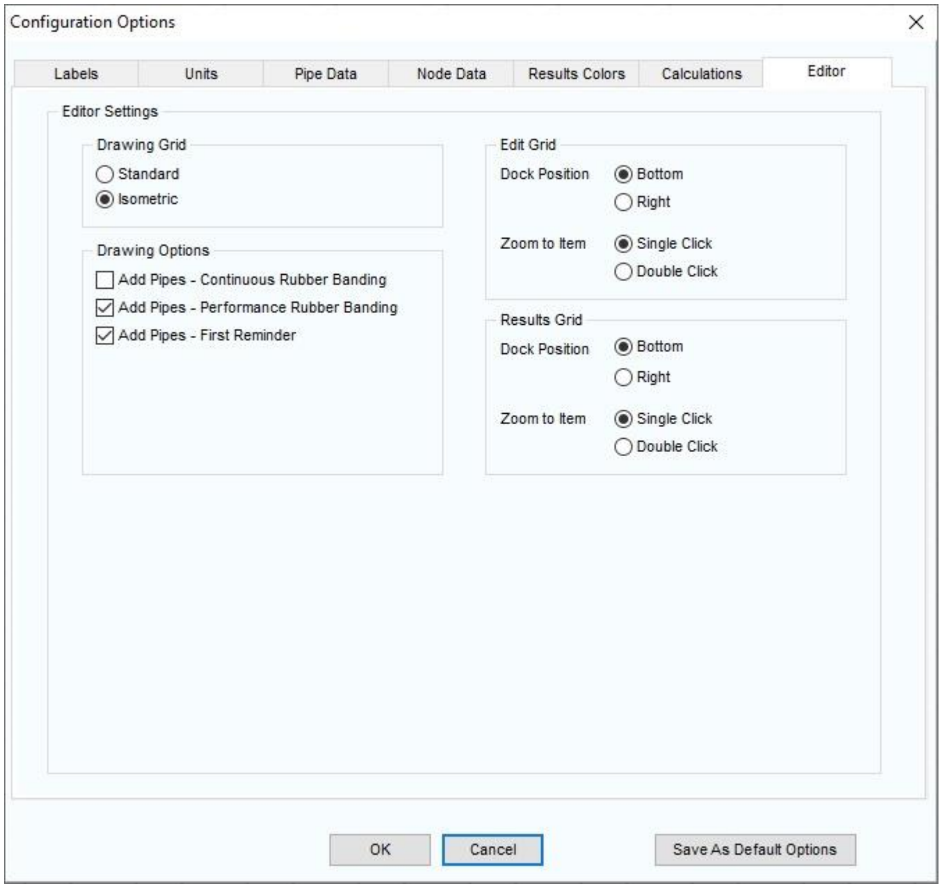


Figure 31 Configuration Options - Editor Tab

Feature	Description
Drawing Grid	Set system drawing to standard or isometric view.
Drawing Options: Add Pipes – Continuous Rubber Banding	Draw pipes using continuous rubber banding.
Drawing Options: Performance Rubber Banding	Draw pipes using performance rubber banding; provides more responsive rubber banding when working with large systems.
Drawing Options: Add Pipes – First Reminder	Display reminder about the default pipe when drawing the first pipe of a new system.
Edit Grid: Dock Position	Position the edit grid at the bottom or right-hand side of the application window.
Edit Grid: Zoom to Item	Set whether the selected item on the edit grid is zoomed to with a single or double mouse click.
Results Grid: Dock Position	Position the results grid at the bottom or right-hand side of the application window.
Results Grid: Zoom to Item	Set whether the selected item on the results grid is zoomed to with a single or double mouse click.
Save As Default Options	Save the current Labels, Units, Pipe Data, Results Colors & Editor options as defaults for a new system. **See System Options – Default Options for further information.**

OK	Confirm changes to editor options.
Cancel	Close the editor options without making any changes.

Results Tables

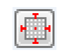
The results of a pipeline system analysis are displayed in the Results tables. Data for flow rates in pipes, velocities in pipes, pressures at join points, tank elevations, hydraulic grade lines, pump operating points and many more calculated results, can all be viewed in the results tables.

The results table interacts with the drawing pane in a two-way manner. If a pipe or a node is selected in the results tables then the pipe or node is selected on the drawing.

To zoom to the highlighted item in the Drawing Pane:

When the Results Grid: Zoom To configuration option is set to Single Click, selecting a cell or row in the results grid will automatically zoom to the highlighted item.

When the Results Grid: Zoom To configuration option is set to Double Click:

1. Select a cell or row in the results grid and click the Zoom to Highlighted item button, , on the tool bar, or

2. Double click in a cell or the row header (the ‘Pipe id’ column) in the edit grid.

If a pipe or node is selected on the drawing then the results table will also change to show the selected item.

The results display units can be altered by choosing the units required from the configuration options units tab. The system results data can also be saved to an Excel spreadsheet file or exported to a new Excel spreadsheet.

Use the tool buttons to carry out the following actions:

- Save the results shown on the active tab to a file.
- Export the active tab results to an Excel spreadsheet (Microsoft Excel is required)
- Paste the active tab results to the Windows clipboard.
- Choose new Unit options.
- Dock the results grid to the base of the drawing window.
- Dock the results grid to the right of the drawing window
- Zoom to the selected pipe or node.

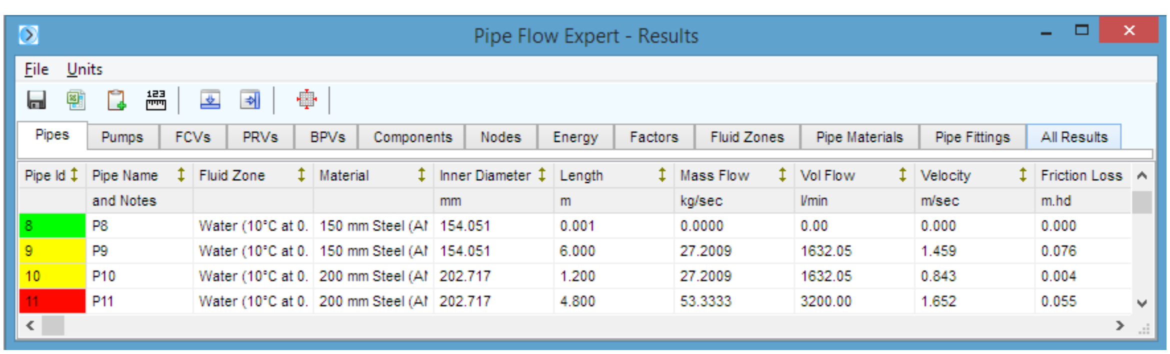


Figure 32 Results tables

Feature	Description
File menu	Select an option to save results in Excel format or to export the results to a new Excel spreadsheet.
Units menu	Select option to display the results in Imperial, Metric or selected units.
Save button	Save the results data in Excel format.
Export to Excel button	Start Excel and display the results data in a new Excel spreadsheet.
Copy to Clipboard	Copy the information from the current sheet to the clipboard.
Choose Units	Select option to display the results in Imperial, Metric or selected units.
Dock Results Sheet Bottom	Dock, auto size and move the Result Sheet to the bottom of the drawing.
Dock Results Sheet Right	Dock, auto size and move the Result Sheet to the right of the drawing.
Zoom to selected item	Zoom the drawing in the background to show the selected item.
Pipes tab	Display the information about the pipes in the system.
Fittings tab	Display the information about the valves and fittings in the system.
Pumps tab	Display the information about the pumps in the system.
Nodes tab	Display the information about the nodes in the system.
Fluids tab	Display the information about the fluids in the system.
Pipe Materials tab	Display the information about the pipe materials in the system.
Energy tab	Display the information about the energy usage in the system.
All Results tab	Display the information about the elements in the system.

Viewing Individual Results

Items are highlighted by a left clicking with the mouse while hovering over an item on the drawing pane. Data for the highlighted item is displayed in the Hint pane.

In **Results** mode, when an item is selected the **Hint** pane is used to display information about the item, which can include flow rates, velocities, pressure drops, pressures at join points, head pressure (grade line), elevations of tanks and nodes, and fluid heads at specific points within the system.

Also in **Results** mode, when the cursor is moved over an item on the drawing, a pop-up panel appears that displays information about the item. When the cursor is moved away from the item the popup panel automatically disappears. This allows a user to inspect a large number of results for different parts of the system without having to continually click the mouse button.

The Results Popup Panel displays a variety of important results data, including:

- Volumetric Flow Rate
- Mass Flow Rate
- Velocity
- Pressure Loss
- Friction Loss
- Fitting Loss

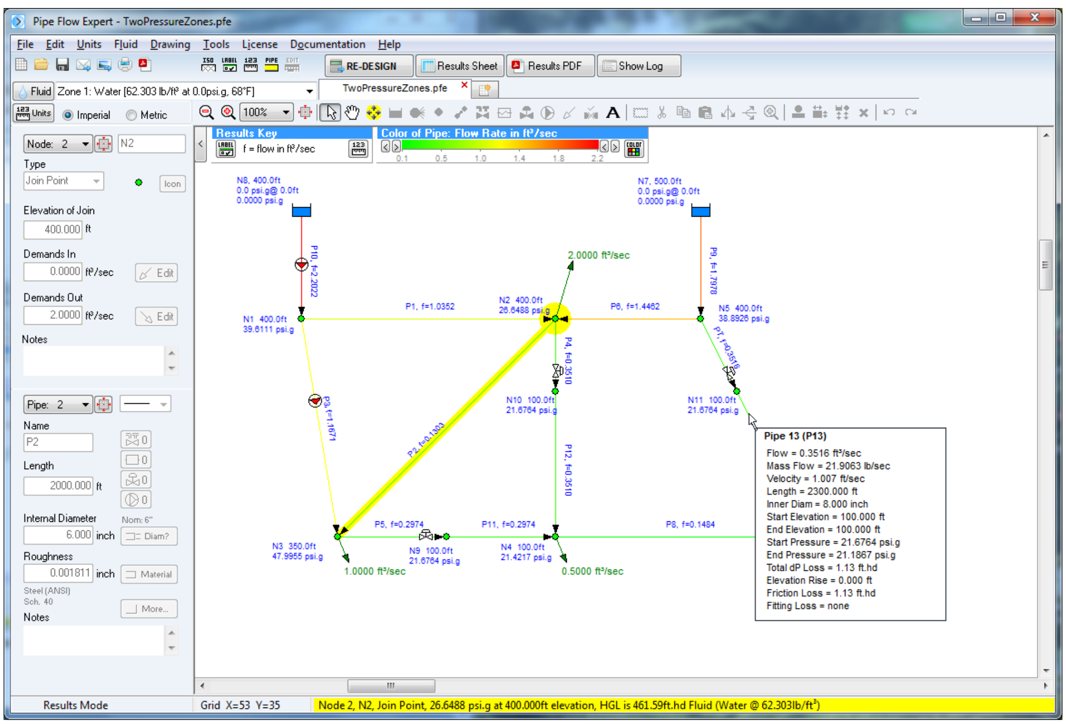


Figure 33 View Individual Results

The **Results** drawing can be configured to display the pipes in various colors based on the value of a chosen category such as Flow Rate, Velocity, Friction Loss, Internal Diameter etc. The color gradient start and end point values may be adjusted to narrow the band over which the colors are displayed.

File and Design Operations

Pipe Flow Expert works with a number of different file types, for example, pump files, component files, and the main system design file (a .pfe file).

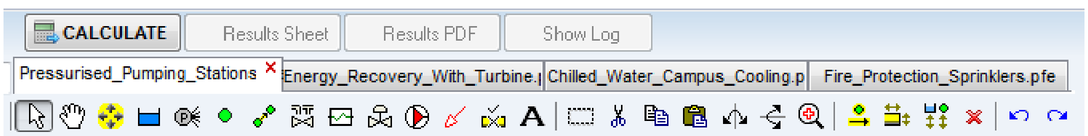
Once a system has been created it can then be saved to a system design file which stores all of the information about the system, including the node and pipe data, the fittings and valve information, the components information, and the pump data.

Individual items in the system such as components and pumps can also be saved to their own file, so that they can be re-used and re-loaded as required (perhaps for use in future systems or to duplicate and copy these items within an existing system). The pump file extension is .pfpm and the component file extension is .pfco.

Images of the drawing screen can be saved and/or attached to an email as a convenient way of sharing a quick snapshot of a system or set of results.

Tabbed Design Sheets

Tabbed Design Sheets let you work on multiple systems simultaneously. Up to 4 different systems may be opened and worked on concurrently using the 'Tabbed' system feature.



- Tabbed Sheets work just like Tabs in an Internet Browser.
- Tabbed Sheets work in both Design View and Results Mode.
- Switch between different system models with a single click.
- Easy to design, view, and compare results across multiple models.
- Easy to copy sections of drawings between different system models.
- Easy to demonstrate and review results for altered designs

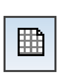
Creating a New Pipe System

To create a new pipe system:

1. Define whether imperial or metric units are used throughout the pipe system.

2. To define the imperial or metric setting, select the **Imperial** or **Metric** option on the toolbar or the **Units** tab in the **Configuration Options** dialog, or you can select the **Imperial** or **Metric** menu items from the **Units** menu.

3. You can customize the units and default values used for the individual items in the system at any time by using the **Configuration Options** dialog.

4.  Click the **New Grid** button or select **File | New** to create a new drawing grid.

5. Pipe Flow Expert asks the user if the grid should be shown for isometric drawing. Click ‘yes’ to display the isometric grid or click ‘no’ to display the standard drawing grid.

After creating a pipe system you should set the preferences of your choice. In Pipe Flow Expert you can configure whether imperial, metric or specifically chosen units are used throughout the system. The labels shown on the pipe system drawing can be altered, so that specific items can be selected to be shown or excluded as required.

The configuration settings for a pipe system can be amended at any time, while you are working in a pipe system, however defining your choices in advance will make it easier when you come to start the drawing process. All configuration options are saved with the system and will be set to their saved values when a system is loaded.

Isometric System Options

When the isometric drawing mode has been selected Pipe Flow Expert will show fittings, valves, components, control valves and pumps in the isometric plane which follows the associated pipe direction

Tanks and end-pressure nodes will also be displayed in the isometric plane. The drawing menu provides options to display an ‘Isometric tank view from the left’ or to display an ‘Isometric tank view from the right’. Selecting either of these options will change the display of all tanks and end pressure on the isometric view.

Designing a Pipe System

To design a pipe system:

1. Define the pipe system’s fluid and the fluid’s properties. For more information about selecting a fluid, see: *Defining the System Fluid*

2. Add the applicable tanks to the system. For more information about adding tanks, see: *Adding a Tank*

3. Add the applicable join points to the system. For more information about adding join points, see*: Adding a Node (Join Point)*

4. Add the applicable pipes to the system. For more information about adding pipes, see: *Adding a Pipe*

5. Add the applicable pumps to the system. For more information about adding pumps, see: *Adding a Pump*

6. Add the applicable fittings to the system. For more information about adding fittings, see: *Adding Fittings and Valves*

7. Add the applicable component pressure losses to the system. For more information about adding component pressure losses, see: *Adding a Component*

8. Add the applicable control valves to the system. Flow Control Valves, Pressure Reducing Valves or Back Pressure Valves can be used in the system. For more information about adding control valves, see: *Adding a Control Valve*

9. Add the applicable demand flows to the system. For more information about adding demand flows, see: *Adding a Demand Flow*

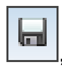
10. Add the applicable demand pressures to the system. For more information about adding demand pressures, see: *Adding a Demand Pressure*

11. Calculate and solve the pipe system's flow and pressure. For more information about calculating and solving, see: *Calculating and Solving the System*

Saving a System

When you save a pipe system in Pipe Flow Expert, a .pfe file is generated that is used exclusively with Pipe Flow Expert. All the system's data is stored within the file. If you need to modify the system, or want to reuse the system for a future system, you can simply open the file in Pipe Flow Expert, make the necessary changes, and resave the file.

To save a pipe system:

1. Click the **Save File** button, , on the toolbar or select **File | Save** to open the **Save As** dialog.

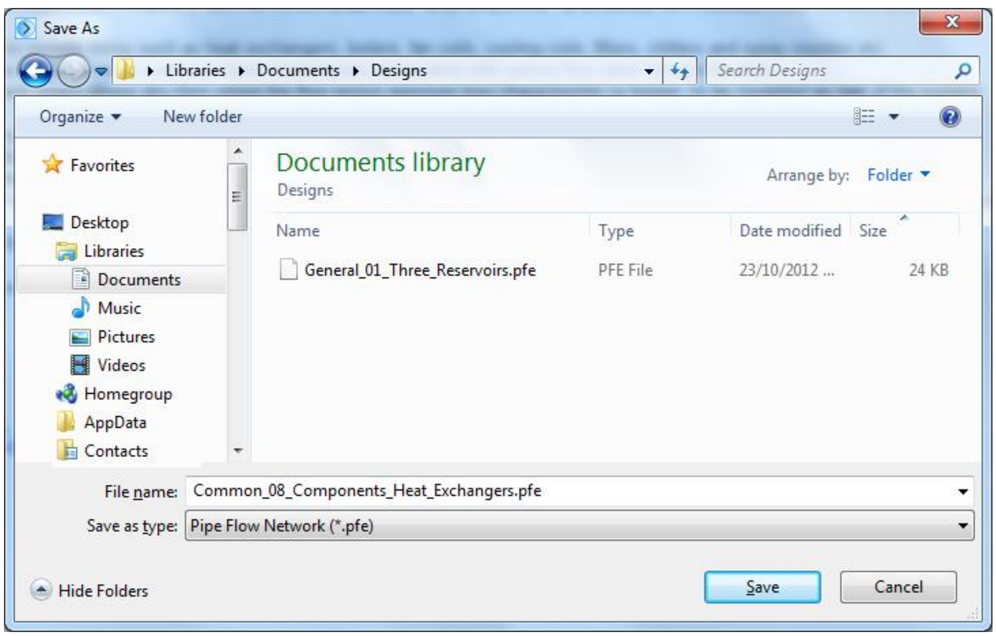
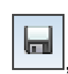


Figure 34 Save As dialog

2. Type a name for the file in the **File Name** field.

3. Click **Save** to save the pipe system.

Once a pipe system has been saved to a .pfe file, you can simply click the **Save** button, , on the toolbar or select **File | Save** to save changes to the system.

Change the System View - Isometric Mode Toggle

A Pipe Flow Expert system can be displayed in Isometric mode or Standard mode. The system information such as pipe sizes, pipe lengths, node elevations etc. do not change when the view of the system is changed.

When an Isometric view is transposed (flattened) to a standard view, any vertical lines from the isometric view will be drawn at 45 degrees. This happens because Pipe Flow Expert does not allow a drawing to contain nodes which sit over other nodes, or nodes which sit over pipes.
This restriction has been applied to avoid confusion.

With the 'Isometric to Standard' type of transposition the result can be visualized as an offset view from a high overhead position, where the vertical lines can still be seen, i.e. the result looks similar to a plan view but it is not an exact plan view (where some of the lines which were vertical in the isometric view would not be seen in an exact plan view).

When a Standard view is transposed to an Isometric view the result will be a flat view of the current drawing in the isometric plane. The user may then adjust/move node positions as required to make the drawing appear 3D on the isometric grid.

To change the system view:

1. Click the **ISO Mode On / Off Toggle** button, 

, on the toolbar.

2. A new view of the system will be displayed.

3. Select the 'Drag and Move' mode and adjust the node positions as required.

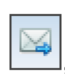
4. Use the drawing menu to select 'Isometric tank View from Right' or 'Isometric Tank View from Left'.

The default drawing grid can for new systems can be set in the Editor Tab of the **Configuration Options** dialog.

Sending a System via E-mail

You can share pipe systems with others via e-mail from within Pipe Flow Expert.

To Email a pipe system:

1. Click the **Email System Information** button, , on the toolbar to open a new mail message in your default e-mail application with the system's .pfe file attached to the e-mail.

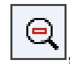
2. Compose and send the email.

Printing a System

We recommend creating a PDF file of the system drawing ('Create PDF' available from the file menu) and then using the PDF viewer (Adobe Acrobat for example) to print out the system as required, however there is also a direct Quick Print option that can send the part of the system displayed on screen, directly to a printer, where it can be scaled to print over x by y pages. This allows a large system to be printed out over multiple pages which can be fitted together.

Direct Print

In Pipe Flow Expert, when you print a pipe system directly to a printer, only the visible part of image, what is currently displayed in the Drawing pane, is printed.

If you wish to print the entire pipe system, click the Zoom Out button, , on the toolbar to zoom out until the area you wish to print is visible in the Drawing pane.

The printed image will be scaled to fit on one page by default, but you can customize the scaling using the ‘Scale to fit’ fields. If you want to change the paper’s orientation to landscape or print to a printer other than your computer’s default printer, open the Page Setup dialog and make your selections before printing the pipe system. To access the Page Setup dialog, select Print Setup from the File menu.

Note: As an alternative, for larger higher quality prints of your complete system you can create a PDF image of your system drawing by using the option provided on the file menu.
The PDF drawing can then be printed at a size to suit your requirements.

To print a pipe system directly to a printer:

1. Click the **Print** button, , on the toolbar or select **File | Print** to open the **Printing Information** dialog.

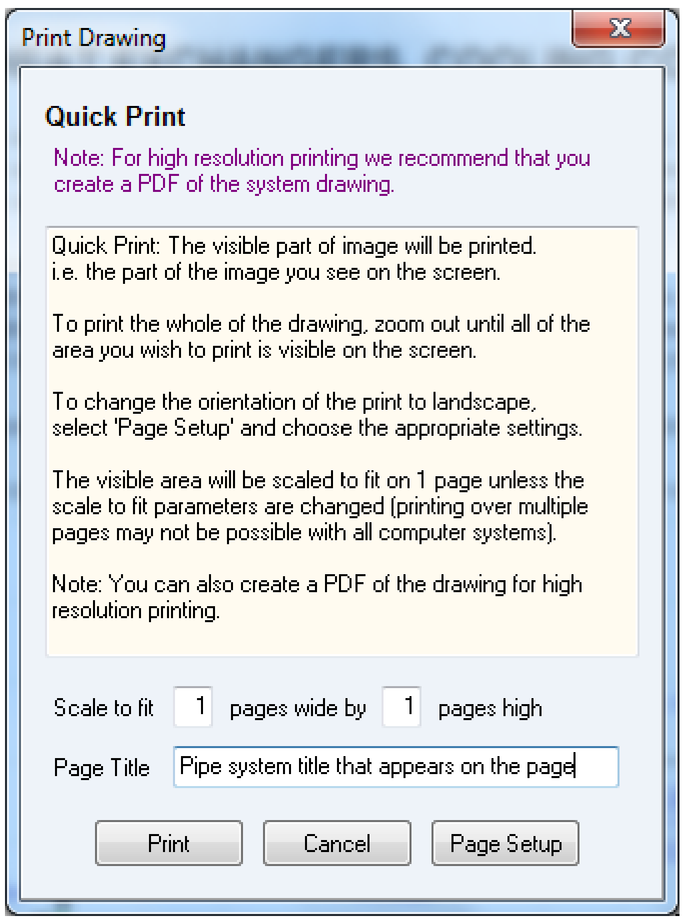


Figure 35 Printing Information dialog

2. If you want to change the printing scale, type the number of pages wide by the number of pages high in the **Scale to fit** fields. **Note:** Printing over multiple pages requires use of additional memory and problems may arise if your system (and printer) do not have enough memory available. The amount of memory required varies considerably depending on the size and number of items on the drawing that is to be printed.

3. Type the title you want displayed at the top of the page.

4. Click the **Print** button to print the pipe system.

Saving a Screen Image

To save the current image displayed in the drawing pane:

1. Click on the File Menu and select, , **Save Screen Image…**
2. A save file dialog window is opened.
3. Navigate the folder hierarchy as required
4. Enter the filename that the image should be saved as.
5. Click on **Save** to store a jpeg image of the drawing.

Saving a Drawing to an EMF Image

To save the current drawing as an Enhanced Metafile Image:

1. Click on the File Menu and select, , **Save Screen as EMF Metafile…**
2. A save file dialog window is opened.
3. Navigate the folder hierarchy as required.
4. Enter the filename that the image should be saved as.
5. Click on **Save** to store a enhanced metafile image of the drawing.

Emailing a Screen Image

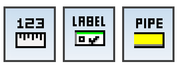
To email the current image displayed in the drawing pane:

1. Click on the File Menu and select, , **Email Screen Image…**
2. Your default email client will be opened with an attached jpeg image named PFEImg.jpg
3. Fill in the normal email details (i.e. the To address, etc).
4. Click to send your email as normal.

System Options

Pipe Flow Expert can be configured and set-up to suit the users working preferences.
The labelling on the drawing can be configured so that specific items can be included or excluded as required. The units in which to display lengths, diameters, head loss, velocity etc. can be chosen. The details of a default pipe can be chosen and these details will be used to draw any new pipes. The calculation tolerances can be changed if necessary (however in most cases these values should be left unchanged).

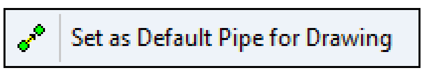
Most configuration settings are defined in the Configuration Options dialog which is made up of a series of tabs, one for each page of configuration options.



You can access the Configuration Options dialog by selecting Specify Units from the Units menu, or by clicking the Units button or Label button on the tool bar.

To select a Default Pipe size for drawing click the Pipe button on the tool bar to choose the pipe material, the pipe diameter and pipe length etc.

To set the Default Pipe attributes to match the currently selected pipe, right click to display the pipe pop-up menu and choose the ‘Set as Default Pipe for Drawing’ option.



Choosing Units (imperial / metric)

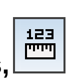
Pipe Flow Expert provides the flexibility to enter the units for the components of a pipe system in imperial (inches, feet, gallons, etc), or metric (millimeters, centimeters, liters, etc) or a combination of imperial and metric units. It is best practice to configure the system’s universal imperial or metric unit setting before configuring individual units and default values. The universal unit setting converts all values to imperial or metric units in a pipe system. If you will be using a combination of imperial and metric units you can change the type of units used for individual items on the Units tab of the Configuration Options dialog once you’ve defined the universal setting.

NOTE: When you change the universal unit setting, it converts all unit values to the unit type selected, and all values on the Units tab of the Configuration Options dialog are converted to the fields’ default values. For example, if you are changing the universal setting from imperial to metric, all values, including any imperial units selected on the Units tab of the Configuration Options dialog, are converted to metric. All field values on the Units tab are also reset to their default metric value.

There are three places in Pipe Flow Expert that you can define the universal unit setting.

To define the universal unit setting:

1. To change the units from the toolbar, click the **Imperial** or **Metric** option on the tool bar.

2. To change the units from the **Units** tab, click the **Units,**  **,** button on the tool bar or select **Specify Units** from the **Units** menu to open the **Units** tab of the **Configuration Options** screen. Click the **Imperial** or **Metric** option.

3. To change the units from the **Units** menu, select the **Imperial Units** or **Metric Units** from the Configuration Options **Units Tab** screen.

Click **Yes** when asked, *Do you wish to proceed*?

Choosing Item Labelling

In **Pipe Flow Expert**, you can configure the labelling of a pipe system displayed in the Drawing pane. The labelling for the pipe system is configured on the Labelling tab of the Configuration Options dialog. The tab is organized by the system components. A series of check boxes are used to configure labelling in drawing/design mode and a series of radio buttons are used to select labelling options when in results mode. The result options determine which calculated results are displayed for a solved pipe system in the Drawing pane. Results are displayed for a calculated solved pipe system after clicking the Calculate button on the toolbar.

For more information about pipe system results, see: *Calculating the System Flow and Pressure*

To configure the pipe system labelling:

Click the Labelling button, 

, on the tool bar to open the Labelling tab of the Configuration Options screen.

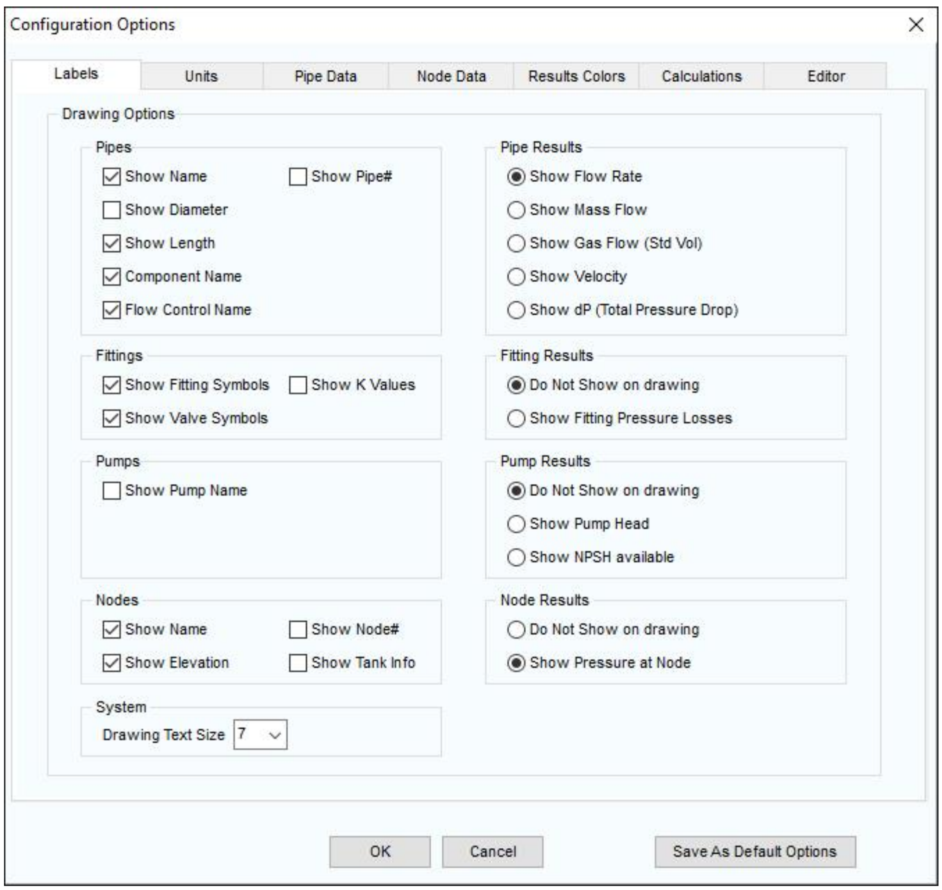


Figure 36 Configuration Options – Labelling tab

1. Select the check boxes next to the labelling options you want to apply to the pipe system in the **Pipes**, **Fittings**, **Pumps**, and **Nodes** sections.

2. Click one of the labelling options in the **Pipe Results**, **Fitting Results**, **Pump Results** and **Node Results** sections to determine which of the pipe system's calculation results are displayed in the Drawing pane.

3. Click **OK** to save the labelling configurations and close the **Configuration Options** dialog, or click another tab in the dialog to continue configuring Pipe Flow Expert and the pipe system.

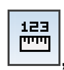
4. To remove a labelling option, clear the check box next to the applicable label in the **Pipes**, **Fittings**, **Pumps**, or **Nodes** section of the **Labelling** tab.

Choosing the System Units

The universal unit setting for imperial or metric units determines which type of unit is used by default throughout Pipe Flow Expert for a specific item. You can also change the types of units used for individual components in the pipe system on the Units tab of the Configuration Options dialog once you have defined the universal unit setting. The units selected on the Units tab determine which units are displayed for the corresponding fields in the Pipe Flow Expert Node and Pipe panes, the various dialogs and the units displayed in the Drawing pane.

For example, if you select *feet* from the Length list, feet will be displayed next to the Length field in the Pipe pane, and if the Show Length check box is selected in the Pipes section of the Labelling tab, each pipe's length in the pipe system is displayed in feet in the Drawing pane.

To define the units used in Pipe Flow Expert and the pipe system:

1. Click the **Choose Units** button, 

, on the tool bar, or select **Specify Units** from the **Units** menu to open the **Units** tab of the **Configuration Options** screen.

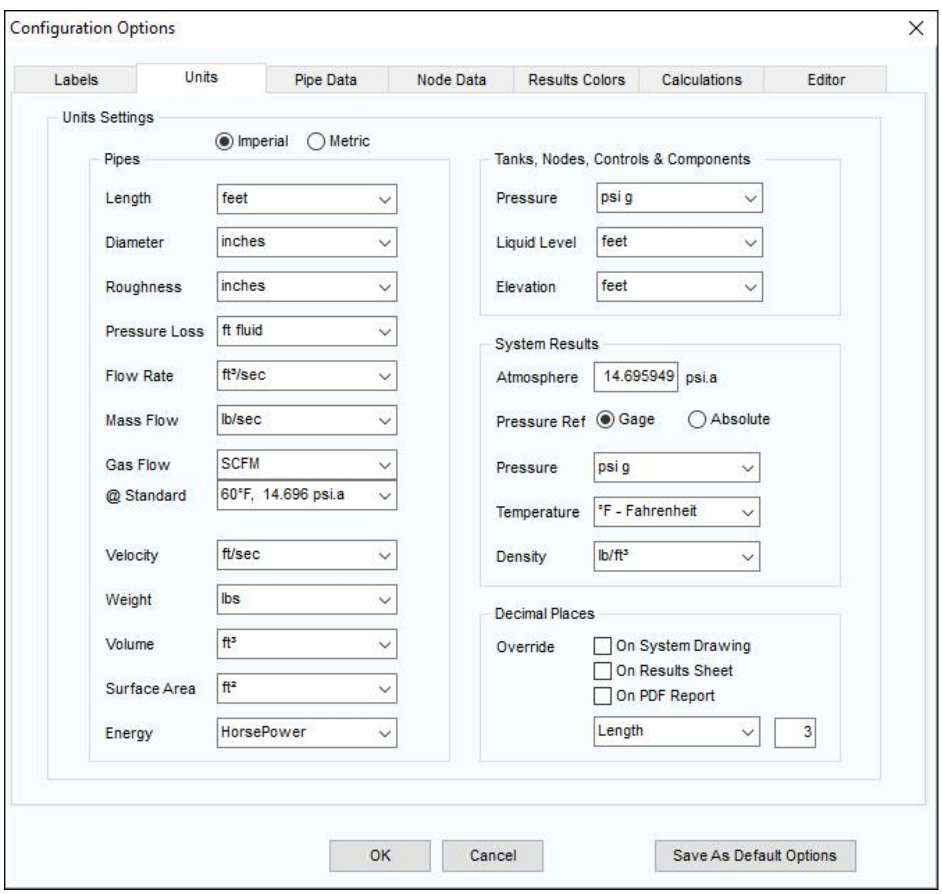


Figure 37 Configuration Options – Units tab

2. Select or enter the units for each component on the tab.

3. Do not click the **Imperial** or **Metric** option in the **Change All Units** section of the **Units** tab, unless you want to change the universal unit setting and reset all the values currently defined on the **Units** tab.

NOTE: When you change the universal unit setting, it converts all unit values to the unit type selected, and all values on the Units tab of the Configuration Options dialog are converted to the fields' default values. For example, if you are changing the universal setting from imperial to metric, all values, including any imperial units selected on the Units tab of the Configuration Options dialog, are converted to metric. All field values on the Units tab are also reset to their default metric value.

Overriding the Default Number of Decimal Places:

In Pipe Flow Expert, by default, each unit has a number of decimal places associated with it. The software tries to be smart, in that it shows a different number of decimal places for a given value based on the display unit. For example, a flow rate value in m^3/s is displayed to 4 decimal places, whereas a flow rate value in US gpm is shown to 2 decimal places.

For gas flow rates the software even adjusts the number of decimal places based on the size of the flow rate value. For example, a gas flow rate displayed in SCMH may be displayed to a different number of decimal places depending on the size of the value and how significant the fractional part of the value may be.

The default 'smart decimals' work well for many users, however, when necessary, it is possible to override the number of decimals that are displayed on the system drawing, the results sheet, and the PDF report.

The decimal places override options can be configured on the Units tab.

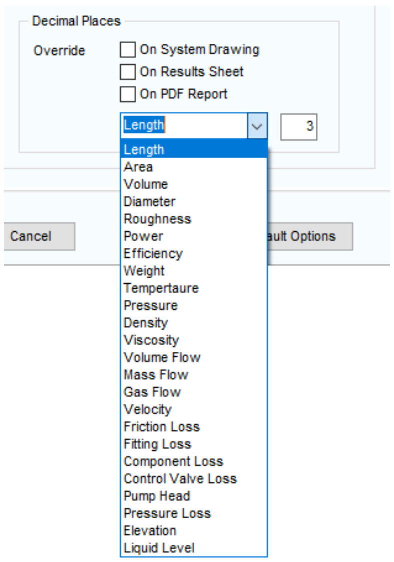


Figure 38 Configuration Options – Decimal Places

The decimal override functionality works slightly differently to the default decimals functionality, in that when overriding the number of decimals to display, this is specified on a per attribute basis (for attributes such as length, diameter, roughness, volume flow) rather than operating in a 'smart' manner based on the display unit.

i.e. a specific attribute, such as volume flow for example, can be configured to display to a set number of decimal places, and this will be applied irrespective of the unit of flow that is chosen.

Select one or more of the options (System Drawing, Results Sheet, PDF Report) where the post decimal override values are to be applied. Once an override option has been selected, a set of default decimal places for each different attribute will be applied, and these can then be updated as required.

To change and set the number of override post decimal places for a specific attribute, select the required attribute from the dropdown list and update the number of post decimals for that attribute.

Note: Once the override decimals has been applied, the default number of decimal places for each attribute will be applied, and these will be different in some cases to the ‘smart’ decimals that were used for individual units when no override was in place. This means that when the override decimals functionality is selected, it may be necessary to review and update as needed, the specified number of decimal places for each and all attributes in the list.

Once the number of decimal places for each attribute has been configured, click OK to save the decimal place configurations and close the Configuration Options dialog or click another tab in the dialog to continue updating other configuring options within the Pipe Flow Expert software.

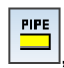
Choosing Pipe Drawing Defaults

If you will be using the same type of pipe throughout a pipe system, you can set up default pipe values. When the pipe default values are defined, the values are automatically populated each time you add a section of pipe to a pipe system. You can set up default pipe values for pipe length, internal diameter and roughness, as well as fittings and the pipe's material and size. Pipe default values are set up on the Default Values tab of the Configuration Options dialog.

The default pipe values are used when adding pipes however you can customize the values for an individual pipe in the system at any time, by editing its values in the Pipe pane or in the Pipe diameter data dialog.

NOTE: Be sure to enter the value for the pipe's properties in the units displayed next to the field. For example, if you are using metric units, and *m* for meter is displayed next to the Length field, enter the pipe length value in meters.

To set up pipe default values for a pipe system:

1. Click the Set Default Pipe for drawing button, 

, on the tool bar to open the Default Values tab of the Configuration Options dialog.

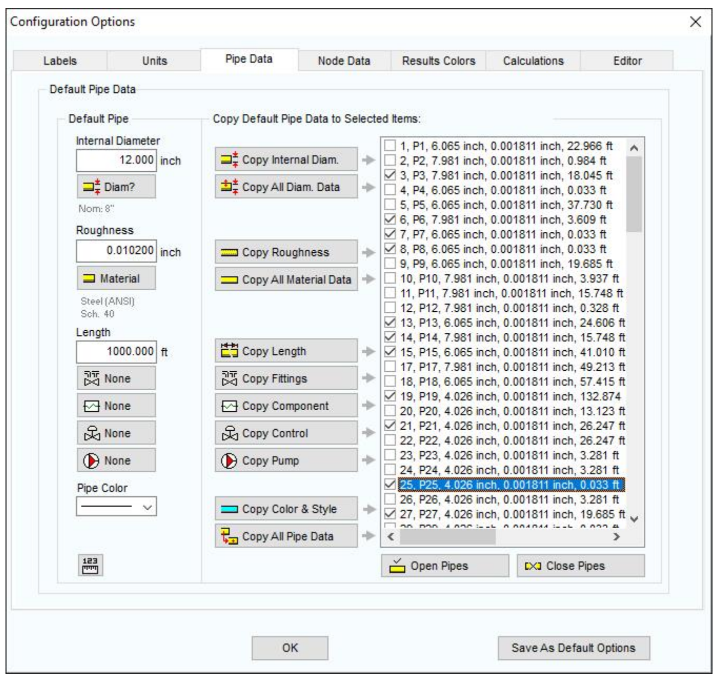


Figure 39 Configuration Options – Pipe Settings tab

To change the pipe material:

1. Click the **Material** button. This opens the **Pipe diameter data screen** A list of all the pipe materials available in the pipe database are displayed in the **Choose new pipe material** list.

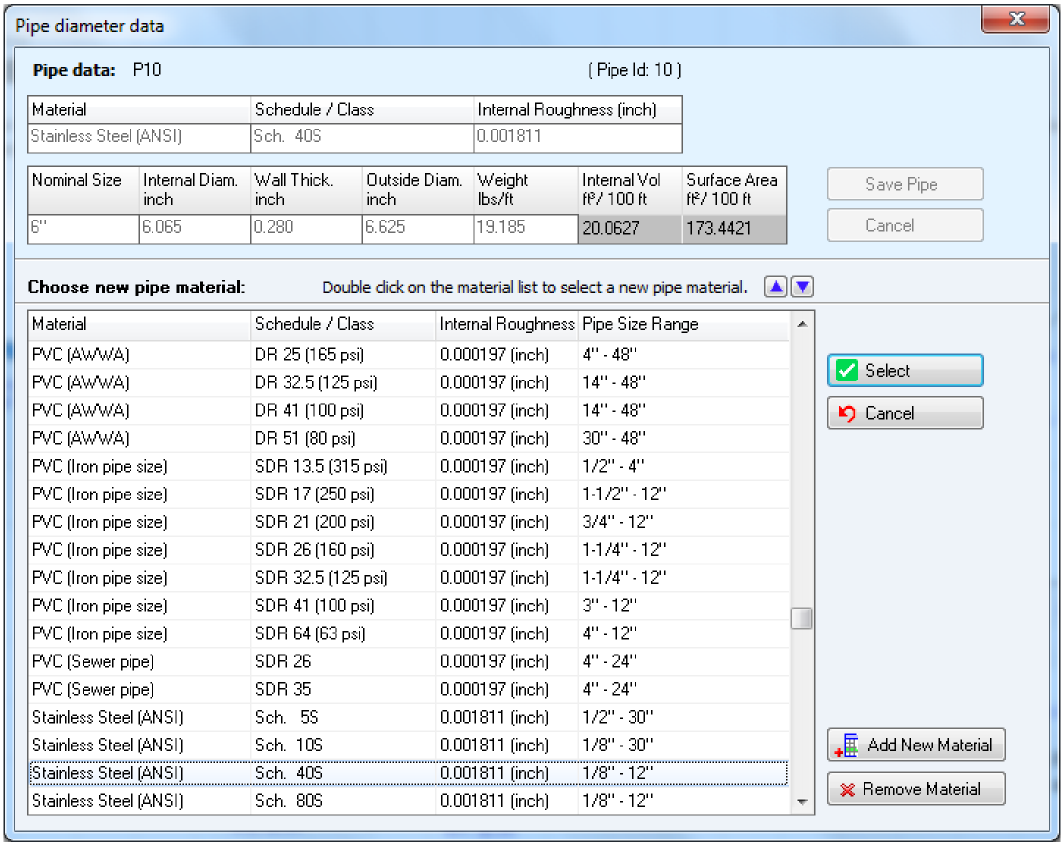


Figure 40 Pipe diameter data - materials list

2. Select the pipe's material from the **Choose new pipe material** list.

3. If the pipe material is not in the **Choose new pipe material** list, click the **Add new material** button to add the new material to the list. For more information about adding a pipe material, see: *Adding a Pipe Material to the Database* section in *Section: Creating a Pipe System*.

4. Click the **Select** button or double-click the row containing the material.

To change the pipe size:

1. The selected material is displayed in the **Pipe data** section of the **Pipe diameter data** dialog, and a list of all the different pipe sizes available for the selected pipe material is displayed in the **Pipe diameter database**.

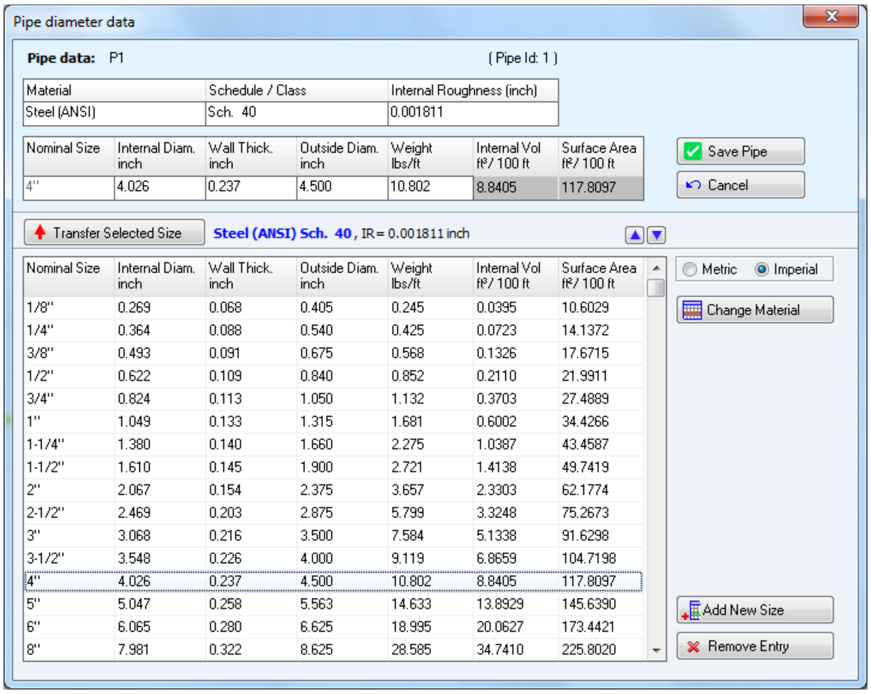


Figure 41 Pipe Diameter Data - Pipe Sizes

2. Select the applicable pipe size from the list.

3. If the pipe size is not in the list, click the **Add new size** button to add the new size to the list. For more information about adding a pipe size, see: *Adding Pipe Size Data to the Database Section: Creating a Pipe System*.

4. Click the **Transfer Selected Size** button or double-click the row containing the size. The selected size is displayed below the pipe material in the **Pipe data** section of the **Pipe diameter data** screen.

5. Edit the applicable pipe properties in the **Pipe data** section. The edits you make only affects the pipe's properties for the system; they do not affect the database data.

6. Click **Save Pipe** to add the pipe's material and size, and to close the **Pipe diameter database**.

The Pipe Default Tab is now populated with the choices that have been made.

The pipe's material is displayed under the **Roughness** value, and the nominal size is displayed under the **Diam** button. The pipe's internal diameter and roughness are displayed in the **Internal Diameter** and **Roughness** fields on the **Default Values** tab.

To change the pipe length:

1. Type the default pipe length into the **Length** field on the pipe pane.

To change the fittings on the pipe:

1. The default number of fittings currently on the selected pipe is displayed on the **Add/Change Fittings** button.

2. Click the **Add/Change Fittings** button, .

3. Clicking the **Add/Change Fittings** button opens the **Pipe fitting friction coefficients** dialog. A list of all the fittings available in the fitting database is displayed in the **Fitting Database** list.

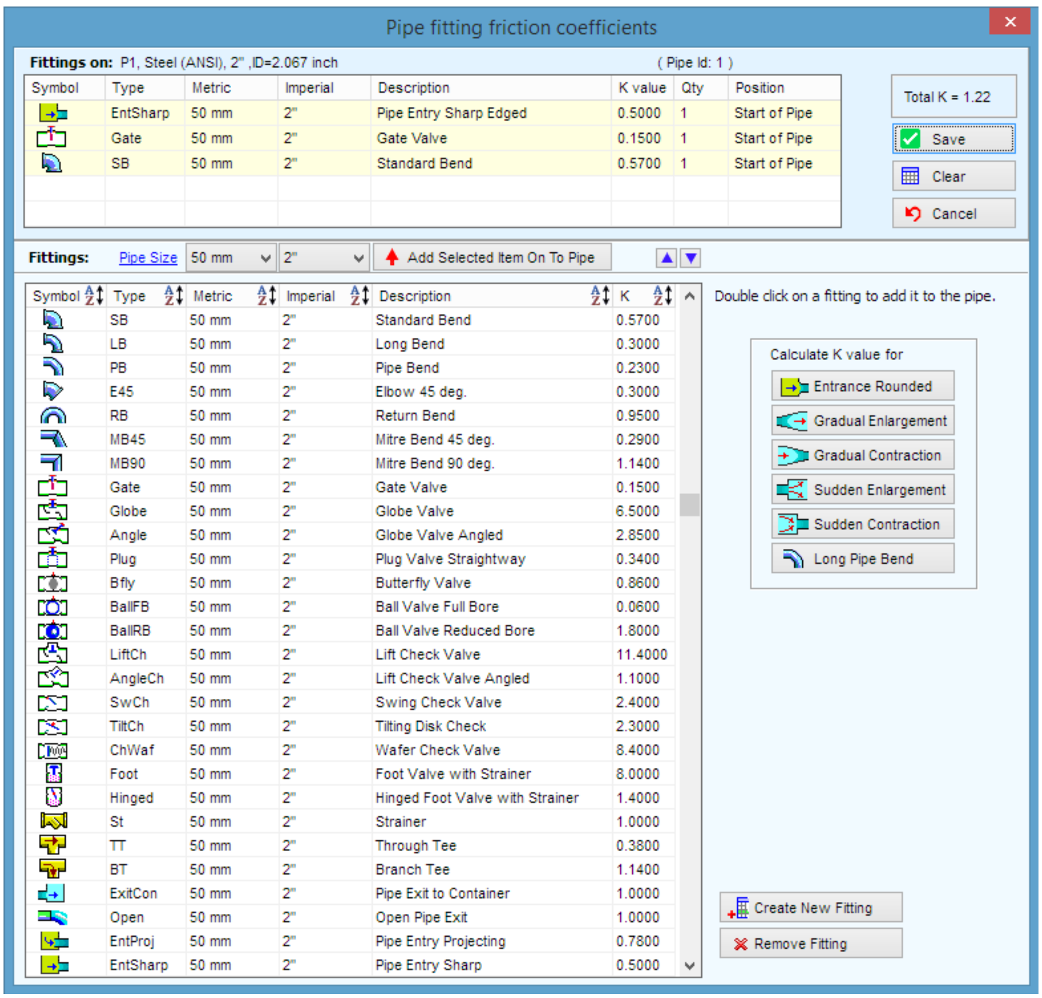


Figure 42 Pipe fitting friction coefficients

4. Select the fitting you want to add to the pipe from the **Fitting Database** list.

5. Use the **Move to fitting size** fields to locate a specific metric or imperial fitting size in the **Fitting Database** list.

6. If the fitting is not in the **Fitting Database** list, click the **Create new fitting** button to add the new fitting to the list. For more information about adding a fitting to the database, see: *Adding a Fitting to the Database* section in *Section: Creating a Pipe System*.

7. Click the **Add Selected Item on to pipe** button or double-click the row containing the fitting.

8. The selected fitting is displayed in the **Fittings on** section of the **Pipe fitting friction coefficients** dialog.

9. Select the quantity of the fitting you are adding to the pipe from the fitting's **Qty** list in the **Fittings on** section.

10. Select the position of the fitting, Start of Pipe or End of Pipe, from the drop down list.

11. To add additional fittings to the pipe, repeat Steps 4 – 9.

12. Click **Save** to add the fitting to the pipe, and to close the **Pipe fitting friction coefficients** screen.

13. Confirm the changes to the default pipe settings:

14. Click **OK** to save the default pipe values and close the **Configuration Options** dialog, or click another tab in the dialog to continue configuring Pipe Flow Expert and the pipe system.

Change attributes of more than one pipe

Some or all of the **Default pipe values** may be copied to other pipes in the system. The **Selection Tool** or the **Individual Item Selection Option** may be used to select a range of pipes on the drawing prior to opening the **Pipe Settings** tab.

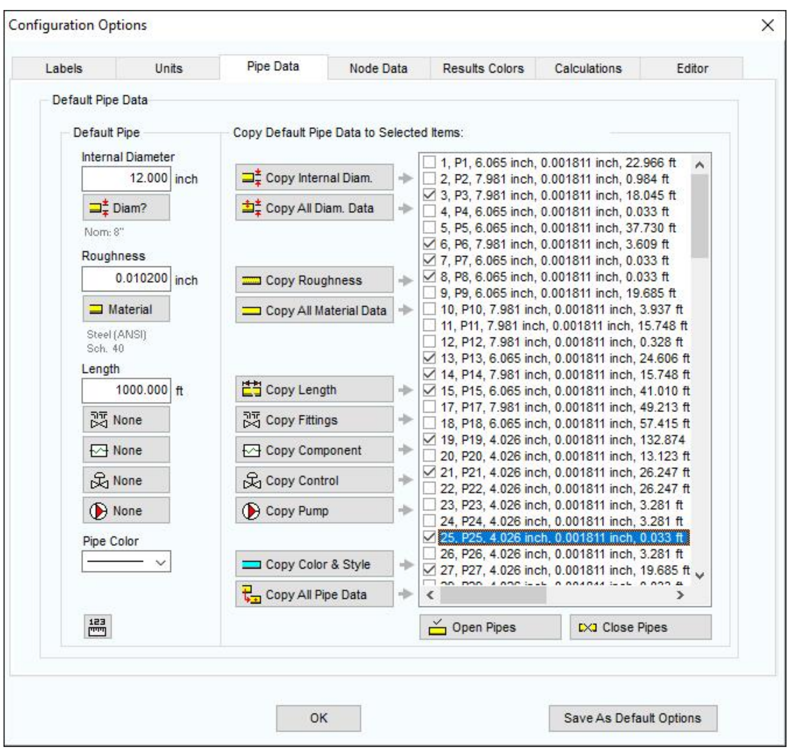
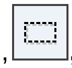
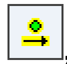
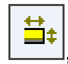


Figure 43 Configuration Options – Pipe Settings Tab

1. Optional: First select a group of pipes to update using the **Selection Tool**, **,** or the **Individual Item Selection Option,** **,**
2. Click the Adjust Attributes of Selected Pipes button, , on the tool bar, to open the Pipe Settings tab of the Configuration Options dialog.
3. Individual pipes can be added or removed from the selected list by checking or un-checking the box adjacent to the pipe description.
4. Click **Copy All Pipe Data** to copy all of the pipe attributes, including valves and fittings and the fluid zone to the selected pipes.
5. Click **Copy All Diam. Data** to copy the default internal and external pipe diameter to the selected pipes.
6. Click **Copy Internal Diam.** to copy the default internal diameter to the selected pipes.
7. Click **Copy All Material Data** to copy the default material, schedule/class and internal roughness to the selected pipes.
8. Click **Copy Internal Diam.** to copy the default internal diameter to the selected pipes.
9. Click **Copy Roughness** to copy the default internal roughness to the selected pipes.
10. Click **Copy Length** to copy the default pipe length to the selected pipes.
11. Click **Copy Fittings** to copy the default pipe fittings to the selected pipes.
12. Click **Copy Component** to copy the default pipe component to the selected pipes.
13. Click **Copy Control** to copy the default pipe control valve to the selected pipes.
14. Click **Copy Pump** to copy the default pipe pump to the selected pipes.
15. Click **Copy Color & Style** to copy the default pipe color and line width to the selected pipes.

Node Updates

The node elevation and the node image on the **Node Updates Tab** may be copied to other nodes in the system. The **Selection Tool** or the **Individual Item Selection Option** may be used to select a range of nodes on the drawing prior to opening the **Node Updates** tab.

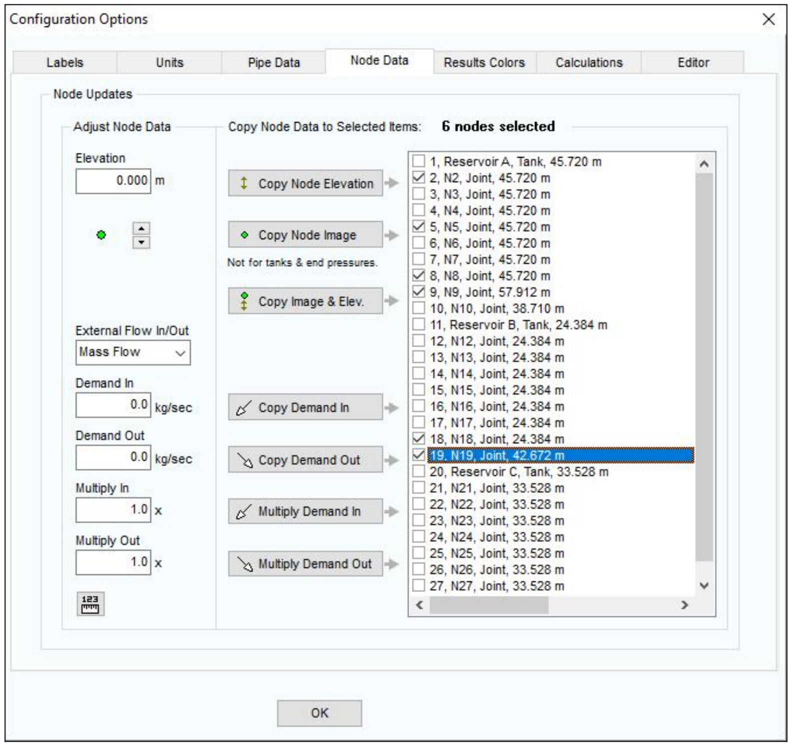
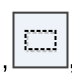
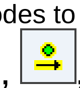
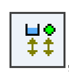


Figure 44 Configuration Options – Node Updates Tab

1. Optional: First select a group of nodes to update using the **Selection Tool**, **,** or the **Individual Item Selection Option,** **,**
2. Click the Adjust Attributes of Selected Nodes button, , on the tool bar, to open the Node Updates tab of the Configuration Options screen.
3. Individual nodes can be added or removed from the selected list by checking or un-checking the box adjacent to the node description.
4. Click **Copy Node Elevation** to copy the node elevation to the selected nodes.
5. Click **Copy Node Image** to copy the node image to the selected nodes.
6. Click **Copy Image & Elev.** to copy the node image and the node elevation to the selected nodes.
7. Click **Copy Demand In** to copy the in-flow values to the selected nodes.
8. Click **Copy Demand Out** to copy out-flow values to the selected nodes.
9. Click **Multiply Demand In** to multiply the in-flow values of the selected nodes by the specified value.

10. Click **Multiply Demand Out** to multiply the out-flow values of the selected nodes by the specified value.

Results Colors

The results screen can display pipes and nodes from the solved network using a color gradient to highlight the pipes or nodes based on the result value for a chosen calculation category.

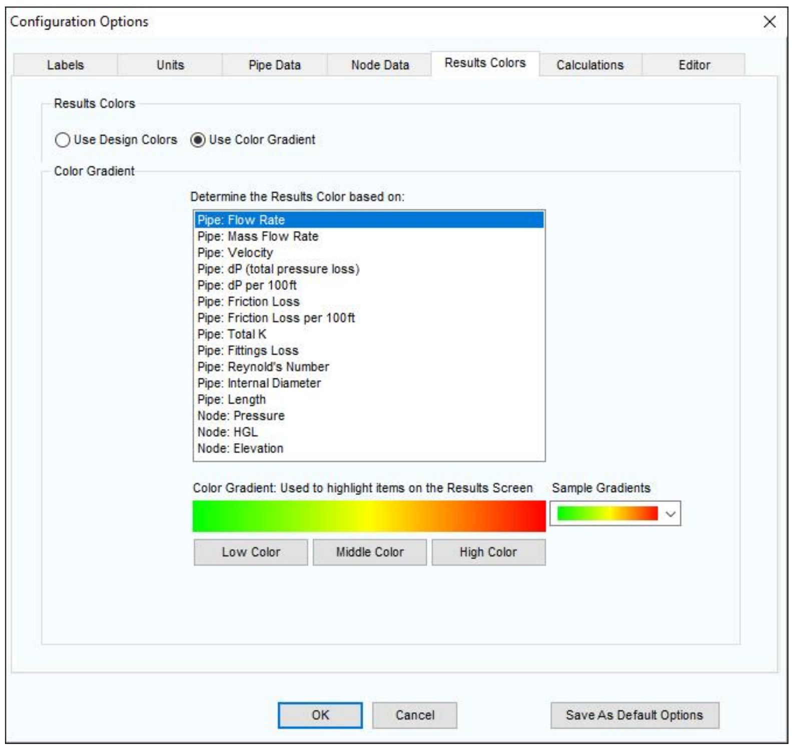


Figure 45 Configuration Options – Results Colors Tab

1. Click the **Color button**, 

, on Results Screen Color Key, to open the **Results Colors** tab of the **Configuration Options** screen.

2. Select a calculation category to be display in color on the Results screen.

3. Choose a color gradient from the drop down of sample gradients, or use the Low, Middle and High color buttons to create your own color gradient.

4. Click OK to update the drawing pane.

Configuring the Calculation Parameters

The accuracy to which a pipeline system is balanced can be changed. The balance options for a pipe system are configured on the Calculations tab of the Configuration Options dialog.

Most systems can be solved without any changes to the Calculation options, however these options are included for experimental use when a balanced solution cannot be found. Widening of these parameters may give results where the flow into the system and out of the system do not balance and/or the pressure results throughout the system will not agree with the sum of the pressure drops through the individual pipes.

A balanced solution may not always be found if the system contains components which 'fight' each other. If a pump is placed in a pipeline, the change in flow rate will be related to the head added by the pump and the extra pipeline friction losses. In some circumstances, when balancing a system that includes a number of pumps, the adjustment of the flow rate in one pipe/pump combination may generate a disproportionate change of flow rate in another pipe/pump combination, which itself may produce a counter acting effect to the previous flow rate adjustment. Thus the program would oscillate between two possible solutions, neither of which will give the normal expected degree of accuracy.

It is normally better to change the system design to overcome these sorts of problem rather than proceed with a design where it is difficult to predict an accurate result (since this likely indicates that there is a true design problem).

Note: We recommend that you do not change these values, unless it becomes necessary to solve a finally balanced system.

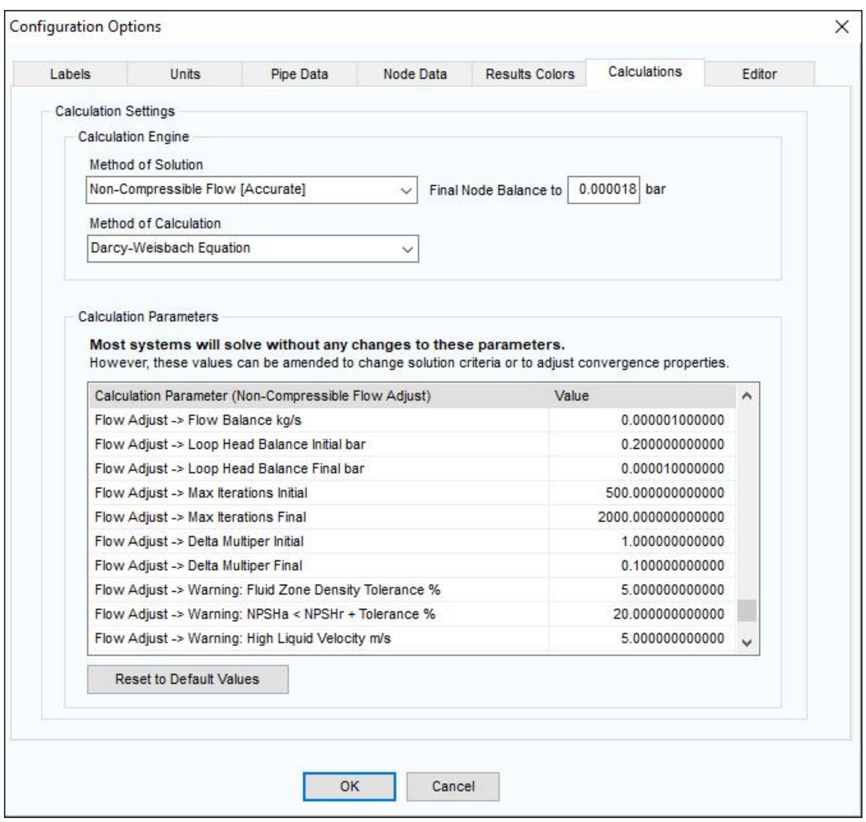


Figure 46 Calculation settings

The following adjustments can be made:

Feature	Description
Method of Solution	Non-Compressible or Compressible flow calculation engine
Method of Calculation	Equation used to calculate friction loss.
Final Node Balance	Balance pressure at all nodes to this accuracy
Calculation Parameters	A set of values that are used during convergence to determine a solution. The actual parameters vary depending on the Method of Solution selected and the Method of Calculation equation that is selected.
Reset All entries to the Default Values	Restore the standard settings for the calculation parameters.

For liquid systems the Method of Solution selected should always be ‘Non-Compressible Flow’ and the Method of Calculation equation should be ‘Darcy-Weisbach’. These give the most accurate results possible for liquid systems.

For gas systems the Method of Solution selected should normally be ‘Compressible Gas Flow’. The Method of Calculation equation should then be selected to meet any specific needs of the system. In most cases the General Fundamental Flow Equation should be selected since this is the most generally applicable, however the user can choose from additional equations, such as the AGA equation, the Weymouth equation, the Panhandle A equation, and others, in order to produce calculation results that are based on the equation that is normally used for their ‘type’ of gas system.

Default Options

Clicking **Save As Default Options** will save the currently defined Label, Units, Pipe Data, Results Colors and Editor options as the default options.

The default options will automatically be applied to new systems.

Fluid Zones

Pipe Flow Expert will allow the user to design pipeline systems with up to 20 different fluid zones. The density and viscosity of each fluid zone must be set by the user. Pipe Flow Expert does not calculate the resultant density or viscosity of any 'mixed' fluids.

Each pipe in a system can be associated with an available fluid zone. The density and viscosity of the associated fluid zone will be used in calculating the flow rate and pressure loss in each pipe.

Gas Systems:

When using the Compressible Gas Flow calculation engine, the density of the fluid will automatically be adjusted based on the pressure in the network at the point of the calculation. i.e. the software will automatically account for any compression and expansion of the gas in the friction loss calculations. This means that the fluid density and viscosity could be defined at normal atmospheric conditions and specified at say 0 bar.g, however the fluid data MUST be defined at the temperature condition that is present in the network, since the software needs to know what temperature the fluid is at.

For example, gas data could be defined at say 0 bar.g and 20 degrees C, and it would be ok to use this for calculating the system even if the pressure in the network were say 10 bar.g, provided that the temperature in the network was 20 degrees C. If the temperature in the network was say 50 degrees C then the fluid data should be specified for some pressure condition at 50 degrees C.

Many systems will only have one Fluid Zone, therefore when a new system is being drawn each new pipe will be associated with the fluid zone associated with the default pipe data. Fluid Zone 1 is the **Default Fluid Zone.** The color of Fluid Zone 1 cannot be changed. Pipes in the default fluid zone are not background highlighted.

Pipes associated with Fluid Zone 2 to Fluid Zone 20 are background highlighted to assist with identification of the different fluid zones. The background highlight color for these fluid zones can be amended by the user. This background highlighting may be toggled on/off from a fluid menu option.

Changing the fluid data of a fluid zone will affect the calculation of flow and pressure loss for all pipes associated with the fluid zone.

Defining Fluid Zones

The **Fluid** button, 

, on the tool bar opens the fluid zone menu.

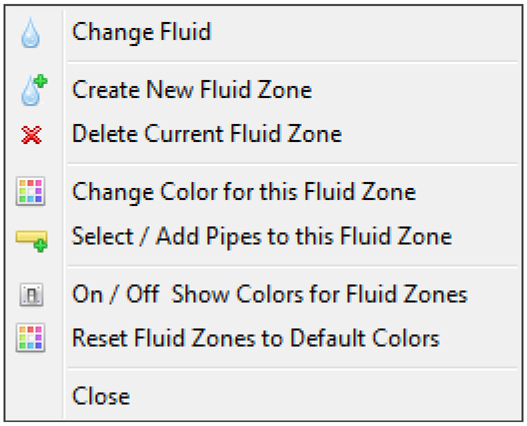


Figure 47 Fluid Zone Menu

Feature	Description
Change Fluid	Change the fluid name, density and viscosity for the currently selected fluid zone.
Create New Fluid Zone	Create a new fluid zone. The fluid database screen will be displayed so that new fluid data can be selected. When the fluid data has been chosen / amended click save to confirm the data for the new fluid zone.
Delete Current Fluid Zone	Delete the currently selected fluid zone.
Change Color for this fluid zone	Change the background highlight color for the currently selected fluid zone
Select / Add pipes to this fluid zone	Select pipes to be associated with the currently selected fluid zone. The drawing cursor will change to a Fluid Zone selection rectangle. Move the drawing cursor to an appropriate position on the system drawing. Click on the drawing pane and hold the mouse button down while dragging the Fluid Zone selection rectangle to enclose the pipes to be associated with current fluid zone. When the mouse button is released the enclosed pipes will be associated with the current fluid zone.
On / Off Show Colors for Fluid Zones	Toggle the fluid zone background highlight colors display on / off.
Reset Fluid Zone to Default Colors	Reset all fluid zone colors to the default color for each fluid zone.
Close	Close the fluid menu.

Properties of Mixed Fluids

Pipe Flow Expert does not calculate the resultant density or viscosity of any 'mixed' fluids. The characteristics of each fluid zone must be set by the user. Establishing the density and viscosity of fluid mixtures can be a complicated and a reliable independent method should be used to calculate the mixed fluid properties.

Two Phase Flow – Additional Pressure Drop

Pipe Flow Expert does not calculate the pressure drop for two-phase flow.

When two different fluids (or two different fluid states) exist together, it is possible that two-phase flow may occur at some point in a system. Flows transitioning from pure liquid to vapor as a result of external heating can lead to two-phase flow. Two-phase flow can produce an extremely high pressure drop that is many times greater than the pressure drop from either individual fluid phase. The user should make due allowance for any two-phase flow pressure drop, possibly by using a component to add an appropriate additional pressure loss.

Fluids Database

Pipe Flow Expert includes a fluid database. The Fluid Properties list displays all of the fluids available in the fluid database for either liquids or gases. When you are selecting a fluid to be used for the current fluid zone you can select an existing fluid, or enter your own fluid data. Once entered, new fluid data will show up in the list of fluids and can then be used just the same as the standard fluid data.

The Fluid Data can be defined and overwritten in the fluid properties section of the Fluid data screen.

New fluid data can be added to the existing database entries by clicking the Add New Fluid button.

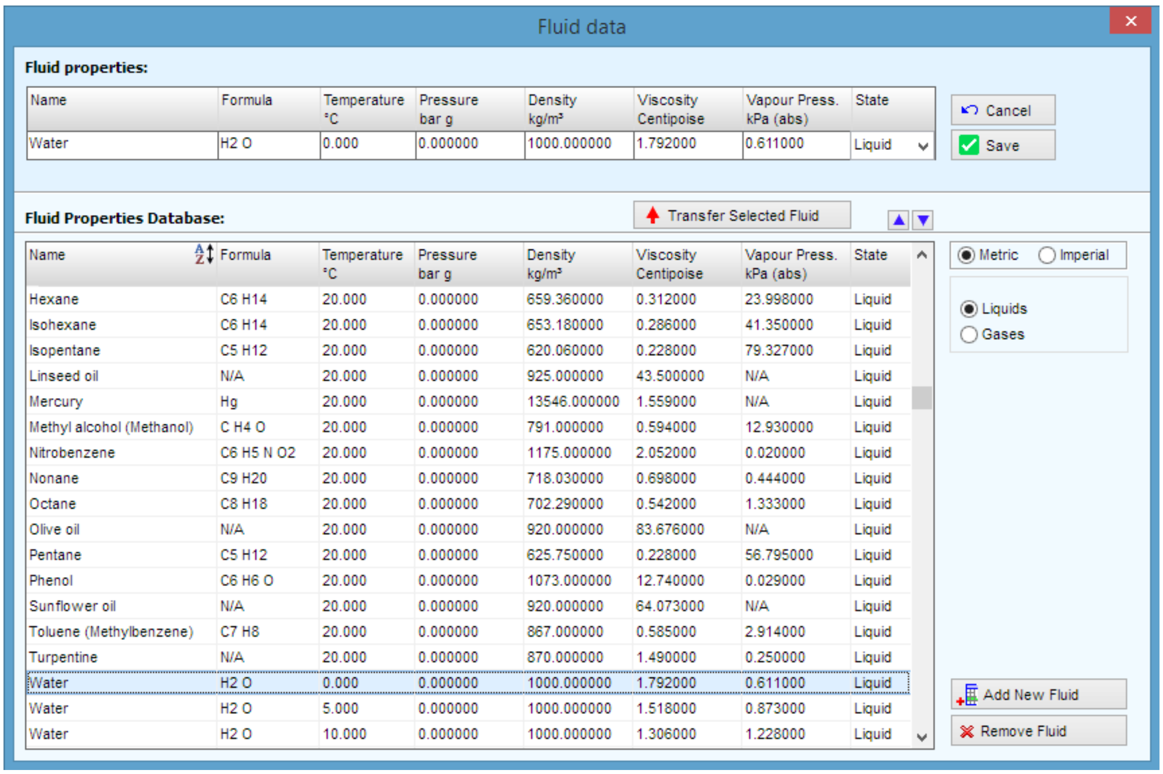


Figure 48 Fluid Database

To define the fluid and fluid properties for the currently selected fluid zone:

1. Click the **Fluid** button,  , on the tool bar and select **'Change Fluid'** from the pop-up menu to open the **Fluid data** dialog.

2. Select the fluid contained in the pipe system from the **Fluid Properties Database** list.

3. If the fluid is not in the **Fluid Properties Database** list, click the **Add new fluid data** button to add the new fluid data to the list. For more information about adding fluid data, see: *Adding Fluids to the Fluid Database*

4. Click the **Use selected fluid** button or double-click the row containing the fluid.

5. The selected fluid is displayed in the **Fluid properties** section of the **Fluid data** dialog.

6. Edit the applicable fluid properties in the **Fluid properties** section.

7. The edits you make only affect the fluid properties for the system; they do not affect the database data.

8. Click **Save** to save the fluid data to be used with the currently selected fluid zone.

The name of the fluid for the currently selected fluid zone is displayed next to the **Fluid** button on the tool bar.

Adding Fluids to the Fluid Database

If the fluid you are using in the pipe system does not exist in the Pipe Flow Expert fluid database, you can quickly add the fluid to the database by clicking the Add new fluid data button on the Fluid data screen.

NOTE: Be sure to enter the value for the fluid's properties in the units displayed on the column header. For example, if you are using imperial units, *ºF* for Fahrenheit is displayed under *Temperature* in the Temperature column. The temperature value in this case should be entered in degrees Fahrenheit.

To add new fluid data to the fluid database:

1. Click the **Fluid** button, 

, on the tool bar and select **'Change Fluid'** from the pop-up menu to open the **Fluid database**.

2. Click the **Add new fluid data** button.

3. Clicking the **Add new fluid data** button opens the **Add fluid data** section in the **Fluid data** screen.

4. Type the fluid's name in the **Name** field.

5. Type the fluid's formula in the **Formula** field.

6. Type the fluid's temperature in the **Temperature** field.

7. Type the fluid's density in the **Density** field.

8. Type the fluid's viscosity centipoise in the **Viscosity Centipoise** field.

9. Type the fluid's vapor pressure in the **Vapour Press** field.

10. If the fluid's state is gas, select the **Gas** check box in the **State** field.

11. Leave the **Gas** check box blank if the fluid's state is liquid.

12. Click **Save** to save the new fluid data to the database and the **Fluid Properties Database** list.

Adding Gas Data to the Fluid Database

If the fluid you are using in the pipe system is one of the most common gases, the viscosity and density of the gas at various temperatures and pressures can be calculated.

To add new gas data to the fluid database:

1. Click the **Fluid data** button, 

, on the tool bar and select **'Change Fluid'** from the pop-up menu to open the **Fluid Data** dialog.

2. Choose the **Gases** option to display the **Calculate Gas Data** button.

3. Click the **Calculate Gas Data** button to display the **Properties of Gas Data** calculator.

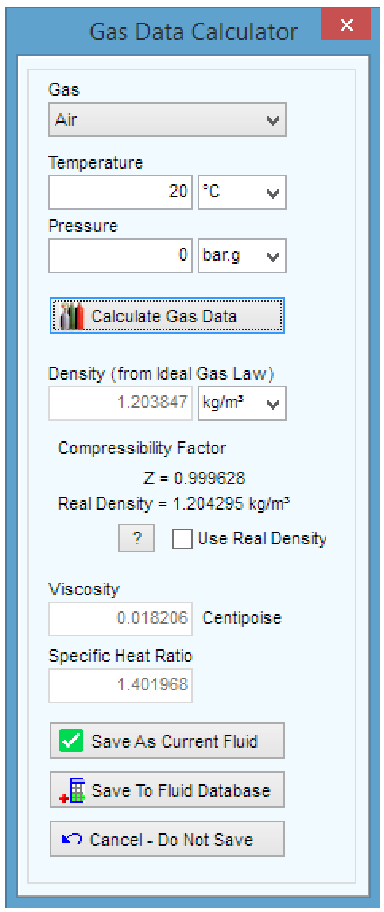


Figure 49 Properties of Gases

4. Choose a Gas from the drop down listing.

5. Enter the temperature of the gas.

6. Enter the pressure of the gas.

7. The density, viscosity and specific heat ratio data for the gas are displayed.

8. Click **Add new entry** to add the data to the fluid data list.

Note on gas compressibility:

When entering gas data, we recommend defining the gas properties for a pressure condition that is on the Ideal Gas Law line. Normally the easiest approach is to always define the gas properties for the specific temperature within your system and at the 0 bar.g pressure (atmospheric condition), even if your system generally operates at a completely different pressure.

The software will automatically account for and calculate the density of the gas as the pressures within your model. If you have a system that operates at a very high-pressure condition where the ideal gas law does not predict the true real density of the gas, then a Z compressibility factor can also be defined.

A Compressibility Factor can be specified on the Calculations Tab. The Pipe Flow Expert software can use the Z compressibility factor to calculate the REAL density of the gas at a high-pressure condition. It does this by calculating the density based on the ideal gas law and then it applies the Z compressibility factor to arrive at the real gas density.

For example, if your system is operating at some high-pressure condition, where the real gas density is different from the one as calculated by the ideal gas law, then you can specify a Z Compressibility Factor on the Calculations Tab (under Config Options in Pipe Flow Expert). The compressibility factor will be applied as the software solves the model, by first calculating the density of the gas at the high-pressure nodes using the ideal gas law and secondly by then applying the Z compressibility factor to arrive at the real density within your model during calculations.

Using the above approach, where the fluid density is defined for the standard atmospheric pressure condition (0 bar.g) at some specific temperature condition, when the standard flow rate (volume of flow at the defined standard condition) is reported, the Pipe Flow Expert software will convert the mass flow to the standard volume using the correct fluid density for the standard pressure condition (even if a Z compressibility factor was specified for use in calculating the real-density at the nodes within a high-pressure system).

i.e. the mass flow rate, even in a system with high pressure conditions, where a z compressibility factor is applied, will convert back to give the correct standard flow volume (the correct SCMH or SCFM value for example).

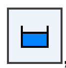
Tanks

In Pipe Flow Expert, a tank can represent a reservoir or any type of fluid supply. It can also model a point of discharge where a fluid exits a pipe system. A tank is a type of node in Pipe Flow Expert. There are three types of nodes used in a pipe system: tanks, demand pressures, and join points. Nodes are located at the beginning and end of a pipe.

NOTE: Be sure to enter the values for the tank's properties in the units displayed next to the field. For example, if you are using imperial units, and *ft* for feet is displayed next to the Liquid Level field, enter the liquid level value in feet. You can change the default units on the Configuration Options Units tab.

Add a Tank

To add a tank to the pipe system:

1. Click the **Add Tank** button, , on the tool bar.
2. When you click the **Add Tank** button, the tank symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click on the place on the Drawing pane where you want to add a tank.
4. The tank is added to the system, and is selected on the Drawing pane. Define the tank's properties in the Node pane.

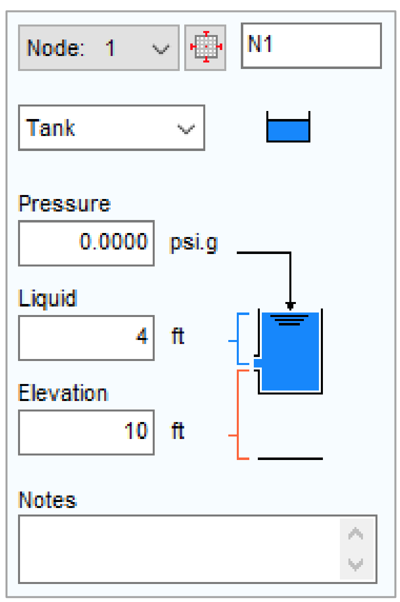


Figure 50 Node Pane for Tanks

5. Type the tank's name in the **Node** field.
6. Tank is automatically selected in the Type list.

7. Click the **Tank Icon** button to select the tank image you want displayed on the pipe system drawing. The **icon size** can be selected from a range of scales. The image you select does not affect any of the tank's properties or values.

8. Type the surface pressure of the fluid in the tank in the **Surface Pressure** field.

9. Type the amount of fluid above the tank exit point in the **Liquid Level** field.

10. Type the exit elevation from the tank in the **Elevation (Exit)** field.

11. Type any applicable notes regarding the tank in the **Notes** field.

12. To add additional tanks to the system, repeat Steps 3 – 12.

Nodes (Join Points)

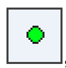
A join point is the place where two or more pipes meet in a pipe system. A circular dot represents each join point in the Drawing pane. In Pipe Flow Expert, you can add join points as you add pipes, or you can add join points first and then connect the join points with pipe. If you add a join point to a pipe, and the location of the join point causes the pipe to be split, you will have to manually reset the pipe details for each section of pipe.

You can define the fitting(s) used at each join point by adding these to the appropriate pipe, specifying if the fitting is to be placed at the start or end of the pipe.

NOTE: Be sure to enter the values for the join point's properties in the units displayed next to the field. For example, if you are using imperial units, and *ft* for feet is displayed next to the Elevation of Join field, enter the elevation in feet.

Adding a Node

To add a join point node to the pipe system:

1. Click the **Add Join Point** button, , on the tool bar.
2. When you click the **Add Join Point** button, the join point symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click on the place on the Drawing pane where you want to add a join point.
4. The join point is added to the system, and is selected on the Drawing pane. Define the join point's name, elevation, and volume demand in and/or demand out in the Node pane.

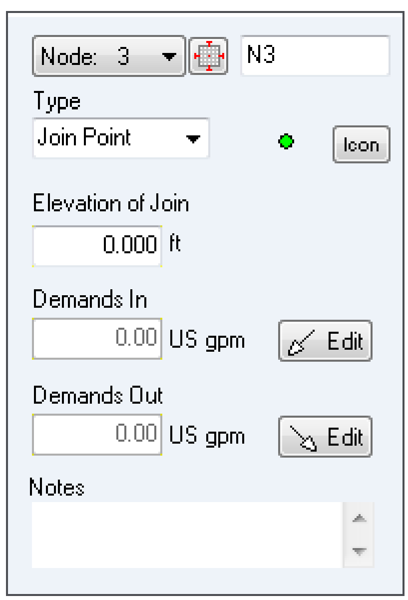


Figure 51 Node Pane for Join Points

5. Type the join point's name in the **Node** field.
6. *Join Point* is automatically selected in the **Type** list.

7. Type the elevation of the join point in the **Elevation of Join** field

8. If the system has a required demand flow at the join point, click the **Edit** button to open the **Set Flow Demands** screen.

9. You can enter either an In-Flow or an Out-Flow demand at the join point on the **Set Flow Demands** screen. Both **Edit** buttons in the Node pane open the **Set Flow Demands** screen.

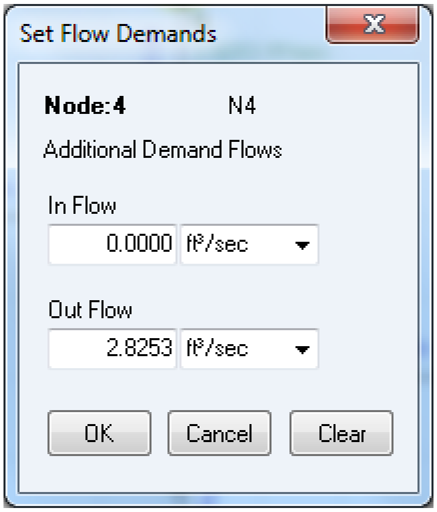


Figure 52 Set Flow Demands

10. Type the required flow rate for the fluid entering the system at the join point in the **In Flow** field. (Usually the out flow field should be set at zero).

11. Type the required flow rate for the fluid leaving the system at the join point in the **Out Flow** field. (if this is a point of discharge then usually the In Flow field here will be set at zero).

12. Click **OK** to add the join point's demand flow.

13. Type any applicable notes regarding the join point in the **Notes** field.

14. To add additional join points to the system, repeat Steps 3 – 12.

Note:

Demand Flows specify fluid flow that is entering the system from an external source or that is leaving the system to an external system (or discharging or being taken out of the existing system to some external system).

Demand Flows DO NOT control the flow at a particular point in the system (FCVs are used for this), unless they are placed on a node which only has one connecting pipe (in this case they the define the flow in or out of that node, which in turn effectively determines the flow rate in the single connecting pipe).

Pipes

If you will be using the same type of pipe throughout your system, you can define the default pipe type and values that are used when drawing pipes, using the Pipe Data tab in the Configuration Options dialog. When the default pipe values are set, each time you add a pipe to the system, the pipe type and values are automatically defined for the pipe. You can customize the values for an individual pipe in the system, by editing its values in the Pipe pane or in the Pipe diameter data dialog. The Pipe diameter data dialog contains a list of the pipe materials and sizes available in the pipe database and additional properties specific to a pipe in the system.

For more information about using default pipe values, see: *Setting up the System Options*

NOTE: Be sure to enter the values for the pipe’s properties in the units displayed next to the field. For example, if you are using metric units, and *m* for meter is displayed next to the Length field, enter the pipe length value in meters.

Adding a Pipe

To add a pipe to the pipe system:

1. Click the **Add Pipes** button, , on the tool bar.
2. When you click the **Add Pipes** button, the pipe symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click on the place on the Drawing pane where you want the pipe to start.
4. Click on the place where you want the pipe to end.
5. The pipe is added to the system, and is selected in the Drawing pane. Define the pipe’s properties in the Pipe Pane.
6. The rubber-banding functionality of the **Add Pipes** feature provides the ability to continue drawing additional connected sections of pipe after adding a section of pipe.
7. If you want to continue adding connected lengths of pipe to the system, move the mouse pointer to draw the pipe, and click where you want the pipes to end. To turn off the rubber-banding, press the right mouse button (right-click).
8. Type the pipe’s name in the **Name** field.
9. If the default pipe type and values for the system are defined on the **Default Values** tab in the **Configuration Options** dialog, you are done adding the pipe.
10. The pipe type, default length, internal diameter and roughness values are displayed in the Pipe pane.
11. If the default pipe values are not being used for the pipe, continue with Steps 12 to 25.
12. Type the length of the pipe into the **Length** field.
13. Click the **Material** button.
14. Clicking the **Material** button opens the **Pipe diameter database**. A list of all the pipe materials available in the pipe database are displayed in the **Choose new pipe material** list.

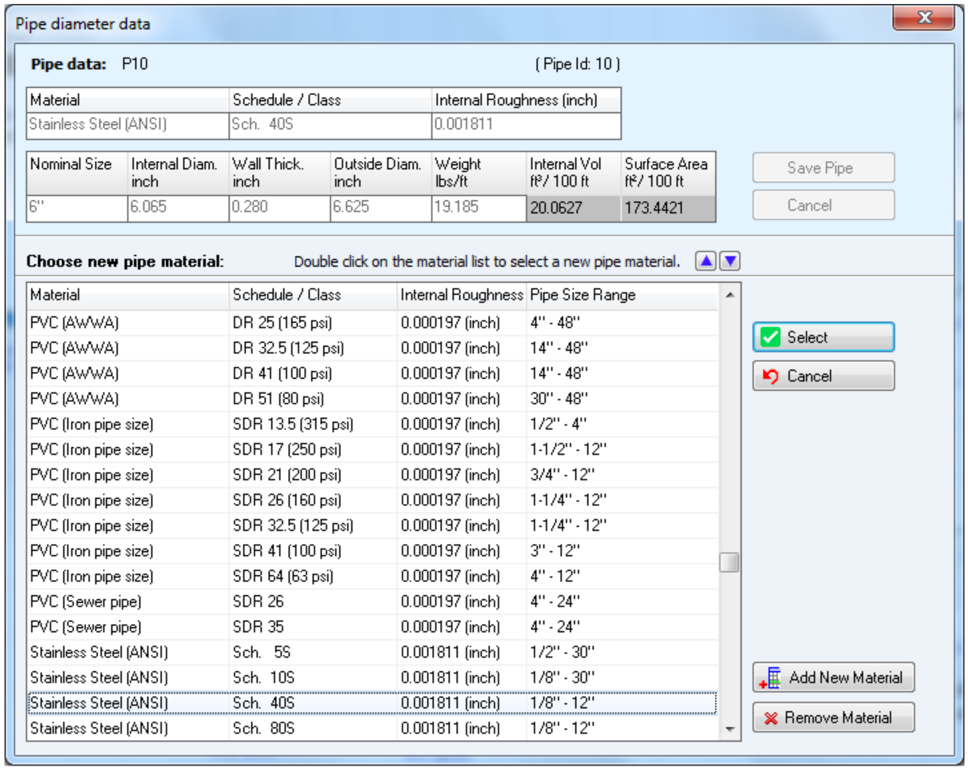


Figure 53 Pipe diameter data - materials list

15. Select the pipe's material from the **Choose new pipe material** list.

16. If the pipe material is not in the **Choose new pipe material** list, click the **Add new material** button to add the new material to the list. For more information about adding a pipe material, see: *Adding a Pipe Material to the Database*

17. Click the **Select** button or double-click the row containing the material.

18. The selected material is displayed in the **Pipe data** section of the **Pipe diameter data** dialog, and a list of all the different pipe sizes available for the selected pipe material is displayed in the **Pipe diameter data** dialog.

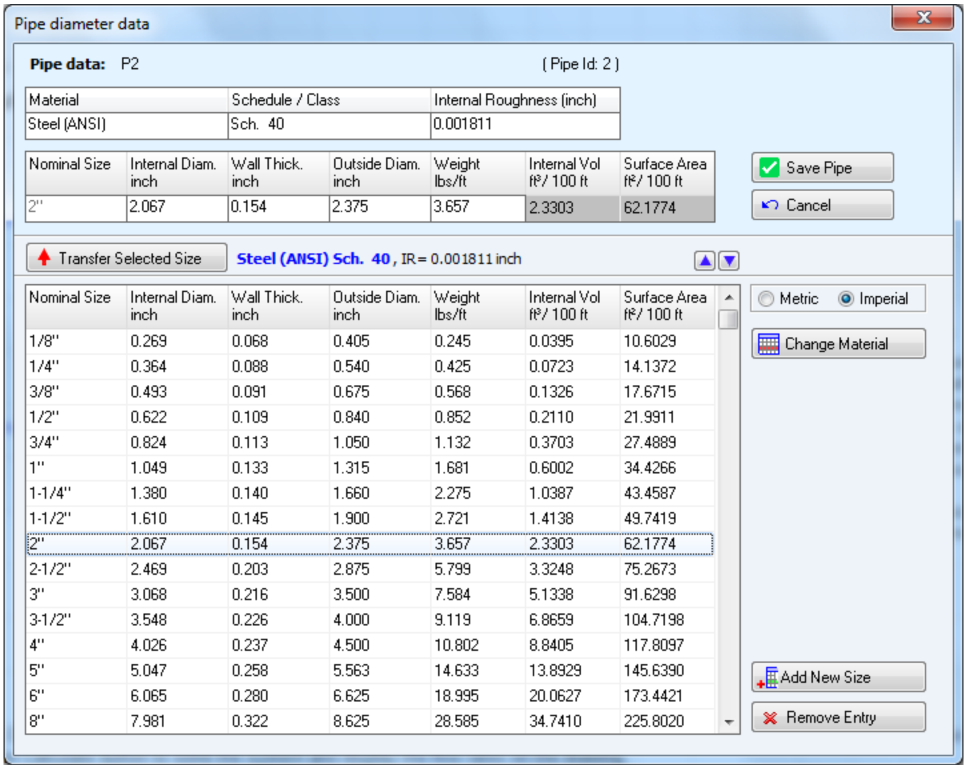


Figure 54 Pipe diameter data - pipe sizes

19. Select the applicable pipe size from the list.

20. If the pipe size is not in the list, click the **Add new size** button to add the new size to the list. For more information about adding a pipe size, see: *Adding Pipe Size Data to the Database*

21. Click the **Transfer Selected Size** button or double-click the row containing the size.

22. The selected size is displayed below the pipe material in the **Pipe data** section of the **Pipe diameter data** screen.

23. Edit the applicable pipe properties in the **Pipe data** section.

24. The edits made only affect the current pipe's properties; they do not affect the database data.

25. Click **Save Pipe** to add the pipe's material and size, and to close the **Pipe diameter data** screen.

The pipe's material is displayed under the **Roughness** field, the nominal size is displayed under the **Diam** button, and the pipe's length, internal diameter, and roughness are displayed in the **Length**, **Internal Diameter**, and **Roughness** fields in the Pipe pane.

Adding a Pipe Material to the Database

If the pipe material you are using in the pipe system does not exist in the Pipe Flow Expert pipe database, you can quickly add the material to the database by clicking the Add new material button on the Pipe diameter data dialog.

NOTE: Be sure to enter the values for the pipe material's properties in the units displayed next to the field. For example, if you are using metric units, and *mm* for millimeter is displayed next to the Internal Roughness field, enter the internal surface roughness in millimeters.

To add new material to the pipe database:

1. Select a pipe in the pipe system and click the **Material** button in the Pipe pane, or click the **Material** button on the **Default Values** tab in the **Configuration Options** screen to open the **Pipe diameter database** with the material list displayed.

2. Click the **Add new material** button.

3. Clicking the **Add new material** button opens the **Add pipe material** section in the **Pipe diameter data** screen.

4. Type the material name in the **Name** field.

5. Type the material's schedule or class in the **Schedule/Class** field.

6. Type the material's internal roughness, expressed in the units displayed in parenthesis, in the **Internal Roughness** field.

7. Click the **Add new material entry** button to save the new material data to the database and the materials list.

Adding Pipe Size Data to the Database

If the pipe size data you are using in the pipe system does not exist in the Pipe Flow Expert pipe database, you can quickly add the size data to the database by clicking the Add new size button on the Pipe diameter data dialog.

NOTE: Be sure to enter the value for the pipe's size data in the units displayed on the column header. For example, if you are using imperial units, and *lbs/ft* for pounds per foot is displayed under *Weight* in the Weight column, the weight value should be entered in pounds per foot.

To add new size data to the pipe database:

1. Select a pipe in the pipe system and click the **Diam?** button in the Pipe pane, or click the **Diam?** button on the **Pipe Data** tab in the **Configuration Options** dialog to open the **Pipe diameter data** dialog with the size list displayed.

2. Click the **Add new size** button.

3. Clicking the **Add new size** button opens the **Add pipe data** section in the **Pipe diameter data** dialog.

4. The pipe's nominal size will be automatically calculated from the pipe outside diameter data and the type of material selected. If the user prefers the **Nominal Size** field can be selected from the drop down list of nominal pipe sizes.

5. Type pipe's internal diameter in the **Internal Diam** field.

6. Type the wall thickness of the pipe in the **Wall Thick** field.

7. Type the outside diameter of the pipe in the **Outside Diam** field.

8. Type the pipe weight in the **Weight** field.

9. The values in the **Internal Vol.** and **Surface Area** fields are automatically calculated.

10. Click the **Add new entry** button to save the new size data to the database size list.

Reversing the Pipe Flow

When you add a pipe to a pipe system, the direction the fluid flows through the pipe always defaults to the direction in which the pipe is drawn. Each pipe in the Drawing pane shows an arrow to indicate the direction of fluid flow. If a pipe is inadvertently drawn in the wrong direction, you can reverse the direction of the pipe and fluid flow.

To reverse the fluid flow through a pipe:

1. Select and highlight a pipe by clicking on the drawing pane.

2. Right Click to display the Pipe Pop-up menu options.

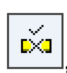
3. Select Reverse Pipe Direction.

The direction of a selected pipe can also be reversed by using the 'More' button on the pipe pane to display the Pipe Pop-up menu options.

Closing a Pipe in the System

The Closed Pipe feature can be used to represent actual pipes in the system where the flow is shut off. In Pipe Flow Expert, you can stop the flow of fluid through any pipe in the system by selecting the Open/Close pipe mode and clicking on a pipe in the Drawing pane to close the pipe. With the Closed Pipe feature, you can evaluate different configuration scenarios within the system when calculating the results.

To close a pipe in the pipe system:

1. Click the **Open/Close Pipe** button, , on the tool bar to select open/close pipe mode.

2. Click any pipe in the Drawing pane which you want to close. The pipe will be closed.

3. To re-open a pipe, click a closed pipe with the Open/Close cursor. The pipe will be re-opened.

4. Click the **Show Item Info** button, , or right click to exit the Open/Close pipe direction mode.

A selected pipe can also be opened or closed by using the Pipe Pop-up menu option. A Closed pipe will be displayed 'grayed out' and 'dotted' on the drawing to indicate that the pipe is no longer part on the functioning system. **Note:** care must be taken with this feature, as a non-functional system may result from this action and Pipe Flow Expert may not be able analyze the resulting network.

Using the Pipe Pop-up menu while drawing

In Pipe Flow Expert, you can change some of the features of the selected pipe by using the Pipe Pop-up menu.

Select a pipe (left click) and then click the right mouse button to display the **Pipe Pop-up menu.**

Select the option that you require and click to perform the action.

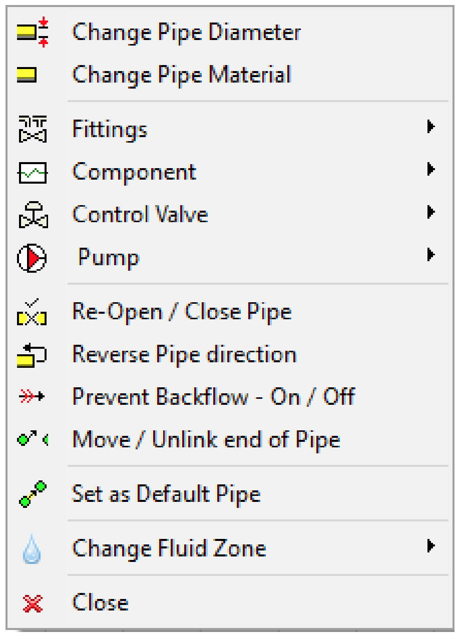


Figure 55 Pipe Pop-up Menu

Feature	Description
Change Pipe Diameter	Opens the pipe diameter data – pipe sizes (for current material).
Change Pipe Material	Opens the pipe diameter data – materials list.
Fittings	Opens the pipe fittings sub-menu.
Component	Opens the component sub-menu.
Control Valve	Opens the control valve sub-menu.
Pump	Opens the pump sub-menu.
Re-Open / Close Pipe	Open a closed pipe / Close a pipe.
Reverse Pipe Direction	Reverse the flow direction of a pipe.
Prevent Backflow – On / Off	Tag a pipe to prevent backflow.
Move / Unlink end of pipe	Unlink the end of the pipe and allow re-positioning
Use Pipe Valves for Drawing	Use the selected pipe data as the default pipe for drawing new pipes.
Change Fluid Zone	Change the fluid zone associated with the pipe.
Close	Close the pipe pop-up menu.

Pipe Pop-up Fittings sub-menu

Select Fittings on the pipe pop-up menu to display the fittings sub-menu.

The sub-menu options are context sensitive and will be visible and/or enabled/disabled depending upon whether there are fittings on the selected pipe.

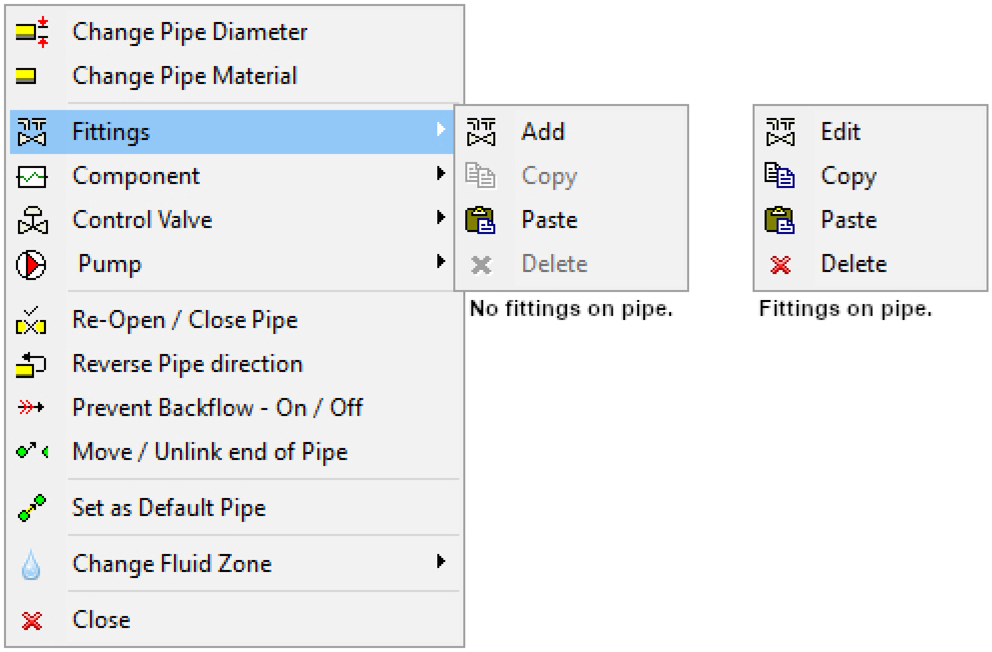


Figure 56 Pipe Pop-up Fittings sub-menu

Feature	Description
Add	Visible when there are no fittings on the pipe. Opens the pipe fittings friction coefficients screen so that new fittings can be added to the pipe.
Edit	Visible when there are fittings on the pipe. Opens the pipe fittings friction coefficients screen so that the existing fittings can be edited.
Copy	Enabled when there are fittings on the pipe. Copies the fittings on the pipe to the clipboard so that they can be pasted to another pipe.
Paste	Enabled when fittings have been copied to the clipboard using the copy option. Pastes the fittings from the clipboard to the current pipe.
Delete	Enabled when there are fittings on the pipe. Deletes the fittings on the pipe.

Pipe Pop-up Component sub-menu

Select Component on the pipe pop-up menu to display the component sub-menu.

The sub-menu options are context sensitive and will be visible and/or enabled/disabled depending upon the whether there is a component on the selected pipe.

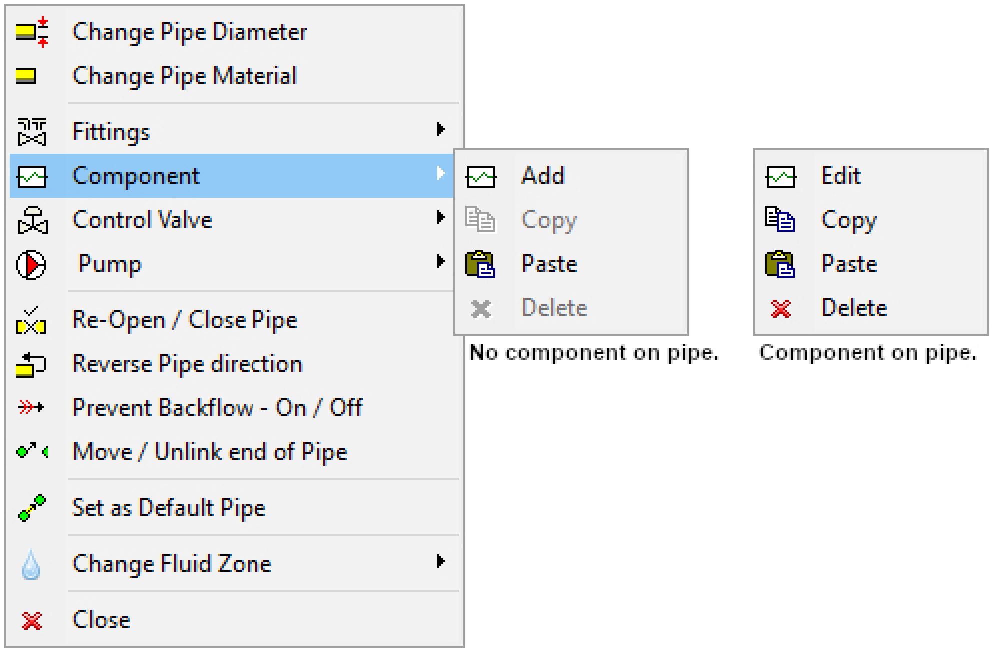


Figure 57 Pipe Pop-up Component sub-menu

Feature	Description
Add	Visible when there is no component on the pipe. Opens the component pressure loss screen so that a new component can be added to the pipe.
Edit	Visible when there is a component on the pipe. Opens the component pressure loss screen so that so that the existing component can be edited.
Copy	Enabled when there is a component on the pipe. Copies the component on the pipe to the clipboard so that it can be pasted to another pipe.
Paste	Enabled when a component has been copied to the clipboard using the copy option. Pastes the component from the clipboard to the current pipe.
Delete	Enabled when there is a component on the pipe. Deletes the component on the pipe.

Pipe Pop-up Control Valve sub-menu

Select Control Valve on the pipe pop-up menu to display the control valve sub-menu.

The sub-menu options are context sensitive and will be visible and/or enabled/disabled depending upon whether there is a control valve on the selected pipe.

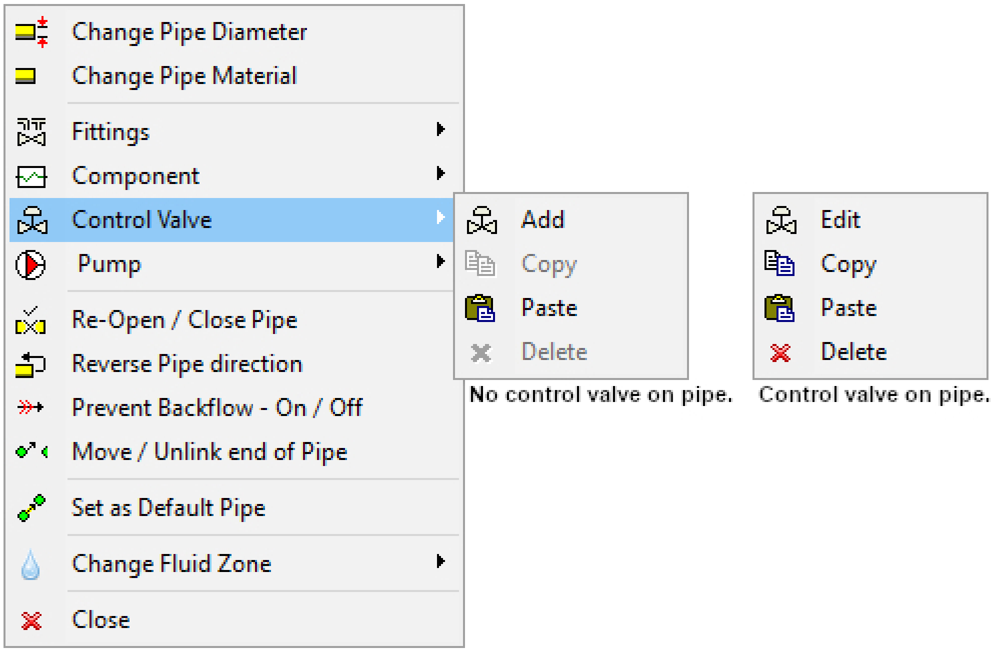


Figure 58 Pipe Pop-up Control Valve sub-menu

Feature	Description
Add	Visible when there is no control valve on the pipe. Opens the control valve data screen so that a new control valve can be added to the pipe.
Edit	Visible when there is a control valve on the pipe. Opens the control valve data screen so that so that the existing control valve can be edited.
Copy	Enabled when there is a control valve on the pipe. Copies the control valve on the pipe to the clipboard so that it can be pasted to another pipe.
Paste	Enabled when a control valve has been copied to the clipboard using the copy option. Pastes the control valve from the clipboard to the current pipe.
Delete	Enabled when there is a control valve on the pipe. Deletes the control valve on the pipe.

Pipe Pop-up Pump sub-menu

Select Pump on the pipe pop-up menu to display the pump sub-menu.

The sub-menu options are context sensitive and will be visible and/or enabled/disabled depending upon whether there is a pump on the selected pipe.

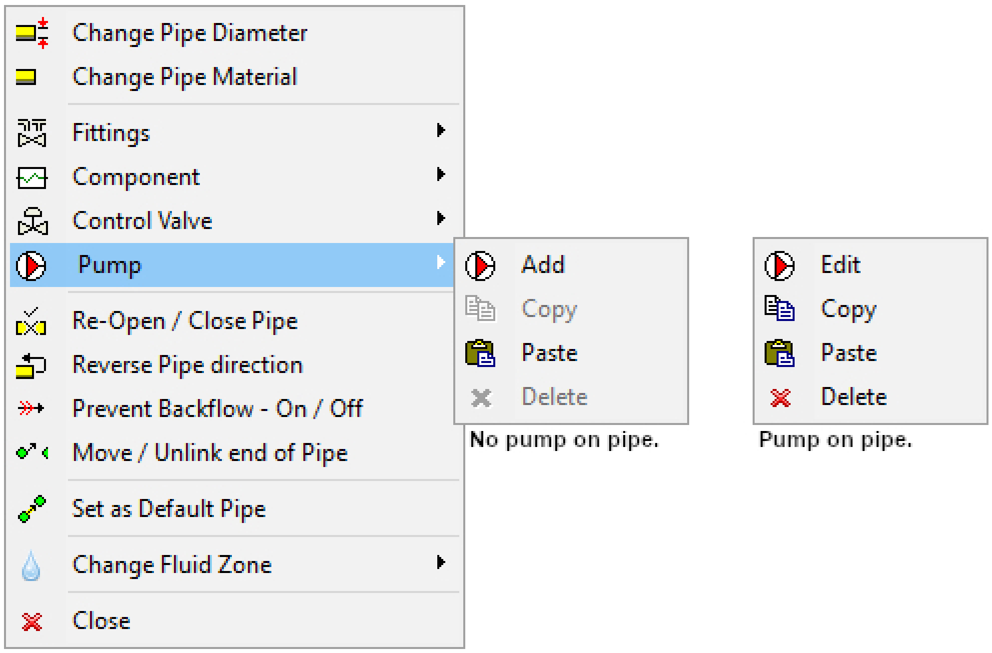


Figure 59 Pipe Pop-up Pump sub-menu

Feature	Description
Add	Visible when there is no pump on the pipe. Opens the pump data screen so that a new pump can be added to the pipe.
Edit	Visible when there is a pump on the pipe. Opens the pump data screen so that so that the existing pump can be edited.
Copy	Enabled when there is a pump on the pipe. Copies the pump on the pipe to the clipboard so that it can be pasted to another pipe.
Paste	Enabled when a pump has been copied to the clipboard using the copy option. Pastes the pump from the clipboard to the current pipe.
Delete	Enabled when there is a pump on the pipe. Deletes the pump on the pipe.

Preventing Backflow in a pipe

A selected pipe can be tagged to prevent backflow when the system is solved. A pipe in which backflow is not permitted is identified with a double arrow marker, .

When solving a system if backflow occurs in the pipe the flow rate will be set to zero.

Use the pipe pop-up menu option to toggle the status of pipes tagged with the prevent backflow symbol.

Using the Default Pipe feature while drawing

In Pipe Flow Expert, you can set the default pipe options for the next pipe to be added to the system by copying the features of the currently selected pipe.

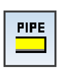
To set the default pipe options:

1. Select a pipe and click the right mouse button to display the **Pipe Pop-up menu.**

2. Click the Use Pipe Values for Drawing option.

Now when the next pipe in the system is drawn it will have the default attributes. You can then change the pipe length as necessary in the Pipe Pane.

To draw a pipe with different attributes, select another pipe that has the attributes that you require and then set that pipe as the default pipe for drawing.



Alternatively use the **Set Default Pipe for drawing** button on the tool bar to open the **Pipe Data tab** on the **Configuration Options screen.** Choose the pipe material, diameter, length and other attributes, to define the Default Pipe values that will be used when drawing.

Fittings and Valves

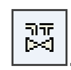
Fittings can be added to a pipe and they are selected to be at the start or end of the pipe (so as to associate the fitting with the join point before or after the end of the pipe) using the Pipe fitting friction coefficients dialog. The Pipe fitting friction coefficients screen contains shows a list of the fittings on a pipe and also display a list of fittings that are available in the fitting database. If you need to model a specific fixed pressure loss then you should choose to include this pressure loss by adding a Component to the pipe and then specifying the fixed pressure loss on the Set Component Pressure Loss screen.

The position of the fittings affects the NPSHa (Net Positive Suction Head available) value at the pump inlet. Fittings that are placed at the start of a pipe will be considered to be in front of the pump and therefore pressure loss through these fittings will be accounted for when calculating the NPSHa at the pump inlet. If you need to move a fitting, you can change its position by clicking on it to open the Fitting Manager and then selecting its position to be either at the start or end of the pipe.

NOTE: Choose a fitting size that matches the size of the pipe. For example, if you are using 4" nominal pipe size select either a 100 *mm* or a 4 *inch* fitting.

Adding a Fitting to a pipe

To add fittings to a pipe:

1. You can add a fitting using the **Add Fittings** button on the tool bar or the **Add/Change Fittings** button in the Pipe Pane.
2. To add a fitting using the **Add Fittings** tool bar button, click the **Add Fittings** button, .
3. When you click the **Add Fittings** button, the fittings symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane
4. Click on the pipe on the Drawing pane where you want to add a fitting.
5. Clicking on the pipe opens the **Pipe fitting friction coefficients** screen with a list of all the fittings available in the fitting database displayed in the **Fitting Database** list.
6. Alternately you can add a fitting by clicking the **Add/Change Fittings** button , 

. in the Pipe pane, once a pipe has been selected on the drawing and its information is displayed in the Pipe pane.
7. The number of fittings currently on the selected pipe is displayed on the **Add/Change Fittings** button. Clicking the **Add/Change Fittings** button opens the **Pipe fitting friction coefficients** dialog showing the fittings on the current pipe and all of the fittings available in the fitting database displayed in the **Fitting Database** list.

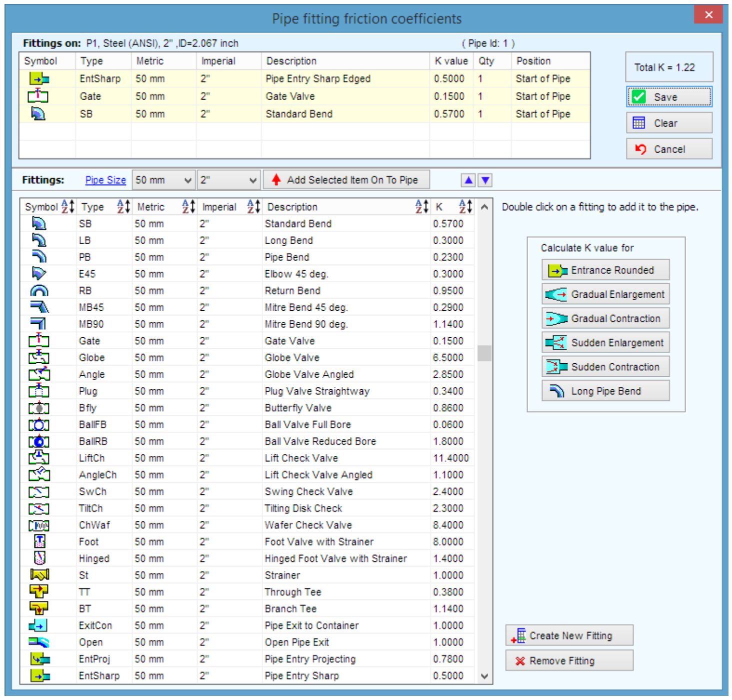


Figure 60 Pipe fitting friction coefficients database

8. Select the nominal metric or imperial pipe size from the drop down list to move to the required fitting sizes in the **Fitting Database**. Click on the Pipe Size link to show the fittings which match the current pipe size.

9. If the fitting is not in the **Fitting Database** list, click the **Create new fitting** button to add the new fitting to the list. For more information about adding a fitting to the database, see: *Adding a Fitting to the Database*

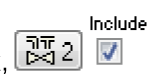
10. **Double click** on a fitting item to add it to the fittings on the pipe.

11. The selected fitting is displayed in the **Fittings on** section of the **Pipe fitting friction coefficients** screen.

12. Select the quantity of the fitting you are adding to the pipe from the fitting's **Qty** list in the **Fittings on** section, and also select the position of the fitting to be either start or end of pipe.

13. To add additional fittings to the pipe, repeat Steps 12 – 14.

14. Click **Save** to add the fitting to the pipe, and to close the **Pipe fitting friction coefficients** screen.

When one or more fittings are added to a pipe, the **Include** check box, 

is displayed and selected next to the **Add/Change Fittings** button. When the **Include** check box is selected, Pipe Flow Expert includes the data from the fittings on the pipe in the calculations for solving the pipe system.

To remove the fittings' data from the calculations, clear the **Include** check box.

Information about calculating and solving pipe systems is available in *Section: Calculating the System Flow and Pressure*.

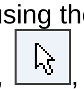
Adding a Fitting to the Database

If the fitting you want to add to a pipe does not exist in the fitting database, you can quickly add the fitting to the database and/or pipe by clicking the Create new fitting button or by clicking one of the Calculate K value buttons in the Pipe fitting friction coefficients dialog.

Adding a fitting involves selecting a fitting from the fitting database and changing the data to a specific K value for your bespoke fitting, or selecting one of the helper buttons from the Calculated K value section that calculates values for a specific type of fitting. The customized fitting data is then entered to match the new fitting and the helper calculates the fitting's K value.

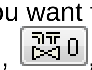
A K value represents a local loss coefficient. This value is supplied by the fitting manufacturer. Most fittings have a fixed K value, but some fittings have a K value that needs to be calculated. The Calculated K value section in the Pipe fitting friction coefficients dialog is used to add fittings with a calculated K value to a pipe system and the fitting database. When you click a fitting button in the Calculated K value section, a dialog opens that contains fields for adding the fitting measurements required to calculate the fitting's K value.

To add a fitting to the fitting database using the Create new fitting button:

1. Choose the **Show Item Info** button, , on the tool bar.

2. Click on a fitting symbol in the Drawing pane to open the **Pipe fitting friction coefficients** screen.

OR

Select the pipe for which you want to create a new fitting in the Drawing pane, and click the Add/Change Fittings button, , in the Pipe pane to open the Pipe fitting friction coefficients screen and then:

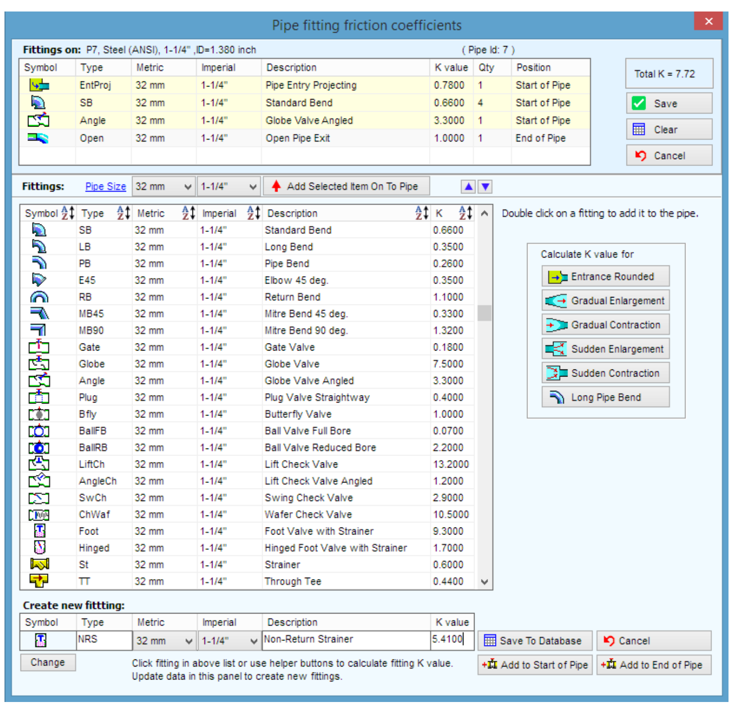


Figure 61 Add a fitting in the Pipe fitting friction coefficients listings

1. Click the **Create new fitting** button. This opens the Create new fitting section in the Pipe fitting friction coefficients screen

2. To change the symbol representing the fitting, click the **Change** button to open the **Choose symbol** screen.

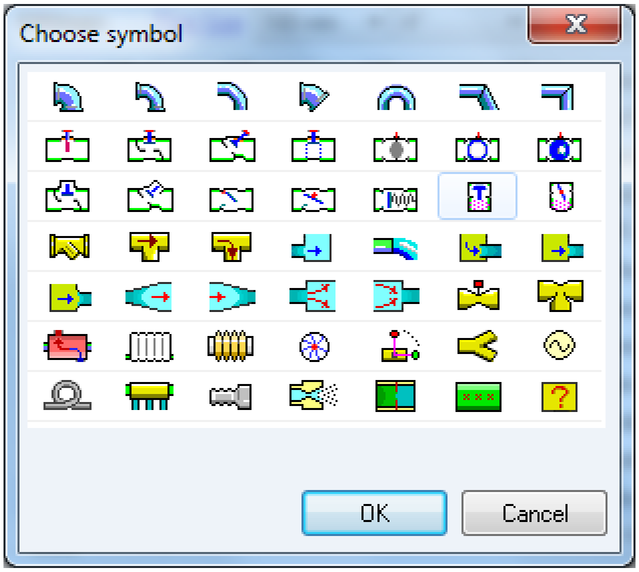


Figure 62 Choose fitting symbol

3. Select the symbol you want to represent the fitting.

4. Click **OK** to add the symbol to the **Symbol** field and close the **Choose symbol** screen.

5. Type the fitting type name in the **Type** field.

6. Select the fitting's metric size from the **Metric** list or select the fitting's **imperial** size from the drop down nominal size listings.

7. You only need to select the metric or imperial fitting size. Pipe Flow Expert automatically populates the other size for you. For example, if you select *32 mm* from the **Metric** list, the imperial equivalent, *1-1/4",* automatically appears in the **Imperial** field.

8. Type a description of the fitting in the **Description** field.

9. Click the **Save this fitting data to list** button to add the fitting to the **Fitting Database** list.

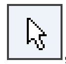
10. If you want to add the fitting to the pipe, as well as the fitting database, click the **Add this fitting to pipe** button to add the new fitting to the pipe.

11. The new fitting is displayed in the **Fittings on** section of the **Pipe fitting friction coefficients** screen.

12. Select the quantity of the fitting you are adding to the pipe from the fitting's **Qty** list in the **Fittings on** section.

13. Click **Save** to add the fitting to the pipe, and to close the **Pipe fitting friction coefficients** screen.

To add a fitting to the fitting database using a Calculate K value button:

1. Click the **Show Item Info** button, , on the tool bar.

2. Select a fitting in the Drawing pane to open the **Pipe fitting friction coefficients** screen.

Or

3. Select the pipe for which you want to create a new fitting in the Drawing pane, and click the **Add/Change Fittings** button, , in the Pipe pane to open the **Pipe fitting friction coefficients** screen.

4. Click the **Calculate K value** button that represents the fitting you are adding.

Clicking one of the **Calculate K value** buttons opens the **Create new fitting** section in the **Pipe fitting friction coefficients** screen, and the **K value** calculation screen for the selected button.

For example, if you selected the **sudden contraction** button, the **Sudden contraction K value** calculation screen opens.

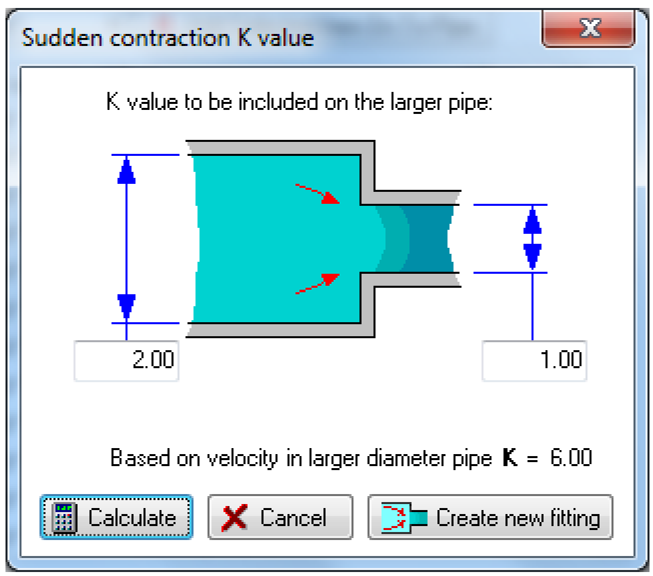


Figure 63 Sudden Contraction K value

5. Enter the applicable values in the **K value** calculation screen.

6. Click the **Calculate** button to calculate the fitting's K value.

7. The calculated K value is displayed in the **K value** dialog.

8. When you have the desired K value, click the **Create new fitting** button to close the **K value** calculation screen, and add the calculated K value to the **K value** field in the **Create new fitting** section in the **Pipe fitting friction coefficients** dialog.

9. The **Symbol** field defaults to the symbol on the **Calculated K value** button you selected.

10. To change the symbol representing the fitting, click the **Change** button to open the **Choose symbol** screen.

11. Select the symbol you want to represent the fitting. Click **OK** to add the symbol to the **Symbol** field and close the **Choose symbol** screen

12. The **Type** field defaults to the type of fitting associated with the **Calculated K value** button selected.

13. To edit the fitting type, type the fitting type name in the **Type** field.

14. The **Metric** and **Imperial** fields default to *N/A* for fittings with a calculated K value. The user can select the nominal metric or imperial pipe size from the drop down listings if appropriate.

15. Type a description of the fitting in the **Description** field.

16. Click the **Save this fitting data** to list button to add the fitting to the fitting database and **Fitting Database** list.

If you want to add the fitting to the pipe, instead of the fitting database:
Click the **Add this fitting to pipe** button to add the new fitting to the pipe.

The new fitting is displayed in the **Fittings on** section of the Pipe fitting friction coefficients screen.

1. Select the quantity of the fitting you are adding to the pipe from the fitting's **Qty** list in the **Fittings on** section.

2. Click **Save** to add the fitting to the pipe, and to close the **Pipe fitting friction coefficients** dialog.

Components

Certain components, such as strainers, boilers, heat exchanges, manifolds etc. cause a pressure loss in a system. In Pipe Flow Expert you can model pressure loss for components in a number of different ways – fixed pressure loss, variable pressure loss specified by a data curve, Cv value, Kv value, Sprinkler K value, and defined Orifice data. Pressure loss for components are entered and maintained on the Component Pressure Loss screen.

The pressure loss defined for a component can be saved to a .pfco file. This file provides a way to share the pressure loss data for a component, and quickly enter the pressure loss data when the same type of component is used again in the pipe system or a future pipe system. The .pfco file also provides a quick and easy way to evaluate different components with pressure loss in the pipe system to determine how each one affects the system.

In Pipe Flow Expert, you can only add one component pressure loss to each pipe. If there is more than one component on a pipe that causes a pressure loss, you can break up the applicable pipe into the number of sections corresponding with the number of components causing a pressure loss.

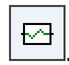
For example, if you have a pipe that is 100 feet long with two components causing a pressure loss, you can break the pipe into two pieces, each 50 feet long, and add one of the component pressure losses to each piece. How you divide a pipe does not matter, only the total length of the pieces added together matters – so that if the pipe is supposed to be 100 feet long, then the lengths of the pieces need to add up to 100 feet.

NOTE: Be sure to enter the value for the pressure loss properties in the units displayed next to the field. For example, if you are using imperial units, and *psi.g* for pounds per square inch is displayed next to the Fixed Loss field, enter the value in pounds per square inch.

Adding a Component with a pressure loss

To add a component's pressure loss to a pipe:

1. You can add the pressure loss using the **Add Component Pressure Loss** button on the toolbar or the **Add/Change Component Pressure Loss** button in the Pipe Pane.

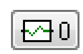
2. To add a pressure loss using the **Add Component Pressure Loss** tool bar button, click the **Add Component Pressure Loss** button, .

3. When you click the **Add Component Pressure Loss** button, the pressure loss symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.

4. Click on the pipe on the Drawing pane where you want to add a component pressure loss.

5. Clicking on the pipe opens the **Component Pressure Loss** dialog.

6. To add a pressure loss using the **Add/Change Component Pressure Loss** button in the Pipe pane, first select the pipe to which you want to add a Component in the Drawing pane, so that the pipe's information is displayed in the Pipe pane.

7. Click the Add/Change Component Pressure Loss button,  in the Pipe pane.

8. The number of component pressure losses currently on the selected pipe is displayed on the **Add/Change Component Pressure Loss** button. Clicking the **Add/Change Component Pressure Loss** button opens the **Set Component Pressure Loss** dialog.

9. To add a component's pressure loss data from an existing .pfco file, click the **Load From File** button, select the applicable .pfco file, click **Open**, edit the applicable fields, and then click the **OK** button. An image of the component pressure loss screen is shown below this step.

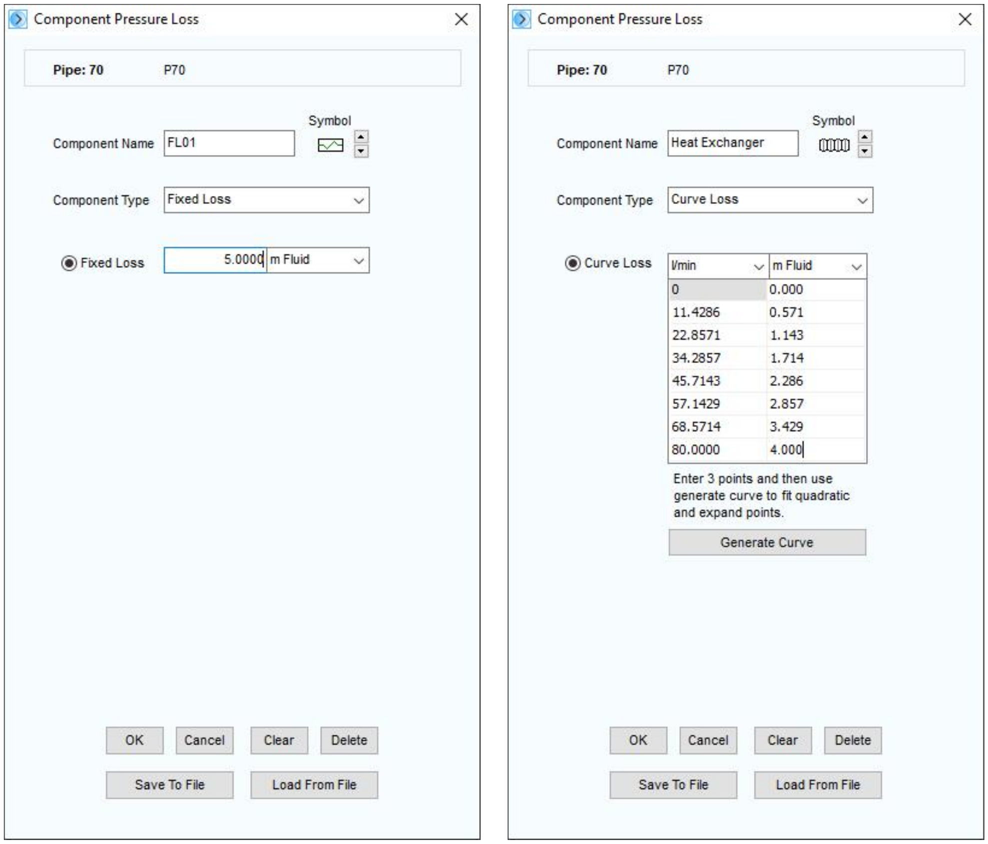


Figure 64 Set Component Pressure Loss

10. Type the name of the component pressure loss in the **Component Name** field.

11. Select the **Fixed Loss** Component Type, if the component's pressure loss is a fixed pressure loss, and type the fixed pressure loss value.

12. Select the **Curve Loss** Component Type, if the component's pressure loss is based on a data curve.

13. Select **Cv** or **Kv** Component Type if the component flow versus pressure loss is defined in this manner, and type the appropriate value. **See the next section about Cv and Kv values** for specific information about entering and using this type of data.

14. Select the **Sprinkler K (imperial)** or **Sprinkler K (metric)** Component Type if the component flow versus pressure loss is defined by this type of value. A sprinkler will often have a specific Sprinkler K value (this is different to a normal K value for head loss through a fitting, and it is different to a Kv value that is normally associated with a valve). **See the later section on Sprinkler K values.**

15. Select the Orifice Component Type if the component pressure loss occurs due to a sharp-edged, round-edged, bevel-edged, or thick-edged orifice. **See the later section on defining Orifice data**.

16. A symbol appropriate for the chosen Component Type is selected by default. The symbol can be changed by clicking the Symbol **Scroll Up** or **Scroll Down** button to select the component pressure loss image you want to be displayed on the pipe system drawing.

17. The image you select does not affect any of the component's pressure loss properties or values.

18. Select the applicable units for the component pressure loss curve from the **Curve Loss** flow and head lists.

19. The left **Curve Loss** column represents the fluid's flow values, and the right **Curve Loss** column represents the fluid head loss or pressure loss values.

20. Determine which points in the component's pressure loss curve data (as provided by the component manufacturer) that you want to include in the component pressure loss curve, and then type the flow and head loss values for each of these points in the **Curve Loss** table.

21. Add a minimum of three and maximum of eight points in the **Curve Loss** table. After adding three curve points to the table, you can have Pipe Flow Expert calculate a 'curve' through these points, to expand the number of data points automatically (fitting points on this curve). Click the **Generate Curve** button to perform this action.

22. Click **OK** to save and add the component pressure loss data to the pipe.

The pressure loss data you enter can be saved to a .pfco file by clicking the **Save To File** button. By saving the data to a .pfco file, you can reuse the component's pressure loss data for other pipes in the pipe system or future pipe systems.

When a component pressure loss is added to a pipe, the **Include** check box is displayed and selected next to the **Add/Change Component Pressure Loss** button, . When the **Include** check box is selected, Pipe Flow Expert includes the data from the component pressure loss on the pipe in the calculations for solving the pipe system. To remove the component pressure loss data from the calculations, clear the **Include** check box.

Information about calculating and solving pipe systems is available in the *Section: Calculating the System Flow and Pressure*.

Cv and Kv Flow Coefficients

There are many different types of components and control valves. Each may have different flow / pressure loss characteristics. Manufacturers of control valves usually publish a Cv flow coefficient or a Kv flow coefficient to describe the flow versus pressure loss characteristics of their control valves in a standardized manner.

A Cv flow coefficient specifies the volume of water in US gpm at 60°F (15.55°C) that will flow through a valve with a 1.0 psi pressure drop across the valve.

Thus a Cv flow coefficient of 10 indicates that a 1.0 psi pressure drop will occur with a flow of 10 US gpm of water through the valve. Pressure drop for the different fluids and different flow rates can be calculated from the Cv flow coefficient. See the 'Calculation Theory and Methods of Solution' section for further details.

A Kv flow coefficient specifies the volume of water in m^3/hour at 20°C (68 °F) that will flow through a valve with a 1.0 bar pressure drop across the valve.

Thus a Kv flow coefficient of 10 indicates that a 1.0 bar pressure drop will occur with a flow of 10m^3/hour of water through the valve. Pressure drop for the different fluids and different flow rates can be calculated from the Kv flow coefficient. See the 'Calculation Theory and Methods of Solution' section for further details.

The component object is used to model the flow rate / pressure loss of control valves with Cv or Kv flow coefficients.

Adding a component/valve with a Cv or Kv value

To add a component/valve with a Cv or Kv flow coefficient:

1. Add a component to a pipe as described in the previous section.

2. Type the name of the control valve in the **Component Name** field.

3. Select the Cv Value or Kv Value Component Type.

4. The symbol appropriate for the chosen Component Type is selected by default. The symbol can be changed by clicking the Symbol **Scroll Up** or **Scroll Down** button to select the component pressure loss image you want displayed on the pipe system drawing.

5. Enter the flow coefficient value for the component.

6. Click **OK** to save and add the Cv/Kv flow coefficient data to the pipe.

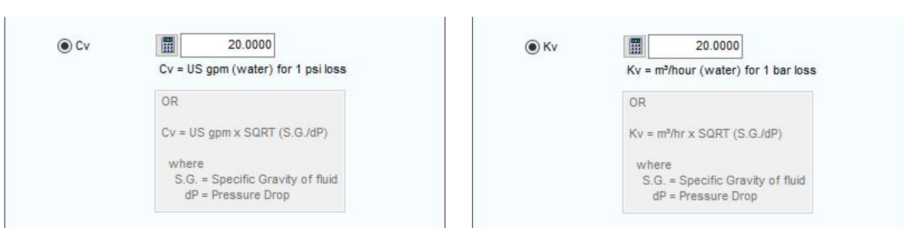


Figure 65 Adding a Cv / Kv flow coefficient value

The component dialog includes 'helpers' to assist in calculating an appropriate Cv or Kv flow coefficient value for a particular flow rate and pressure drop (based on the current fluid). Click the calculator button to display the calculator.

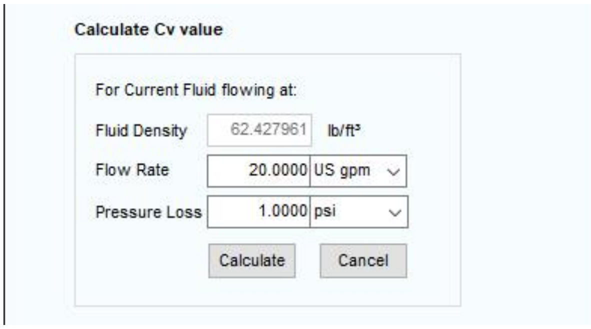


Figure 66 Helper to calculate a Cv value

The user should be aware that the Cv or Kv flow coefficient specifies the flow rate of water for a particular pressure loss.

When the fluid density is greater or less than water, a different flow rate of the fluid will be required to produce a 1.00 psi or a 1 bar pressure loss through the valve.

CAUTIONS for Compressible Gas Flow:

If the fluid is a gas, the flow rate entered into the calculator must represent the actual flow rate for the gas at the density that is defined in the fluid data for this fluid zone, i.e. the volumetric flow rate of the gas based on the gas density defined in the current fluid. The equivalent mass flow will then be calculated and this will be converted to an equivalent volume of water at normal conditions. The volume of water and the stated pressure drop will then be used to calculate a Cv value (which by definition is based on a volume of water flow for a certain pressure drop). Generally, for gas systems, it would be better to use a mass flow rate entry for the calculation instead of a volumetric flow rate.

If the fluid is a gas and the pressure drop exceeds the critical pressure ratio when compared to the inlet pressure of the valve then **the flow will become choked and it will not be possible to achieve the calculated flow rate**. If this occurs, Pipe Flow Expert will warn of a problem in the Result Log (if the Component option to check for choked flow has been selected).

Gas Flow Calculation with the Compressible Flow Calculation Engine

Gas systems should generally be solved using the Compressible flow calculation engine, which takes account of the pressure condition at the component and adjusts the density of the gas as appropriate when performing the component pressure loss calculation.

The fluid properties should be defined at the operating temperature for a particular section of the system (while the software adjusts fluid properties for changes of pressure as they are used in calculations, it assumes the temperature of the fluid remains the same and therefore the user must define the fluid properties for the appropriate temperature conditions within the pipe network). The software uses the viscosity of the gas as defined in the current fluid data and this is not adjusted for changes in pressure (pressure changes generally have a small impact on viscosity in comparison to temperature changes).

See the section on *Working with Compressible Fluids* for more information.

Gas Flow Calculation with the Non-Compressible Flow Calculation Engine:

Prior to version 7, Pipe Flow Expert did not include a specialist Compressible Flow calculation engine, however it allowed calculation of gas systems using the non-compressible flow calculation engine that used the Darcy-Weisbach equation, which used a constant density and viscosity across the calculations. The user was required to define the fluid data to represent the density at the approximate average pressure condition in a section of the network.

When using the Non-Compressible calculation engine with the Darcy-Weisbach equation to solve gas systems, Pipe Flow Expert will calculate the pressure drop through the component/valve for a particular flow rate at a given density as defined by the current fluid zone based on its Cv or Kv flow coefficient value.

If the fluid zone associated with the control valve does not represent the pressure condition at the outlet of the valve/component, it may be necessary to use an adjusted Cv (or Kv) value for valve selection to take in to account the effect of the gas expansion. The adjusted Cv (or Kv) value should be based on the Cv or Kv formula for sub critical gas pressure drop.

We recommend you use the Compressible Flow calculation engine to solve gas systems.

See ‘Calculation Theory and Methods of Solution’ section for further details about Cv and Kv flow coefficient calculations for systems where the fluid is a gas.

Cv & Kv Control Valve Sizing and Open Position

The Cv (or Kv) flow coefficient of a component/valve is usually stated for the fully open flow condition. The Cv (or Kv) flow coefficient will be less when the valve is partly closed. In an actual system it is important to select a control valve which has an appropriate Cv or Kv flow coefficient for the actual valve position that will be used.

A control valve that is too small or too large will not be able to provide the correct control characteristics in a pipe system.

Most control valve manufacturers recommend that you should select a valve where the required Cv (or Kv) value matches the given valve Cv (or Kv) value when operating within 20% - 80% of the valve’s range.

When selecting a valve to control a ‘top end’ pressure loss you should select a valve where the required Cv (or Kv) value is in the higher operating range of the valve, normally where the required Cv (or Kv) value matches the given valve Cv (or Kv) value when operating at about 70% of the valve’s range.

Some control valve manufacturers recommend that an allowance of 30% should be added to the required Cv (or Kv) flow coefficient to obtain the minimum Cv (or Kv) flow coefficient rating that the selected valve should have.

Please check your control valve selection with the control valve manufacturer.

Sprinkler K factors

Manufactures of fire sprinkler nozzles sometimes publish sprinkler K factor coefficients which describe the flow rate and pressure loss relationship through the nozzle. Usually the sprinkler k factor is determined by the relationship of flow rate through the nozzle divided by the square root of the loss through the nozzle. K = Flow / √dP

The sprinkler K value can describe the sprinkler performance in either imperial or metric flow rates (there is a metric K value and an imperial K value for the same sprinkler and these will be different values, hence it is important to check a sprinkler K value to confirm if it is a metric value or an imperial value).

The imperial sprinkler K value is calculated from US gpm (water) / √pressure loss (psi)

The metric sprinkler K value is calculated from L/min (water) / √pressure loss (bar)

When a sprinkler K (imperial or metric) radio button is selected a calculator button is enabled to allow an appropriate sprinkler K value to be obtained from a published flow rate and pressure loss.

Note: Sprinkler K factor coefficients are not the same as 'K' factors for standard fittings & valves.

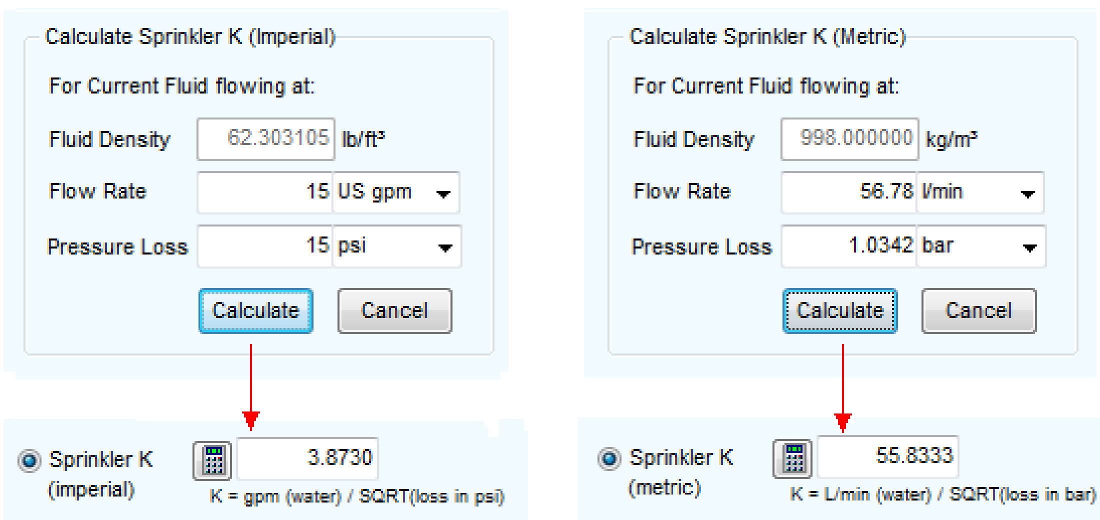


Figure 67 Calculate Sprinkler K value

The component object can be used to model the flow rate versus pressure loss of a sprinkler component either by using a single Sprinkler K value, or by specifying a set of flow versus pressure loss curve points that defines the performance of the sprinkler.

Modeling a Spray Nozzle with a Sprinkler K Factor

Many systems include points where flow will discharge to atmosphere. When a spray nozzle is included in the design it is usual to use the flow versus pressure loss characteristic of the spray nozzle to establish the flow leaving the system due the pressure differential across the nozzle.

A combination of a Component and an End Pressure node can be used to model the performance of a spray nozzle.

The End Pressure node should be set to 0.000 psi.g, if the spray discharges to atmosphere. If the spay discharges to a pressurized tank then the End Pressure node should be set to the pressure in the tank.

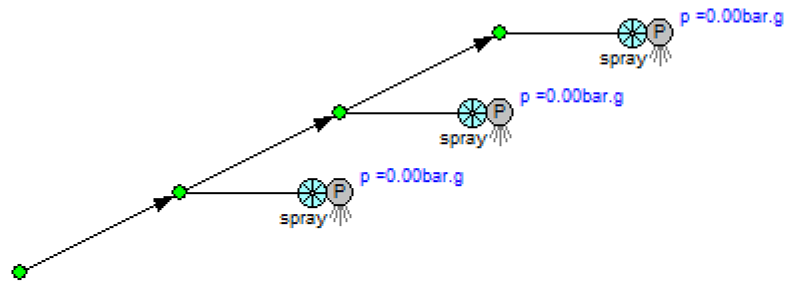


Figure 68 Modeling a Spray Nozzle

To add a component with a Sprinkler K value:

1. Add a component to a pipe as described in the previous sections.
2. Type the name of the control valve in the **Component Name** field.
3. The symbol appropriate for the chosen Component Type is selected by default. The symbol can be changed by clicking the Symbol **Scroll Up** or **Scroll Down** button to select the component pressure loss image you want displayed on the pipe system drawing.
4. Select the Sprinkler K radio button (for imperial or metric values) and enter the Sprinkler K factor value (or use the calculator helper to compute a sprinkler K factor for a given flow rate and pressure drop characteristic of the sprinkler).
5. Click **OK** to save and add the Sprinkler K factor data to the pipe.

Note: The component dialog includes ‘helpers’ to assist in calculating an appropriate Sprinkler K value for a particular flow rate and pressure drop (based on the current fluid).

Orifices

Orifices are widely installed in piping systems to provide a regular and reproducible loss of pressure. Orifices are often used to limit flow, or in branching systems, to balance or distribute flow as required through different parts of the system.

Pipe Flow Expert provides support for the following orifice types:

- Sharp Edged
- Round Edged
- Bevel Edged
- Thick Edged

To add a component with an Orifice:

1. Add a component to a pipe as described in the previous sections.

2. Type the name of the orifice in the **Component Name** field.

3. Select the Sharp Edged, Round Edged, Bevel Edged or Thick Edged orifice type.

4. The symbol appropriate for the chosen Component Type is selected by default. The symbol can be changed by clicking the Symbol **Scroll Up** or **Scroll Down** button to select the component pressure loss image you want displayed on the pipe system drawing.

5. Enter the appropriate data for the orifice type (see following sections).

6. Click **OK** to save and add the orifice to the pipe.

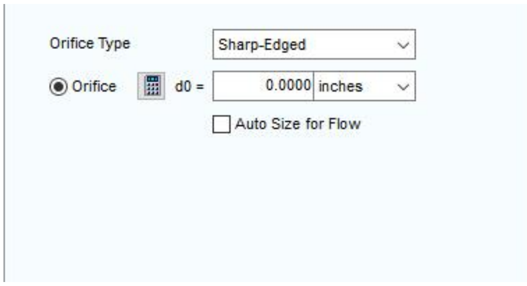
Notes:

In Pipe Flow Expert, orifice components are always positioned at the end of the associated pipe. The associated pipe defines the pipe diameter preceding the orifice. The pipe following the orifice defines the pipe diameter that follows the orifice. The preceding and following pipe may have the same diameter and this would represent an orifice in a straight pipe. If the preceding and following pipe have different diameters then this represents a transition section, where the pipe diameter may get smaller or larger following the orifice. The Pipe Flow Expert software supports all of these cases when performing orifice calculations.

The component orifice screen includes a ‘helper’ to assist in calculating an appropriate orifice diameter (d_0) value based on the flow rate at some inlet pressure, and the required pressure drop, as specified by the user.

In addition, there is an option that allows the software to automatically size the orifice when the whole system is calculated. In this case, the orifice diameter is automatically calculated such that it produces the required pressure drop in order to achieve a specified flow rate in the pipe, when considering all of the other flows and pressures throughout the system. The user can then review the size of orifice required, together with the additional technical information provided such as inlet and outlet velocities, the jet velocity ratio, and the inlet and outlet pressures.

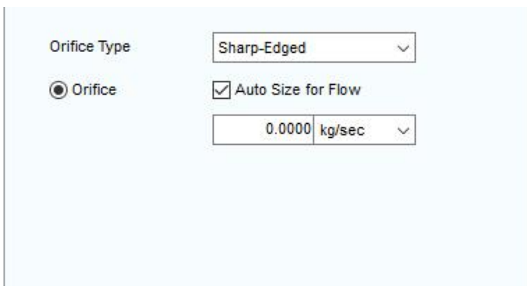
Adding a Sharp Edged Orifice



To define a Sharp Edged orifice with a specific diameter (d_0):

1. Select the Sharp Edged Orifice Type.

2. Enter the Orifice Diameter (d_0) as required.

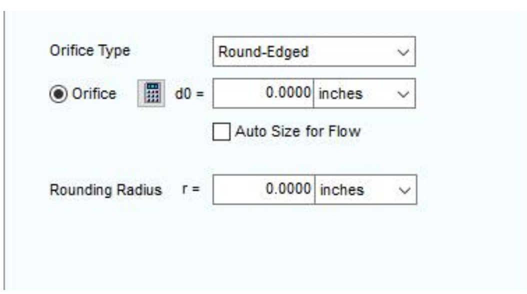


To define a Sharp Edged orifice with a diameter (d_0) for a specific flow rate:

1. Select the Sharp Edged Orifice Type.

2. Select the Auto Size for Flow checkbox and enter the flow rate as required.

Adding a Round-Edged Orifice

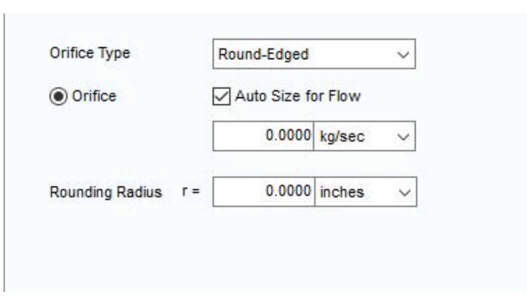


To define a Round-Edged orifice:

1. Select the Round-Edged Orifice Type.

2. Enter the Orifice Diameter (d_0) as required.

3. Enter the Rounding Radius (r) as required.



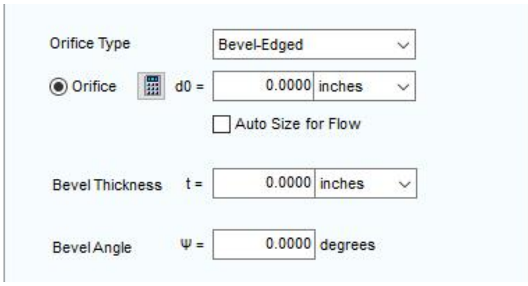
To define a Round-Edged orifice with a diameter (d_0) for a specific flow rate:

1. Select the Round-Edged Orifice Type.

2. Select the Auto Size for Flow checkbox and enter the flow rate as required.

3. Enter the Rounding Radius (r) as required.

Adding a Bevel-Edged Orifice



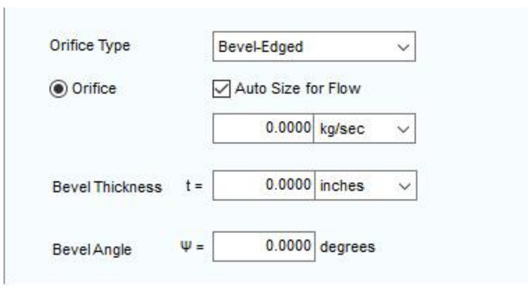
To define a Bevel Edged orifice:

1. Select the Bevel-Edged Orifice Type.

2. Enter the Orifice Diameter (d_0) as required.

3. Enter the Bevel Thickness (t) as required.

4. Enter the Bevel Angle (ψ) as required.



To define a Bevel-Edged orifice with a diameter (d_0) for a specific flow rate:

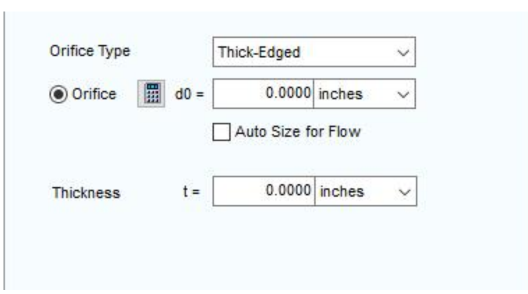
1. Select the Bevel-Edged Orifice Type.

2. Select the Auto Size for Flow checkbox and enter the flow rate as required.

3. Enter the Bevel Thickness (t) as required.

4. Enter the Bevel Angle (ψ) as required.

Adding a Thick-Edged Orifice

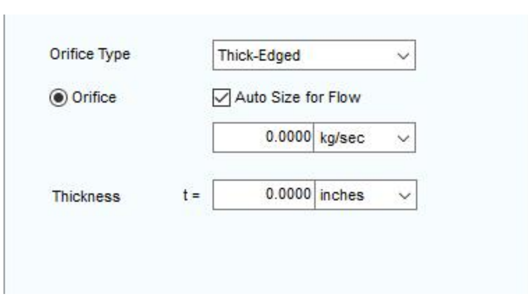


To define a Thick-Edged orifice:

1. Select the Thick-Edged Orifice Type.

2. Enter the Orifice Diameter (d_0) as required.

3. Enter the Thickness (t) as required.



To define a Thick-Edged orifice with a diameter (d_0) for a specific flow rate:

1. Select the Thick-Edged Orifice Type.

2. Select the Auto Size for Flow checkbox and enter the flow rate as required,

3. Enter the Thickness (t) as required.

Control Valves (FCV, PRV, BPV)

Control valves may be used to control flow or pressure at various points in a system.

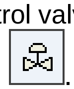
- A flow control valve (FCV) determines the required flow rate for the fluid in a pipe.
- A pressure reducing valve (PRV) controls the pressure at the end of a pipe.
- A back pressure valve (BPV) controls the pressure at the start of a pipe.

The modeling of Flow Control Valves, Pressure Reducing Valves and Back Pressure Valves requires that the flow rates and pressures from adjacent pipes must be used as a reference to establish the pressure loss that control valve must introduce.

To allow this modeling a control valve cannot be added to any pipe where a joining pipe already includes a control valve.

To add a control valve to a pipe:

1. You can add a flow control using the **Add Control Valve** button on the tool bar or the **Add/Change Control Valve** button in the Pipe Pane.

2. To add a control valve using the **Add Control Valve** tool bar button, click the **Add Control Valve** button, .

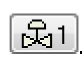
3. When you click the **Add Control Valve** button, the control valve symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.

4. Click on the pipe on the Drawing pane where you want to add a control valve.

5. Clicking on the pipe opens the **Set Control Data** screen.

6. To add a flow control using the **Add/Change Control Valve** button in the Pipe pane, select the pipe to which you want to add a control valve in the Drawing pane.

7. The pipe's information is displayed in the Pipe pane.

8. Click the Add/Change Control Valve button, .

9. If there is a control valve currently on the selected pipe a number 1 is displayed on the **Add/Change Control Valve** button. Clicking the **Add/Change Control Valve** button opens the **Set Control Data** screen.

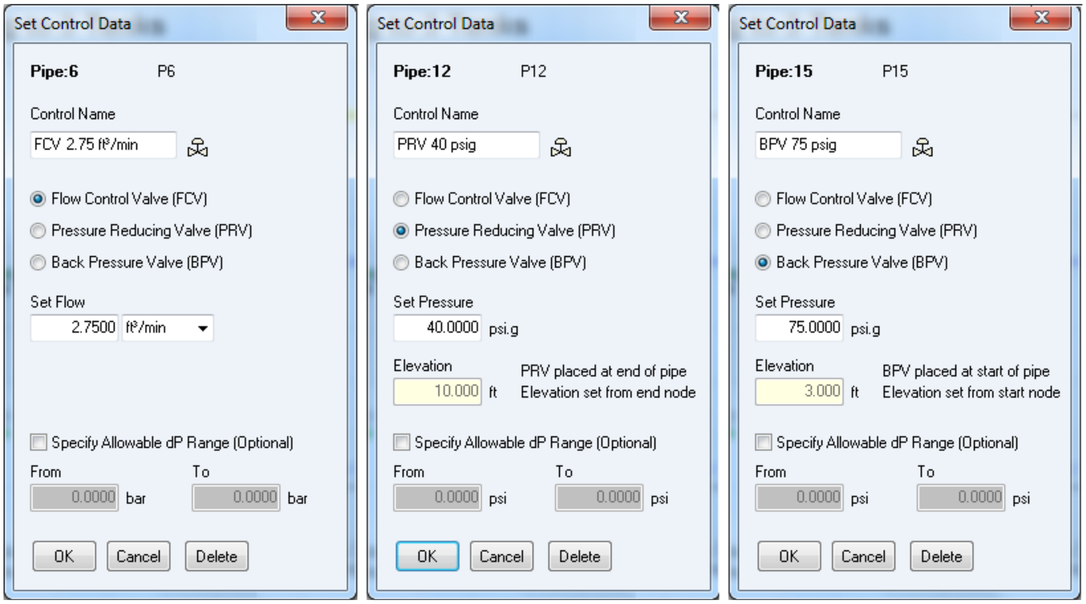


Figure 69 Set Control Valve Data

10. Select the type of control valve to be added (FCV, PRV or BPV) by clicking one of the radio buttons.

11. The screen will change to allow the entry of data for the type of valve selected.

12. Enter the appropriate data for the control valve type.

13. If you wish to enter an allowable differential pressure range for the valve operation click the check box and enter the 'From' and 'To' pressure values. When the system is solved a warning will be issued if the differential pressure across the control valve is outside the entered range.

14. Click **OK** to add the control valve to the pipe.

When a control valve of any type is added to a pipe, the **Include** check box is displayed and selected next to the **Add/Change Flow Control** button.

When the **Include** check box is selected, Pipe Flow Expert includes the data from the flow controls on the pipe in the calculations for solving the pipe system. To remove the flow control data from the calculations, clear the **Include** check box.

Information about calculating and solving pipe systems is available in *Section: Calculating the System Flow and Pressure*.

Adding a Flow Control Valve

A flow control valve determines the required flow rate for the fluid in a pipe.

1. Open the **Set Control Data** dialog as described in the previous section.

2. Click the flow control valve (FCV) radio button.

3. Type the name of the flow control valve in the **Control Name** field.

4. Select the appropriate units of flow from the drop down list.

5. Type the required flow rate for the fluid in the pipe in the **Set Flow** field.

6. Enter the allowable differential pressure range for the valve operation if appropriate.

7. Click **OK** to add the flow control valve to the pipe.

NOTE: The flow control valve (FCV) introduces an additional pressure loss in the pipe to control the flow to the value specified by the user. The additional pressure loss is reported in the results tables.

Using the flow rate and the additional pressure loss the user can calculate the Cv or Kv value for a suitable valve which will allow the flow to be controlled at the required flow rate.

The flow control valve (FCV) cannot introduce a negative pressure loss. The FCV would need to act as pump to add pressure, which it cannot do. If this situation is encountered when a system is analyzed it will not be possible to identify a valid solution. The flow control valve setting will have to be amended or the flow control valve may have to be removed to allow a solution to be calculated.

Adding a Pressure Reducing Valve

A pressure reducing valve controls and regulates the pressure downstream of the valve.

1. Open the **Set Control Data** dialog as described in the previous section.

2. Click the pressure reducing valve (PRV) radio button.

3. Type the name of the pressure reducing valve in the **Control Name** field.

4. Type the required reduced pressure in the **Set Pressure** field.

5. Enter the allowable differential pressure range for the valve operation if appropriate.

6. Click **OK** to add the pressure reducing valve to the pipe.

NOTE: The pressure reducing valve (PRV) introduces an additional pressure loss in the pipe to control the downstream pressure at the end of the pipe to the value specified by the user. The additional pressure loss across the control valve is reported in the results tables.

A PRV can operate under three different conditions: (1) regulating, (2) fully closed, and (3) fully open. How the valve operates depends on the defined set pressure value for the valve. The fully open and fully closed positions represent the extreme operations of the valve. Each of the valve positions is described below:

(1) Regulating The valve maintains the downstream pressure to the set value by introducing a pressure loss across the valve, thus throttling the flow rate through the PRV.

(2) Fully Closed This mode of operation occurs if the valve’s set pressure is less than the pressure downstream of the valve for the case where the valve is closed. When this occurs in an actual pipe system, the flow through the PRV reverses and the PRV acts as a check valve, closing the pipe. In PipeFlow Expert, this method of operation is detected and reported but the system is not then solved for this scenario. The user must decide if this method of operation is what they intended and if so then they can close the pipe and continue to solve the system.

(3) Fully Open This mode of operation occurs if the valve's set pressure is greater than the pressure upstream of the valve for the case where the valve is fully open. When this occurs in an actual pipe system, the PRV maintains a fully open position and it has no effect on the flow conditions (except to add a frictional loss through the valve). In PipeFlow Expert, this method of operation is detected and reported. But the system is not solved because the differential pressure across the valve would have to be negative, i.e. the valve would be acting like a pump rather than a pressure control.

Pipe Flow Expert will only solve a system when the PRV is operating in Regulating mode.

AVOIDING PRV OPERATION PROBLEMS: In general, PRV operation problems can be avoided by finding the valve's pressure regulating range and specifying the valve's set pressure to a value within this range such that the mode of operation is 'Regulating'. First, solve the system without the PRV control and note the pressure at the node downstream of the pipe which previously contained the PRV. This is the maximum pressure the PRV can be set to (i.e. it is equivalent to finding the valve's inlet pressure for the case where the valve is fully open). Secondly, solve the system after closing the pipe that contains the PRV and note the pressure at the node downstream of the closed pipe. This is the minimum pressure the PRV can be set to (i.e. it is equivalent to finding the pressure downstream of the valve for the case where the valve is fully closed).

Adding a Back Pressure Valve

A back pressure valve controls and regulates the pressure upstream of the valve.

1. Open the **Set Control Data** dialog as described in the previous section.
2. Click the back pressure valve (BPV) radio button.
3. Type the name of the back pressure valve in the **Control Name** field.
4. Type the required back pressure in the **Set Pressure** field.
5. Enter the allowable differential pressure range for the valve operation if appropriate.
6. Click **OK** to add the back pressure valve to the pipe.

NOTE: The back pressure valve (BPV) introduces an additional pressure loss in the pipe to control the upstream pressure at the start of the pipe to the value specified by the user. The additional pressure loss across the control valve is reported in the results tables.

A BPV can operate under three different conditions: (1) regulating, (2) fully closed, and (3) fully open. How the valve operates depends on the defined set pressure value for the valve. The fully open and fully closed positions represent the extreme operations of the valve. Each of the valve positions is described below:

(1) Regulating The valve maintains the upstream pressure to the set value by introducing a pressure loss across the valve, thus reducing the flow rate through the BPV.

(2) Fully Closed This mode of operation occurs if the valve's set pressure is greater than the pressure upstream of the valve for the case where the valve is closed. When this occurs in an actual pipe system, the flow through the BPV reverses and the BPV acts as a check valve, closing the pipe. In PipeFlow Expert, this method of operation is detected and reported but the system is not then solved for this scenario. The user must decide if this method of operation is what they intended and if so then they can close the pipe and continue to solve the system.

(3) Fully Open This mode of operation occurs if the valve’s set pressure is less than the pressure downstream of the valve for the case where the valve is fully open. When this occurs in an actual pipe system, the BPV maintains a fully open position and it has no effect on the flow conditions (except to add a frictional loss through the valve). In Pipe Flow Expert, this method of operation is detected and reported but the system is not solved because the differential pressure across the valve would have to be negative, i.e. the valve would be acting like a pump rather than a pressure control.

Pipe Flow Expert will only solve a system when the BPV is operating in regulating mode.

AVOIDING BPV OPERATION PROBLEMS: In general, BPV operation problems can be avoided by finding the valve's pressure regulating range and specifying the valve’s set pressure to a value within this range such that the mode of operation is ‘Regulating’. First, solve the system without the BPV control and note the pressure at the node upstream of the pipe which previously contained the BPV. This is the minimum pressure the BPV can be set to (i.e. it is equivalent to finding the pressure at the valve outlet for the case where the valve is fully open). Secondly, solve the system after closing the pipe that contains the BPV and note the pressure at the node upstream of the closed pipe. This is the maximum pressure the BPV can be set to (i.e. it is equivalent to finding the pressure at the valve inlet for the case where the valve is fully closed).

Pumps

A pump can be added to any pipe, except a pipe that contains a control valve.

Options are provided to choose how the pump should be modeled:

- A Fixed Flow Rate Pump
- A Fixed Head / Pressure Rise Pump
- A Fixed Speed Pump (with a flow versus head performance curve)

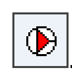
A pump's properties, including its pump curve, are defined and graphed on the Pump Data screen. The Pump Data screen has an interactive graph that allows you to determine which elements of the pump graph are displayed. It also provides functionality to calculate pump values for different points on the graph, including the pump's flow rate, head added, operating efficiency percentage and NPSHr (Net Positive Suction Head requirement).

You can also save the data associated with a Pump to the Pump Database or a .pfpm file. This provides a way to share the pump data, and quickly load the pump data, when the same type of pump is used again in the pipe system or a future pipe system. Saving and loading pump data also provides a quick and easy way to evaluate different pumps at a specific point in the pipe system to determine how each one affects the system.

NOTE: Be sure to enter the values for the pump data in the correct units (you can select which units you wish to use). For example, if you are using metric units, and *m* for meter is displayed next to a field, enter the pump value in meters.

Adding a Pump

To add a pump to a pipe:

1. You can add a pump using the **Add Pump** button on the tool bar or the **Add/Change Pump** button in the Pipe Pane.
2. To add a pump using the **Add Pump** tool bar button, click the **Add Pump** button, .
3. When you click the **Add Pump** button, the pump symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
4. Click on the pipe on the Drawing pane where you want to add a pump.
5. You can add a pump to a pipe in the system. Clicking on the pipe opens a **Confirm** dialog.
6. The pump is positioned at the start of the pipe, essentially 0m in distance along the pipe, immediately after any fittings that are included at the start of the pipe. The **Elevation** field is automatically populated and set to be the same as the elevation of the node at the start of the pipe.

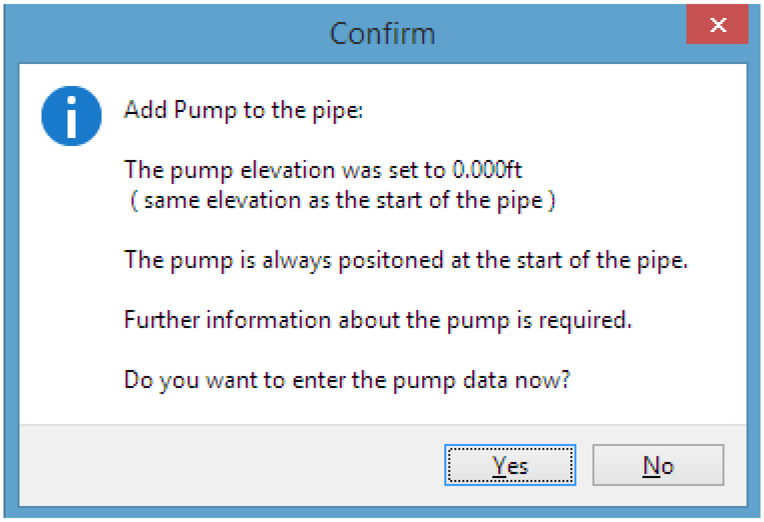


Figure 70 Add Pump Confirm dialog

7. Click **Yes** when asked, Do you want to enter the pump data now? Clicking **Yes** adds the pump to the system, and opens the **Pump Data** screen.

8. To add a pump using the **Add/Change Pump** button in the Pipe pane, select the pipe to which you want to add a pump in the Drawing pane.

9. The pipe's information is displayed in the Pipe pane.

10. Click the **Add/Change Pump** button, .

11. The number of pumps currently on the selected pipe is displayed on the **Add/Change Pump** button. Clicking the **Add/Change Pump** button opens the **Pump Data** screen.

12. The value for the **Elevation** field will be set automatically to elevation of the node at the start of the pipe. If the node's elevation is updated later on then the pump's elevation will automatically be updated to match this value before the system is 'Calculated'..

13. Click the pump **Icon** button to popup a list of pump images. Select the pump symbol you want displayed on the pipe system drawing and choose its size and color.

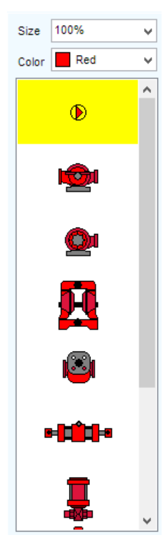


Figure 71 Pump Icons

14. The image you select does not affect any of the pump's properties or values.
15. Use the **Pump Data** screen to choose the Pump Type to be modelled and to define the properties of the pump.

Adding a Fixed Flow Rate Pump

This pump type is used to specify the pump flow rate for a positive displacement pump, such as a piston pump or a peristaltic pump. The system will be solved and the required head from the pump will be calculated.

If the performance curve for a pump is not available, this pump-type can also be used to establish the flow / head requirements for a centrifugal pump. Setting the pump to a particular flow rate allows the system to be solved and the required pump head is then calculated. A user can then select a centrifugal pump that meets the flow and head requirement, after which chosen pump's flow / head performance data can be entered, and the system can be re-solved to find the exact operating point on the pump curve.

See the note about using fixed flow rate pumps after this section.

To model a fixed flow rate pump condition:

1. Add a pump to a pipe (as described in the previous Adding a Pump section).
2. The pump is positioned at the start of the pipe, immediately after any fittings that are included at the start of the pipe. The **Elevation** field is automatically populated and set to be the same as the elevation of the node at the start of the pipe. Click the Icon button to select the pump **symbol** you want displayed on the pipe system drawing.
3. Type the pump's name in the **Name** field.
4. Click the Set Flow Rate radio button.
5. Select the flow rate units and enter the flow rate required.
6. Click the **Save** button to save the pump flow rate data to the pipe.

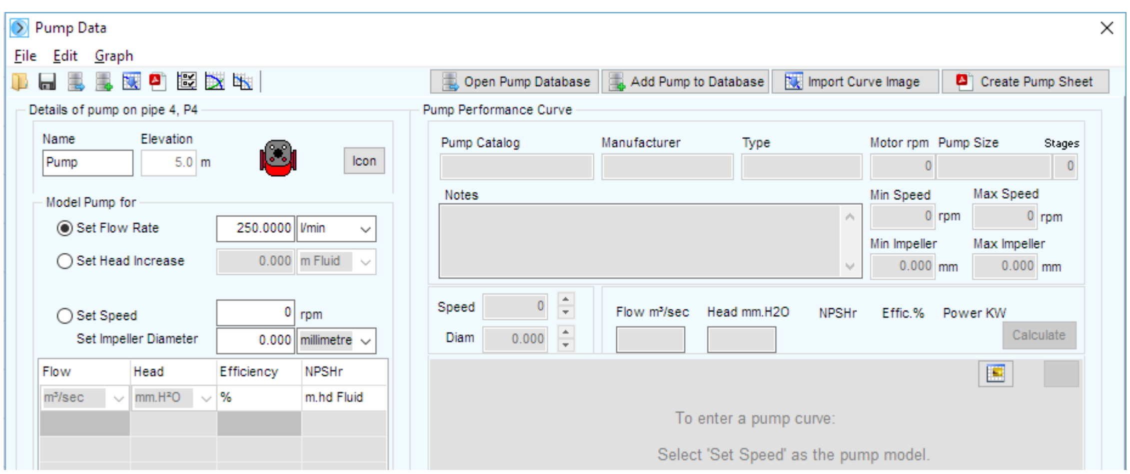


Figure 72 Fixed Flow Rate Pump

The system can now be solved, using the fixed flow rate from the pump.

Note about Fixed Flow Rate Pumps

When the flow rate of the pump is fixed, it is possible, with the addition of multiple fixed flow rate pumps to produce a system where the in-flow and out-flow of a particular node are not the same and this would result in an invalid system where a flow balance could never be achieved.

The fixed flow rate pump defines the flow rate and this cannot vary to give a flow / pressure balance within the system, so some other part of the system must be allowed to vary. If flow control valves are also used, at least one flow path must have no control device fitted. If all flow paths are controlled, it will not be possible to solve the system, and in this case the system is termed over-controlled.

If a pump with a fixed flow rate is used on a ‘main line’ that branches to other paths with flow control valves then at least one path on the branch lines must have no flow control valve. The flow rate along this path will be set to the difference between the total flow in all other paths and the flow rate produced by the fixed flow rate pump.

Flow control valves need to have a positive pressure at their inlet that is sufficient to meet the system pressure losses after the flow control valve (through pipe friction, fittings, components, changes in elevation) plus the pressure loss that the flow control valve itself needs to introduce to achieve a balance within the pipe system. If the required pressure cannot be achieved, it will not be possible to solve the system.

Adding a Fixed Head Increase Pump

If the performance curve for a pump is not available, the analysis of the pipeline system can be carried out using a fixed value for the pump head / pressure rise to be added by the pump.

This pump type is used to specify the additional increase in head (pressure) that a pump will add in to the system. When the system is solved, the flow produced by the pump will be calculated.

When difficulties are experienced in solving a system, perhaps because a chosen pump is not capable of producing the required flow, it can be useful to solve the system using a fixed head pump that adds a high pressure. The fixed head added by the pump can then be progressively reduced, and the system re-solved, to identify the minimum pressure needed to produce the required flow through the system

Using a fixed head pump as described above can often help identify bottlenecks in the system, where perhaps design changes can be made to achieve the required flow rate with a lower pump head

To model a fixed head / pressure rise pump condition:

1. Add a pump to a pipe (as described in the previous Adding a Pump section).
2. The pump is positioned at the start of the pipe, immediately after any fittings that are included at the start of the pipe. The **Elevation** field is automatically populated and set to be the same as the elevation of the node at the start of the pipe. Click the Icon button to select the pump **symbol** you want displayed on the pipe system drawing.
3. Type the pump’s name in the **Name** field.
4. Click the Set Head Increase radio button.

5. Select the head / pressure units and enter the head / pressure to be added.

6. Click the **Save** button to save the pump head / pressure data to the pipe.

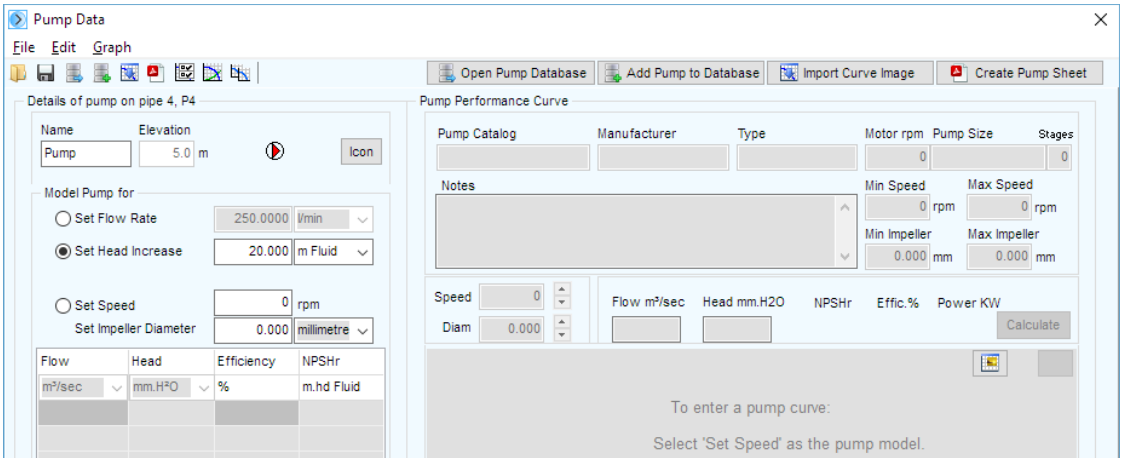


Figure 73 Fixed Head / Pressure Pump

The system can now be solved, using the fixed head / pressure added by the pump.

Adding a Fixed Speed Pump Curve

To add a fixed speed pump to a pipe:

1. Add a pump to a pipe (as described in the previous Adding a Pump section).

2. The pump is positioned at the start of the pipe, immediately after any fittings that are included at the start of the pipe. The **Elevation** field is automatically populated and set to be the same as the elevation of the node at the start of the pipe. Click the Icon button to select the pump **symbol** you want displayed on the pipe system drawing.

3. Type the pump's name in the **Name** field.

4. Click the Set Speed radio button.

5. Type the pump's number of rotations per minute (rpm) in the **Speed** field.

6. Choose the diameter units and enter the pump impeller diameter (if applicable).

7. Select the applicable units for the pump curve from the **Flow** and **Head** drop down lists.

8. Determine the points from the manufacturer's pump graph that you wish to include on the Pump Curve graph, and enter the head, efficiency and NPSHr values for each of these points in the **Pump Curve** section of the **Pump Data** performance table.

9. Add a minimum of 4 and maximum of 10 points to the **Pump Curve** section.

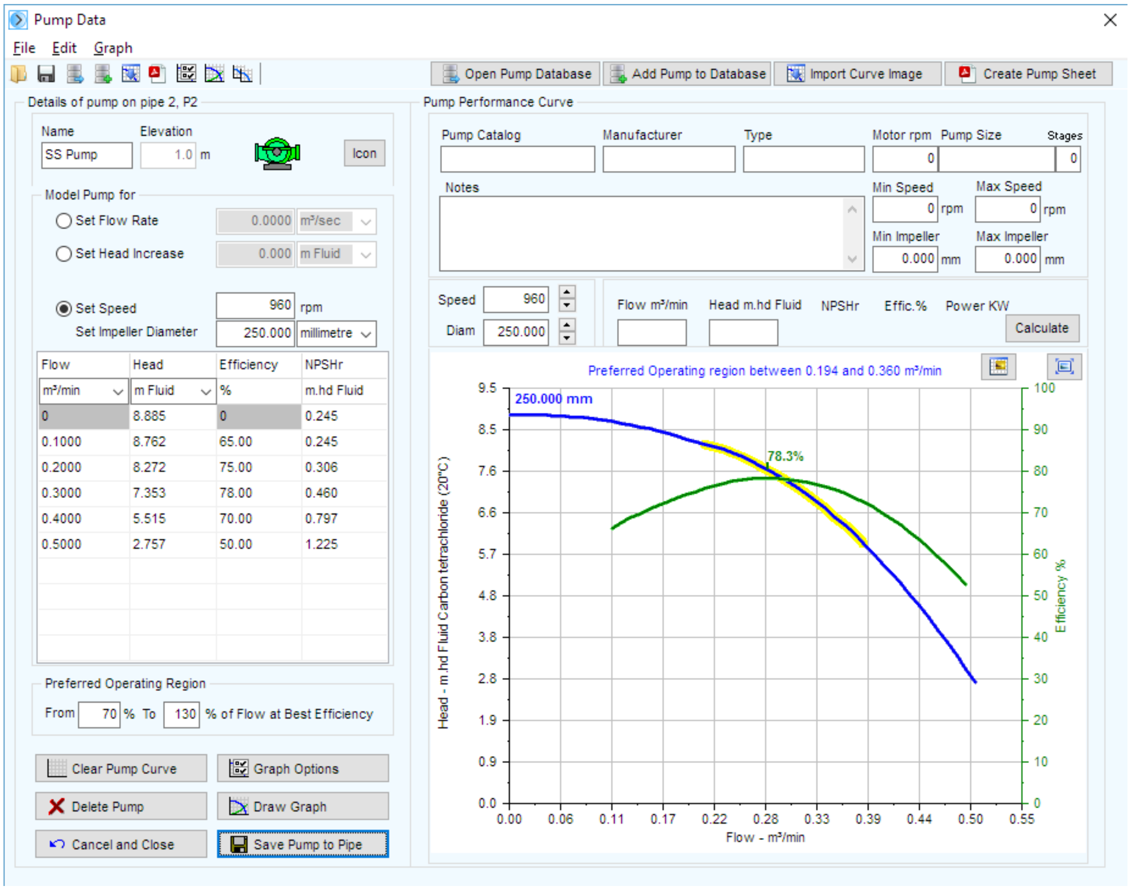


Figure 74 Pump Data screen

10. Type Define the Preferred Operating Region around the pump's best efficiency point by specifying the preferred flow rate range between a **From** percentage and a **To** percentage of the flow at the best efficiency point.

11. Once the operating points have been entered, click the 'Draw Graph' button to produce an operating curve for the pump.

12. Click the **Save Pump to Pipe** to add the pump data to the Pipe Flow Expert system.

13. The system can now be solved and the balanced operating point of the centrifugal pump will be determined.

14. The pump will produce a flow rate and pump head to match the frictional resistance throughout the system. The duty point will be identified on the performance curve for the operating speed of the pump

15. More complex pump graphs can be produced which include speed & diameter adjustments, together with iso lines that display an efficiency map for the pump.

16. The pump data that has been entered can be saved to a **Standalone Pump File** with a pfpm extension by clicking the **Save** button on the toolbar on the pump screen or by selecting the Save Standalone Pump File from the pump screen file menu options.

17. By saving the data to a Standalone Pump File (with a .pfpm extension), you can reuse the pump data for other pumps in the pipe system or future pipe systems.

18. To add a pump's data using an existing .pfpm file, click the **Load Pump from File** button or select the **Load Standalone Pump File** from the pump screen menu options. Select the applicable .pfpm file, click **Open**, edit the applicable fields, and then click the **Save Pump to Pipe** button.

When a pump is added to a pipe, the **Include** check box is displayed and set as selected, next to the **Add/Change Pump** button, .

When the **Include** check box is selected, Pipe Flow Expert includes the data from the pumps on the pipe in the calculations for solving the pipe system. To remove the pump data from the calculations, clear the **Include** check box.

Working with Pump Curve Graph

Once the data for a centrifugal pump has been added, you can view an interactive graphical representation of the data in the graph section of the Pump Data screen.

To view the Pump Curve Graph:

1. Click on a pump in a Pipe Flow Expert system to open the **Pump Data** screen. Click the **Draw Graph** button to generate and display the Pump Curve graph.

2. Click the **Graph Options** button to open the graph options screen. Select the check boxes for the elements you wish to be displayed on the graph.

3. Clear the check boxes for the elements that you do not want displayed on the graph.

4. When you select a check box, the element appears on the graph. When you clear a check box, the element is removed from the graph.

5. To view the pump head, NPSHr, efficiency percentage and power for a specific flow rate, type the flow rate in the **Flow** field above the graph drawing.

6. Click the **Calculate** button.

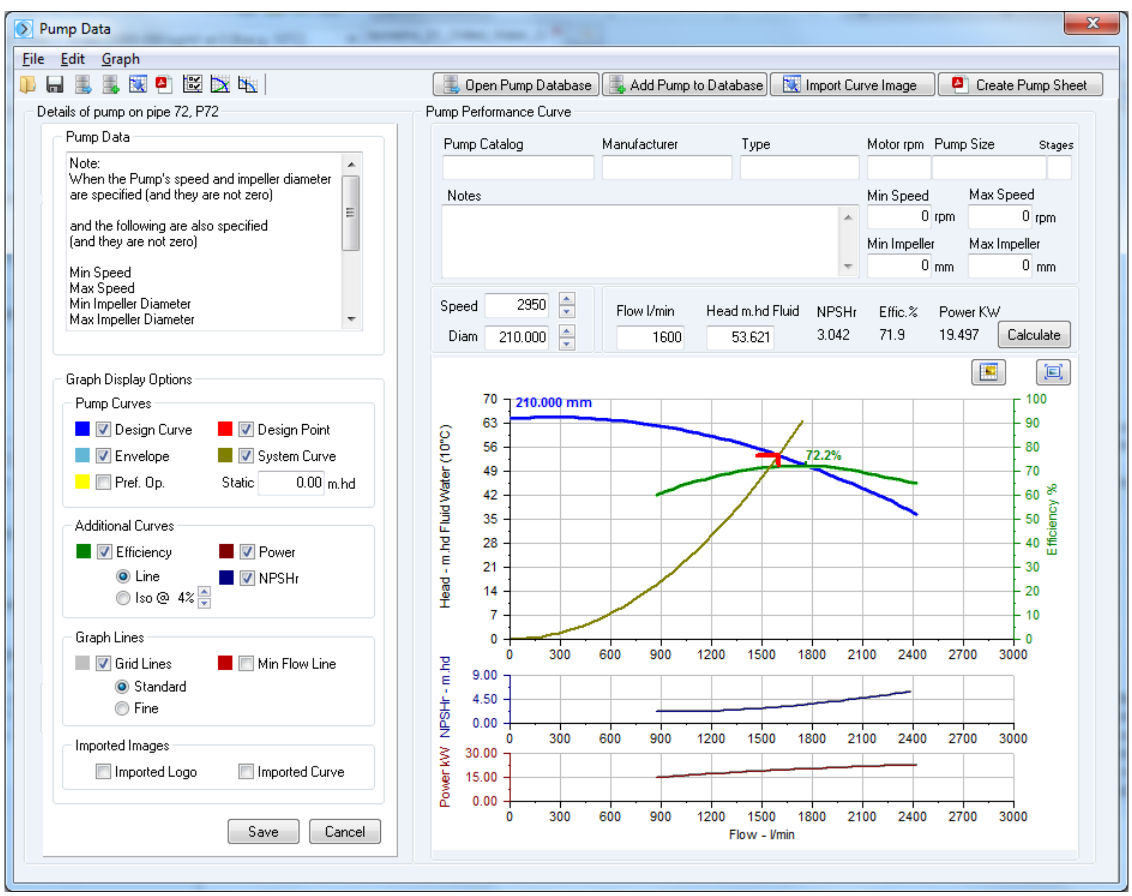


Figure 75 Pump Graph with NPSHr and Power

7. The calculated pump head, NPSHr, efficiency percentage, and power for the entered flow rate are displayed, and the point is mapped on the Pump Curve graph. The design point is represented by a red marker on the graph.

8. To view the flow rate, NPSHr, efficiency percentage and power for a specific head, type the pump head in the **Head** field above the graph drawing.

9. Click the **Calculate** button.

10. The calculated flow rate, NPSHr, efficiency percentage, and power for the entered pump head are displayed, and the point is mapped on the Pump Curve graph. The design point is represented by a red marker on the graph.

11. A logo may be added to the pump graph. Click the **Add Logo** button on the pump graph area to load a logo image. Click the **Expand** button to view a larger graph.

12. Any relevant notes may be added to the **Notes** field.

13. The Pump Data screen menu provides options to:

    - Save the pump data to a standalone pump file.
    - Add the pump to the pump database.
    - Open the graph options to allow selection of the items to be included on the graph.
    - Copy the graph image.
    - Create a PDF pump sheet with a high resolution performance graph.

14. Further functionality allows the import of a scanned image of a manufacturer’s pump curve, plus modelling of performance predictions for changes in pump operating speed and variations in impeller diameters. The following sections describe the use of these further options.

Import Pump Curve Image

Pipe Flow Expert provides functionally to import and create a mathematical pump model which allows adjustment of the pump performance for different operating speeds & impeller diameters.

To import a pump curve image and plot Flow v Head points:

Click on a pump in a Pipe Flow Expert system to open the **Pump Data** screen

Click the 'Import Curve Image' button to open the import screen.

Follow these steps to create a mathematical model of the pump performance.

Step 1:

Import an image of a pump manufacturer's performance graph

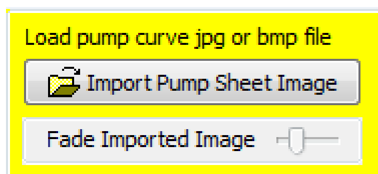


Figure 76 Import Curve Image Step 1

Step 2:

Set the range of the X axis (flow) & the flow units.
Now click the start & end points of the X axis on the imported image.

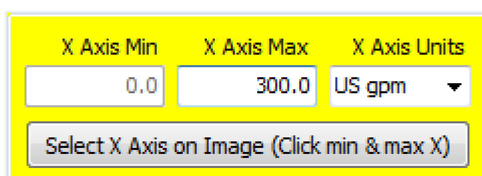


Figure 77 Import Curve Image Step 2

Step 3:

Set the range of the Y axis (head) & the head units.
Now click the start & end points of the Y axis on the imported image.

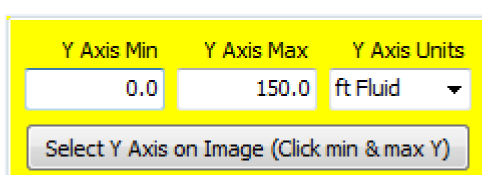


Figure 78 Import Curve Image Step 3

Step 4:

Click on the ‘Select Points on Curve’ button.
Click the shut off head point at zero flow for the chosen curve.
A red cross shows the point that has been selected.

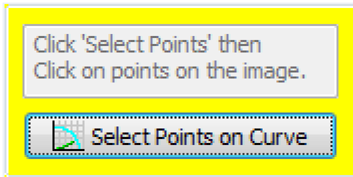


Figure 79 Import Curve Image Step 4

Step 5:

Click additional points on the chosen pump curve
(in order to define it across the flow range of the pump).
Enter the Efficiency & NPSHr associated with each operating point.
Repeat for up to 10 points to build the pump model.

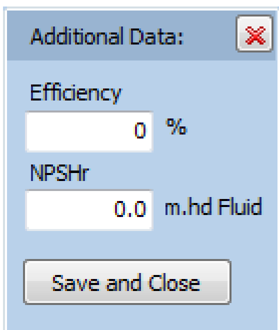


Figure 80 Import Curve Image Step 5

Step 6:

This input table can be moved around the screen in order to allow viewing of any parts of the pump curve which may be obscured. Click on the header bar and drag it to a new position as required.

The table shows Flow, Head, Efficiency and NPSHr entries.

Select from the heading drop down to switch between display of Head, Efficiency and NPSHr values. Entries can be updated and deleted as required.

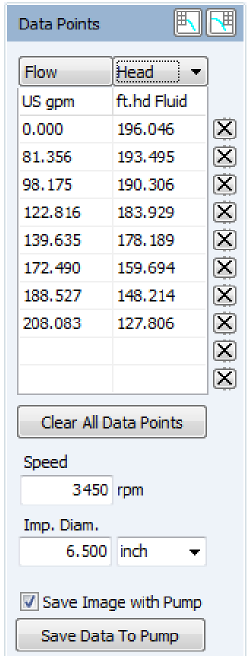


Figure 81 Import Curve Image Step 6

Step 7:

Enter the pump speed, impeller diameter and specify the impeller units.

Click 'Save Data to Pump' to return to the main Pump Screen.

The imported data produces a pump graph with a flow versus head performance curve. This simple pump object can be 'Saved' on to a pipe within a system.

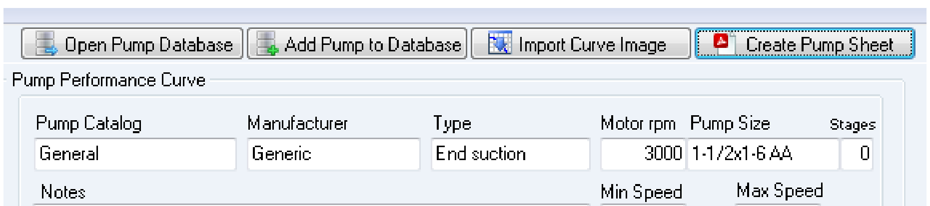


Figure 82 Pump Data for Saving

The Pump Catalog, Manufacturer, Pump Type, Motor rpm, Pump Size, Stages and Notes fields should be filled in before saving the pump to a system or saving to the pump database.

This supplementary information will be included on the Pump Data Sheet PDF and on the system's Results PDF Report when these documents are produced.

The supplementary information will also be available in the future when pump data is loaded from the pump database.

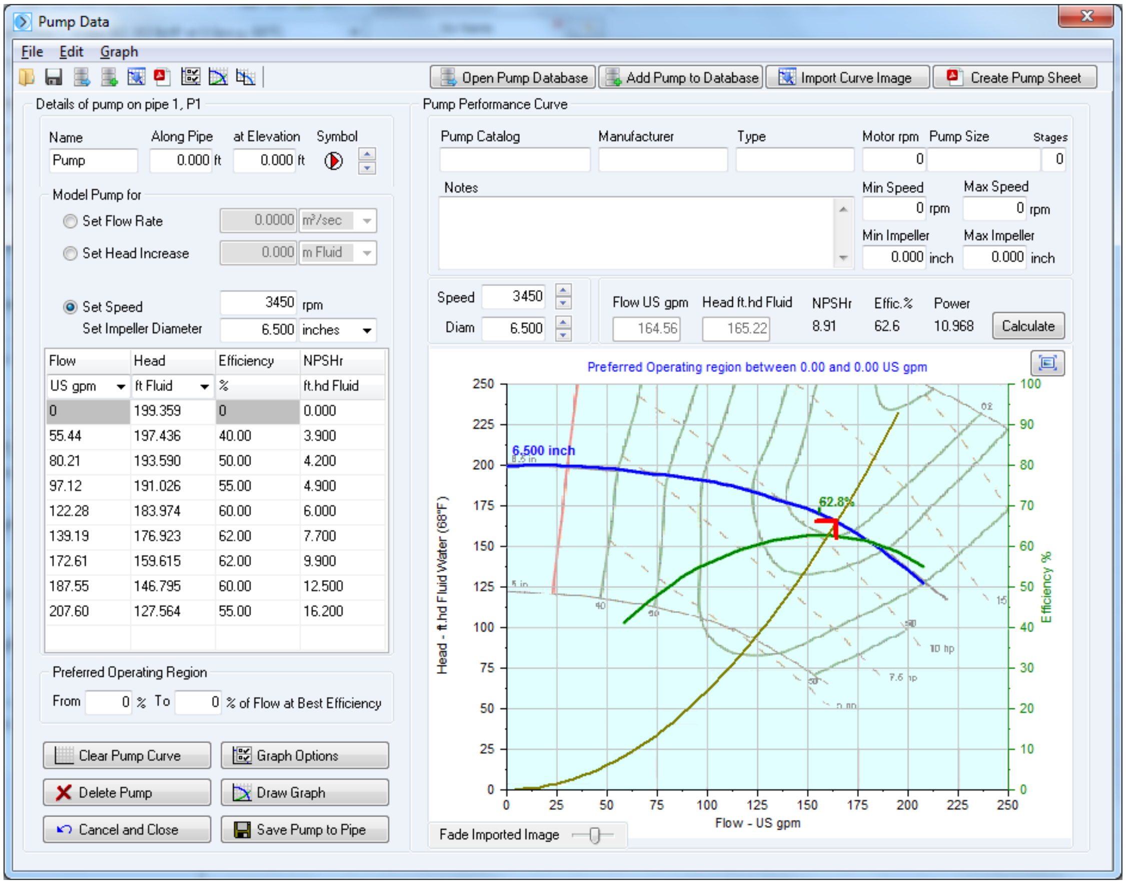


Figure 83 Simple Pump Graph from imported points

When a system is solved the pump's operating point will be calculated and the Flow Rate, Head, Efficiency and NPSHr values will be displayed together with shaft power.

A system curve can also be plotted through the operating point.

Note: If the static head in the system is not zero then the user can enter a static head value and the system curve will automatically redraw taking account of this. The software cannot automatically calculate a static head because if a network has many loops and paths then the static head would depend on which section and run of pipe was being considered. To change the static head, click on the Graph Options button and then enter the Static head under the checkbox for the system curve.

Further options are available for the modelling of predicted pump performance when changes in pump operating speed and variations to pump impeller diameters are applied. The following sections describe the use of these further options.

Variable Speed and Change to Impeller Size

The pump model may be enhanced to predict the pump performance for a range different operating speeds & different impeller diameters.

To define the limits for the pump performance, enter the minimum & maximum operating speed data, and the minimum & maximum size of the impeller diameter.

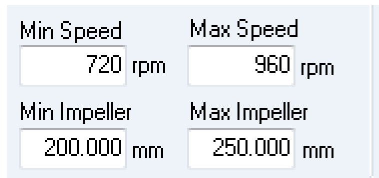
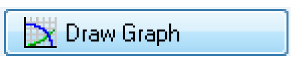


Figure 84 Pump Speed and Impeller Data

Click the ‘Draw Graph’ button to re-draw and show the new operating range.



The pump graph will be re-drawn by using standard affinity laws to predict the flow & head performance for the maximum & minimum impeller diameters.

The ‘Graph Options’ button provides access to additional settings that configure the color and display of items on the graph, and it also provides access to allow bespoke configuration of the predicted pump performance data, so that it can be matched to a pump’s real-world performance as detailed on a pump manufacturers graph and data sheet.

The pump graph can be set to display the Design Curve, Design Point, Operating Envelope, System Curve, Preferred Operating Region, and Efficiency Curve.

Shaft Power and NPSHr graphs can be displayed below the main performance graph

Efficiency data can also be displayed on the graph, either as a single line for a specific pump curve, or as a series of iso efficiency lines that show efficiency performance across a range of impeller sizes

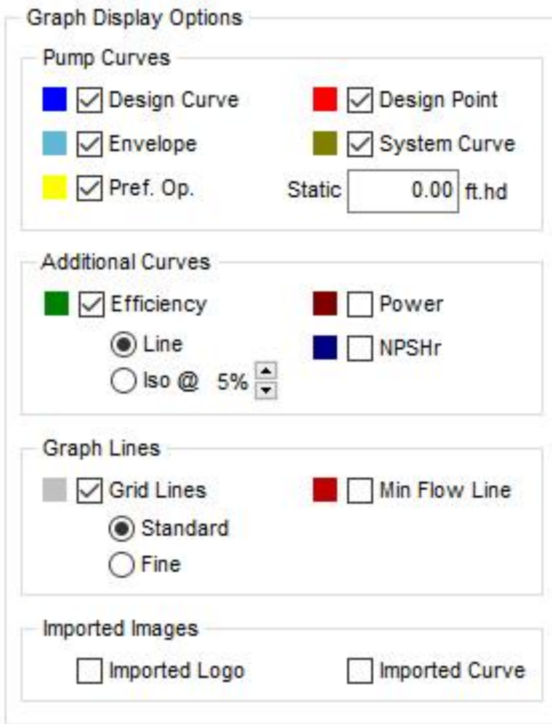


Figure 85 Pump Graph Options

If a logo image has been imported, this can be included on or excluded from the pump graph by selecting the 'Imported Logo' checkbox in Graph Options.

If a pump curve image has been imported, this can be included on or excluded from the background of the pump graph by selecting the 'Imported Curve' checkbox in Graph Options. On the Pump Graph screen itself, the imported pump curve image can also be faded in and out as required.

Predicted Performance Data

Initially the pump's predicted performance is based on the standard affinity laws.

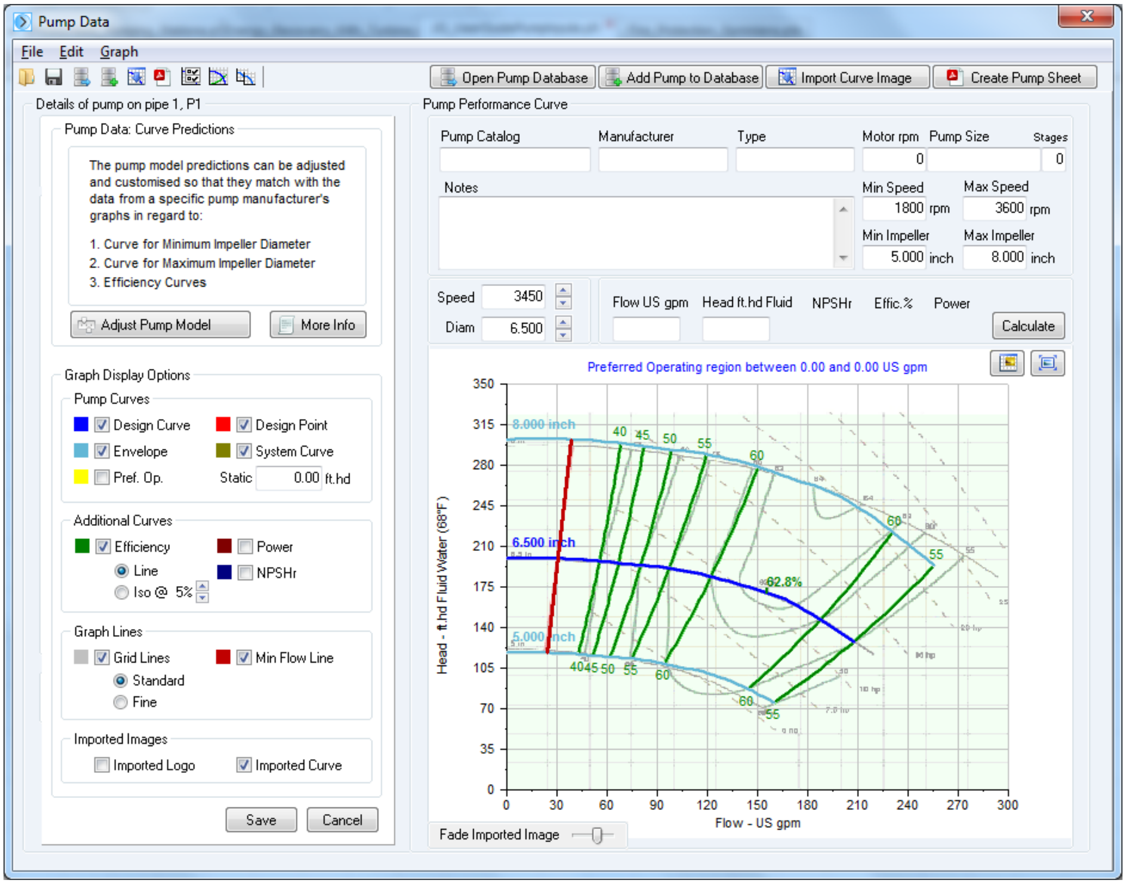


Figure 86 Pump Initial Affinity Predictions

In many cases the predicted performance curves will agree closely with a pump manufacturer's published graph, however for more accurate modelling of pump performance, it may be necessary to adjust the predicted shut-off head, and adjust the curvature & efficiency points for the predicted curves at the maximum and minimum impeller diameters.

The 'Adjust Pump Model' button provides access to functionality that allows customization of the predicted pump performance curves.

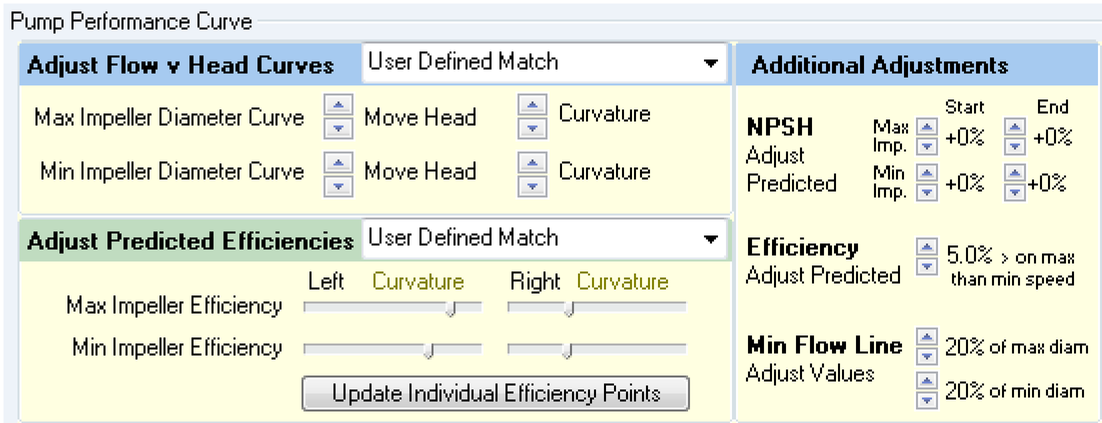


Figure 87 Pump Performance Curve

The 'Adjust Flow v Head' options can be used to adjust the shut-off head & the rate of curvature for the minimum & maximum impeller diameter curves, to match them to the published performance data.

The 'Adjust Predicted Efficiencies' options allow the predicted efficiency values to be modified to match the published performance data.

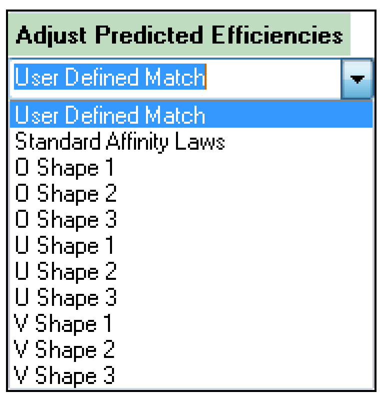


Figure 88 Adjust Predicted Efficiencies

A number of iso efficiency 'shapes' are pre-defined in the dropdown options.

These choices provide an easy method of selecting an initial starting position for the iso efficiency map. Once a further adjustment has been made to the data, the iso efficiency shape selection type will become user defined match.

The slide adjustments allow the curvature of groups of iso efficiency lines to be moved as a block to the left or right. The BEP (best efficiency point) on the base data performance curve is used to divide the performance graph into different areas.

These adjustments should be used to match the efficiency points on the maximum and minimum impeller performance curves.

Final adjustment of individual efficiency points on the maximum and minimum performance curves can be made (if necessary) by selecting the 'Update Individual Efficiency Points' option.

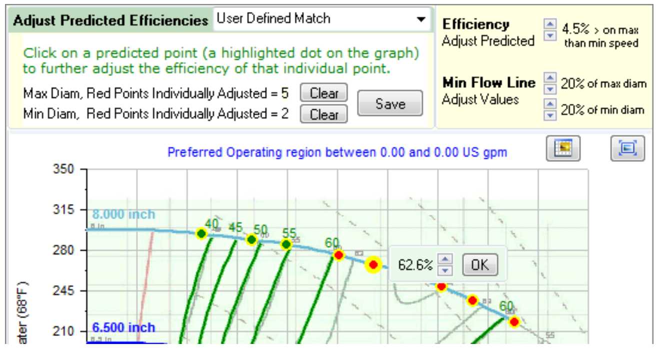


Figure 89 Adjust Individual Efficiency Points

To adjust an individual efficiency point, click on a ‘Green’ point on the performance curves and use the up/down adjustment to set the efficiency value for the point.

The point will be colored ‘Red’ to indicate an adjustment has been made.

These adjustments may be reset by clicking the ‘Clear’ button

Additional adjustments for NPSHr and efficiency predictions at different operating speeds can also be made in order that the performance of the pump across a range of speeds and impeller diameters is accurately mapped.

The efficiency of a centrifugal pump generally reduces as the operating speed is reduced. It can be seen from many pump manufacturers performance graphs that a speed reduction of 50% will reduce the pump efficiency by around 5%.

When a pump is required to operate at low flow rates and/or in low efficiency areas a centrifugal pump may not perform as expected. Operation in these areas may not be recommended.
The minimum flow line can be set as a percentage of the maximum flow on a particular curve.

Pipe Flow Expert will not include pumps which operate below the minimum flow line setting when searching the pump database for suitable pumps to match a particular flow rate.

Once a pump has been modelled to match the published data, the complete pump model can be saved to the searchable Pump Database for future use. A pump data sheet in PDF format can also be produced. This shows detailed information about the pump and provides a scalable high resolution image of the pump performance graph.

Figure 90 Pump Options

The adjusted pump graph can be used in a Pipe Flow Expert system to model the complete performance of the pump over a range of speeds and impeller sizes.

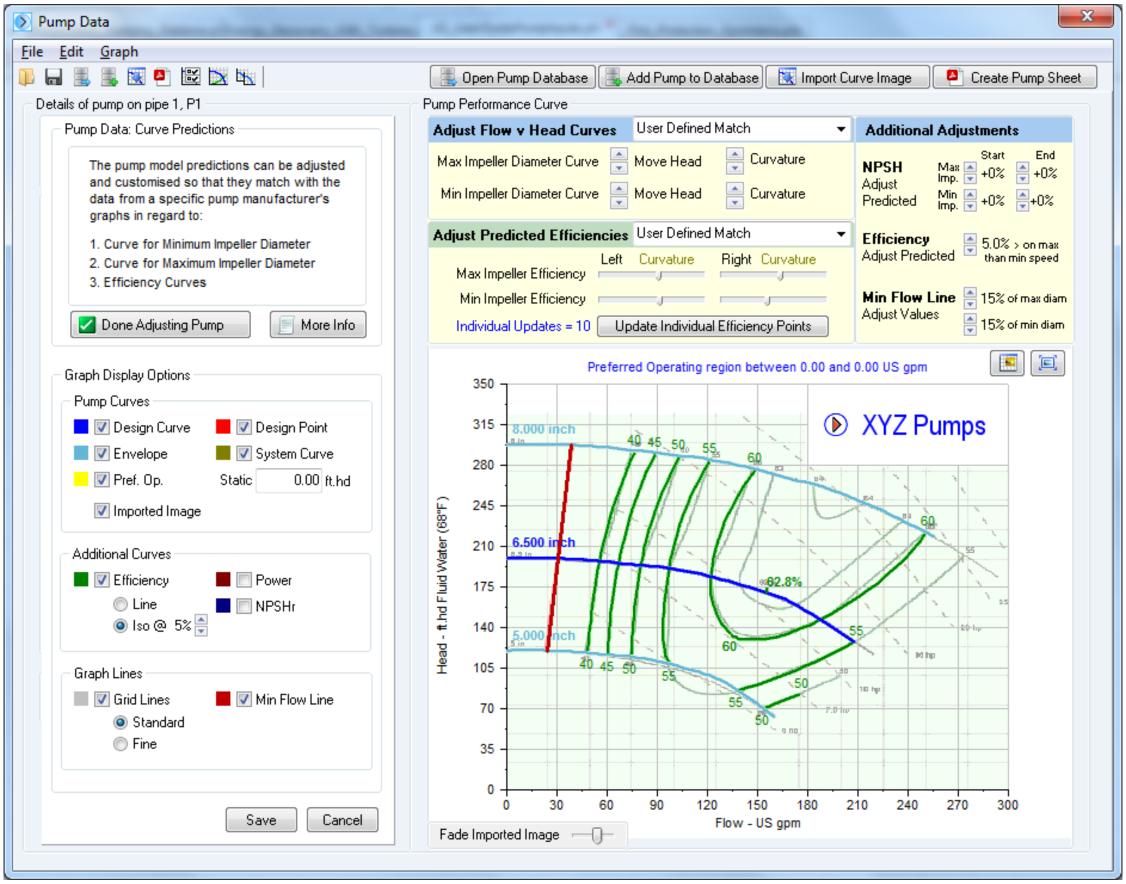


Figure 91 Adjusted Pump Model

A logo image can also be added to the pump graph to assist with identification of the pump.

Search Pump Database

The Pump Database can be used to search for pumps that match a set of specified criteria, which include flow & head requirements within a given speed range. There are further search refinements that filter the results to look for solutions with the best efficiencies, largest impellers or fastest operating speeds.

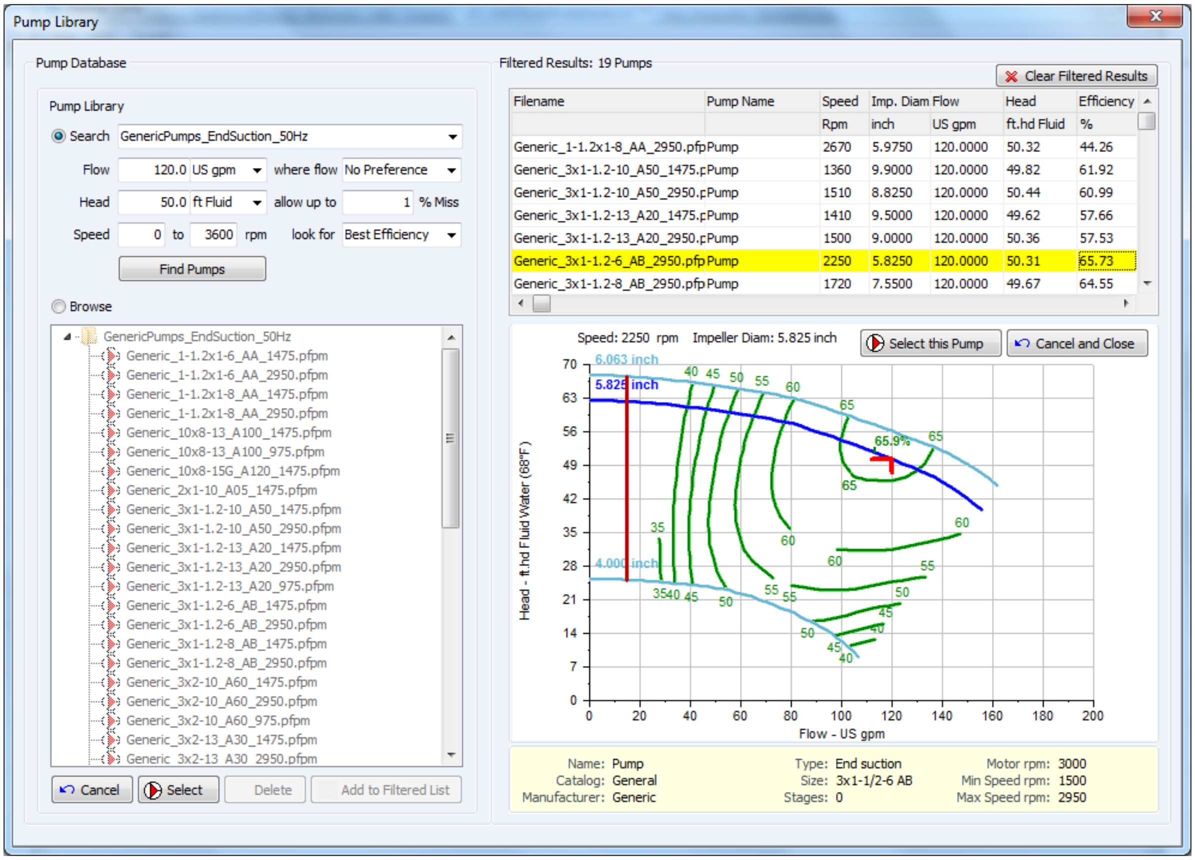
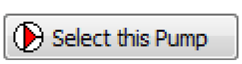


Figure 92 Pump Database

The user can work through the pump search results by clicking on a row in the Filtered Results Grid or by using the up and down cursor keys to scroll through rows in the grid. As a row on information about a pump is highlighted, the corresponding pump performance graph is shown

Click the ‘Select this Pump’ button to transfer the chosen pump details to the pump data screen.



Browse Pump Database

The Pump Database can also be switched in to Browse mode, which allows a user to select a Pump Library from a dropdown list of folders. This is automatically populated with a list of available pump folders from within the Pipe Flow Expert installation.

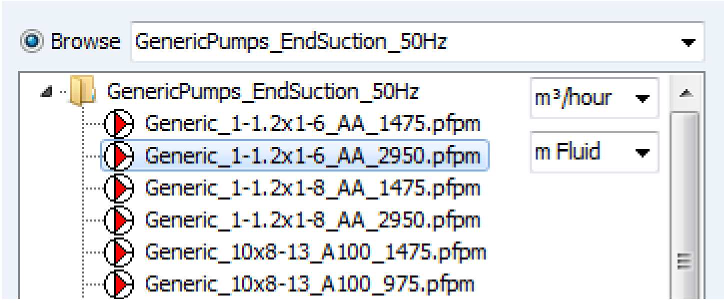


Figure 93 Pump Database Browser

Once a folder is selected, the pumps within the chosen pump library folder are displayed in the Browse List, where the user can click on a pump to show its corresponding pump performance graph.

The units used to display the flow rate and pump head on the performance graph should be selected from the dropdown options in the Browse window.

The pump data may have originally been created using flow rate units and head units that are different from those now required. Selection of new flow and head units will produce a new performance graph in the chosen units, with the software automatically converting the original data to the new flow and head units.

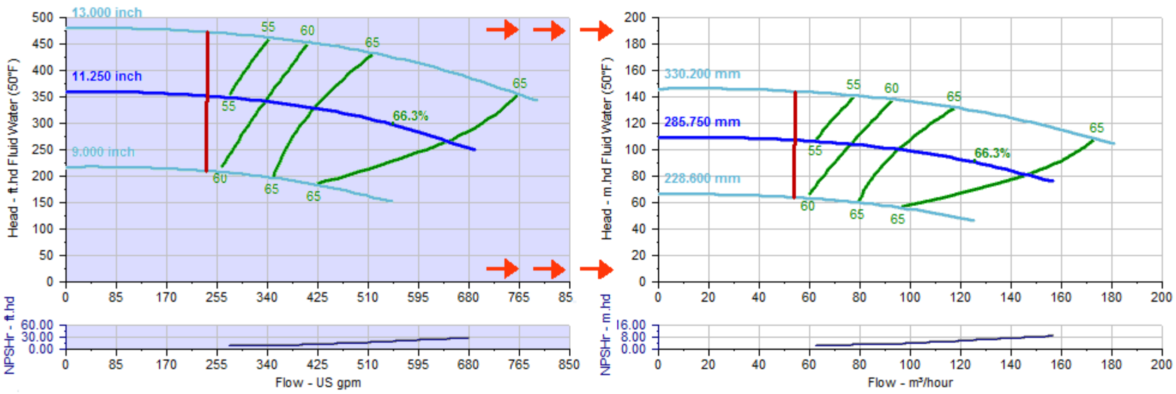


Figure 94 Changing Units of Pump Performance Graph

Both the performance curve and the NPSHr curve can be shown in the pump graph window.

As the user browses a Pump Library folder, pumps can 'saved' for further consideration by clicking the button to add them to the filtered list.

Once a user has found a suitable pump, it can be selected for use in the pipe system by clicking the 'Select' button, or by double clicking on the chosen pump.

Example Pumps (with Flow versus Head curve)

Two sets of example pump files are included in the sub-folders 'GenericPumps_EndSuction_50Hz' and 'GenericPumps_EndSuction_60Hz'. These sub-folders are located in the 'Pumps' folder that is created as part of a Pipe Flow Expert installation.

When a user opens the Pump Database all sub-folders within the main 'Pumps' folder will be displayed within the Library dropdowns of the Search and Browse sections. An appropriate folder should be chosen on which to perform the Search or Browse.

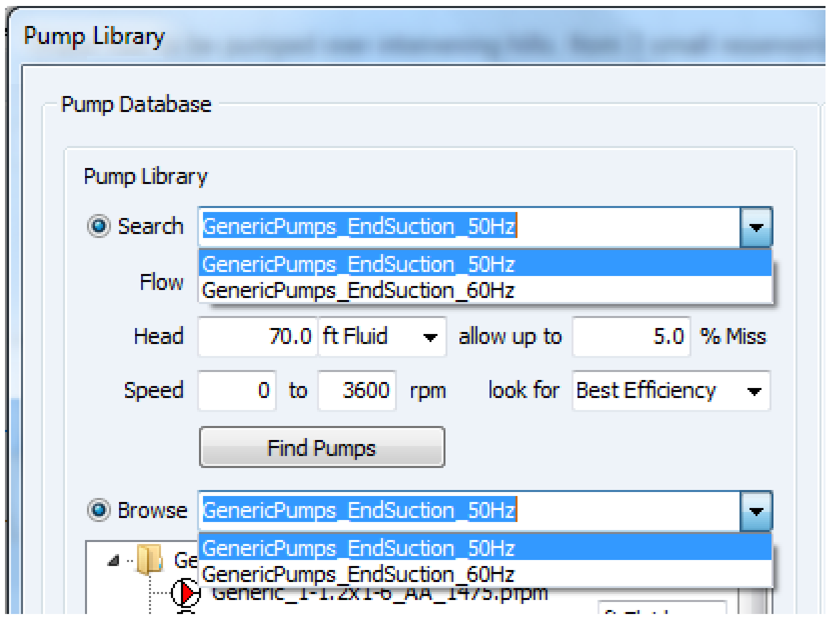


Figure 95 Pump Database - Example Pumps

The example pumps can be loaded and used to see the results that are produced by using different pump operating data. This data is provided solely as example data and is not related to any specific pump manufacturer. When adding further pump data to the pump database, we recommend that a new folder is created for each pump manufacturer. The pumps can then be organized and saved in named folders.

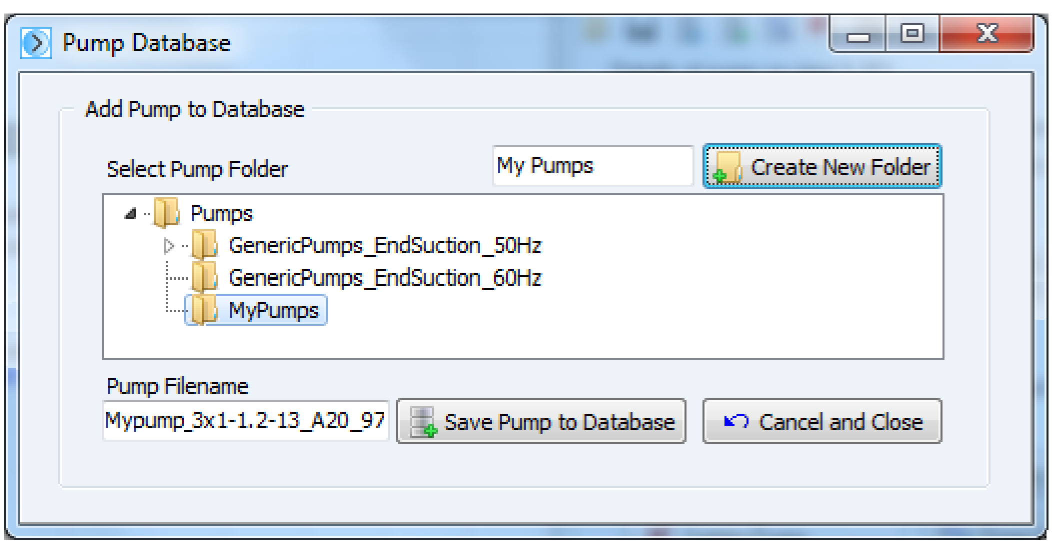


Figure 96 Pump Database - New Folders

Demand Flows

A demand in-flow or demand out-flow can be set at any join point in the system, or at the end of a single pipe if flow is entering or leaving the system at that point. A demand can also be the pressure required at an end node, however a node cannot have both a demand flow and a demand pressure.

A demand **in-flow** is the volume of fluid that enters the system at a join point (from an external source).

A demand **out-flow** is the volume of fluid that leaves the system at a join point (discharge or flows in to some external system).

Demand flows may also be used to specify the required flow rate at the end of a pipe where fluid is flowing out of the system, or to specify the flow rate at the start of a pipe where fluid is entering the system (from some external source). A pipe can have a demand flow or a demand pressure at the end of a pipe but it cannot have both.

Demand flows can be added to a join point from the Node pane or the Drawing pane. Demand flow values are set on the Set Flow Demands dialog. The demand flows are then shown at the appropriate node on the Drawing pane.

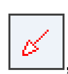
NOTE: Be sure to enter the value for the required fluid flow rate in the units displayed next to the field. For example, if you are using imperial units, and ft^3/sec for cubic feet per second is displayed next to the In Flow and Out Flow fields, enter the value in cubic feet per second.

Compressible Gas Flow:

If you are working with a compressible gas and you want to set the demand flows in volume of gas at standard conditions such as SCFM, SCMH, or other standard volume units, then you must ensure that the current fluid is a gas. If the current fluid data is a liquid then the units for gas flow at standard volume do not appear in the drop down list of units for demand flows.

Adding a Demand Flow at a join point

To add a demand flow to a join point in the pipe system:

1. Click the **Add Demand (Flow)** button, , on the tool bar.
2. When you click the **Add Demand (Flow)** button, the demand flow symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click on the join point on the Drawing pane where you want to add a demand flow.
4. The **Set Flow Demands** dialog opens.
5. To add a demand flow from the Node pane, select a join point on the Drawing pane to display the join point's properties in the Node pane, and then click one of the **Edit** buttons in the Node pane to open the **Set Flow Demands** dialog. Both **Edit** buttons in the Node pane open the **Set Flow Demands** dialog.

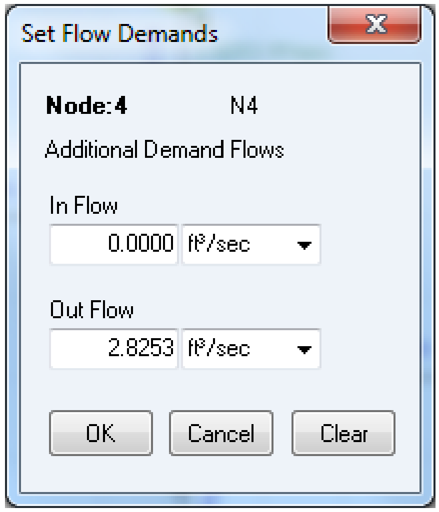


Figure 97 Set Flow Demands dialog

6. Select the flow units from the drop down list of available units.
7. Type the required flow rate for the fluid **ENTERING** the system at the join point in the **In Flow** field, if applicable.
8. Type the required flow rate for the fluid **LEAVING** the system from the join point in the **Out Flow** field, if applicable.
9. If you select a different flow unit, the value entry will be converted automatically to the equivalent value for the current selected flow unit.
10. Click **OK** to add the join point's demand flow.

Demand Pressures

A demand pressure can only be set at exit points from the system, i.e. the boundaries or end nodes.

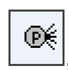
A demand pressure, referred to as an End Pressure in the Node pane, is the required pressure demand at the end of a pipe when the fluid is discharged. For example, in a sprinkler system, each sprinkler head may have a required minimum pressure. To find the fluid flow rate exiting the pipe system, a demand pressure is drawn and the required pressure set for each sprinkler head. Demand pressures can only be set at the end of a pipe that does not join into any other pipes.

A pipe can have a demand flow or a demand pressure at the end of a pipe but it cannot have both.

In the real-world it is the difference in pressure between two points that determine the flow rate that will occur and hence this is why you cannot define a flow rate and a pressure condition at the same point. If you define a flow rate then the software will calculate what pressure is required at that point in order to get that flow rate when considering all of the other flows and pressures throughout the system. If you define a demand pressure then the software will calculate what flow rate will occur.

Adding a Demand Pressure at an end node

To add a demand pressure to an end node:

1. Click the **Add Demand (Pressure)** button, , on the tool bar.
2. When you click the **Add Demand (Pressure)** button, the demand pressure symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click on the end of a pipe on the Drawing pane where you want to add a demand pressure.
4. Click **Yes** when asked ‘Do you wish to change the type of node to an end demand pressure?’
5. The demand pressure is added to the pipe system, and is selected on the Drawing pane. Define the demand pressure’s properties in the Node pane.

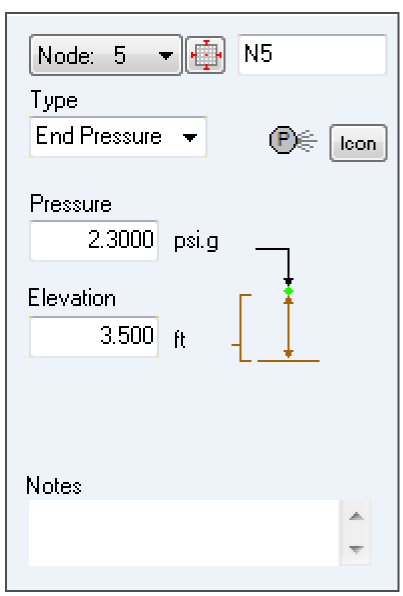
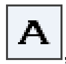


Figure 98 Node pane for end pressures

6. Type the demand pressure’s name in the **Node** field.
7. *End Pressure* is automatically selected in the **Type** list.
8. Click the **Demand Pressure Scroll Up** or **Scroll Down** button to select the demand pressure image you want displayed on the pipe system drawing.
9. The image you select does not affect any of the demand pressure’s properties or values.
10. Type the required pressure for the end of the pipe in the **Pressure** field.
11. Type the elevation at the demand pressure in the **Elevation** field.
12. Type any applicable notes regarding the demand pressure in the **Notes** field.
13. To add additional demand pressures to the system, repeat Steps 3 – 12.

Text Items

Text items allow you to add free text labels at any position on the pipe system drawing. These are useful for annotating a system during and after the design process and provide an easy and clear way to provide additional details about specific items on the drawing.

Using the Add Text button, , on the tool bar, you can add text to the drawing and/or modify text that has been previously added.

Adding Text to the drawing

To add and/or modify text:

1. Click the **Add Text** button, , on the tool bar.
2. A circle will appear at the top left of any text that has been previously added.
3. To modify previously added text click this circle to display the **Add Free Text Label** dialog.
4. To add new text click in a clear space on the drawing pane to display the **Add Free Text Label** dialog.

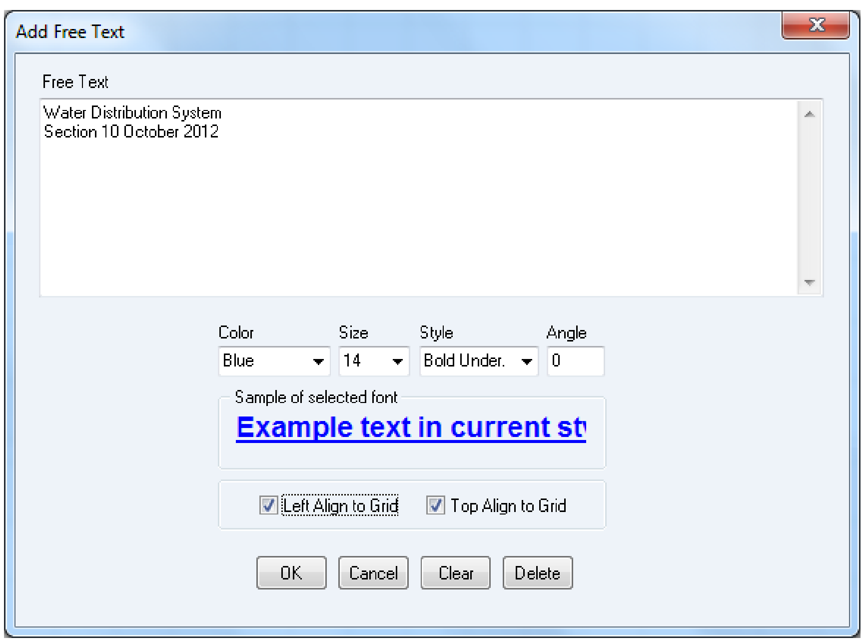
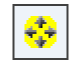


Figure 99 Add Free Text Label Dialog

5. Enter or amend the text, choose the color, size, style and angle for the text.
6. Click OK to place the text on the drawing at the position that was clicked.

7. To align the text to the nearest grid intersection check the Left Align to Grid / Top Align to Grid check boxes before clicking the OK button

8. Click Delete to remove the text from the drawing.

9. To move the individual text to a new position, use the Drag and Move menu button  to highlight all items on the drawing. Select a highlighted item of text, and while holding down the left mouse button, drag the text to its new position on the drawing.

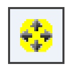
10. The text may also be moved as part of a selected group. Use the selection tool to enclose the items to be moved, and include the top left area of any text to be moved.

Images

Images can be imported on the pipe system drawing. The images can then be resized and repositioned as required. The images appear behind the main drawing of the pipe system. They can be set to appear either on top of or behind the grid lines.

Images are useful for annotating a system to provide visual information that enhances the design data. For example, an imported image could show a map of geographical information behind the system drawing, or it could show a technical drawing that the system design has been traced over. They can also be used for visual impact to add company logos or other imagery.

Use the Import Image button, , on the tool bar, to import an image to the drawing. You can also use the Import Image option from the File menu.

Use the Drag and Move Items button,  , to position, resize and modify images that are already on the drawing.

Importing Images on to the drawing

To add and/or modify images:

1. Click the **Import Image** button, , on the tool bar.
2. A dialog will open that will allow you to browse and select an image file from your computer.
3. Select the image file and click Open. The image can be of type gif, jpg, jpeg, bmp, tif, tiff, ico or vectorized formats emf and wmf. Vectorized formats allow you to zoom in and out whilst maintaining image quality without loss of resolution.
4. The image will now be imported and may automatically be resized as appropriate if it is a very large image.
5. The image is shown on the drawing and a box is drawn around its edges, with particular points highlighted (as shown below).

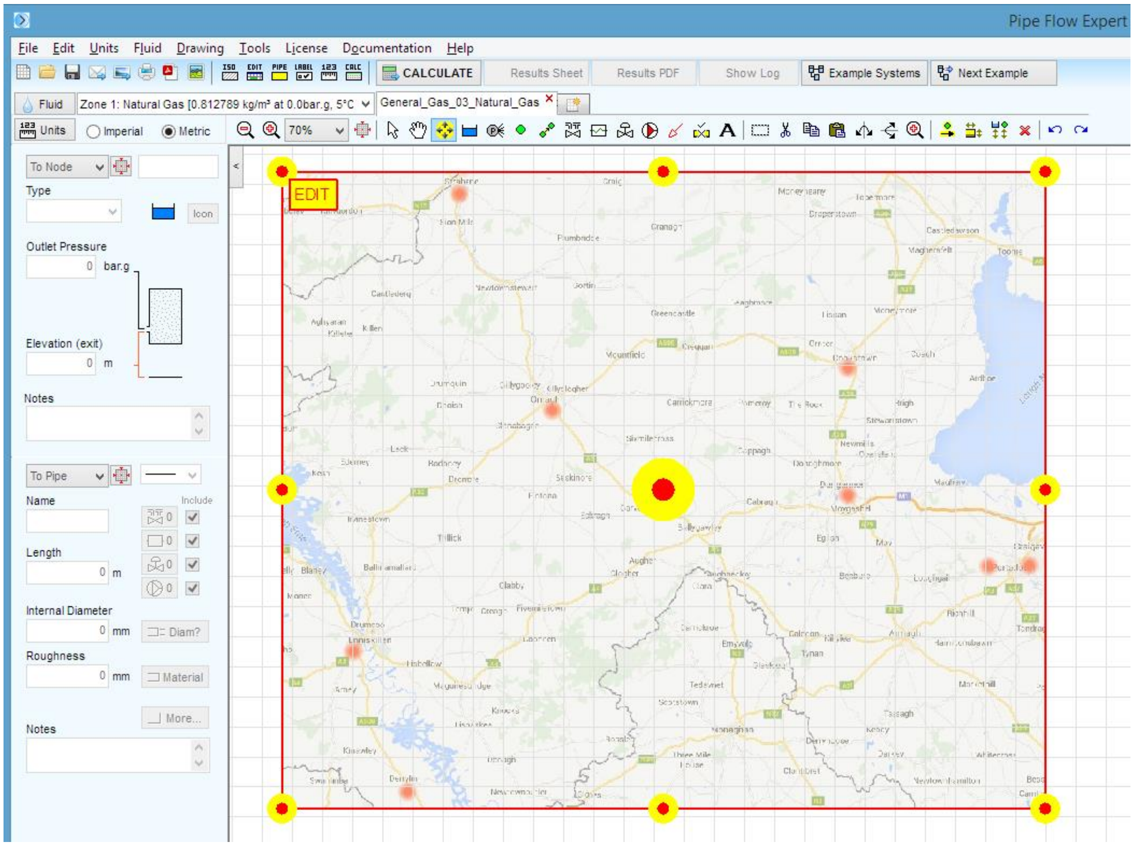
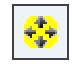


Figure 100 Importing an image on to the drawing

6. Left click on a highlighted point and drag to resize the image (or use the highlighted point in the center of the image to drag and move the position of the image). Right click (or select the Show Item Info tool) to turn off the highlighting and image editing functionality.

7. Click the Edit button, EDIT, at the top left of the image to adjust its properties.

Adjusting image properties

To display the Image Properties dialog, click the Drag and Move Items button, , followed by the Edit button, , at the top left of the image to be modified.

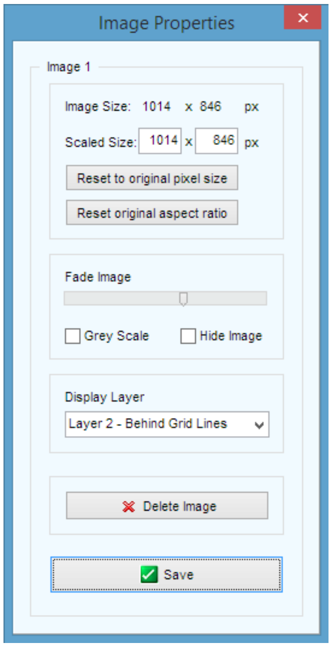


Figure 101 Image Properties

Enter the desired width and height (in pixels) to scale the image. The image will be dynamically scaled.

The original image size and aspect ratio can be restored by clicking the appropriate Reset button.

The image transparency can be adjusted by using the Fade image slider control. Moving the slider to the left will increase the transparency of the image, whilst moving the slider to the right will decrease the transparency.

A colored image can be made black and white (Grey Scale) by selecting the Grey Scale check box.

Select the Hide Image checkbox to conceal the image (whilst keeping the image object on the drawing).

To change how the image appears relative to other system drawing elements, click Display Layer and select the required option as follows:

Display Layer	Description
Layer 1 – Above Grid Lines	The image will be positioned on layer 1, above the grid lines. The grid lines will not be visible regardless of the image transparency.
Layer 2 – Behind Grid Lines	The image will be positioned on layer 2, behind the grid lines. The grid lines will always be visible over the image.
Layer 3 – Behind Layer 2	The image will be positioned on layer 3, behind layer 2. As images are added to the drawing they are layered over the previous image(s). Layer 3 allows the user to change the layering (z index) of the images on the system drawing.

The image can be permanently removed from the system drawing by clicking the Delete Image button.

Viewing, Modifying, and Deleting Items

Pipe Flow Expert comes with a number of tools for viewing and modifying a pipe system from the drawing pane. The units and labelling options selected in the Configuration Options dialog determine which values are displayed on the pipe system in the Drawing pane.

Additional details can be viewed for components by clicking the Show Item Info button, , on the tool bar and selecting a specific component in the Drawing pane.

The following sub-sections describe zooming, viewing, panning, modifying and moving items.

Zooming in or Out in the Drawing Pane

The zoom function in Pipe Flow Expert provides the ability to view a pipe system up close or to see more of the pipe system at a reduced size. The zoom function can also be used as part of the printing process. When you print a pipe system in Pipe Flow Expert (rather than creating a PDF), only the visible part of image that is currently displayed in the Drawing pane is printed. The zoom function helps you define how much of a system is printed. Alternately you can create a PDF document and print from within Acrobat (or other PDF reader).

If your mouse has a scroll button, you can also use the scroll button to zoom in and out in the Drawing pane. Scroll down to zoom out, and scroll up to zoom in.

To zoom in or out in the Drawing pane:

1. Click the, , **Zoom Out** button on the tool bar to view the system farther out in the Drawing pane.

2. Click the, , **Zoom In** button on the tool bar to magnify the system in the Drawing pane.

3. To view a large pipe system in its entirety, you can click the , **Zoom Drawing to Fit Screen** button to see the whole system in the Drawing pane.

4. Click a zoom percentage from the, 100%, **Zoom** list on the tool bar to zoom in or out in the Drawing pane.

Zooming in to a Selected Area

The selected area zoom function provides the ability to quickly and automatically zoom in to a selected area of the pipe system.

To Zoom In to a Selected Area:

1. Click the **Selection tool** button, , on the tool bar.

2. When you click the **Selection tool** button, the selection icon is displayed next to your mouse pointer when the pointer is in the Drawing pane.

3. Draw a box around the node or group of components you want to view in the Drawing pane.

4. To draw a box, click where you want to start the box in the Drawing pane and hold down the left mouse button, then drag the mouse until you have drawn the box.

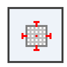
5. Release the left mouse button.

6. Click the selected area ,  , **Zoom Tool Button**, located among the ‘Selection Tool Buttons’.

7. The selected area will be displayed in the center of the drawing pane.

If no selection has been made the selection zoom tool button will increase the display by one scale increment.

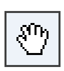
Viewing the whole System in the Drawing Pane

To view the whole of a pipework system click the,  , **Zoom Drawing to fit Screen** button, on the tool bar, to fit the system in the visible Drawing pane.

Panning a System in the Drawing Pane

The pan function in Pipe Flow Expert provides the ability to move an entire pipe system in the Drawing pane, without making any changes to the pipe system components, or to pan a pipe system to quickly view different areas of the system in the Drawing pane.

To move or pan a pipe system in the Drawing pane:

1. Click the,  , **Pan the drawing** button on the tool bar.

2. When you click the **Pan the drawing** button, the hand symbol is displayed next to your mouse pointer when the pointer is in the Drawing pane.

3. Click and hold down the left mouse button anywhere in the Drawing pane.

4. Drag the pipe system to where you want to move it.

5. Release the left mouse button when you are done moving or panning the pipe system.

Tip: The pan function can also be accessed by clicking in ‘white space’, when either the ‘Show mode’ or the ‘Move mode’ has been selected.

Finding a Pipe or a Node

Finding a pipe or a node on a large system schematic is made easy by selection of the item from the drop down selection boxes on the Pipe Pane and the Node Pane. This allows a selection of a specific pipe or node and it will then be highlighted on the drawing.

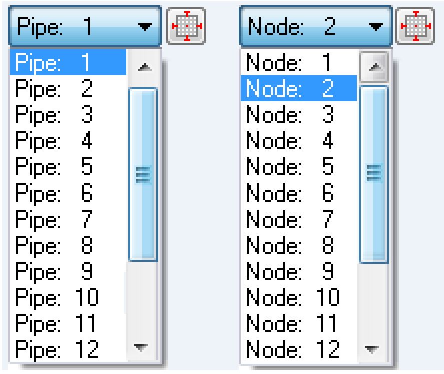
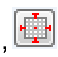


Figure 102 Pipe and Node Panes

If the drawing scale is too small to see the highlighted pipe or node, then click the zoom button , , to zoom the display to show the selected pipe or node in the visible drawing window at an increased scale.

Mirror View of the Pipe System

The drawing menu items provide an option to produce a Mirror View (Left / Right) of the complete pipe system without making any changes to the pipe system components.

Where a system contains free text entries, the starting position of the free text is mirrored but the direction of the text is not changed (the user may have to re-position the text as necessary).

Information about applying the mirror feature to selected items within the system is available in Section: *Group updates on the drawing*.

Inverted View of the Pipe System

The drawing menu items provide an option to produce an Inverted View (Up / Down) of the complete pipe system without making any changes to the pipe system components

Where a system contains free text entries the starting positions of the free text is inverted but multiple lines of text still progress downwards line after line (the user may have to re-position the text as necessary.

Information about applying the invert feature to selected items within the system is available in Section: *Group Updates on the Drawing*.

System Amendments and Group Updates

Making amendments to your system usually involves changing the diameter, length or material of a pipe or group of pipes. It may also involve changing the node elevations where the pipes are linked.

Amendments can be made in a number of ways:

a. Use the Edit Grid to display, filter and edit the data for Pipes, Nodes, and other items.

b. Click on the pipe or node and update the entries shown on the Pipe Pane or the Node Pane.

c. Set the Default Pipe data, select a group of items to be amended, or select a collection of individual items and then copy some or all of the Default Pipe attributes to the selected items.

Using the Edit Grid

1. Click the Edit Network Data tool button 

to display the Edit Grid.

2. Choose a tab to display data for Pipes, Tanks, End Pressures, Nodes, FCVs, PRVs, BPVs, Fixed Losses, Cv Values, Kv Values, Sprinklers K (Imperial), Sprinklers K (Metric), Orifices, Pumps (Fixed Flow), Pumps (Fixed Head) and Pumps (Curve).

3. Select the cell that contains the data you want to edit.

4. The Edit Grid can be over-typed and updated directly (changes are applied immediately).

Note: The changes will be applied immediately and the drawing will be updated. An **Undo** option is **not available** hence amendments cannot be cancelled; the original data must be re-entered.

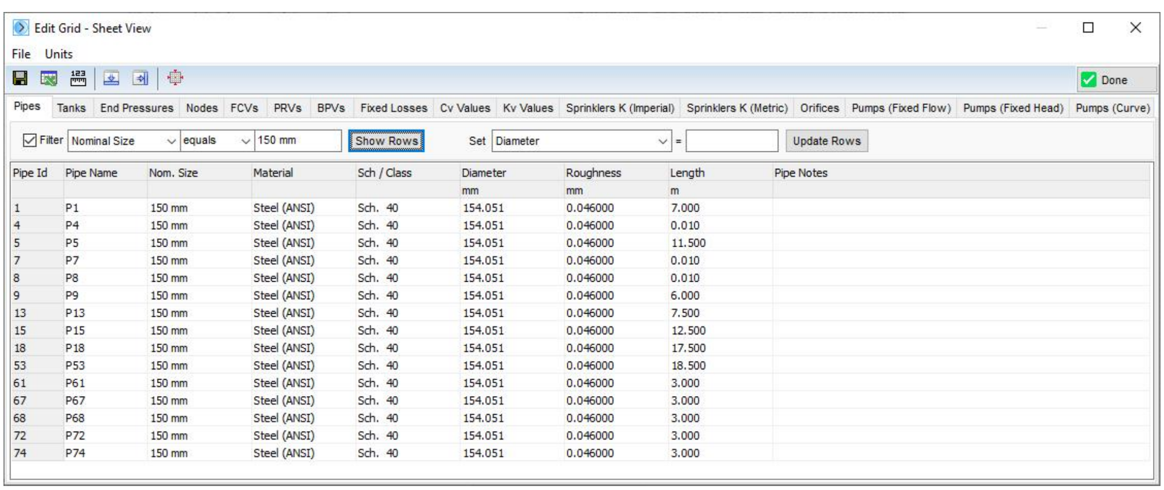


Figure 103 The Edit Grid (in filter mode)

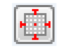
Item Selection:

Selecting a cell in the edit grid will highlight the row and highlight the item in the Drawing Pane.

To zoom to the highlighted item in the Drawing Pane:

When the **Edit Grid: Zoom To configuration option** is set to Single Click, selecting a cell or row in the edit grid will automatically zoom to the highlighted item.

When the **Edit Grid: Zoom To configuration option** is set to Double Click:

1. Select a cell or row in the edit grid and click the Zoom to Highlighted item button, , on the tool bar, or

2. Double click in a cell or the row header (the ID column) in the edit grid.

To de-select an item in the edit grid:

1. Right click in the row header (the ID column), or

2. Change to a different tab.

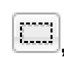
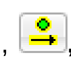
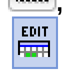
Filtering Items:

By default, the Edit Grid will display all the pipes and nodes in the current system. The Edit Grid can show a subset of the pipes and nodes in the system using one of the following two methods:

1. Using the Filter to only show rows that match certain specified criteria. For example, the data shown on the Pipe Tab can be filtered by: Pipe ID, Pipe Name, Nominal Size, Material/Sch., Diameter, Roughness or length.

   When a filter has been applied, the items displayed in the Edit Grid will also be selected in the Drawing Pane.

   Note: When closing the Edit Grid after applying a filter, the items displayed in the Edit Grid will remain selected in the Drawing pane and the individual selection tool, **,** will be enabled thereby enabling the selection to be refined using the network diagram.

2. Select the required items in the Drawing Pane (using the group, **,** or individual, **,** selection tools) and then Click the Edit Network Data tool button  to display the Edit Grid. Only the selected items will be displayed in the Edit Grid.

   Note: The filter by criteria capability is not available when the Edit Grid is opened when items are selected in the Drawing Pane.

Updating Items:

Data in the Edit Grid can be updated by selecting individual cells and editing the value directly. Alternatively, multiple rows can be amended as a group, in a single operation, by selecting an item to be updated from the drop down list, entering a value to 'Set', and clicking the 'Update Rows' button.

Individual Item Viewing and Modifying

Using the Show Item Info button, , on the tool bar, you can view and/or modify the specific details about any component in a pipe system. Depending on the type of component you select, clicking a component in the Drawing either displays the component's information in the Node or Pipe pane, or opens up the dialog specific to the selected component.

To view and modify an individual item:

1. Click the **Show Item Info** button, , on the tool bar.
2. When you click the **Show Item Info** button, SHOW is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click a component in the Drawing pane that you want to view and/or edit.
4. When you select a tank, join point, or demand pressure, its details are displayed, and can be edited in the Node pane.
5. When you select a demand flow, the **Set Flow Demands** dialog opens, and you can view and edit the demand flow values.
6. When you select a pipe, the pipe's details are displayed, and can be edited in the Pipe pane. To view and edit a pipe's material and/or size, click the **Diam?** or **Material** buttons in the Pipe pane to open the **Pipe diameter data** screen.
7. When you click a fitting symbol or a valve symbol , on a pipe the **Pipe fitting friction coefficients** screen opens. You can view and edit all fittings on the pipe containing the fitting you selected in the Drawing pane. You can also select a pipe in the Drawing pane, and then click the **Add/Change Fittings** button, , in the Pipe pane to open the **Pipe fitting friction coefficients** screen.
8. When you click a component symbol on a pipe, , the **Set Component Pressure Loss** dialog opens, and you can view and edit the component's pressure loss values. You can also select a pipe in the Drawing pane, and then click the **Add/Change Component Pressure Loss** button, , in the Pipe pane to open the **Set Component Pressure Loss** dialog.
9. When you click a control valve symbol on a pipe, , the **Control Valve Data** screen opens, and you can view and edit the control valve type or values. You can also select a pipe in the Drawing pane, and then click the **Add/Change Control Valve** button, , in the Pipe pane to open the **Control Valve Data** screen.
10. When you click a pump symbol on a pipe, , the **Pump Data** screen opens, and you can view and edit the pump's properties. You can also select a pipe in the Drawing pane, and then click the **Add/Change Pump** button, , in the Pipe pane to open the **Pump Data** screen.

Group Updates on the Drawing

The attributes of a group of pipes or nodes can be amended in a single update operation.

The Selection tool button  allows the user to select a group of items which are located within an area defined by a rectangle. When using the isometric grid, clicking the Selection tool button will display a sub menu to enable items to be selected using either a rectangle or isometric (rhombus) selection.

Figure 104 Isometric Selection Menu

While holding down the left mouse button, drag the mouse to create a rectangle around the item(s) you want to select.

The selected items can then be amended using the following operations:

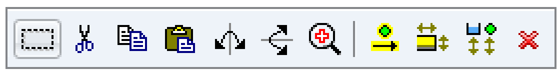
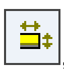


Figure 105 Selection Amendment Toolbar

Cut, Copy, Paste, Mirror, Invert, Zoom in, Adjust Attributes (Pipes), Adjust Attributes (Nodes), & Delete.

When the, , 'Adjust Attributes of Selected Pipes' button is clicked, a screen showing the current 'Default Pipe' attributes and a list of pipes is displayed with the selected pipes shown as checked. The 'Default Pipe' data can now be amended as required and the buttons in the middle of the screen can then be used to copy item values to the group of selected pipes.

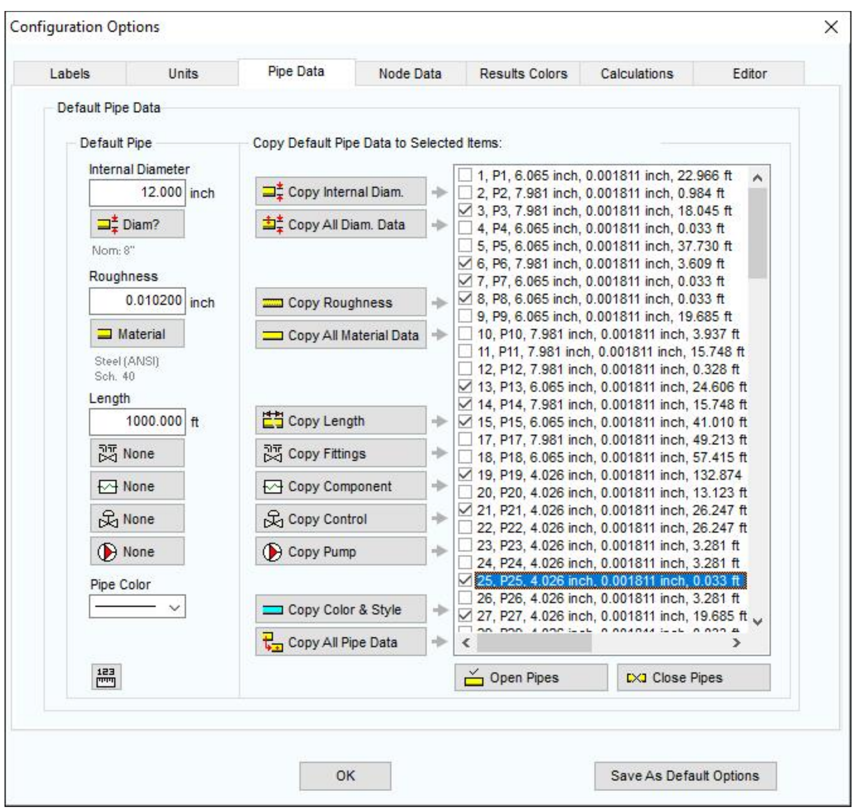
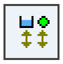
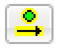


Figure 106 Pipe data Group Updates

When the,  , 'Adjust Attributes of Selected Nodes' button is clicked, a similar screen to the one above is shown, except on the left hand side it shows Default Node values and on the right a list of selected nodes is displayed. Data values can be copied from the default node to the selected nodes using the 'copy' buttons in the middle of the screen.

The individual Selection tool button,  , allows the user to select an ad hoc group of pipes and nodes, on a one-by-one basis. This is often useful when the group of items to be updated are positioned within different parts of the drawing, such that they cannot be selected inside a standard rectangle (without including other items that are not wanted).

Again the characteristics for these individually selected 'groups' of items can be updated in one operation by using the 'Adjust Attributes of Selected Pipes' and 'Adjust Attributes of Selected Nodes' buttons.

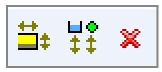
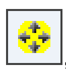
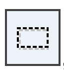


Figure 107 Pipe & Node Adjustment Toolbar

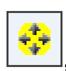
Moving Components in a System

In Pipe Flow Expert, you can move sections of a pipe system by dragging a node or a group of components to another place in the drawing, and you can move individual components on a pipe to a different location along the same pipe. You can also cut, copy, and paste pipes and nodes to move pipes and nodes in the system.

Using the Drag and Move Items button, , on the tool bar, you can move a node, and all the components connected to the node, or you can move individual components on a pipe in the Drawing pane.

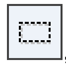
Using the Selection tool button, , on the tool bar, you can move nodes or groups of components in the Drawing pane.

To move system components using the Drag and Move Items button:

1. Click the **Drag and Move Items** button, , on the tool bar.
2. When you click the **Drag and Move Items** button, each of the pipe system's components are highlighted in the Drawing pane. MOVE is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Click the component you want to move and hold down the left mouse button in the Drawing pane.
4. Drag the component where you want to move it.
5. Release the left mouse button when you are done moving the component.

When you move components using the **Drag and Move Items** button, the values for the moved components do not change. If you want to change the value of a moved component, such as pipe length, the value needs to be manually updated in Pipe Flow Expert.

To move system components using the Selection tool:

1. Click the **Selection tool** button, , on the tool bar.
2. When you click the **Selection tool** button, the selection icon is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Draw a box around the node or group of components you want to move in the Drawing pane.
4. To draw a box, click where you want to start the box in the Drawing pane and hold down the left mouse button, then drag the mouse until you have drawn the box.
5. Release the left mouse button.
6. Click in the box and hold down the left mouse button, then drag the box to the new location in the Drawing pane. Moving the box automatically moves the node or group of components within the box.
7. Release the left mouse button and right-click when you are done moving the component.

When you move components using the **Selection tool** button, the values for the components that are moved do not change. If you want to change the value of a moved component, such as pipe length, the value needs to be manually updated in Pipe Flow Expert.

Cutting, Copying, and Pasting Nodes & Pipes

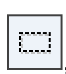
Nodes (tanks, demand pressures, or join points) and the connecting pipes can be cut, copied or pasted in the Drawing pane. A pipe cannot exist unless it is connected between two nodes. When you cut a node from a pipe system in the Drawing pane, any pipes that have one end connected to a selected node but which are not linked at the other end to another selected node (i.e. the pipe is not fully contained within the selection rectangle), will be permanently removed from the system (and the components on those deleted pipes will also be removed from the system).

When you paste a group of objects on to the Drawing pane, these items will initially appear slightly offset from the position of the original copy operation. The group of objects will be contained in a selection rectangle, to allow the objects to be moved to a new position. Once the selected items have been positioned, click the right mouse button to complete the paste operation.

If you try to paste a node over the top of an existing node or an existing pipe, the paste action will be disallowed.

NOTE: A cut, copy, or paste action in the Drawing pane cannot be undone.

To cut, copy, and paste nodes and pipes in the Drawing pane:

1. Click the **Selection tool** button, , on the tool bar.
2. When you click the **Selection tool** button, the selection icon is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Draw a box around the pipes and nodes you want to cut or copy in the Drawing pane. To draw a box, click where you want to start the box in the Drawing pane and hold down the left

mouse button, then drag the mouse until you have drawn the box and then release the left mouse button.

4. To cut the nodes and pipes, click the **Cut** button, , on the tool bar or select **Edit | Cut** Any pipes which have one end connected to a selected node but are not linked at the other end to another selected node, will be permanently removed from the system.

5. After clicking the **Cut** button or selecting **Edit | Cut**, a message asking if you are sure you wish to cut the nodes and all pipes that connect to these nodes will be displayed.

6. Click **Yes**.

7. You may wish to copy rather than cut a set of selected items. To copy the nodes and pipes, click the **Copy** button, , on the toolbar or select **Edit | Copy**.

8. Click the **Paste** button, , on the tool bar or select **Edit | Paste** to paste the cut or copied nodes and pipes to the Drawing pane.

9. When you paste nodes and pipes to the Drawing pane, these items will be pasted slightly offset from the position of the original copy operation. The group of new objects will be contained in a selection rectangle.

10. Click in the box and hold down the left mouse button, then drag the objects to their new location in the Drawing pane.

11. Release the left mouse button, to allow the objects to locate on the drawing grid.

12. Click the right mouse button to complete the paste operation.

Attaching Nodes

Nodes (tanks, demand pressures, or join points) can be attached onto other nodes in the Drawing Pane. When the node being moved (source node) is attached to another node (target node) the source node is deleted and all pipes connected to the source node are connected to the target node.

NOTE: Attaching nodes in the Drawing pane cannot be undone.

To attach a node using the Drag and Move Items tool:

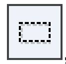
1. Click the **Drag and Move Items** button, , on the tool bar.

2. When you click the **Drag and Move Items** button, each of the pipe system's components are highlighted in the Drawing pane. MOVE is displayed next to your mouse pointer when the pointer is in the Drawing pane.

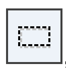
3. Click the node you want to move (source node) and hold down the left mouse button in the Drawing pane.

4. Drag the source node over the target node that you wish to attach to and release the left mouse button.

To attach one or more nodes using the Selection tool:

1. Click the **Selection tool** button, , on the tool bar.
2. When the **Selection tool** is active, the selection icon is displayed next to your mouse pointer when the pointer is in the Drawing pane.
3. Draw a box around the nodes (source nodes) you want to move in the Drawing pane. To draw a box, click where you want to start the box in the Drawing pane and hold down the left mouse button, then drag the mouse until you have drawn the box and then release the left mouse button.
4. Click in the box and hold down the left mouse button in the Drawing pane.
5. Drag the selected source nodes over one or more target nodes that you wish to attach to and release the left mouse button.
6. Click the right mouse button to complete the attach operation.

Copying Between Drawings

The **Selection tool** button, , allows a group of nodes and pipes to be copied, as described in the previous section. It is possible to use the group copy and the paste feature to transfer a group of nodes and pipes from one drawing to another drawing.

Nodes and pipes on the current drawing are selected and copied.
A new grid is then chosen or a different drawing is opened.

The **Paste** button, , can be used to place the copied nodes and pipes on to the new drawing

Opening Additional Systems

Tabbed Design Sheets let you work on multiple systems simultaneously. Each tab operates independently, enabling instant switching between different systems, in both design mode and results mode, for easy comparison of models and calculation results.

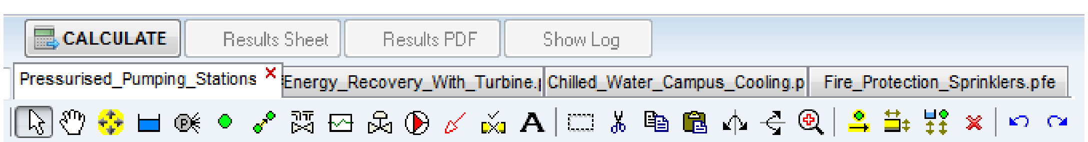
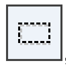


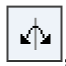
Figure 108 Tabs

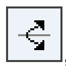
- Tabbed Sheets work just like Tabs in an Internet Browser.
- Tabbed Sheets work in both Design View and Results Mode.
- Switch between different system models with a single click.
- Easy to design, view, and compare results across multiple models.

- Easy to copy sections of drawings between different system models.
- Easy to demonstrate and review results for altered designs

Rotating Selected Items

The **Selection tool** button, , allows a group of nodes and pipes to be selected, as described in the previous section. It is possible to use the options on the edit menu to rotate the selected group either 90 degrees clockwise or 90 degrees anticlockwise. The mirror or invert features can also be applied to the selected group.

The Mirror Selection button, , can be used to mirror the selected group.

The Invert Selection button, , can be used to invert the selected group.

Pipe Flow Expert does not allow one node to be placed over the top of another node, or over the top of an un-related pipe. This restriction is applied to prevent confusion.

When the rotated, or mirrored, or inverted group are added back to the drawing the action may be disallowed if the above conditions apply. The selected group will have to be re-positioned to allow the items to be ‘dropped’ back onto the drawing pane.

Moving a Pipe to link at a new position

If a pipe has been drawn incorrectly between two nodes, it is possible to unlink the end of the pipe and enter the pipe drawing mode to allow the existing pipe to be connected at a new position.

To unlink / move an existing pipe:

1. Select the pipe to be moved by clicking on it in the drawing pane.
2. Click the right mouse button to display the **Pipe Pop-up menu.**
3. Choose the ‘Move / Unlink end of Pipe’ option.
4. The pipe drawing mode will be selected.
5. The ‘direction arrow’ end of the pipe will be unlinked to allow you to connect the pipe at a new position.

The ‘Move / Unlink end of Pipe’ option can also be performed when in ‘Add Pipes’ drawing mode, by ‘clicking’ on a pipe to first highlight it, then ‘clicking’ again on the highlighted pipe to ‘unlink’ it.

Using the Undo and Redo Functions

When you add a single pipe or a single node in the Drawing pane, you can undo the add pipe or node action to remove the pipe or node from the Drawing pane. If you change your mind about an undo action, you can redo the pipe or node in the Drawing pane.

To undo an add pipe or node action in the Drawing pane, click the Undo button, , on the tool bar or select Edit | Undo.

To redo an undo action in the Drawing pane, click the Redo button, , on the tool bar or select Edit | Redo.

NOTE: If you added new components, such as fittings or flow demands, to a pipe or node, and you then click the Undo button or select Edit | Undo, and then click the Redo button or select Edit | Redo, then the pipe or node will be re-added to the Drawing pane without the components. When you use the undo action for a pipe or node with components, any components that were not drawn as part of the default pipe are permanently removed from the pipe.

Deleting Components in a System

Individual components or groups of components can be deleted from a pipe system in the Drawing pane. When you delete a node from a pipe system in the Drawing pane, all the pipes connected to the node, and the components on those pipes, are deleted from the system. When you delete an individual pipe, all the components on the pipe are also deleted.

Highlight the item to be deleted, and then click the Delete button, , on the tool bar. You can also use Ctrl-Delete on the keyboard instead of clicking the Delete button.

An item can be selected and highlighted by clicking on it. A group of items can be selected using the selection tool. This allow selection of groups of Nodes (tanks, demand pressures, or join points) and individual pipes.

Other pipe system components (Pumps, Fittings, Control Valves, & Components) are deleted on the dialog associated with that component. For example, fittings on a pipe are deleted from the Pipe fitting friction coefficients dialog.

NOTE: Deletions in the Drawing pane cannot be undone.

Deleting a Node or Pipe

To delete a node or pipe from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.
2. Click the node or pipe you want to delete in the Drawing pane.
3. You can also select a node or pipe for deletion by using the **Selection** tool button, , on the tool bar.
4. Click the **Delete** button, , on the tool bar, or select **Edit | Delete**, or click Ctrl-Delete on the keyboard.

5. If you are deleting a node, the message, *Are you sure you wish to delete the current highlighted node and all pipes that connect to the highlighted node?,* is displayed after clicking the **Delete** button or selecting **Edit | Delete**.

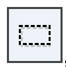
6. If you are deleting a pipe, the message, *Are you sure you wish to delete the current highlighted pipe?,* is displayed after clicking the **Delete** button or selecting **Edit | Delete**.

7. Click **Yes**.

8. The selected node or pipe is deleted. When you delete a node, all pipes and components connected to the node are deleted. When you delete a pipe, all components on the pipe are deleted with the pipe.

Deleting a Group of Components

To delete a group of components from a pipe system:

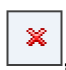
1. Click the **Selection tool** button, , on the tool bar.

2. When you click the **Selection tool** button, the selection icon is displayed next to your mouse pointer when the pointer is in the Drawing pane.

3. Draw a box around the group of components that you want to delete in the Drawing pane.

4. To draw a box, click where you want to start the box in the Drawing pane and hold down the left mouse button, then drag the mouse until you have drawn the box.

5. Release the left mouse button.

6. Click the **Delete** button, , on the tool bar or select **Edit | Delete**, or click Ctrl-Delete on the keyboard.

7. The message, Are you sure you wish to delete the selected nodes and all pipes that connect to these nodes?, is displayed after clicking the **Delete** button or selecting **Edit | Delete**.

8. Click **Yes.**

9. The selected nodes and all the pipes and their components that are connected to the nodes are deleted.

Deleting a Demand Flow

To delete a demand flow from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.

2. Click the node containing the demand flow that you want to delete in the Drawing pane.

3. Click one of the **Edit** buttons in the Node pane to open the **Set Demand Flow** dialog.

4. If the **In Flow** field contains a value other than **0.000**, type **0**.

5. If the **Out Flow** field contains a value other than **0.000**, type **0**.

6. Click **OK** to save your changes and close the **Set Demand Flow** dialog.

7. The demand flow icon and value are removed from the selected node in the Drawing pane.

Deleting Fittings

To delete a fitting from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.

2. Click the fitting symbol, , on the pipe containing the fitting that you want to delete in the Drawing pane.

3. Clicking the fitting symbol opens the **Pipe fitting friction coefficients** dialog with all the fittings on the pipe listed in the **Fittings on** section.

4. You can also select the pipe containing the fitting that you want to delete in the Drawing pane, and then click the **Add/Change Fittings** button, , in the Pipe pane to open the **Pipe fitting friction coefficients** dialog.

5. Select the row containing the fitting that you want to delete in the **Fittings on** section.

6. Click the **Delete** button, 0.3000 1, next to the row that shows the K value for the fitting, to delete the fitting(s) from the selected pipe.

7. Click the **Save** button in the **Pipe fitting friction coefficients** dialog to save your changes and close the **Pipe fitting friction coefficients** dialog.

8. If you deleted all the fittings on the pipe, the fittings symbol on the selected pipe is deleted in the Drawing pane.

Deleting a Component Pressure Loss

To delete a component's pressure loss from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.

2. Click the component pressure loss symbol, , on the pipe containing the component pressure loss that you want to delete in the Drawing pane.

3. Clicking the component pressure loss symbol opens the **Set Component Pressure Loss** dialog.

4. You can also select the pipe containing the component pressure loss that you want to delete in the Drawing pane, and then click the **Add/Change Component Pressure Loss** button, , in the Pipe pane to open the **Set Component Pressure Loss** dialog.

5. If you want to save and reuse the component pressure loss data for other pipes in the pipe system or future pipe systems, be sure to click the **Save To File** button and save the pressure loss data to a .pfco file before deleting the component pressure loss from the selected pipe.

6. Click the **Delete** button.

7. The message, Are you sure you wish to delete the component and its data from this pipe? is displayed after clicking the **Delete** button.

8. Click **Yes**.

9. The component pressure loss icon and values are removed from the pipe in the Drawing pane.

Deleting a Control Valve

To delete a control valve from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.

2. Click the flow control symbol, , on the pipe containing the flow control that you want to delete in the Drawing pane.

3. Clicking the control valve symbol opens the **Control Valve Data** dialog.

4. You can also select the pipe containing the control valve that you want to delete in the Drawing pane, and then click the **Add/Change Control Valve** button, , in the Pipe pane to open the **Control Valve Data** dialog.

5. Click the **Delete** button.

6. The control valve icon and value are removed from the pipe in the Drawing pane.

Deleting a Pump

To delete a pump from a pipe system:

1. Click the **Show Item Info** button, , on the tool bar.

2. Click the pump symbol, , on the pipe containing the pump that you want to delete in the Drawing pane.

3. Clicking the pump symbol opens the **Pump Data** dialog.

4. You can also select the pipe containing the pump that you want to delete in the Drawing pane, and then click the **Add/Change Pump** button, , in the Pipe pane to open the **Pump Data** dialog.

5. If you want to save and reuse the pump data for other pumps in the pipe system or future pipe systems, be sure to click the **Save To File** button and save the pump data to a .pfpm file before deleting the pump from the selected pipe.

6. Click the **Delete** button.

7. The message, Are you sure you wish to delete the pump and all pump data from this pipe?, is displayed after clicking the **Delete** button.

8. Click **Yes**.

9. The pump icon and value are removed from the pipe in the Drawing pane.

Calculations and Results

Once you have finished constructing a pipe system, you can calculate and solve the system to determine whether the system is valid. If the system is valid, the calculated values for the system's flow rates and pressures are displayed in the Drawing pane and the Result tables.

Results drawing:

The calculated values displayed in the Drawing pane are determined by the options selected on the Labels and Units tab of the Configuration Options dialog. The user can choose which labels to include on the drawing, and for some labels, which attribute value is displayed within the label.

Results Sheet:

The Results tables contain all of the calculated values for the whole pipe system, including pipe data, fitting data, component data, pump data, fluid data and node data.

Results PDF:

The Results PDF can be generated once a system has been solved. The data shown in the Results PDF is configurable and can be selected at a section levels, such as pipe data and node data, and then further refined at an attribute level, as to which attributes are shown within the various results tables.

Results Log:

When you calculate a pipe system, a log is generated and displayed in the Results Log dialog. If a pipe system cannot be solved, the log provides additional information and may offer some suggestions as to why the system cannot be solved.

Calculations:

To calculate the balanced results for a pipe system in the Drawing pane, click the Calculate button.

The 'Include' check boxes in the Pipe pane allow you to include or exclude the fittings, component pressure losses, control valves, and pumps when solving the system. The check boxes allow you to view and re-run the system calculations without having to delete and re-add the components to the system.

Select the Include check box to include a component's values in a system's calculations. Clear an Include check box to exclude a component's values from a system's calculations.

Automatic Checks and Updates

Before a pipe system is solved, the software performs some automatic checking of the system data and allows users to accept various recommended updates to make the system information consistent.

The software interface is intuitive and it provides easy access to amend the system data as required, however standalone changes to pipe diameters, pipe lengths and node elevations can sometimes create inconsistence’s in the design data.

Three types of checks are performed before a system solution is calculated.

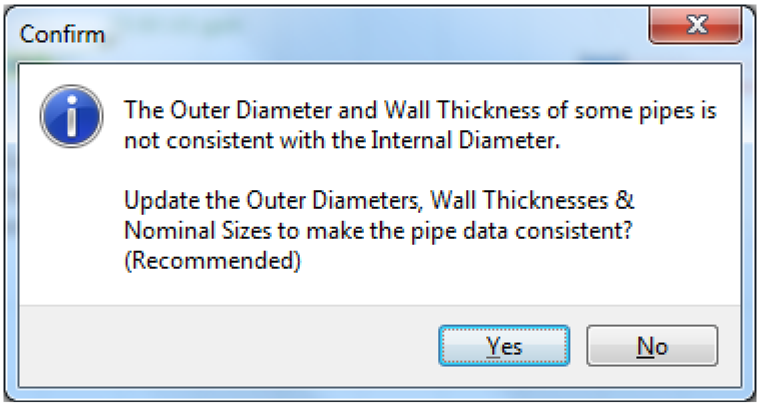


Figure 109 Pipe Attribute Inconsistency

Where an Internal Diameter has been amended by over typing the data in the pipe pane or the edit grid the Outer Diameter, Wall Thickness & Internal Diameter may no longer be consistent with each other.

The option to update Outer Diameters and Wall Thicknesses should normally be accepted. This will allow the diameter data to be made consistent and the nominal pipe size will then be recalculated.

Fitting sizes will be checked and matched to the nominal pipe size.

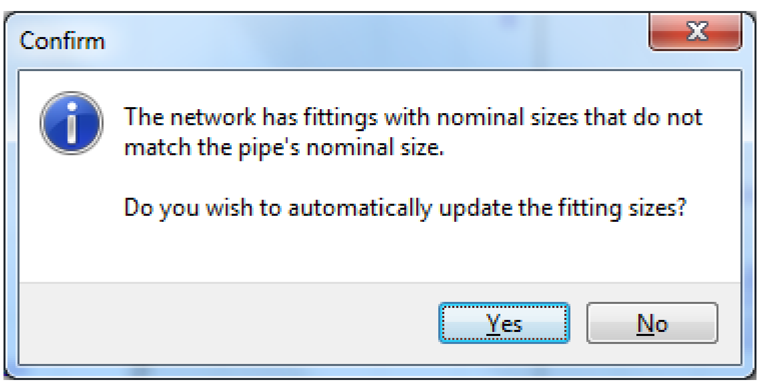


Figure 110 Fitting Size Inconsistency

If the nominal pipe size does not match some of the fitting sizes associated with the pipe, then the option to update the fitting sizes should normally be accepted. This will update the fitting sizes and the fitting 'K' values so that the fitting pressure losses will be calculated correctly.

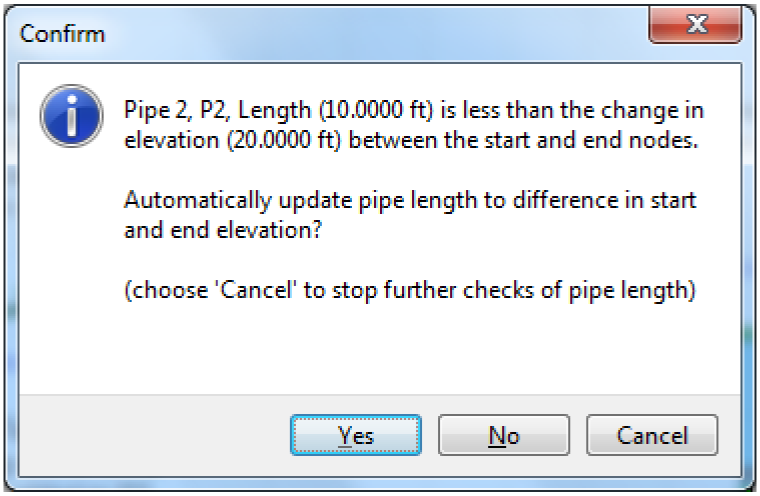


Figure 111 Pipe Length & Elevation Inconsistency

Where pipe lengths are not sufficient to span the difference in node elevations, these pipes will be identified and the user will be asked to confirm an update for each pipe. The pipe length can automatically be set to the difference in start and end elevations of the pipe.

The changes in pipe length will be noted in the results log when the system is solved. The user may wish to check that the lengths of these pipes are also sufficient to cover any additional length that may be required if the pipe is not vertical (a non-vertical, sloped pipe, would need to have a length that was greater than just the change in elevation between its start and end node). Where the pipe direction is not vertical and the length has been set to less than the difference in elevation between its start node and end node, it will be necessary to amend each pipe length manually, in the pipe pane or edit grid.

Calculating and Solving a System

To calculate and solve a system:

1. Open the .pfe file for the pipe system you want to calculate in Pipe Flow Expert.

2. The pipe system is displayed in the Drawing pane. In order to calculate a system's values, the pipe system must be open in the Drawing pane.

3. Click the **Calculate** button on the tool bar.

4. The system results are calculated and the **Results Log** dialog is shown.

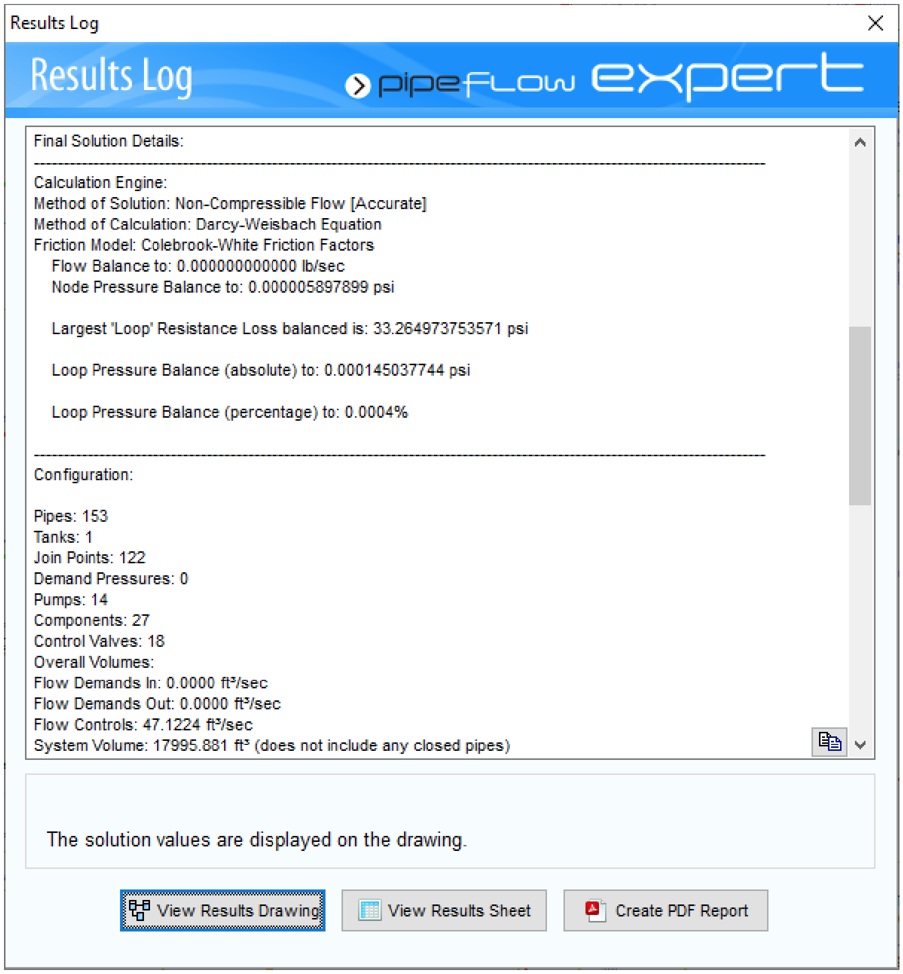


Figure 112 Result Log dialog

If the pipe system is solved, the message '*The network was solved successfully'* is displayed in the **Results Log** dialog.

If the pipe system is not solved, the log offers some suggestions as to why the system could not be solved, and the message '*The network was not solved. See log details above.'* is displayed in the **Results Log** dialog. When this happens, review the log and click **OK** to return to the Drawing pane in design mode. Make the necessary changes to the system to resolve the design errors in the system, and click the **Calculate** button on the tool bar to recalculate the revised system.

If the Results Log has been closed, it can be re-opened by clicking the **Show Log** button on the toolbar.

To change back to Design Mode, close the Results Log and then click on the **Re-Design** button on the toolbar.

To view the Results Drawing:

Click the **View Results Drawing** button on the results log to display the drawing with results.

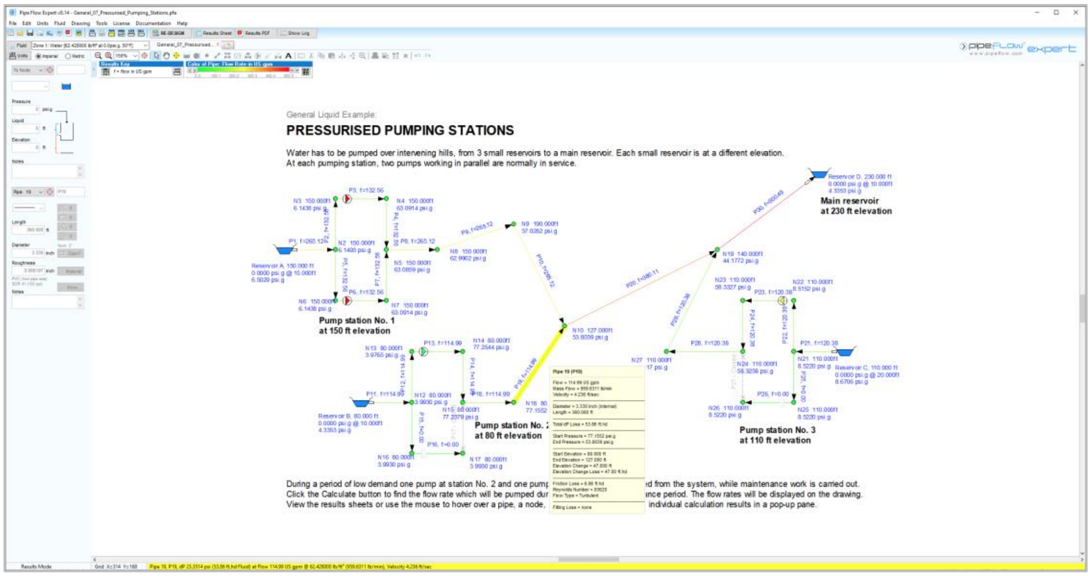
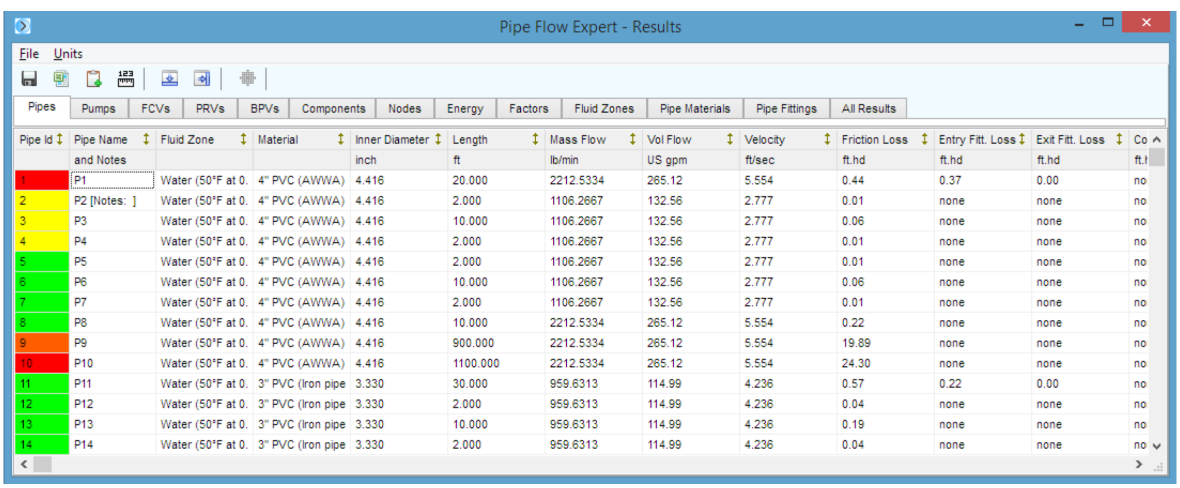


Figure 113 Pipe Flow Expert interface in Results mode

The details for the last node or pipe selected with a mouse click will be displayed in the hint pane. Moving the cursor over an item on the drawing will allow the user to view the calculated results in a pop-up data pane.

To view the Results Table:

Click the **View Results Sheet** button on the results log to open the Results Table.



Pipe Id	Pipe Name and Notes	Fluid Zone	Material	Inner Diameter inch	Length ft	Mass Flow lb/min	Vol Flow US gpm	Velocity ft/sec	Friction Loss ft.hd	Entry Fitt. Loss ft.hd	Exit Fitt. Loss ft.hd	Co ft.h
1	P1	Water (50°F at 0.	4" PVC (AWWA)	4.416	20.000	2212.5334	265.12	5.554	0.44	0.37	0.00	no
2	P2 [Notes:]	Water (50°F at 0.	4" PVC (AWWA)	4.416	2.000	1106.2667	132.56	2.777	0.01	none	none	no
3	P3	Water (50°F at 0.	4" PVC (AWWA)	4.416	10.000	1106.2667	132.56	2.777	0.06	none	none	no
4	P4	Water (50°F at 0.	4" PVC (AWWA)	4.416	2.000	1106.2667	132.56	2.777	0.01	none	none	no
5	P5	Water (50°F at 0.	4" PVC (AWWA)	4.416	2.000	1106.2667	132.56	2.777	0.01	none	none	no
6	P6	Water (50°F at 0.	4" PVC (AWWA)	4.416	10.000	1106.2667	132.56	2.777	0.06	none	none	no
7	P7	Water (50°F at 0.	4" PVC (AWWA)	4.416	2.000	1106.2667	132.56	2.777	0.01	none	none	no
8	P8	Water (50°F at 0.	4" PVC (AWWA)	4.416	10.000	2212.5334	265.12	5.554	0.22	none	none	no
9	P9	Water (50°F at 0.	4" PVC (AWWA)	4.416	900.000	2212.5334	265.12	5.554	19.89	none	none	no
10	P10	Water (50°F at 0.	4" PVC (AWWA)	4.416	1100.000	2212.5334	265.12	5.554	24.30	none	none	no
11	P11	Water (50°F at 0.	3" PVC (Iron pipe	3.330	30.000	959.6313	114.99	4.236	0.57	0.22	0.00	no
12	P12	Water (50°F at 0.	3" PVC (Iron pipe	3.330	2.000	959.6313	114.99	4.236	0.04	none	none	no
13	P13	Water (50°F at 0.	3" PVC (Iron pipe	3.330	10.000	959.6313	114.99	4.236	0.19	none	none	no
14	P14	Water (50°F at 0.	3" PVC (Iron pipe	3.330	2.000	959.6313	114.99	4.236	0.04	none	none	no

Figure 114 Results Tables

When an item is highlighted in the results sheet table, the selected item is also highlighted on the drawing pane and data for the highlighted item is displayed in the Hint pane.

If the results tables have been closed, click the **Result Sheet** button on the toolbar to reopen the **Results Tables.**

To Create a PDF Report Document:

Units and Labels Note: First configure the labels and units to be used, as described in the section ‘Configuring the System Results’.

Click the **Create PDF Report** button to open the PDF report dialog.

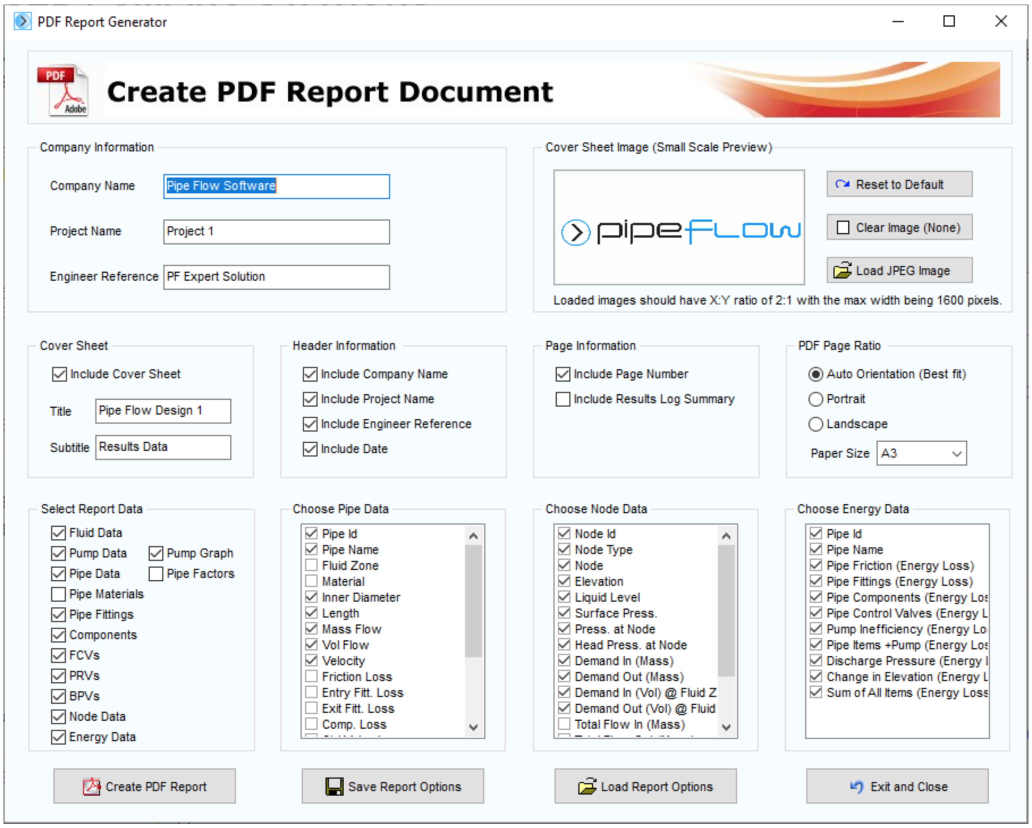


Figure 115 PDF Report Options Dialog

1. Enter the Company Information.
2. Load a cover sheet image.
3. Select the Cover Sheet, Header Information and Page Information options.
4. Select the Report Data options.
5. Choose the Pipe Data, Node Data and Energy Data to be included.
6. Click the **Create PDF Report** button to generate a PDF results document.

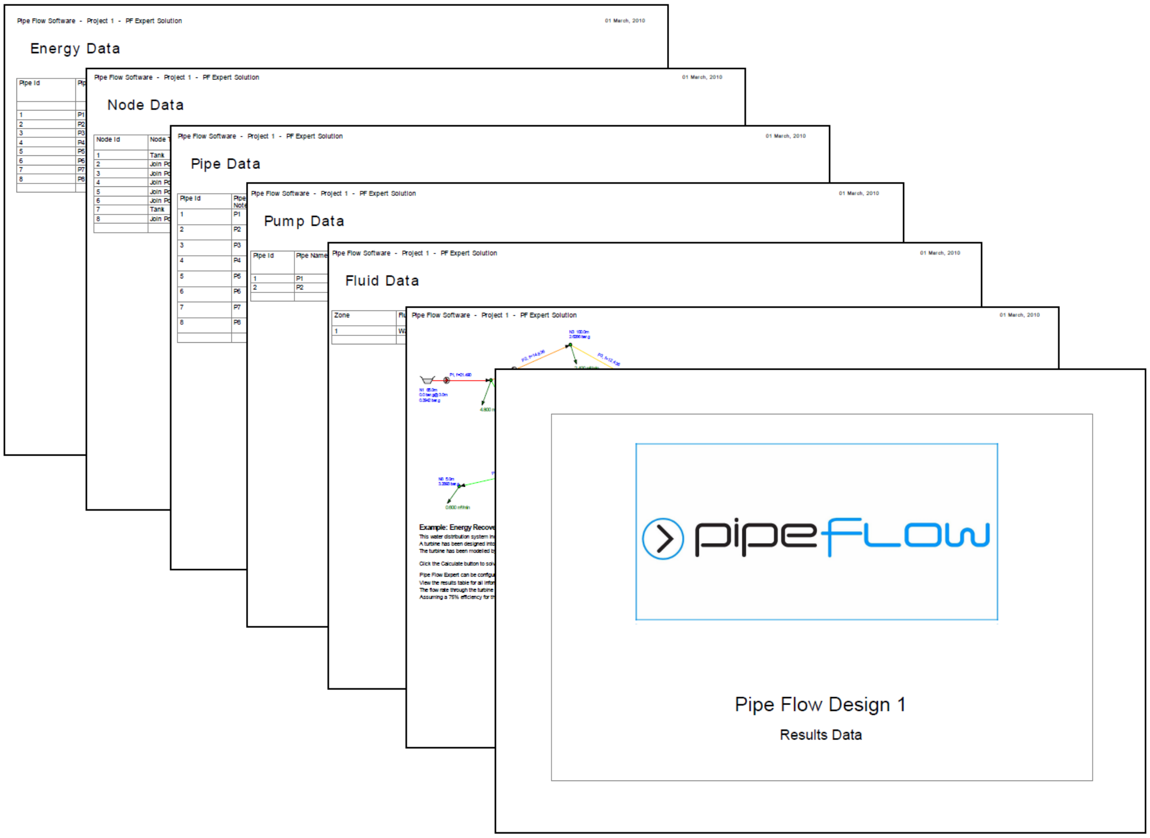


Figure 116 PDF Report Pages

Configuring the System Results

Once a pipe system is calculated and solved, the calculated results are displayed with the pipe system in the Drawing pane. The options selected on the Labels tab of the Configuration Options dialog determine which result values are displayed in the Drawing pane. By default, the calculated flow and pressure values are displayed for each pipe in the system.

You can also reconfigure the units for the pipe system results on the Units tab of the Configuration Options dialog. You can access the Units tab of the Configuration Options dialog by selecting

Specify Units from the Units menu, or by clicking the Choose Units button, , on the toolbar, or clicking the Choose Units button in the Results dialog.

To configure the pipe system labelling for results:

1. Click the **Choose Labelling** button, , on the tool bar to open the **Labelling** tab of the **Configuration Options** screen.

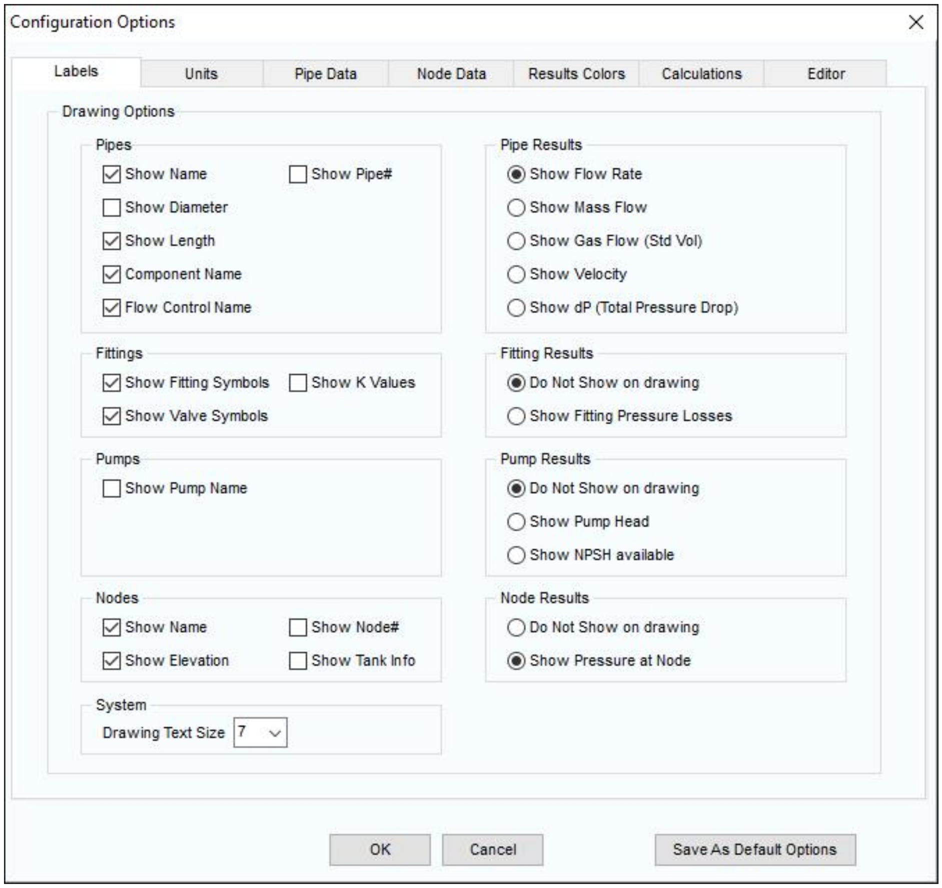


Figure 117 Configuration Options dialog – Labelling tab

2. Click one of the labelling options in the **Pipe Results**, **Fitting Results**, **Pump Results** and **Node Results** sections to determine which of the pipe system's calculation results are displayed in the Drawing pane.

3. Click **OK** to close the **Configuration Options** dialog and apply your changes to the results displayed in the Drawing pane.

To reconfigure the units used in a pipe system and its calculated results:

1. Click the **Choose Units** button, , in the **Results** dialog or on the tool bar, or select **Specify Units** from the **Units** menu to open the **Units** tab of the **Configuration Options** screen.

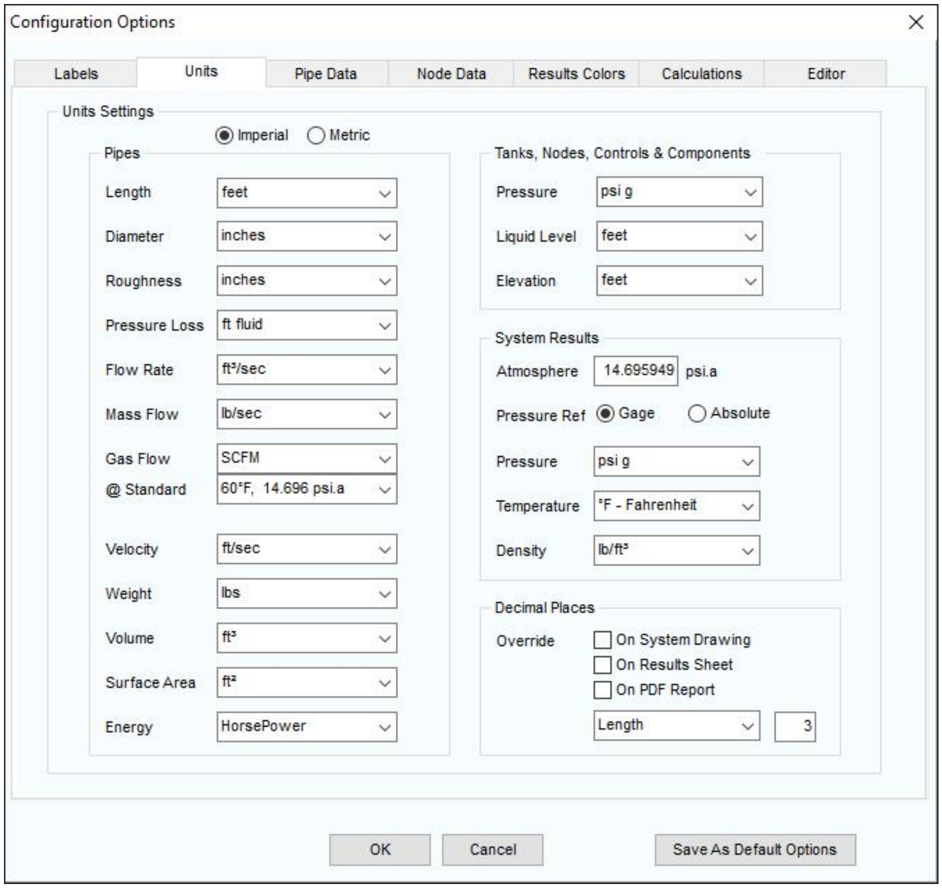


Figure 118 Configuration Options Screen – Units tab

2. Select the required units for each item.

Do not click the **Imperial** or **Metric** option in the **Change All Units** section of the **Units** tab, unless you want to change the universal unit setting and reset all the values currently defined on the **Units** tab.

NOTE: When you change the universal unit setting, it converts all unit values to the unit type selected, and all values on the **Units** tab of the **Configuration Options** screen are converted to the fields' default values. For example, if you are changing the universal setting from imperial to metric, all values, including any imperial units selected on the **Units** tab of the **Configuration Options** screen, are converted to metric. All field values on the **Units** tab are also reset to their default metric value.

Click **OK** to close the **Configuration Options** dialog and apply the unit configurations to the pipe system and its results.

Viewing the System Results

After a pipe system is solved, you have the option of opening the Results Tables to view all the calculated results, or viewing the solved pipe system in the Drawing pane with specific results displayed with the pipe system. To view only the solved system in the Drawing pane, click the 'View Results Drawing' button in the Results Form dialog after the system is calculated and solved. To view the Results Tables, click the 'View Results Sheet' button.

The Results Drawing and the Results Sheet can be viewed at the same time, and they are interactive in that clicking on an item on the Results Drawing will automatically highlight and move to that item within the Results Sheet. The reverse is also true, in that selecting a row in the Results Sheet will highlight and zoom to that item on the Results Drawing. The user can switch between the Results Drawing and the Results Sheet as needed. In addition, the user can review the output in the Results Log dialog, by clicking the Show Log button.

The Results Tables contain a series of tabs that display the calculated results for each component of the pipe system in a table. The tabs are organized by component type, except for the All Results tab. The All Results tab displays all of the calculated results by component type. All values displayed in the Results Tables are read-only.

To view a pipe system's calculated results in the Results Tables:

1. After a pipe system has been calculated and solved, click the **View Results Sheet** button on the **Results Log** dialog to open the **Results Tables** or close the results log and click the **Results Sheet** button on the tool bar.

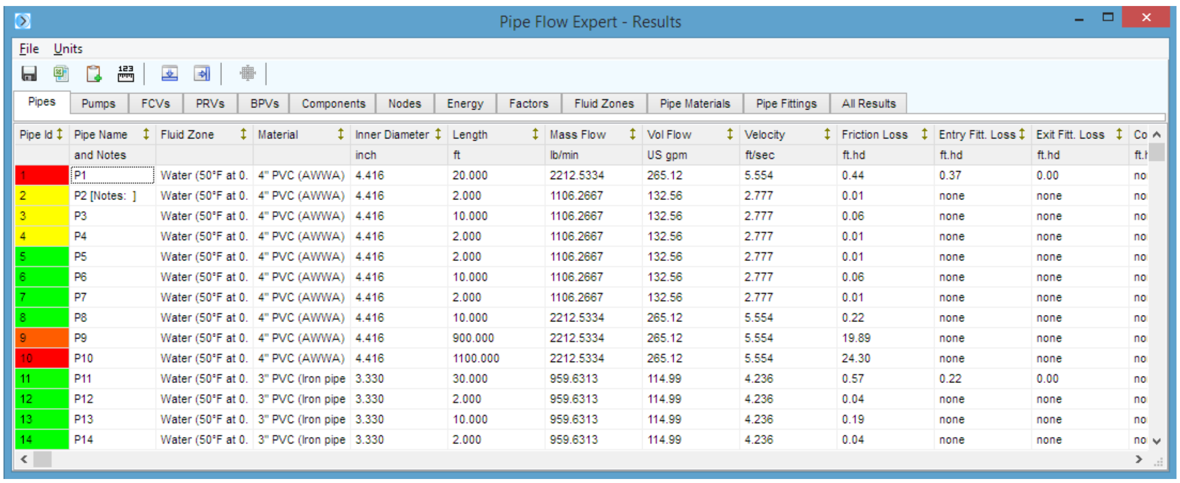


Figure 119 Results Tables – All Results tab

2. Click the tab(s) containing the data you want to view.

3. Click the **Close** button, , to close the **Results Tables** and return to the Drawing pane.

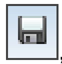
Saving the System Results

The values displayed in the Results Tables can be saved to a Microsoft® Excel® .xlsx file. When you are saving from the Results Tables, only the values displayed on the currently selected tab are saved to the .xlsx file. To save all the result values, click the All Results tab before saving.

To save a pipe system's results to an .xlsx file:

1. After a pipe system has been calculated and solved, click the **View Results Sheet** button on the **Results Log** dialog to open the **Results Tables** or close the results log and click the **Results Sheet** button on the tool bar.

2. Click the tab containing the values you want to save.

3. Click the **Save** button, , or select **File | Save to Excel** to open the **Save As** dialog.

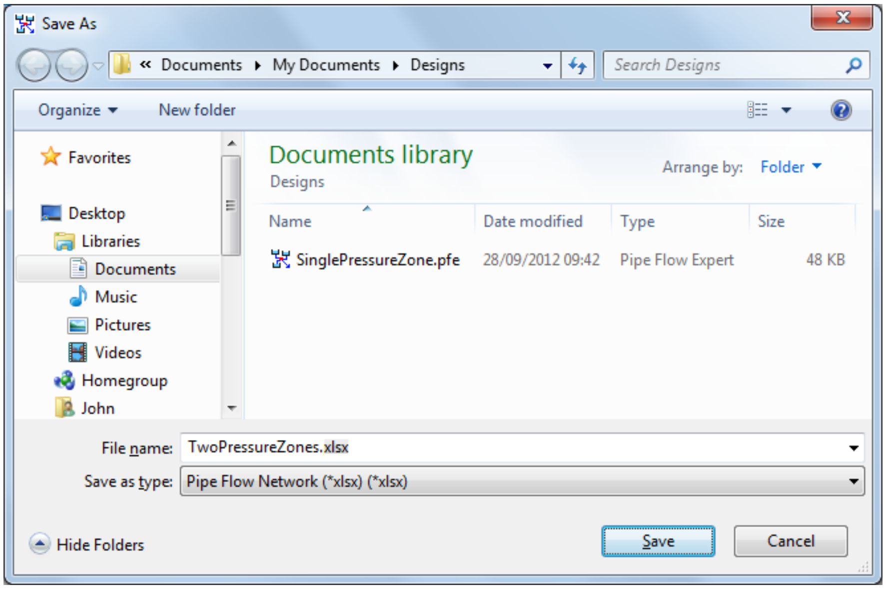


Figure 120 Save Results as Excel File dialog

4. Type a name for the file in the **File Name** field.

5. Select Pipe Flow Network (*xlsx) from the Save as type list.

6. Click **Save** to save the pipe system results from the selected tab.

Exporting the System Results

The values displayed in the Results Tables can be exported directly into Microsoft Excel. When you are exporting to Microsoft Excel from the Results Tables, only the values displayed on the currently selected tab are exported. To export all the result values, click the All Results tab before exporting.

To export a pipe system's results into a .xlsx file:

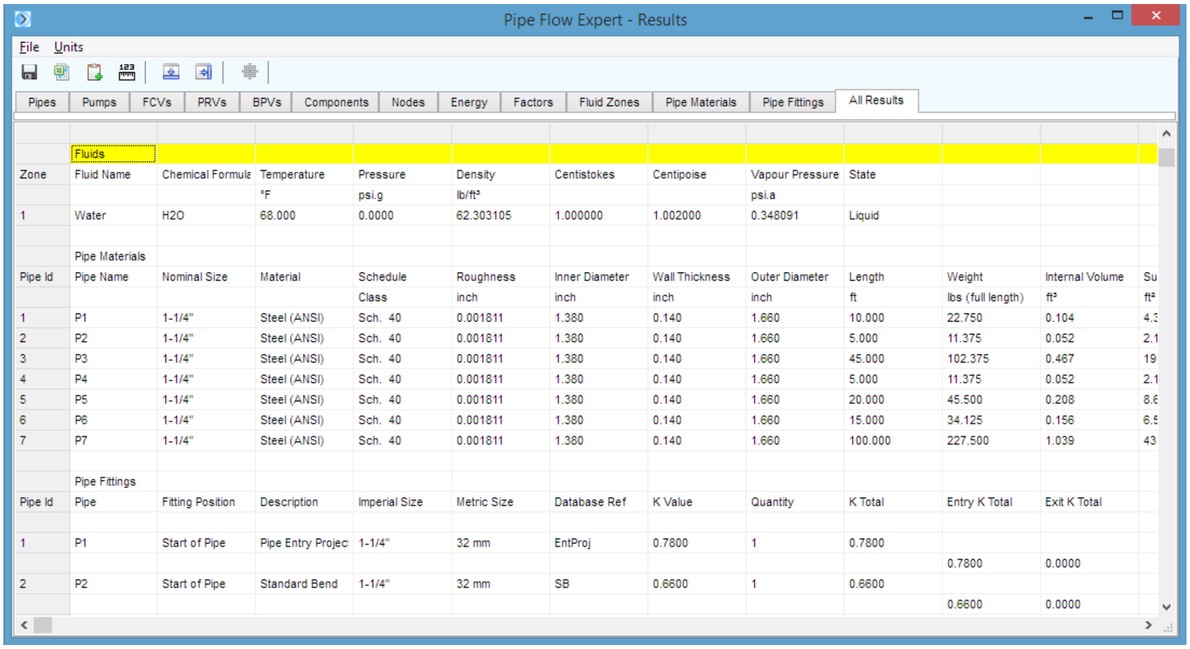
1. After a pipe system has been calculated and solved, click the **View Results Sheet** button on the **Results Log** dialog to open the **Results Tables** or close the results log and click the **Results Sheet** button on the tool bar.

2. Click the tab containing the values you want to export.

3. Click the Export to Excel button, , or select File | Export to Excel.

4. Clicking the **Export to Excel** button or menu selection opens the pipe system results from the selected tab in a worksheet in Microsoft Excel.

If Microsoft Excel is not available the results can be copied to the Windows clipboard, so that the data can be pasted into other spreadsheet applications.



	Fluids											
Zone	Fluid Name	Chemical Formula	Temperature	Pressure	Density	Centistokes	Centipoise	Vapour Pressure	State			
			°F	psi.g	lb/ft³			psi.a				
1	Water	H2O	68.000	0.0000	62.303105	1.000000	1.002000	0.348091	Liquid			
	Pipe Materials											
Pipe Id	Pipe Name	Nominal Size	Material	Schedule	Roughness	Inner Diameter	Wall Thickness	Outer Diameter	Length	Weight	Internal Volume	Su
				Class	inch	inch	inch	inch	ft	lbs (full length)	ft³	ft²
1	P1	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	10.000	22.750	0.104	4.3
2	P2	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	5.000	11.375	0.052	2.1
3	P3	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	45.000	102.375	0.467	19
4	P4	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	5.000	11.375	0.052	2.1
5	P5	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	20.000	45.500	0.208	8.6
6	P6	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	15.000	34.125	0.156	6.5
7	P7	1-1/4"	Steel (ANSI)	Sch. 40	0.001811	1.380	0.140	1.660	100.000	227.500	1.039	43
	Pipe Fittings											
Pipe Id	Pipe	Fitting Position	Description	Imperial Size	Metric Size	Database Ref	K Value	Quantity	K Total	Entry K Total	Exit K Total	
1	P1	Start of Pipe	Pipe Entry Projec	1-1/4"	32 mm	EntProj	0.7800	1	0.7800			
										0.7800	0.0000	
2	P2	Start of Pipe	Standard Bend	1-1/4"	32 mm	SB	0.6600	1	0.6600			
										0.6600	0.0000	

Figure 121 pipe system results in Microsoft Excel

Redesigning the System

After a pipe system is solved, the pipe system is read-only in the Drawing pane. If you decide that you want to modify a solved pipe system, you can return to the Drawing pane in design mode by clicking the Re-Design button on the toolbar.

If you want to retain the original solved pipe system and the calculated values for the system before you redesign the pipe system, be sure to save the pipe system and results before clicking the Re Design button.

When a pipe system cannot be solved, additional information is displayed in the Results Log and the message, '*The network was not solved. See log details above.'*, is shown. Clicking OK on the Results Log automatically returns you to the Drawing pane in design mode.

Amending the System

Making amendments to your system usually involves changing the diameter, length or material of a pipe or group of pipes. It may also involve changing the node elevations where the pipes are linked.

Amendments can be made in a number of ways:

a. Use the Edit Grid to display, filter and edit the data for Pipes, Nodes, and other items.

b. Click on the pipe or node and update the entries shown on the Pipe Pane or the Node Pane.

c. Set the Default Pipe data, select a group of items to be amended, or select a collection of individual items and then copy some or all of the Default Pipe attributes to the selected items.

These amendment options are described in the 'System Amendments and Group Updates' section of this document.

Create a PDF Report of the System Results

When a system has been solved successfully a customized PDF report can be created.

The report can contain a cover sheet with your logo, a scalable results drawing for high resolution printing and a selection of tabulated results data, which provides a detailed breakdown of the calculated results.

Note: First configure the results drawing labelling and the results units to be used, as described in the section 'Configuring the System Results'.

To display the Create PDF Report options dialog choose one of these actions:

- Click the 'Create PDF Report' button on the Results Log (after solving a system) OR
- Click the 'Results PDF' Tool Button in the tool bar at the top of the screen OR
- Click the 'Create PDF – Customized Report' option from the File menu

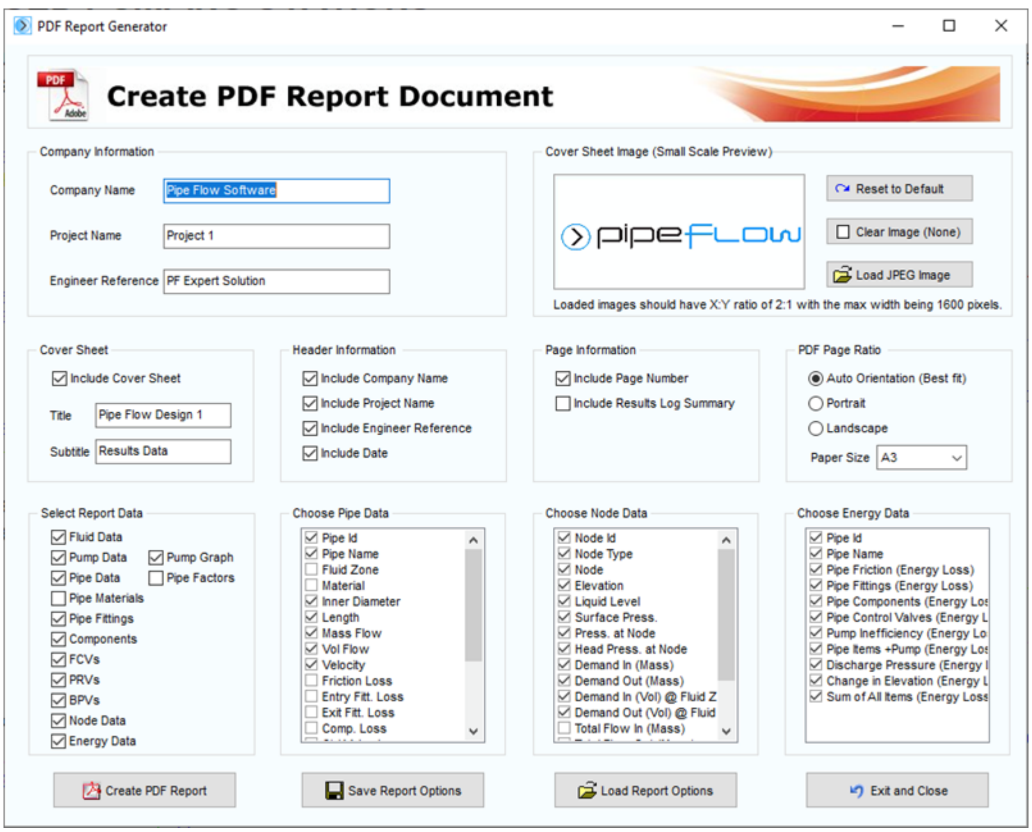


Figure 122 Create PDF Report Options

Choose the options for the PDF Report by selecting and checking the appropriate items from the PDF Report Generator screen.

The PDF report can include:

- A Cover Sheet including Company Logo, Title and Sub-Title.
- Page Header Information including Company Name, Project Name and Engineer Reference
- Page Information including page number
- Tabulated Results for Fluid Data

- Tabulated Results for Pump Data
- Tabulated Results for Pipe Data with up to 12 selected data items for each pipe
- Tabulated Results for Node Data with up to 12 selected data items for each node
- Tabulated Results for Energy Data
- Results Log Summary (if option selected)

The company information, the cover sheet image, and all the other selected options can be saved for future use. Click the ‘Save Report Options’ button to display the ‘Save As’ dialog.

Previously saved report options can be reloaded. Click the ‘Load Report Options’ button to display the ‘Open dialog.

Figure 123 PDF Results Report Pages

To Create a PDF Report Document:

1. Enter the Company Information.
2. Load a cover sheet image.
3. Select the Cover Sheet, Header Information and Page Information options.
4. Select the Report Data options.
5. Choose the Pipe Data, Node Data and Energy Data to be included.
6. Click the Create PDF Report button to generate a PDF results document

The generated PDF report file will be displayed in your PDF Adobe Reader program.
The PDF file will be automatically stored in the user’s local temporary folder, however the user may wish to save a copy of this document to a different location using the ‘Save a Copy’ option from Adobe Reader file menu.

Calculation Theory and Method of Solution

The Pipe Flow Expert program will allow you to draw a complex pipeline system and analyze the features of the system when flow is occurring. Pipe Flow Expert calculates the balanced steady flow and pressure conditions of the system. The reported results include the flow rates for each pipe, the fluid velocities for each pipe, Reynolds numbers, friction factors, friction pressure loss for each pipe, fittings, pressures losses, pressure at join points (nodes), head pressure (grade line), pump operating points and more.

The pipeline system is modeled by drawing the join points and the connecting pipes on a drawing pane. Horizontal, vertical or sloping lines can be used to connect one node to another node.

The physical data about the system is then entered, and typically this includes:

- The internal size, internal roughness and length of each connecting pipe.
- The elevation of each node (join point).
- The In-flow and the Out-flow at each node (if applicable).
- The elevation, liquid level and surface pressure data for each tank.
- The performance data for each pump.

Fluid Flow States

Fluids in motion are subjected to various resistance forces, which are due to friction. Friction may occur between the fluid and the pipe work, but friction also occurs within the fluid as sliding between adjacent layers of fluid takes place.

The friction within the fluid is due to the fluid's viscosity. When fluids have a high viscosity, the speed of flow tends to be low, and resistance to flow becomes almost totally dependent on the viscosity of the fluid. This condition is known as 'Laminar flow'.

Fluids which have a low viscosity are usually moved at higher velocities. The flow characteristics change, small eddy currents occur within the flow stream, and the friction between the pipe work and the fluid becomes a factor to be considered. This type of flow is known as 'Turbulent flow'.

Fluid Viscosity

A fluid viscosity can be described by its Dynamic viscosity (sometimes called Absolute viscosity), or it’s Kinematic viscosity. These two expressions of viscosity are not the same, but are linked via the fluid density.

Kinematic viscosity = Dynamic viscosity / fluid density

Dynamic Viscosity
Water @ 20°C has a viscosity of 1.00 x 10^{-3} Pa • s or 1.00 Centipoise
Water @ 70°F has a viscosity of 2.04 x 10^{-5} lb •s/ft^2

Kinematic Viscosity
Water @ 20°C has a viscosity of 1.004 x 10^{-6} m^2 /s or 1.004000 Centistokes
Water @ 70°F has a viscosity of 10.5900 x 10^{-6} ft^2 /s

Pipe Flow Expert has a database of viscosities and densities for common fluids.

Reynolds Numbers

Reynolds numbers (Re) describe the relationship between a fluid's velocity, the internal pipe diameter and the fluid's Kinematic viscosity.

Reynolds number = Fluid velocity x Internal pipe diameter / Kinematic viscosity

Note: Kinematic viscosity (not Dynamic viscosity) must be used to calculate Reynolds numbers.
It is generally accepted that the 'changeover' point between laminar flow and turbulent flow, in a circular pipe, occurs when the Reynolds number (Re) is approximately 2100.

i.e. Laminar flow occurs when the Re is less than 2100. Turbulent flow occurs when the Re is greater than 4000. Between the Laminar and Turbulent flow conditions the flow may be neither wholly laminar nor wholly turbulent. In this transition region there is no exact equation to calculate the friction factor and therefore only an approximate friction factor can be used by way of appropriate interpolation.

Friction Factors

Many formulas have been developed to model the flow of fluids. The Hazen-Williams formula has been a popular method of estimating the head loss in piping systems for a number of years. However, this empirical formula will only give reasonable accuracy if the fluid is water at 60°F or similar. The Hazen-Williams formula is therefore not useful in analyzing a complex pipe system.

The Colebrook-White formula may be used with confidence to calculate an accurate friction factor applicable to the turbulent flow of fluids. The Colebrook-White formula is applicable over a whole range of fluid densities and viscosities, provided that the fluid flow is turbulent.

Colebrook-White Formula

The Colebrook-White formula:

$1/\text{sqrt}(f) = 1.14 – 2 \log_{10} [e/D + 9.35/(Re \text{ x } \text{sqrt}(f))]$

f = friction factor

e = internal roughness of pipe

D = internal diameter of pipe

Re = Reynolds number

Friction factors for turbulent flow calculated by Pipe Flow Expert are based on the Colebrook-White formula.

The friction factor for Laminar flow is calculated from *f* = 64/Re

Friction Losses (resistance to flow)

The resistance to fluid flow is usually expressed in fluid head. This is the height of a column of fluid which would exert enough pressure on the fluid at the bottom of the column to make the fluid flow within the system.

If the level of fluid (fluid head) is increased in a supply container, the volume of fluid entering the system from the supply container will increase due to the increase in pressure (force).

Darcy-Weisbach Formula

Fluid head resistance can be calculated by using the Darcy-Weisbach formula.

$h \text{ fluid} = f\,(L/D) \times (v^2/2g)$

f = friction factor
L = length of pipe work
D = inner diameter of pipe work
v = velocity of fluid
g = acceleration due to gravity

Fluid head loss calculated by Pipe Flow Expert when using the non-compressible calculation engine is based on the Darcy-Weisbach formula.

Compressible Gas Flow Equations

Friction loss for a compressible fluid such as a gas needs to account for the change in density of the gas as it moves along the pipe and loss of pressure occurs. When using the compressible flow calculation engine Pipe Flow Expert can be set to use one of the following compressible flow equations:

- General Fundamental Isothermal Flow Equation
- Complete Isothermal Flow Equation
- AGA Isothermal Flow Equation
- Panhandle A Isothermal Flow Equation
- Panhandle B Isothermal Flow Equation
- IGT Isothermal Flow Equation
- Weymouth Isothermal Flow Equation

The formula for each of the above equations is detailed in a separate PDF document named ‘Compressible Flow Equations’. This PDF is available from the ‘Documentation’ menu inside the Pipe Flow Expert software.

The compressible flow equations calculate a gas flow rate for a difference in pressure between two points. The Pipe Flow Expert software first solves the compressible network using non-compressible flow equations to find an approximate solution and then it employs the compressible flow engine to converge this to an accurate solution.

The non-compressible calculation engine operates by adjusting flows within the network to achieve a pressure balance, whilst the switch to the compressible flow calculation engine operates by adjusting node pressures to achieve a flow balance within the pipe network.

The compressible calculation engine takes account of friction loss through fittings at the start and end of a pipe, where a lower pressure but higher velocity at the end of pipe will results in a higher pressure loss through the fitting then if the same fitting occurred at the start of the pipe.

In addition the compressible flow calculation engine handles complexities that do not occur for non-compressible calculations, such as when considering a fan or blower with a flow versus head curve, where the actual operating point (flow and head value) has to balance out even when taking account of the compression and then expansion of the gas through the pipe.

Fitting Head Loss

The fluid head resistance through various pipe work fittings can be calculated if the 'K' factor of the fitting is known. Many manufacturers of pipe work fittings and valves publish 'K' factors for their products.

’K’ Factor fitting head loss calculation

Fluid head loss through fitting and valves can be calculated from:

h_{fluid} = 'K' x v^2 / 2g

'K' = manufacturer’s published 'K' factor for the fitting
v = velocity of fluid
g = acceleration due to gravity

In many systems where pipe lengths are relatively long, the effect of the fitting losses may be considered to be minor losses, and could be ignored during initial assessment.

If a partially open valve is part of the design, the effect of the valve should always be considered as the valve loss may be large.

Pipe Flow Expert has a database of valve and fittings ‘K’ factors and calculation wizards for:

- gradual enlargements
- gradual contractions
- sudden enlargements
- sudden contractions
- rounded entrances
- long pipe bends

For further information on this subject please refer to ‘Flow of Fluids through valves, fittings and pipe’ - Crane Technical Paper No. 410

Calculate Total Pressure Loss

The fluid head resistance can also be expressed a pressure loss.

Metric units: bar = h_{fluid} x p x g / 100000

h = head loss (m)
p = fluid density (kg/m^3)
g = acceleration due to gravity (m/s^2)

Imperial units: psi = h_{fluid} x SG x 2.311

h = head loss (ft)
SG = specific gravity of the fluid

Energy and Hydraulic Grade Lines

The Energy Grade Line is a plot of the sum of the three terms from the Work-Energy equation or Bernoulli theorem:

EGL (Total Fluid Head) = Velocity Head + Pressure Head + Elevation head

$EGL = v^2/2g + P/y + elevation$

v = velocity
g = acceleration due to gravity
P = pressure
y = fluid density

The Hydraulic Grade Line is a plot of the sum of two terms from the Work-Energy equation or Bernoulli theorem, namely the Pressure head and Elevation head:

HGL (Total Fluid Head – Velocity Head) = Pressure Head + Elevation head

$HGL = P/y + elevation$

P = pressure
y = fluid density

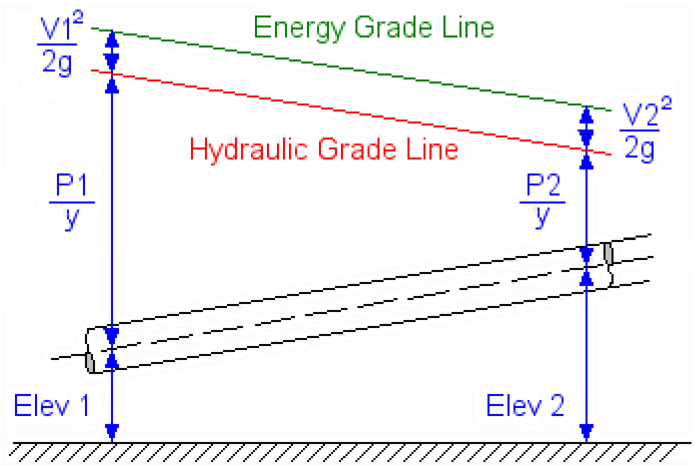


Figure 124 Hydraulic Grade Line

Balanced Flow State

Network analysis is based on two major principles:

Mass flow into a joint in the network must be equal to mass flow out of the joint. Assuming that the fluid density does not change, the total flow rate entering the node must equal the total flow rate leaving the node.

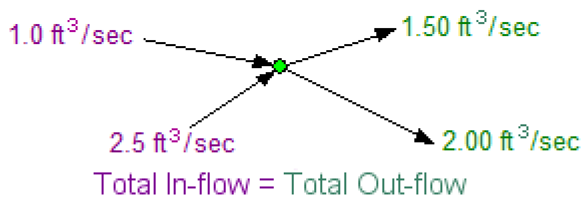


Figure 125 Flow balance at a join point

The pressure drop in the pipes around a loop must be equal to zero. In a recirculation loop the pressure loss in each pipe must be the total of the fluid head energy being provided by the driving force (normally a pump). In a loop where the flows are in opposite directions, the total pressure loss in each pipe with a clockwise flow must equal the total pressure loss in each pipe with an anticlockwise flow.

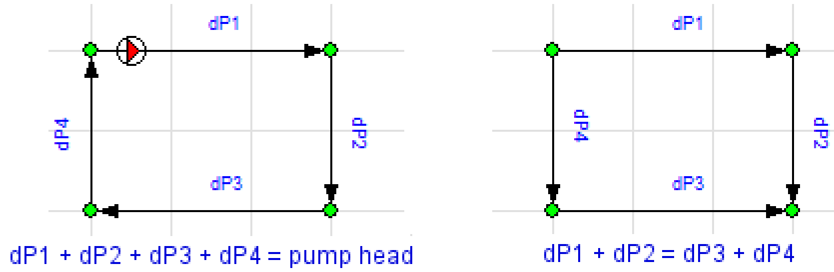


Figure 126 Pressure balance around a loop

Loops, Nodes and Pipes

To solve the unknowns in any pipeline network, a system of nonlinear equations must be developed to mathematically describe the pipeline network in terms of the flows and pressures that must balance at specific points within the system. These equations must be solved simultaneously in order to find a balanced solution.

Loops within a pipeline network consist of a series of connected pipes that return to the start point (the node where the loop started), however pseudo loops can exist between a supply or discharge source, between two supply sources or between two discharge sources.

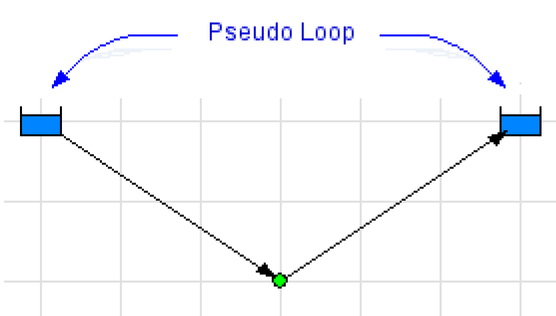


Figure 127 Pseudo Loop

A series of pressure equations describing the pressure loss in each loop must be generated. The same pipe may be included in many different loops.

Where a loop includes a number of different fluid zones the elevation differences of the nodes and the density of the fluid in each pipe along the path in the loop must be used to establish the difference in pressures due to fluid density at the end points of the loop. This pressure difference is used together with the fluid surface pressure difference to establish the net pressure difference around the loop.

Pressure losses and pressure gains within each pipe must take account of changes in elevation, fittings, control valves (including FCVs, PRVs, & BPVs), components (including valves defined with Cv, Kv, & Sprinkler K values, and different types of Orifices), and pumps.

A series of mass flow continuity equations must be generated for the mass flow rate balance at each node in the pipeline system.

The Pipe Flow Expert software will automatically analyze a system, identify all loops and pseudo-loops, generate the pressure equations throughout the network and remove redundant loop equations (those which are already fully described by combining other equations). It will also generate the mass flow balance equations for each node in a pipeline system.

Solving the Flow versus Pressure Loss Balance

Pipe Flow Expert generates initial estimates for the flow rate in each pipe. The pressure losses within the system are then calculated using friction factors obtained from the Colebrook-White equation, and the friction pressure loss for each pipe is obtained from the Darcy-Weisbach equation.

The initial flow estimates are unlikely to give a balanced pressure result over the whole system and these values must be iteratively adjusted using a variation on the Newton method for complex systems, in order to converge to a final result where all of the flow rates and pressures within the system are balanced.

The Pipe Flow Expert software defines the elements of the pipeline system in a series of highly non-linear matrix equations. Once an approximate solution has been obtained, the initial results are refined using developed mathematically techniques and proprietary algorithms, to ensure the final results converge to a balanced flow and pressure result.

Note: Finding the solution to a highly non-linear system of equations (which a complex pipe system inevitably generates) is a difficult mathematical problem. The Pipe Flow Expert software uses advanced mathematical methods together with novel algorithms to converge the equations to a solution.

For compressible gas systems, the algorithms are extended further, to switch to iterative adjustment of nodes pressures, once the solution using equations as described above has been found. This allows the Pipe Flow Expert software to use compressible gas flow equations to calculate flow rate between two pressure conditions while taking account of density changes that occur with any compression or expansion of the gas as it travels down the pipe.

System Calculation Tolerances

Pipe Flow Expert uses a two-step approach to calculate a balanced steady flow state for a system.

An initial approximate solution is obtained using Linear Theory methods and an iterative approach to adjust the flow rates until an approximate pressure balance is achieved. This approach provides estimates for the initial flow rate in each pipe, and these are then used as a starting point to move towards a more accurate and final solution. The initial flow estimates are unlikely to give an accurate pressure result over the whole system and they must be further refined to adjust the flow rates until a final pressure balance is achieved.

Pipe Flow Expert defines the elements of the pipeline system in a series of matrix equations and uses variations on the Newton method to adjust the estimates for the flow rate in each pipe.

Once an approximate solution has been obtained, the results are refined using a variation of the Newton method to ensure convergence until a balanced pressure result is obtained.

A flow balance tolerance and an approximate pressure balance tolerance are used to allow Pipe Flow Expert to find an approximate solution quickly. This approximate solution is then refined until the system is solved within the final pressure balance tolerance.

The flow and pressure balance tolerances and the maximum number of iterations allowed, have been carefully chosen to provide the best overall performance in finding a solution. It is recommended that these values are not changed, as most systems will solve without adjustment to these parameters.

System Components

The user is able to enter the flow rate and head loss information about a component which is to be included in the pipeline system. Pipe Flow Expert generates a performance curve for the component to allow the effect of the component to be modeled.

Cv and Kv Flow Coefficients

Many manufacturers of control valves choose to publish a Cv flow coefficient or a Kv flow coefficient to describe the flow / pressure loss characteristics of their control valves in a standardized manner.

Cv Flow Coefficients:

A Cv flow coefficient specifies the amount of water at 60°F (15.55 °C) in US gpm that will flow through a valve and produce a 1.0 psi pressure drop. Thus a Cv flow coefficient of 10 indicates that a 1.0 psi pressure drop will occur with a 10 US gpm of water throughput through the valve.

The Cv flow coefficient of a control valve can be calculated from the flow rate and the pressure drop through the valve. For liquids other than water the ratio of the fluid density to the density of water must also be used in the calculation.

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

Where:
Cv = flow coefficient
Q = flow rate in US gpm
ΔP = pressure loss in psi across the valve
SG = the ratio of the fluid density to the density of water

With a known Cv flow coefficient the above formula can be re-arranged to calculate the pressure loss for a particular flow rate thus:

$$\Delta p = \frac{SG \times Q^2}{C_v^2}$$

Where:
Cv = flow coefficient
Q = flow rate in US gpm
ΔP = pressure loss in psi across the valve
SG = the ratio of the fluid density to the density of water

The pressure loss through a fitting or valve may also be calculated from:

$$h_{fluid} = \frac{K \times V^2}{2g}$$

Where:
h_{fluid} = head of fluid in ft.
K = flow coefficient of a valve or fitting
V = fluid velocity entering the fitting in ft/s
g = acceleration due to gravity in ft/sec^2

also

$$p = \frac{h_{fluid} \times D}{144}$$

Where:
P = pressure in psi
h_{fluid} = head of fluid in ft.
D = density of fluid in lbs/ft^3

Thus a 1.0 psi pressure drop is equivalent to 2.31 ft head of water at Normal Temperature & Pressure (NTP) or 62.3 lbs/ft^3.

When a pipe diameter is known it is possible to establish a flow velocity from the Cv flow coefficient in US gpm for a 1 psi pressure drop. Thus it is possible to calculate an equivalent fitting ‘K’ factor which will produce the same pressure loss as the control valve Cv rating.

Pipe Flow Expert uses the equivalent fitting ‘K’ factor method to model the flow and pressure loss through a control valve where a Cv flow coefficient is used to specify the control valve characteristics.

A change to the pipe diameter would result in a change to the value of the equivalent fitting ‘K’ factor.

Pipe Flow Expert re-calculates the equivalent fitting 'K' factor for the current pipe diameter and the fluid density at the start of the solution calculation.

The calculation helper provided on the Cv component dialog uses the flow rate and pressure loss entered by the user, together with the current fluid density to calculate a Cv value to match the specified requirements.

Kv = 0.865 Cv (or more accurately Kv = 0.86497767 Cv)

Kv Flow Coefficients:

A Kv flow coefficient specifies the amount of water at 20°C (68 °F) in m^3/hour that will flow through a valve and produce a 1.0 bar pressure drop. Thus a Kv flow coefficient of 10 indicates that a 1.0 bar pressure drop will occur with a 10 m^3/hour of water throughput through the valve.

The Kv flow coefficient of a control valve can be calculated from the flow rate and the pressure drop through the valve. The density of the liquid in kg/m^3 must also be used in the calculation.

$$K_V = Q\sqrt{\frac{D}{1000 \times \Delta p}}$$

Where:
Kv = flow coefficient
Q = flow rate in m^3/hr
ΔP = pressure loss in bar across the valve
D = the density of the fluid in kg/m^3
1000 = the density of water in kg/m^3

The usual arrangement of the formula for calculation of Kv is shown above.
It can be seen that this formula is similar to the one which is used for calculation of Cv values.

With a known Kv flow coefficient, the above formula can be re-arranged to calculate the pressure loss for a particular flow rate thus:

$$\Delta p = \frac{D \times Q^2}{1000 \times K_v^2}$$

Where:
Kv = flow coefficient
Q = flow rate in m^3/hr
ΔP = pressure loss in bar across the valve
D = the density of the fluid in kg/m^3
1000 = the density of water in kg/m^3

The pressure loss through a fitting or valve may also be calculated from:

$$h_{fluid} = \frac{K \times V^2}{2g}$$

Where:
h_{fluid} = head of fluid in meters.
K = flow coefficient of a valve or fitting
V = fluid velocity entering the fitting in m/s
g = acceleration due to gravity in meters/sec^2

also

$$p = \frac{h_{fluid} \times D \times g}{100000}$$

Where:
P = pressure in bar
h_{fluid} = head of fluid in meters
D = density of fluid in kg/m^3
g = acceleration due to gravity in meters/sec^2

When a pipe diameter is known it is possible to establish a flow velocity from the Kv flow coefficient in m^3/hr for a 1 bar pressure drop. Thus it is possible to calculate an equivalent fitting 'K' factor which will produce the same pressure loss as the control valve Kv rating.

Pipe Flow Expert uses the equivalent fitting 'K' factor method to model the flow and pressure loss through a control valve where a Kv flow coefficient is used to specify the control valve characteristics.

A change to the pipe diameter would result in a change the matching equivalent fitting 'K' factor. Pipe Flow Expert re-calculates the equivalent fitting 'K' factor for the current pipe diameter and the fluid density at the start of the solution calculation.

The calculation helper provided on the Kv component dialog uses the flow rate and pressure loss entered by the user, together with the current fluid density to calculate a Kv value to match the specified requirements.

1.000 Cv = 1.156 Kv

The user should be aware that the Cv or Kv flow coefficient specifies the flow rate of water for a particular pressure loss.

When the fluid density is greater or less than water, a different flow rate will be required to produce a 1.00 psi or a 1 bar pressure loss through the valve.

CAUTIONS:

Choked Flow:

If the fluid is a gas, when using the Compressible Flow calculation engine, if the pressure drop exceeds the critical pressure ratio when compared to the inlet pressure of the valve then **the flow will become choked and it will not be possible to achieve the calculated flow rate**. If this occurs, Pipe Flow Expert will warn of a problem in the Result Log (if the Component option to check for choked flow has been selected).

Gas Flow Calculation with the Compressible Flow Calculation Engine (recommend):

Gas systems should generally be solved using the Compressible flow calculation engine, which takes account of the pressure condition at the component and adjusts the density of the gas as appropriate when performing the component pressure loss calculation.

The fluid properties should be defined at the operating temperature for a particular section of the system (while the software adjusts fluid properties for changes of pressure as they are used in calculations, it assumes the temperature of the fluid remains the same and therefore the user must define the fluid properties for the appropriate temperature conditions within the pipe network). The software uses the viscosity of the gas as defined in the current fluid data and this is not adjusted for changes in pressure (pressure changes generally have a small impact on viscosity in comparison to temperature changes). See the section on *Working with Compressible Fluids* for more information and notes about calculating gas system with the non-compressible calculation engine (even though we recommend you ***use the Compressible Flow calculation engine to solve gas systems***).

Gas Flow Calculation with the Non-Compressible Flow Calculation Engine (not recommended):

If using the Non-Compressible calculation engine, and the fluid zone associated with the control valve does not represent the pressure condition at the outlet of the valve/component, it may be necessary to use an adjusted Cv (or Kv) value for valve selection to take in to account the effect of the gas expansion.

The adjusted Cv (or Kv) value should be based on the Cv or Kv formula for sub critical gas pressure drop.

A simplified version of the Cv formula for sub critical gas pressure drop is shown below:

$$C_V = \frac{SCFH}{1360} \sqrt{\frac{D_n \times (^{\circ}F + 460)}{\Delta p \times p_o}}$$

Where:
Cv = flow coefficient
SCFH = flow rate in ft^3/hr (NTP)
D_n = the gas density in lbs/ft^3 at 0.00 psig
$^{\circ}F$ = gas temperature
Δp = pressure loss in psi absolute
p_o = valve outlet pressure in psi absolute

A simplified version of the Kv formula for sub critical gas pressure drop is shown below:

$$K_V = \frac{Q}{514} \sqrt{\frac{D_n \times (^{\circ}C + 273)}{\Delta p \times p_o}}$$

Where:
Kv = flow coefficient
Q = flow rate in m^3/hr (NTP)
D_n = the gas density in kg/m^3 at 0.00 barg

^{o}C = gas temperature
Δp = pressure loss in bar absolute
p_o = valve outlet pressure in bar absolute

Please refer to an appropriate text book for a more detailed formula to take account of piping geometry or gas compressibility, should this be necessary.

Cv / Kv Control valve selection:

The Cv (or Kv) flow coefficient of a control valve is usually stated for the fully open flow condition. The Cv (or Kv) flow coefficient will be less when the valve is partly closed.

In an actual system it is important to select a control valve which has an appropriate Cv (or Kv) flow coefficient for the actual valve position that will be used. A control valve that is too small or too large will never be able to provide the correct control in a system.

Most control valve manufacturers recommend that you should select a valve where the required Cv (or Kv) value falls between 20% - 80% of the port opening.
Some control valve manufacturers recommend that an allowance of 30% should be added to the calculated Cv (or Kv) flow coefficient to obtain the minimum full flow Cv (or Kv) flow coefficient rating which the selected valve should have (when fully open).

Please check your control valve selection with the control valve manufacturer.

Sprinkler K Value Coefficients

Many manufacturers of spray nozzles publish a Sprinkler K value to describe the flow / pressure loss characteristics of their spray nozzle in a standardized manner.

Sprinkler K Value Coefficient:

A sprinkler K value can describe the sprinkler performance in either imperial or metric flow rates (there is a metric K value and an imperial K value for the same sprinkler and these will be different values, hence it is important to check a sprinkler K value to confirm if it is a metric value or an imperial value).

The imperial sprinkler K value is calculated as follows:

$$Sprinkler\ K\ (imperial) = \frac{US\ gpm\ (water)}{\sqrt{Pressure\ Loss\ (psi)}}$$

The metric sprinkler K value is calculated as follows:

$$Sprinkler\ K\ (metric) = \frac{L/min\ (water)}{\sqrt{Pressure\ Loss\ (bar)}}$$

The imperial and metric Sprinkler K values are not interchangeable. When a Sprinkler K value is used for a flow versus pressure drop calculation, it is important to know if the value specified is based on the imperial formula or the metric formula, since this will affect the calculated results.

When a pipe diameter is known it is possible to establish a flow velocity from the Sprinkler K Value for a particular pressure drop and therefore it is possible to calculate an equivalent fitting ‘K’ factor which will produce the same pressure loss as the Sprinkler K Value.

Pipe Flow Expert uses the equivalent fitting ‘K’ factor method to model the flow and pressure loss through a spray nozzle where a Sprinkler K Value is used to specify the nozzle characteristics.

A change to the pipe diameter will result in a change to the matching equivalent fitting ‘K’ factor. Pipe Flow Expert re-calculates the equivalent fitting ‘K’ factor for the current pipe diameter using the fluid density of water.

Orifices Loss Coefficient

Orifices are widely installed in piping systems and hydraulic machinery to produce a regular and reproducible loss of pressure. Orifices are used to limit flow, or, in branching systems, to balance distributed flow, or to measure flow.

There are several different approaches and equations that are used in literature to estimate the pressure loss through an orifice for a specified flow rate. In many cases the methods do not treat the pressure loss calculation consistently, and common physical properties of the orifice are not used in a standard manner. In some cases, the pressure drop estimate relies on a discharge coefficient, and in other cases it is simply presented as graph plot of pressure drop versus flow rate (or pressure drop derived as a percentage of differential pressure across flow measurement).

The Pipe Flow Expert software orifice calculations use a more generalized model, which utilizes the broad physical attributes of the orifice to determine the flow characteristics. This generates a K_0 value (a k factor) that can be applied in a standard method as follows:

$$H_0 = K_0 \frac{{V_0}^2}{2g}$$

Where H_0 = Head Loss (in ft or m of fluid, depending on the units of velocity, ft/s or m/s)
K_0 = K factor (for the density and velocity of fluid flow through the orifice diameter)
g = acceleration due to gravity (in ft/s^2 or m/ s^2)

While this final equation provides a standard method for calculating the pressure drop through different types of orifice, deriving the K_0 factor and then applying it to orifice calculations in a complex pipe network (particularly in compressible gas systems where the density and velocity at the orifice inlet varies with pressure changes in the system), is a complex task, which also then relies on the development of further algorithms that allow convergence to balanced solution across the whole model.

The Pipe Flow Expert software handles the complexity of the calculations and makes it easy to perform calculations with orifices in pipes systems. The following sections detail the equations used to derived the orifice K_0 value.

Sharp-Edged Orifice in a Straight Pipe

A sharp-edged orifice in a straight pipe, where the upstream and downstream pipe sizes are the same, as illustrated in Figure 128.

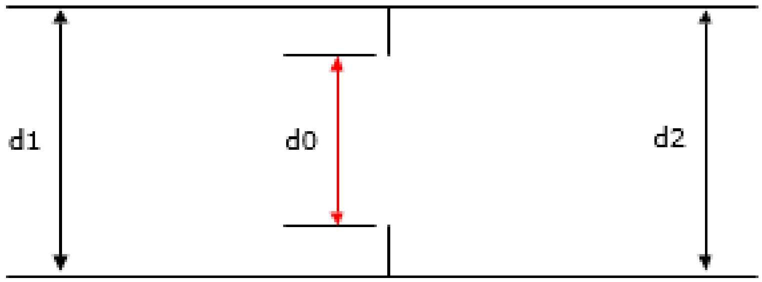


Figure 128 Sharp Edged Orifice in a Straight Pipe

The loss coefficient (K_0) of a straight edged orifice in a straight pipe is calculated as follows:

$$K_0 = 0.0696(1 - \beta^5)\lambda^2 + (\lambda - \beta^2)^2$$

Where the diameter ratio $\beta = d_0/d_1$ and the jet velocity λ is given by:

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5).$$

Sharp-Edged Orifice in a Transition Section

A sharp-edged orifice in a transition section, where the upstream and downstream pipe sizes are *not* the same, are illustrated in Figure 129 and Figure 130.

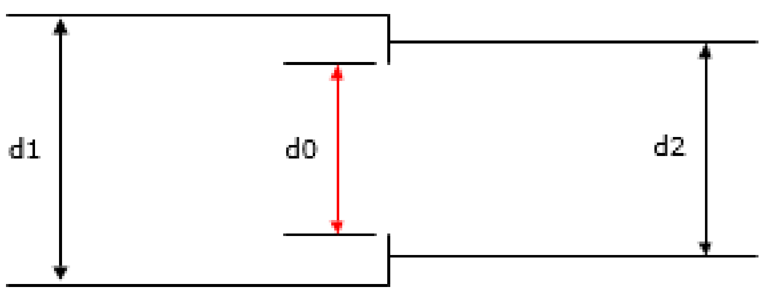


Figure 129 Sharp Edged Orifice in a Transition Section (Contraction)

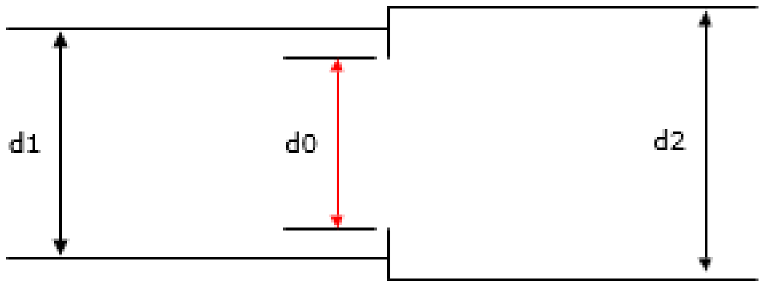


Figure 130 Sharp Edged Orifice in a Transition Section (Expansion)

The loss coefficient (K_0) of a sharp-edged orifice in a transition section is calculated as follows:

$$K_0 = 0.0696(1-\beta^5)\lambda^2 + \left(\lambda - \left(\frac{d_0}{d_2}\right)^2\right)^2$$

Where the diameter ratio $\beta = d_0/d_1$ and the jet velocity λ is given by:

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5).$$

Sharp-Edged Orifice Discharging to Atmosphere

A sharp-edged orifice that discharges to atmosphere is illustrated in Figure 131.

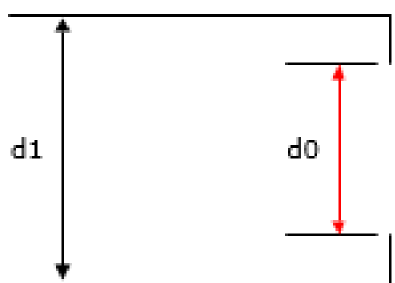


Figure 131 Sharp Edge Orifice Discharging to Atmosphere

For this orifice configuration, the loss coefficient (K_0) equation for a sharp-edged orifice in an expanding transition section can be used, however the downstream pipe diameter (d_2) is effectively infinite and therefore the equation can be transformed into:

$$K_0 = 0.0696(1-\beta^5)\lambda^2 + \lambda^2$$

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5)$$

Round Edged Orifice in a Straight Pipe

A round edged orifice in a straight pipe, where the upstream and downstream pipe sizes are the same, is illustrated in Figure 132.

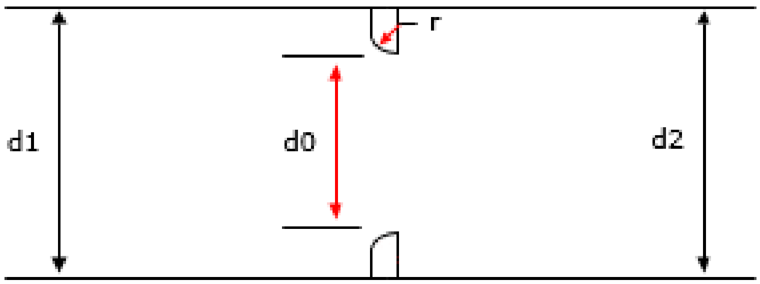


Figure 132 Round Edged Orifice in a Straight Pipe

The loss coefficient (K_0) of a round edged orifice in a straight pipe is calculated as follows:

$$K_0 = 0.0696\left(1 - 0.569\frac{r}{d_0}\right)\left(1 - \sqrt{\frac{r}{d_0}}\beta\right)(1-\beta^5)\lambda^2 + (\lambda - \beta^2)^2$$

$$\left(\frac{r}{d_0}\right) \leq 1$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left(1 - 0.30\sqrt{\frac{r}{d_0}} - 0.70\frac{r}{d_0}\right)^4 (1 - 0.215\beta^2 - 0.785\beta^5)$$

In the case of a generously rounded orifice where r/d_0 is equal to or greater than 1, the jet contraction ratio λ equals 1 and the loss coefficient becomes:

$$K_0 = 0.030(1-\beta)(1-\beta^5) + (1-\beta^2)^2$$

$$\left(\frac{r}{d_0}\right) \geq 1$$

Round Edged Orifice in a Transition Section

A round edged orifice in a transition section, where the upstream and downstream pipe sizes are *not* the same, are illustrated in Figure 133 and Figure 134.

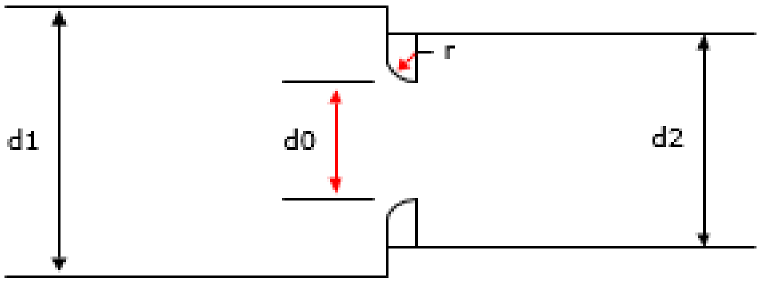


Figure 133 Round Edged Orifice in a Transition Section (Contraction)

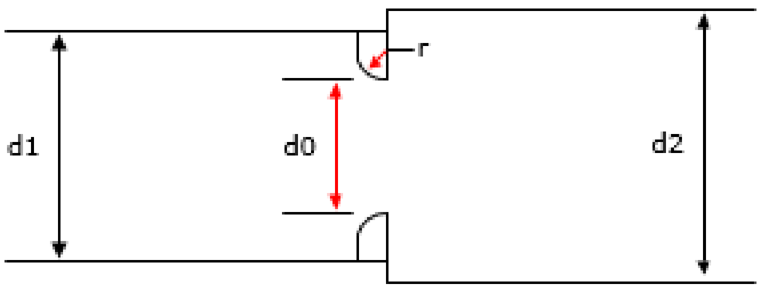


Figure 134 Round Edged Orifice in a Transition Section (Expansion)

The loss coefficient (K_0) of a round edged orifice in a transition section is calculated as follows:

$$K_0 = 0.0696\left(1 - 0.569\frac{r}{d_0}\right)\left(1 - \sqrt{\frac{r}{d_0}}\beta\right)(1-\beta^5)\lambda^2 + \left(\lambda - \left(\frac{d_0}{d_2}\right)^2\right)^2$$

$$\left(\frac{r}{d_0}\right) \le 1$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left(1 - 0.30\sqrt{\frac{r}{d_0}} - 0.70\frac{r}{d_0}\right)^4 (1 - 0.215\beta^2 - 0.785\beta^5)$$

In the case of a generously rounded orifice where r/d_0 is equal to or greater than 1, the jet contraction ratio λ equals 1 and the loss coefficient becomes:

$$K_0 = 0.030(1-\beta)(1-\beta^5) + \left[1 - \left(\frac{d_0}{d_2}\right)^2\right]^2$$

$$\left(\frac{r}{d_0}\right) \ge 1$$

Round Edged Orifice Discharging To Atmosphere

A round edged orifice that discharges to atmosphere is illustrated in Figure 135.

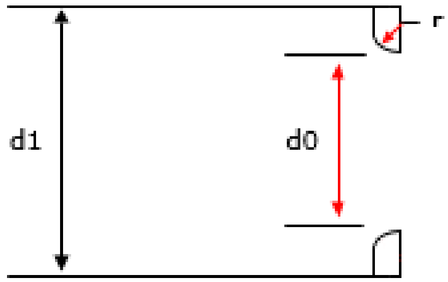


Figure 135 Round Edged Orifice Discharging to Atmosphere

For this orifice configuration, the loss coefficient (K_0) equation for a round edged orifice in an expanding transition section can be used, however the downstream pipe diameter (d_2) is effectively infinite and therefore the equation can be transformed into:

$$K_0 = 0.0696\left(1 - 0.569\frac{r}{d_0}\right)\left(1 - \sqrt{\frac{r}{d_0}}\beta\right)(1 - \beta^5)\lambda^2 + \lambda^2$$

where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left(1 - 0.30\sqrt{\frac{r}{d_0}} - 0.70\frac{r}{d_0}\right)^4 (1 - 0.215\beta^2 - 0.785\beta^5)$$

$$\left(\frac{r}{d_0}\right) \leq 1$$

Bevel Edged Orifice in a Straight Pipe

A bevel edged orifice in a straight pipe, where the upstream and downstream pipe sizes are the same, is illustrated in Figure 136.

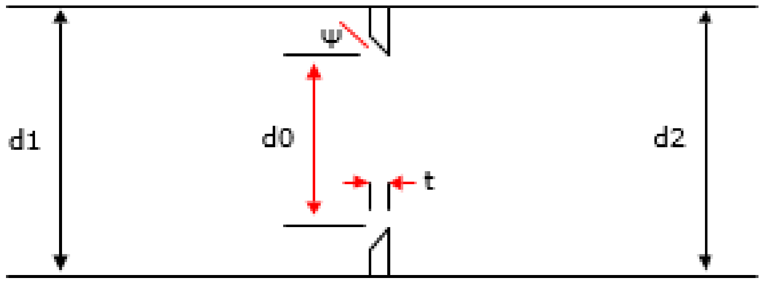


Figure 136 Bevel Edged Orifice in a Straight Pipe

The loss coefficient (K_0) of a bevel edged orifice in a straight pipe is calculated as follows:

$$K_0 = 0.0696\left(1 - C_b\frac{t}{d_0}\right)\left(1 - 0.42\sqrt{\frac{t}{d_0}}\beta^2\right)(1-\beta^5)\lambda^2 + (\lambda - \beta^2)^2$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left[1 - C_b\left(\frac{t}{d_0}\right)^{\frac{1-\sqrt[4]{\frac{t}{d_0}}}{2}}\right](1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_b is a function of bevel angle ψ in degrees, and bevel thickness to diameter ratio t/d_0 is given by:

$$C_b = \left(1 - \frac{\psi}{90}\right)\left(\frac{\psi}{90}\right)^{\frac{1}{2+\frac{t}{d_0}}}$$

Bevel Edged Orifice in a Transition Section

A bevel edged orifice in a transition section, where the upstream and downstream pipe sizes are *not* the same, are illustrated in Figure 137 and Figure 138.

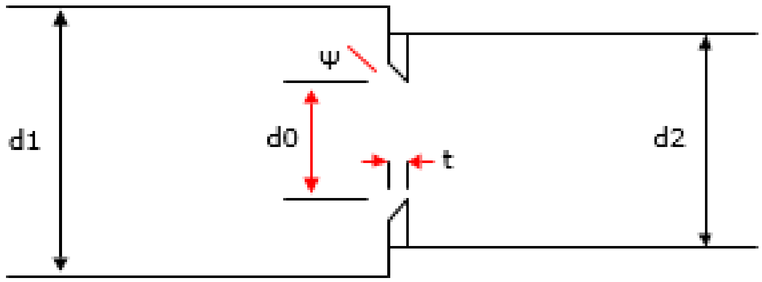


Figure 137 Bevel Edged Orifice in a Transition Section (Contraction)

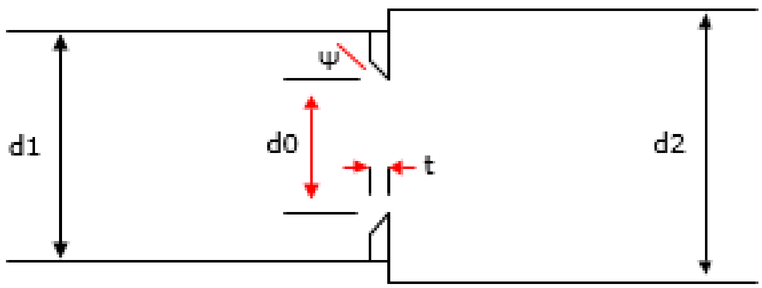


Figure 138 Bevel Edged Orifice in a Transition Section (Expansion)

The loss coefficient (K_0) of a bevel edged orifice in a transition section is calculated as follows:

$$K_0 = 0.0696\left(1 - C_b\frac{t}{d_0}\right)\left(1 - 0.42\sqrt{\frac{t}{d_0}}\beta^2\right)(1-\beta^5)\lambda^2 + \left(\lambda - \left(\frac{d_0}{d_2}\right)^2\right)^2$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left[1 - C_b\left(\frac{t}{d_0}\right)^{\frac{1-\sqrt[4]{\frac{t}{d_0}}}{2}}\right](1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_b is a function of bevel angle ψ in degrees, and bevel thickness to diameter ratio t/d_0 is given by:

$$C_b = \left(1 - \frac{\psi}{90}\right)\left(\frac{\psi}{90}\right)^{\frac{1}{2+\frac{t}{d_0}}}$$

Bevel Edged Orifice Discharging To Atmosphere

A bevel edged orifice that discharges to atmosphere is illustrated in Figure 139.

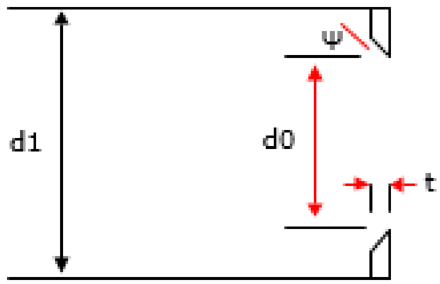


Figure 139 Bevel Edged Orifice Discharging to Atmosphere

For this orifice configuration, the loss coefficient (K_0) equation for a bevel edged orifice in an expanding transition section can be used, however the downstream pipe diameter (d_2) is effectively infinite and therefore the equation can be transformed into:

$$K_0 = 0.0696\left(1 - C_b\frac{t}{d_0}\right)\left(1 - 0.42\sqrt{\frac{t}{d_0}}\beta^2\right)(1-\beta^5)\lambda^2 + \lambda^2$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622\left[1 - C_b\left(\frac{t}{d_0}\right)^{\frac{1-\sqrt[4]{\frac{t}{d_0}}}{2}}\right](1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_b is a function of bevel angle ψ in degrees, and bevel thickness to diameter ratio t/d_0 is given by:

$$C_b = \left(1 - \frac{\psi}{90}\right)\left(\frac{\psi}{90}\right)^{\frac{1}{2+\frac{t}{d_0}}}$$

Thick Edged Orifice in a Straight Pipe

A thick edged orifice in a straight pipe, where the upstream and downstream pipe sizes are the same, is illustrated in Figure 140.

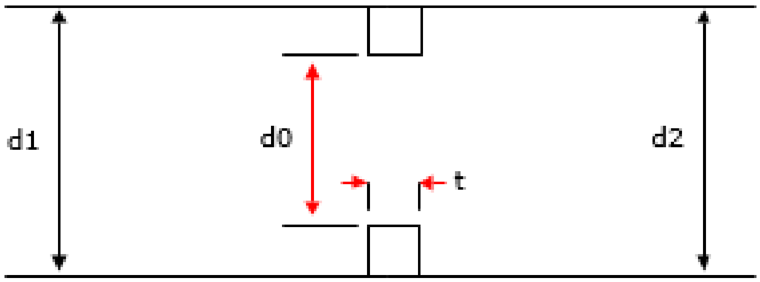


Figure 140 Thick Edged Orifice in a Straight Pipe

The loss coefficient (K_0) of local resistance for thickness t equal to or less than 1.4d of a thick edged orifice in a straight pipe is calculated as follows:

$$K_0 = 0.0696(1-\beta^5)\lambda^2 + C_{th}(\lambda-\beta^2)^2 + (1-C_{th})[(\lambda-1)^2 + (1-\beta^2)^2]$$

$$\left(\frac{t}{d_0}\right) \leq 1.4$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_{th} is given by:

$$C_{th} = \left[1 - 0.50\left(\frac{t}{1.4d_0}\right)^{2.5} - 0.50\left(\frac{t}{1.4d_0}\right)^{3}\right]^{4.5}$$

Thick Edged Orifice in a Transition Section

A thick edged orifice in a transition section, where the upstream and downstream pipe sizes are *not* the same, are illustrated in Figure 141 and Figure 142.

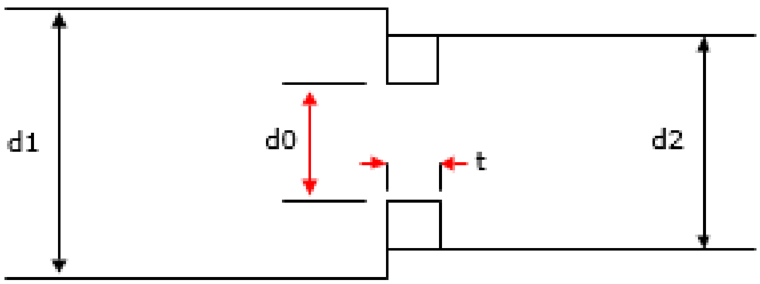


Figure 141 Thick Edged Orifice in a Transition Section (Contraction)

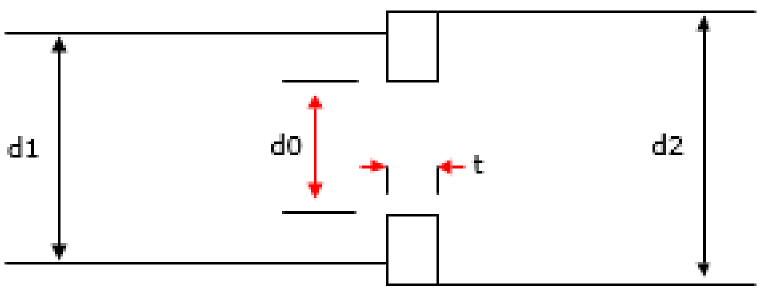


Figure 142 Thick Edged Orifice in a Transition Section (Expansion)

The loss coefficient (K_0) of a thick edged orifice in a transition section is calculated as follows:

$$K_0 = 0.0696(1-\beta^5)\lambda^2 + C_{th}\left(\lambda - \left(\frac{d_0}{d_2}\right)^2\right)^2 + (1-C_{th})\left[(\lambda-1)^2 + \left(1-\left(\frac{d_0}{d_2}\right)^2\right)^2\right]$$

$$\left(\frac{t}{d_0}\right) \leq 1.4$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_{th} is given by:

$$C_{th} = \left[1 - 0.50\left(\frac{t}{1.4d_0}\right)^{2.5} - 0.50\left(\frac{t}{1.4d_0}\right)^3\right]^{4.5}$$

Thick Edged Orifice Discharging to Atmosphere Section

A thick edged orifice that discharges to atmosphere is illustrated in Figure 143.

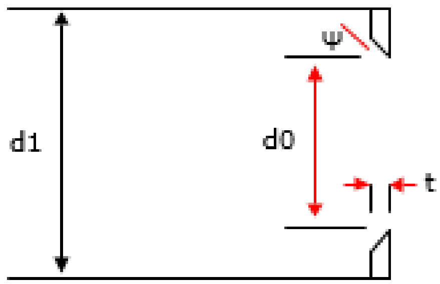


Figure 143 Thick Edged Orifice Discharging to Atmosphere

For this orifice configuration, the loss coefficient (K_0) equation for a thick edged orifice in an expanding transition section can be used, however the downstream pipe diameter (d_2) is effectively infinite and therefore the equation can be transformed into:

$$K_0 = 0.0696(1 - \beta^5)\lambda^2 + C_{th}\lambda^2 + (1 - C_{th})[(\lambda - 1)^2 + 1]$$

$$\left(\frac{t}{d_0}\right) \leq 1.4$$

Where the diameter ratio $\beta = d_0/d_1$ and where the jet contraction coefficient λ is given by:

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5)$$

And where C_{th} is given by:

$$C_{th} = \left[1 - 0.50\left(\frac{t}{1.4d_0}\right)^{2.5} - 0.50\left(\frac{t}{1.4d_0}\right)^{3}\right]^{4.5}$$

Flow Control Valves

Flow Control Valves allow the user to set a flow rate in a particular pipe. Pipe Flow Expert removes the pipe from the system and sets an Out-flow demand at the 'From node' and an equal In-flow demand at the 'To node'. Thus the pipe is replaced by these flow demands while the system is being solved.

No other pipe which connects to a pipe with a Flow Control Valve fitted can contain a control valve (FCV, PRV or BPV).

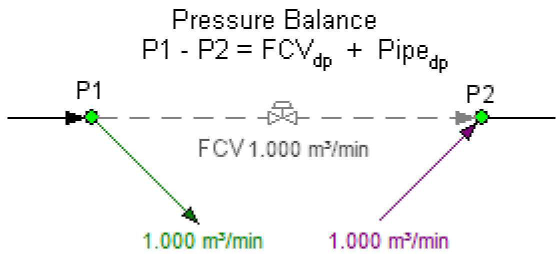


Figure 144 Flow Control Valve replacement

The pressure difference between the 'From node' and the 'To node' must equal the pressure loss introduced by the flow control valve plus the pressure loss that the flow in the pipe will produce, plus the pressure loss that any other component on the pipe produces. The system balance will be maintained when the pipe is reinstated along with the pressure loss introduced by the flow control valve.

If the pressure difference between the 'From node' and the 'To node' is not great enough then the pressure loss in the pipe and the flow control valve pressure loss cannot be set. A warning will be displayed that the pressure in the pipe is not sufficient to deliver the set flow.

Pressure Reducing Valves

Pressure Reducing Valves allow the user to set a pressure at the node downstream of the valve (i.e. at the end of the pipe). The pressure reducing valve (PRV) introduces an additional pressure loss in the pipe to control the pressure at the node downstream of the valve to the value specified by the user.

A pipe with a Pressure Reducing Valve fitted cannot have a tank or pressure demand set on either end. No other pipe which connects to a pipe with a Pressure Reducing Valve fitted can contain a control valve (FCV, PRV or BPV).

Pipe Flow Expert removes the pipe from the system and sets the pressure at the downstream node (N2) by replacing it with an appropriately defined tank. The tank elevation is set to equal the node elevation, the liquid level is set to zero and the fluid surface pressure is set to the pressure reducing valve setting. At the upstream node, an out-flow demand is then set equal to the flow from N2. Hence the pipe is replaced by a tank at the downstream node and an out-flow demand at the upstream node while the system is being solved. The outflow demand at the upstream node must equal the flow rate out of the downstream node that is now represented by the tank.

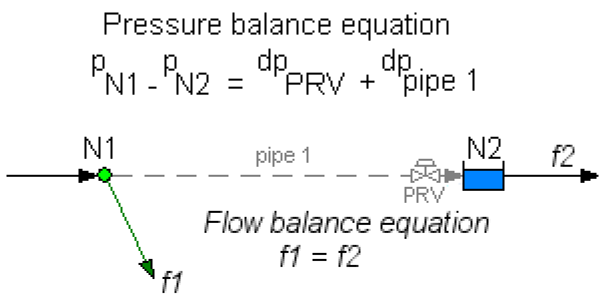


Figure 145 Pressure Reducing Valve replacement

The pressure difference between the upstream node P1 and the downstream node P2 must equal the pressure loss through the pipe, fittings and any components on the pipe plus the pressure loss introduced by the PRV. The pressure balance is then maintained after the system is solved, when the pipe is reinstated along with the pressure loss introduced by the pressure reducing valve.

MODES OF OPERATION: A PRV can operate under three different conditions: (1) regulating, (2) fully closed, and (3) fully open. How the valve operates depends on the defined set pressure value for the valve. The fully open and fully closed positions represent the extreme operations of the valve. Each of the valve positions is described below:

(1) Regulating The valve maintains the downstream pressure to the set value by introducing a pressure loss across the valve, thus throttling the flow rate through the PRV.

(2) Fully Closed This mode of operation occurs if the valve's set pressure is less than the pressure downstream of the valve for the case where the valve is closed. When this occurs in an actual pipe system, the flow through the PRV reverses and the PRV acts as a check valve, closing the pipe. In PipeFlow Expert, this method of operation is detected and reported but the system is not then solved for this scenario. The user must decide if this method of operation is what they intended and if so then they can close the pipe and continue to solve the system.

(3) Fully Open This mode of operation occurs if the valve's set pressure is greater than the pressure upstream of the valve for the case where the valve is fully open. When this occurs in an actual pipe system, the PRV maintains a fully open position and it has no effect on the flow conditions (except to add a frictional loss through the valve). In PipeFlow Expert, this method of operation is detected and reported. But the system is not solved because the differential pressure across the valve would have to be negative, i.e. the valve would be acting like a pump rather than a pressure control.

Pipe Flow Expert will only solve a system when the PRV is operating in Regulating mode.

AVOIDING PRV OPERATION PROBLEMS: In general, PRV operation problems can be avoided by finding the valve's pressure regulating range and specifying the valve's set pressure to a value within this range, such that the mode of operation is 'Regulating'. First, solve the system without the PRV control and note the pressure at the node downstream of the pipe which previously contained the PRV. This is the maximum pressure the PRV can be set to (i.e. it is equivalent to finding the valve's inlet pressure for the case where the valve is fully open). Secondly, solve the system after closing the pipe that contains the PRV and note the pressure at the node downstream of the closed pipe. This is the minimum pressure the PRV can be set to (i.e. it is equivalent to finding the pressure downstream of the valve for the case where the valve is fully closed).

Back Pressure Valves

Back Pressure Valves allow the user to set a pressure at a node upstream of the valve (i.e. at the start of the pipe). The back pressure valve (BPV) introduces an additional pressure loss in the pipe to control the pressure at the node upstream of valve to the value specified by the user.

A pipe with a Back Pressure Valve fitted cannot have a tank or pressure demand set on either end. No other pipe which connects to a pipe with a Back Pressure Valve fitted can contain a control valve (FCV, PRV or BPV).

Pipe Flow Expert removes the pipe from the system and sets the pressure at the upstream node (N1) by replacing it with an appropriately defined tank. The tank elevation is set to equal the node elevation, the liquid level is set to zero and the fluid surface pressure is set to the back pressure valve setting. . At the downstream node, an in-flow demand is then set equal to the flow into the upstream node N1. Hence the pipe is replaced by a tank at the upstream node and an in-flow at the downstream node while the system is being solved. The in-flow at the downstream node must equal the flow rate into the upstream node that is now represented by a pressurized tank.

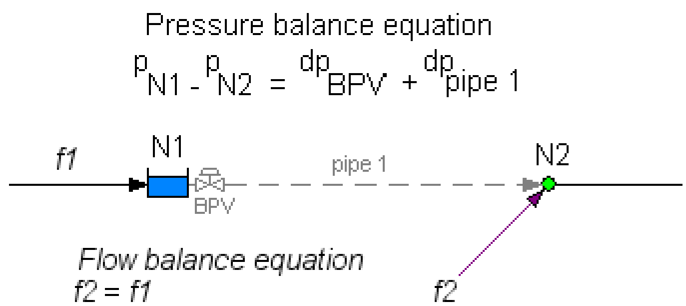


Figure 146 Back Pressure Valve replacement

The pressure difference between the upstream node P1 and the downstream node P2 must equal the pressure loss through the pipe, fittings and any components on the pipe plus the pressure loss introduced by the BPV. The pressure balance is then maintained after the system is solved, when the pipe is reinstated along with the pressure loss introduced by the pressure reducing valve.

MODES OF OPERATION: A BPV can operate under three different conditions: (1) regulating, (2) fully closed, and (3) fully open. How the valve operates depends on the defined set pressure value for the valve. The fully open and fully closed positions represent the extreme operations of the valve. Each of the valve positions is described below:

(1) Regulating The valve maintains the upstream pressure to the set value by introducing a pressure loss across the valve, thus reducing the flow rate through the BPV.

(2) Fully Closed This mode of operation occurs if the valve's set pressure is greater than the pressure upstream of the valve for the case where the valve is closed. When this occurs in an actual pipe system, the flow through the BPV reverses and the BPV acts as a check valve, closing the pipe. In PipeFlow Expert, this method of operation is detected and reported but the system is not then solved for this scenario. The user must decide if this method of operation is what they intended and if so then they can close the pipe and continue to solve the system.

(3) Fully Open This mode of operation occurs if the valve's set pressure is less than the pressure downstream of the valve for the case where the valve is fully open. When this occurs in an actual pipe system, the BPV maintains a fully open position and it has no effect on the flow conditions (except to add a frictional loss through the valve). In PipeFlow Expert, this method of operation is detected and reported but the system is not solved because the differential pressure across the valve would have to be negative, i.e. the valve would be acting like a pump rather than a pressure control.

Pipe Flow Expert will only solve a system when the BPV is operating in Regulating mode.

AVOIDING BPV OPERATION PROBLEMS: In general, BPV operation problems can be avoided by finding the valve's pressure regulating range and specifying the valve's set pressure to a value within this range, such that the mode of operation is 'Regulating'. First, solve the system without the BPV control and note the pressure at the node upstream of the pipe which previously contained the BPV. This is the minimum pressure the BPV can be set to (i.e. it is equivalent to finding the pressure at the valve outlet for the case where the valve is fully open). Secondly, solve the system after closing the pipe that contains the BPV and note the pressure at the node upstream of the closed pipe. This is the maximum pressure the BPV can be set to (i.e. it is equivalent to finding the pressure at the valve inlet for the case where the valve is fully closed).

Pumps (with Flow versus Head Curve)

The user is able to enter flow rate and head loss information about a pump which is to be used in the pipeline system. Pipe Flow Expert generates a performance curve for the pump to allow the pump performance to be modeled as part of the pipeline system.

Fixed Flow Rate Pumps

The user is able to elect to enter a fixed flow rate for a pump. This situation may occur when the natural flow distribution to different parts of the network has to be determined, or when the system has a variable speed pump that has been set to produce a set flow rate. The Fixed Flow Rate Pump is modeled in a similar manner to the Flow Control Valve described above. Pipe Flow Expert removes the pipe on which the pump is mounted and sets an Out-flow demand at the 'From node' and an equal In-flow demand at the 'To node'. The pipe and the pump are replaced by these flow demands while the system is being solved.

Figure 147 Fixed Flow Rate Pump replacement

Since the head pressure provided by the pump is unknown, the system balance cannot be 'tuned' by iterating along the pump performance curve. For this reason it is not possible to include pressure control devices such as flow control valves on all outlet paths through the system when a fixed flow rate pump has been selected (this would result in an over-controlled system).

Fixed Head / Pressure Rise Pumps

The user is able to elect to enter a fixed head or pressure rise for a pump. This situation may be useful when trying to estimate the pump head required to produce the flow distribution to different parts of the network, where a pump flow versus head curve has not been established.
The flow rates would be set by flow control valves on the outlet legs.
The minimum pump head would be determined by subtracting the smallest pressure introduced by the flow control valves from the fixed head supplied by the pump.

Net Positive Suction Head available

Pipe Flow Expert is able to calculate the pressure available at a pump inlet. This pressure will change if the mounting position of the pump along a pipe is changed. The vapor pressure of the fluid being pumped is subtracted from the pump inlet pressure to obtain the Net Positive Suction Head available. The NPSHa must be higher than the pump manufacturer's NPSHr (Net Positive Suction Head requirement for the flow rate) in order to prevent gas bubbles forming in the fluid.

Two Phase Flow

Pipe Flow Expert is not able to model two phase flow, such as gas/liquid mixtures.
When two different fluids are mixed together it is possible that two-phase flow may occur at some point in a system. Pipe Flow Expert does not calculate the pressure drop for two-phase flow.

Two-phase flow can produce an extremely high pressure drop many times greater than the pressure drop of either individual fluid. The user should make due allowance for the two-phase flow pressure drop by using a component to add an appropriate additional pressure loss.

Slurries

Slurries which have a constant density and a constant viscosity are able to be modeled, provided the minimum velocity to keep the solids in suspension is maintained throughout the pipeline system. The density and viscosity of the slurry mixture must be entered as the fluid data.

Working with Compressible Fluids

The Pipe Flow Expert software now contains a Compressible Isothermal Flow Calculation Engine (from version 7 and later).

In a gas system, as pressure loss occurs along a pipe, the gas density will decrease and the volume of the gas will expand. As the volume of gas increases, the velocity of the gas in the pipe will increase. Although the volume of gas and velocity in the pipe changes, the mass flow (weight of flow) in the pipe will remain constant.

Gas flow rates are therefore often referred to in terms of mass flow (weight of flow) or standard volume (which is the volume of gas at standard conditions, normally atmospheric pressure and some common temperature reference, since this standard volume also define a constant mass flow).

Pipe Flow Expert provides a choice of standard volume units for gas flow rate which include:

- SCCM (Standard Cubic Centimeters per Minute),
- SLM (Standard Liters per Minute),
- SCMH (Standard Cubic Meters per Hour)
- MMSCMH (Million Standard Cubic Meter per Hour)
- MMSCMD (Million Standard Cubic Meter per Day)
- SCFM (Standard Cubic Feet per Minute)
- SCFH (Standard Cubic Feet per Hour)
- SCFD (Standard Cubic Feet per Day)
- MMSCFH (Million Standard Cubic Feet per Hour)
- MMSCFD (Million Standard Cubic Feet per Day)

Each of the standard volume units for gas flow relate to the gas at a standard condition, however there are a number of slightly different standard reference conditions that are used worldwide depending on country and location. Pipe Flow Expert provides a choice of standard reference conditions to be used when referring to the standard volume flow rate of gas and these include:

- 0°C, 100.000 kPa.a
- 0°C, 101.325 kPa.a
- 15°C, 101.325 kPa.a
- 20°C, 101.325 kPa.a
- 25°C, 101.225 kPa.a
- 60°F, 14.696 psi.a
- 68°F, 14.696 psi.a

Mass flow can also be used to refer to an amount of gas flow and the units for mass flow include:

- Kgs/sec
- Kgs/min
- Kgs/hour
- Lb/sec
- Lb/min
- Lb/hour

Using Compressible Flow Equations

There are a number of different equations that can be used to calculate flow rate and pressure loss in a compressible gas system and the type of design and user preference often determines which equation they use to calculate the results.

Pipe Flow Expert allows for the selection of a specific compressible isothermal flow equation from a list that includes:

- General Fundamental Isothermal Flow Equation
- Complete Isothermal Flow Equation
- AGA Isothermal Flow Equation
- Panhandle A Isothermal Flow Equation
- Panhandle B Isothermal Flow Equation
- IGT Isothermal Flow Equation
- Weymouth Isothermal Flow Equation

The General Fundamental Isothermal Flow Equation (sometimes known as just the General Flow equation or the Fundamental Flow equation) provides perhaps the most universal method for calculating isothermal flow rates, however it relies on the inclusion of an accurate friction factor. The Pipe Flow Expert software provides such a friction factor by calculating this using the Colebrook-White equation.

For complex interconnected pipe systems the General Flow equation will often provide the best overall calculation result, however this approach is only made possible by the advanced software algorithms and the power of computer calculation.

The preferred method of calculation can be selected from the Calculations Tab in Configuration Options.

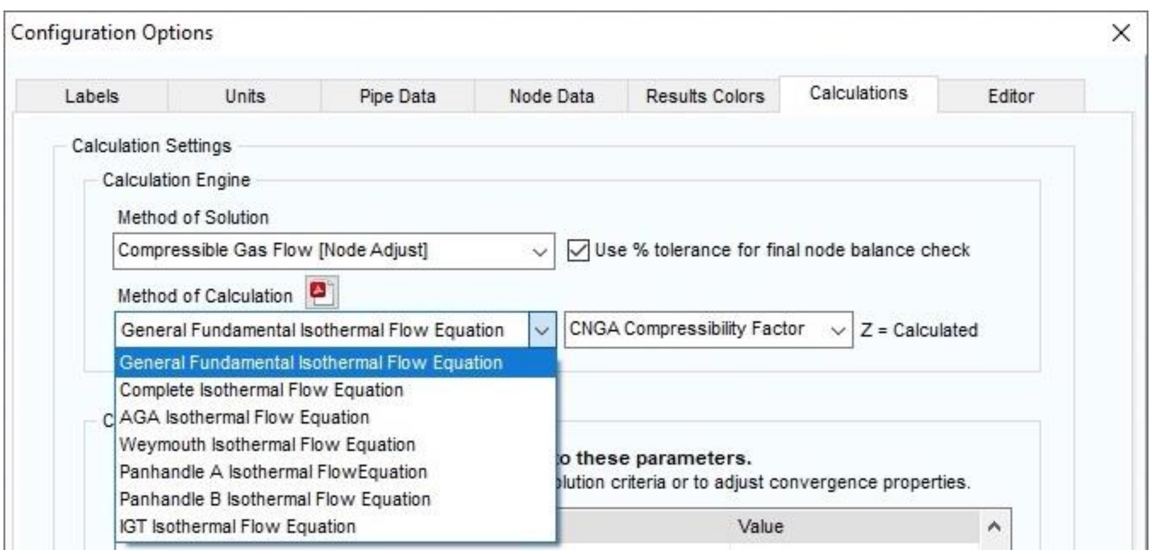


Figure 148 Compressible Flow Calculation Options

The calculations can incorporate the Ideal Gas Law, a custom Compressibility Factor (applied to the whole network) or the CNGA Compressibility Factor that is calculated for each pipe based on the pressures at the start and end of each pipe.

The Compressible Flow Calculation Engine will automatically take account of pressure changes within the pipe network and will automatically adjust the density properties of the gas as appropriate when performing the gas flow rate and pressure loss calculations. The equations used in the calculations currently assume isothermal flow where there is no change in temperature.

If the pipe system contains a compressor, component or valve that either significantly increase the gas pressure or significantly reduces the gas pressure then an additional fluid zone should be defined to specify the density properties of the gas at the required temperature condition.

Pipe Flow Expert will automatically account for changes in pressure within the system, however the user must define fluid properties and gas data for the operating temperature within the pipe system (or within each part of the pipe network if zone of different temperature exist).

When a user clicks to ‘Calculate’ a compressible gas system , if the calculation engine method of solution is not set to ‘Compressible Gas Flow’ then the software will prompt the user to ask if they wish to automatically switch to the Compressible Flow Calculation Engine (which is recommended).

If the user chooses not to switch to the Compressible Flow Calculation Engine then they can continue to solve their system using the standard Non-Compressible Calculation Engine, which uses the Darcy-Weisbach equation that assumes a constant density and viscosity for the gas as defined in the fluid data.

We recommend using the Compressible Flow Calculation Engine to calculate gas systems, however see the next section for considerations when using the non-compressible Darcy-Weisbach equation to calculate a compressible gas system.

Using Non-Compressible Darcy-Weisbach Equation

Important Note:

We would always recommend using the Compressible Flow Calculation Engine to solve compressible gas systems (rather than using the non-compressible calculation engine with the Darcy-Weisbach equation).

The following notes in this section are included for completeness only.

The Darcy-Weisbach equation is normally applicable to incompressible Newtonian fluids, since the density of these fluids can be considered to be constant even with changes in pressure, however this equation is also sometimes used for compressible systems provided that they operate within certain criteria.

When the Non-Compressible Calculation Engine is selected on the Calculations tab in Configuration Options then Pipe Flow Expert uses the Colebrook-White equation to calculate friction factors and the Darcy-Weisbach equation to calculate the friction loss in a pipe.

These equations assume a constant fluid density & viscosity and they provide a very accurate solution when working with non-compressible fluids (liquids) and they are sometime used for low velocity gas systems where there is a relatively small amount of pressure loss.

If the user select the Non-Compressible Calculation Engine when using the software to model gas systems then there are certain criteria that the model must operate within to ensure the solution is of acceptable accuracy.

Changes in pressure and temperature will affect the gas density and viscosity. These property changes affect the actual pressure drop and are not automatically accounted for in the Darcy-Weisbach equation, therefore some limitations must be applied to ensure that the calculated results are within an acceptable accuracy.

Generally systems which involve gases fall into two categories:

Low pressure loss gas systems:

Where the pressure loss is less than 10% of the highest absolute pressure in the system, if the pressure drop is calculated using the entering fluid density then good reliability of the results can be expected.

High pressure loss gas systems:

Where the pressure loss is more than 10% but less than 40% of the highest absolute pressure in the system, if the pressure drop is calculated using the average fluid density then good reliability of the results can be expected.

Where a system has a total pressure loss which exceeds 40% of the highest absolute pressure in the system, then if you are using the Non-Compressible Flow Calculation Engine (which is not recommended for gas systems) it will be necessary to model the system using a number of different fluid zones with the fluid density data defined for different pressure conditions). Up to 20 fluid zones can be used in a Pipe Flow Expert model.

Note: We include the above information for completeness, however we would recommend using the Compressible Flow Calculation Engine to solve gas system, which will automatically account for density changes with pressure losses (even for losses higher than 40% of the highest absolute pressure) and these systems can then be modeled with a single fluid zone (provided that there are no significant temperature changes).

Using Fluid Zones with gas systems:

Where a gas system has been split into a number of separate fluid zones, the fluid density and the fluid viscosity for each fluid zone should be set independently.

The density of each fluid zone in the gas system must reflect the average density of the compressed fluid condition in that fluid zone, in order for the Darcy-Weisbach equation to give a reasonably accurate result.

When a gas system is solved with the Non-Compressible Calculation Engine (and the Darcy-Weisbach equation), if the fluid density is not within 5% of the average fluid density for a particular zone, a warning will be issued in the results log. Suggestions to Update Fluid Zone Data to have a particular density based on the average pressure within the fluid zone will be issued in the results log.

General Suggestions:

For systems that contain compressible fluids the following should be noted.

The mass flow rates entering the system and the mass flow rates leaving the system must be balanced. Normally In-Flow or Out-Flow values are entered using units of mass flow or units of gas flow at standard volume (volume at standard temperature and pressure). Users should avoid specifying gas flow rates in regular volumetric units such as m^3/hour since these refer to actual volume of gas at whatever density has been defined in the fluid data (i.e. the density at some pressure condition). Volume in standard units such as SCMH (Standard Cubic Meters per Hour) should be used instead.

Where volumetric In-Flow rates entering the system are to be used, these values should generally be specified in gas flow standard volume units, such as SCFM, or in mass flow units, such as lb/min. If the flow rates are entered in regular volume, such as ft^3/min, then this defines the actual flow rate of the fluid at the density specified in the fluid data for the current fluid zone (and not the uncompressed volumetric flow rate of the gas at standard conditions).

Where volumetric Out-Flow rates leaving the system are to be used then these values should generally be specified in gas flow standard volume units, such as SCFM, or in mass flow units, such as lb/min (the same as for In-Flow values as described above).

With the Non-Compressible Flow Calculation Engine, Pipe Flow Expert uses a constant value for the fluid density throughout each individual fluid zone in the pipeline system. Where regular volumetric flow rates are used to specify the In-Flows and Out-Flows to the system, the individual density for each fluid zone is used to convert from volumetric flow rate units to the mass flow rate units used internally by Pipe Flow Expert.

As described above, we would recommend not specifying flow rates in volume units related to the density of the fluid zone, but rather use mass flow unit or gas flow standard volume units (volume at standard conditions).

The calculations are performed using mass flow rates to achieve mass flow rate continuity at each node and an overall pressure balance within the pipeline system.

When using the Non-Compressible Flow Calculation Engine, the effects of pressure changes on the fluid density are not modeled.

Note 1:

The Fluid density at the compressed fluid condition can be calculated using the normal density of the compressible fluid and the fluid pressure.

Compressed fluid density =
Normal fluid density x (Fluid pressure + Atmospheric pressure) /Atmospheric pressure

Example: If a volume of 10 m^3 of air at normal temperature and pressure is compressed to 6 bar g

The Fluid density would be: 1.2047 x (6.000 + 1.01325) / 1.01325 = 8.3384 kg/m^3

Note 2:

The Actual volumetric flow rate of the fluid at the compressed fluid condition can be calculated using the uncompressed volume of the fluid at standard condition and the compressed fluid pressure.

Actual flow rate =
Uncompressed fluid volume x (Atmospheric pressure / (Fluid pressure + Atmospheric pressure))

Example: If a flow of 10 m^3/s of air at normal temperature and pressure is compressed to 6 bar g

The Actual flow rate would be: 10 x (1.01325 / (6.000 +1.01325)) = 1.445 m^3/s

Glossary

Term	Description
Absolute pressure	Pressure measured with respect to zero pressure.
Atmosphere	A standard atmospheric pressure of 1.01325 bar a or 14.696 psi a.
Centipoise	Absolute viscosity of a fluid expressed in Pa • s x 10^{-3}
Centistokes	Kinematic viscosity of a fluid expressed in m^2/s x 10^{-6} Note: Pipe Flow Expert requires fluid viscosity to be entered in Centipoise.
Colebrook-White equation	An equation used to calculate accurate friction factors from the internal diameter and internal roughness of a pipe and the Reynolds number for the flow conditions.
Darcy-Weisbach equation	An equation used to calculate the frictional head loss due to fluid flow from the friction factor, the length and diameter of the pipe, the velocity of the fluid and the gravitational constant.
Demand flow	The In-Flow entering the system or the Out-Flow leaving the system.
Demand pressure	The pressure at a point of exit from the system.
Discharge	Out-Flow leaving the system.
Dynamic viscosity	The absolute viscosity of a fluid.
Friction factor	A factor to be used in the Darcy-Weisbach equation. Either calculated from the Colebrook-White equation or read from the Moody diagram.
Elevation	The height above sea level of a node or tank.
End pressure	The pressure at a point where fluid leaves the system.
Fixed pressure	A static pressure loss which is independent of the flow rate.
Fluid head	The resistance to flow expressed in height of fluid as a motive force.
Gage pressure	Pressure measured with respect to atmospheric pressure.
Hydraulic grade line	The pressure at a point in the system expressed in height of fluid plus the elevation above sea level of the node or the tank.
in Hg	Height of a column of mercury in a barometer expressed in inches.
K value	Coefficient of frictional loss through valve or pipe fittings.
Kinematic viscosity	The absolute viscosity of a fluid divided by the fluid density.
Moody diagram	A graphical representation of the relationship between Reynolds number, relative roughness and Friction factor.
mm Hg	Height of a column of mercury in a barometer expressed in millimeters.
NPSHa	Net Positive Such Head available.
NPSHr	Net Positive Such Head requirement.
Pressure loss	The friction loss due to fluid flow expressed in fluid head or Gage pressure.
Pump head	The motive force developed by a pump expressed in height of fluid. (Pump performance graphs usually show pump head for water).
Relative roughness	A dimensionless number expressing the internal roughness of a pipe divided by the internal diameter of a pipe.
Reynolds number	A dimensionless number derived from the fluid velocity, the internal diameter of the pipe and the Kinematic viscosity of the fluid.
Supply	In-Flow entering the system.
Vapor pressure	The absolute pressure at which a liquid will start to evaporate.
Viscosity	A measure of a fluid‘s resistance to flow. See absolute viscosity and Kinematic viscosity.

Index