PIPE-FLO Professional

User's Manual & Method of Solution

Engineered Software, Inc.

TABLE OF CONTENTS

INTRODUCTION

PIPE-FLO Overview

PIPE-FLO is a comprehensive piping design and analysis software package that provides you with a clear picture of the entire system. Using PIPE-FLO you can:

- Draw a piping system schematic on the FLO-Sheet showing all the pumps, components, tanks, control valves and interconnecting pipelines.

- Size the individual pipelines using electronic pipe, valve, and fluid data tables.

- Select pumps and control valves from manufacturer's electronic catalogs, optimizing pump and system operation.

- Calculate how the system operates including pressures, flow rates, and net positive suction head available (NPSHa).

- Calculate the annual operating costs for pumps in the system.

- Create FLO-Links to provide immediate access to supporting documents needed to design, build, and operate the piping system.

- Share the piping system information with others through reports and the PIPE-FLO Viewer.

New in this version

- Operating cost calculator – Calculate and compare the operating costs of various pumps for any configuration or lineup.

- Rapid entry of control valve data – Generate complete characteristic curve data from a single flow coefficient and valve position by selecting from a list of valve design and flow characteristic types.

- Control valve operations – Automatically change control valves from a set flow rate to the calculated manual position.

- Atmospheric pressure calculator – Calculate the atmospheric pressure at any specific elevation.

- FLO-Sheet improvements – Place any number of background images on the FLO-Sheet.

- Printed report improvements – Add an image to the top of printed reports and personalize your cutom reports.

- Presentation mode – Displays the FLO-Sheet with a white background for enhanced viewing.

- Copy & paste improvements – Copy and paste using standard shortcut keys.

- Device symbols – New tank and component symbols are available to choose from.

- New X-Link data codes – Access more flow meter, pump, component, and control valve design data.

Getting Started

Installation

PIPE-FLO comes in both stand-alone and network versions. Stand-alone versions are installed in the standard manner (insert the CD and run Setup. exe). Network versions include special licensing software and setup options. Stand-alone and network versions both require a hardware key that must be attached to the computer or license server before PIPE-FLO will run properly. Please consult the network.pdf document on the root of the installation CD for detailed information about configuring and administering PIPE-FLO network packages.

Learning to use the software

Learning new software can be a daunting experience. To speed up the process, PIPE-FLO includes a comprehensive tuturial system accessible directly from the programs startup screen. Simply launch the software, click the Software Tutorial button and choose the tutorial appropriate for you.

These tutorials are designed to walk users through all aspects of the software as quickly as possible. We recommend that new users begin with the Basic Tutorial to learn about standard PIPE-FLO operation and terminology. Users already familiar with the software may wish to scan the Mini Tutorials section

for information on specific program features. This is a great way to learn about new features added to the most recent version of the software.

Using Program Help

PIPE-FLO's help system has been designed to answer the most common types of questions as they arise. Help may be accessed in the following ways:

- When viewing any dialog box, click it's Help button to bring up a description of the dialog and its choices.

- At any time, pressing the **F1** key, clicking the Help (question mark) button on the toolbar, or selecting **Contents** or **Index** items from the program's Help menu will bring up the full Help file.

- From within the Help interface you may use the **Contents** tab to browse the help file by category, use the **Index** tab to browse for specific topic items by name, or use the **Search** tab to look for any help topics that contain specific key words.

- The Help interface's, **Glossary** tab lists and defines many of the technical terms that are used throughout the software.

- Many Help topics also contain a **Related Topics** button. This button cross-references other topics which are related to the topic you are currently viewing.

Technical Support

The purchase or upgrade of your PIPE-FLO software includes a one year of Engineered Software's TechNet technical support service. This service includes:

- All program upgrades

- Web-based software training

- Discounts on FLO-Master training classes

- Unlimited access to Engineered Software's online knowledge base (**www.eng-software.com/kb**) and other web-based support services.

- Email and telephone support for installation and program troubleshooting issues

- Email and telephone support from Engineered Software's engineering staff for questions about your own piping system models

- Access to new fluid data as it is added to the MKS Fluid Compilation

Limited support for issues with program operation and access to the online knowledge base are still available after your TechNet subscription expires. For information on renewing a lapsed TechNet subscription, please call our sales department at 800-786-8545.

Contacting Engineered Software

When you can't find the answers to your questions in PIPE-FLO's Online Help or in Engineered Software's knowledge base (www.eng-software.com/kb), you can contact our technical support in one of the following ways:

- By e-mail at **solutions@eng-software.com**

- By phone at 360-292-4060

- By fax at 360-412-0672

When contacting us, you should include the following information:

- Your name

- Company name

- Program serial number

- Program version number

- Your phone number, fax number, and e-mail address

- A detailed description of your question or problem

PIPING SYSTEM ANALYSIS

Introduction

This section of the reference book describes the engineering methods used and the assumptions made by the PIPE-FLO program. The calculations associated with system elements, such as pipes, pumps, components, and controls, is discussed, along with the methodology applied in performing the piping system calculations.

The Darcy-Weisbach Method

Users have the option of using the Hazen-Williams method if they prefer to. See The Hazen-Williams Method section below.

The flow of real fluids is greatly influenced by the viscous nature of the fluid. The viscosity of a fluid introduces shear forces between fluid particles as well as between the boundary walls and the fluid particles. The energy which is expended to overcome the fluid shear forces is converted into heat. This conversion of fluid energy to heat results in a fluid pressure drop.

When the fluid shear stress is entered into the Euler equation for ideal flow, the result is a partial differential equation for which no general solution is known (refer to a fluid dynamics textbook for this exercise). This requires engineers to rely on empirical formulas derived from experimental observations for the solution of fluid dynamics problems.

One of the earliest methods developed for determining fluid pressure drops in piping systems was the Hazen-Williams method. However, this formula has severe restrictions in its use because it is only valid for fluids with physical properties similar to that of water at 60°F and a limited number of pipe materials.

The Darcy-Weisbach method takes into account fluid viscosity and pipe roughness, providing valid results for incompressible Newtonian fluids flowing in any round fully charged pipe. This formula can also be extended to compressible fluids with some restrictions.

The Darcy-Weisbach equation is as follows:

$$dP = \rho f(L/D)v^2/2g$$

<div align="right">*equation 1*</div>

dP = pressure drop
ρ = fluid density
f = Darcy friction factor
L = length of pipe
D = pipe diameter
v = mean fluid velocity
g = gravitational constant

Often, the Darcy-Weisbach is expressed in the following way:

$$dP = K\rho v^2/2g \qquad \text{where} \quad K = f(L/D)$$

<div align="right">*equation 2*</div>

The K in the above equation is the total resistance coefficient for the pipeline. This "total K" is a combination of the K value for the pipe and the K value for the valves and fittings in the pipeline. Therefore, the pipeline pressure drop calculated is a combination of the pressure drop due to the pipe and valves.

The only factor in the Darcy-Weisbach equation that is not readily available is the unitless Darcy friction factor, f. Friction factor values have been arrived at by experimentation and empirical formulas have been developed to accurately predict the friction factor under a wide range of fluid conditions.

There are two commonly used friction factors in hydraulics publications, the Darcy friction factor and the Fanning friction factor. They are related as follows:

$$f_{Darcy} = 4f_{Fanning}$$

<div align="right">*equation 3*</div>

The Darcy friction factor is used throughout the PIPE-FLO program.

Determining the Pipe Friction Factor

For both laminar and non-laminar flow, the friction factor is a function of the Reynolds number. The Reynolds number is a dimensionless parameter which describes the characteristics of the fluid flowing in the piping system.

Reynolds developed the following relationship:

$$R_e = 354W/(d\mu)$$

equation 4

W = flow rate, kg/hr
d = pipe diameter, mm
μ = fluid dynamic viscosity, centipoise

It was determined that for Newtonian fluids with a Reynolds number below a specific value, the fluid particles move in slip streams or laminar layers. Above a critical value of Reynolds number, the motion of the fluid particles becomes random or turbulent.

By default, PIPE-FLO uses 2100 as the upper limit of the laminar region. Users have the option of changing this cutoff value.

For engineering calculations, the upper practical limit of laminar flow has been set at a Reynolds number of $R_e = 2100$. Above the laminar flow region, the flow starts to become turbulent. As the Reynolds number of the system increases, the flow becomes more turbulent, until the motion of the fluid particles is completely turbulent. The range between laminar flow and fully turbulent flow is referred to as the transition region.

The friction factor for laminar flow is dependent only on the Reynolds number and is expressed by the following relationship:

$$f = 64/R_e$$

equation 5

If PIPE-FLO calculates a Reynolds number less than 2100, it uses equation 5 to determine the friction factor when performing pressure drop calculations.

A word of caution: when determining pressure drop around the critical Reynolds number of 2100, the flow may shift from laminar to turbulent depending on the piping system. For this reason, a pipeline should not be designed close to the critical Reynolds number.

The friction factor calculation in the transition and turbulent range is more complex. It is a function of the Reynolds number and the ratio of the pipe material surface roughness to the pipe inside diameter.

Nikuradse performed a series of experiments in order to develop a relationship between the friction factor and Reynolds number in pipes with turbulent flow. The value of material surface roughness was arrived at by coating the interior of a smooth pipe with uniform grains of sand. The results of his experimentation presented some valuable relationships. He made the following discoveries:

1 At high Reynolds numbers the pipe friction factor becomes constant.

2 For rough pipes the ratio of surface material roughness to pipe diameter, or relative roughness, is more important than the Reynolds number for determining the friction factor.

Since Nikuradse used pipes with an artificial roughness applied, his friction factor values had little direct application for engineering materials. C. F. Colebrook experimented with commercial pipes of various materials and roughness and developed the following equation for pipes in the transition region to the complete turbulence zone:

$$1/(f^{1/2}) = -0.869\ln[(e/D)/3.7 + 2.523/(R_e f^{1/2})]$$

equation 6

Since this relationship has the friction factor term on both sides of the equation, it must be solved by iteration.

Iterative equations are easily solved by computer, but they take longer to solve than a straightforward relationship. For this reason, PIPE-FLO uses an equation from Reference 7 that provides a direct calculation of the friction factor and is within 1% of the Colebrook equation:

$$f = 1.325/[\ln(e/(3.7D) + 5.74/Re^{0.9})]^2$$

equation 7

PIPE-FLO uses equation 7 to calculate the friction factor in pipelines with Reynolds numbers greater than 2100. This equation gives accurate values of the friction factor and can be solved quickly without performing iterative calculations.

Valves and Fittings

In order to include valves and fittings into the calculation of the pipeline pressure drop, a K value must be computed for each valve and fitting and added to the total K value of the pipe used in the Darcy equation. Valve manufacturers have performed experiments on various valves and fittings in order to arrive at empirical pressure drop expressions for specific types of piping system components. The relationship between the K value and pipe length is as follows:

$$K = f_T(1000L/d)$$

equation 8

f_T = turbulent friction factor
L = length, m
d = pipe diameter, mm

As mentioned above, the K value for various fittings has been arrived at through experimentation. For any valve, a length over diameter (L/D) coefficient can be determined by equating the pressure drop through the valve to the equivalent length of pipe measured in pipe diameters. When the (L/D) coefficient is multiplied by the turbulent friction factor for clean commercial steel pipe at various diameters, the K value for the valve is determined. To determine the turbulent friction factor, PIPE-FLO uses the following equation from Nikuradse:

$$f_T = 8[2.457\ln(3.707d/e)]^{-2}$$

equation 9

There are some resistances to flow in piping systems that are independent of the friction factor. These resistances are determined only by the change in fluid velocity and the change in direction of flow. Losses associated with changes in pipe diameter (both gradual and abrupt) or pipeline entrances or exits fall into this category.

Differential Pressure Calculator

In some cases, it may be necessary to insert a resistance in the pipeline to restrict the flow to a certain value at a given pressure drop. The dP Calculator feature of PIPE-FLO calculates the K value needed to achieve a differential pressure for a specified flow rate. The K value is calculated using the following formula:

$$K = 1.59923 \Delta P d^4 \rho / W^2$$

equation 10

The inside diameter is used when calculating a K value with equations 10 and 11. If you later change the pipe diameter, you should reinstall the K value.

ΔP = Change in pressure, bars
d = Pipe diameter, mm
ρ = fluid density kg/m³
W = Mass flow rate, kg/hr

Cv Values

Many valve manufacturers express the pressure drop characteristics of their valves using a flow coefficient (Cv) rather than a K value. By definition, Cv is the number of gallons per minute of 60°F water which will pass through a valve with a fixed pressure drop of 1 psi.

When users specify a Cv value for a pipeline, PIPE-FLO calculates the K value using the following correlation:

Users can also specify a Kv value for a pipeline. Kv is related to Cv by the following conversion factor: Kv = 0.86477*Cv

$$K \cong 891 d^4 / C_v{}^2$$

equation 11

d = inside pipe diameter, inches

Reducers

A change in fitting diameter causes a pressure loss due to the change in velocity of the fluid as it passes through the fitting. The pressure drop is also dependent on the rate at which the change of direction in the fitting occurs.

For example, a 12x6 reducer with a 12 inch approach length has a smaller pressure drop than a 12x6 reducer with a 6 inch approach length. This holds true when the flow in the reducer is either contracting or expanding.

The equations used by PIPE-FLO to calculate reducer K values are listed below. Notice that these equations vary with the angle of approach, which is determined from the approach length specified by the user (see Figure 1 below).

Reducer Fitting

θ = Angle of approach
L = Approach length
1 = Smaller diameter subscript
2 = Larger diameter subscript
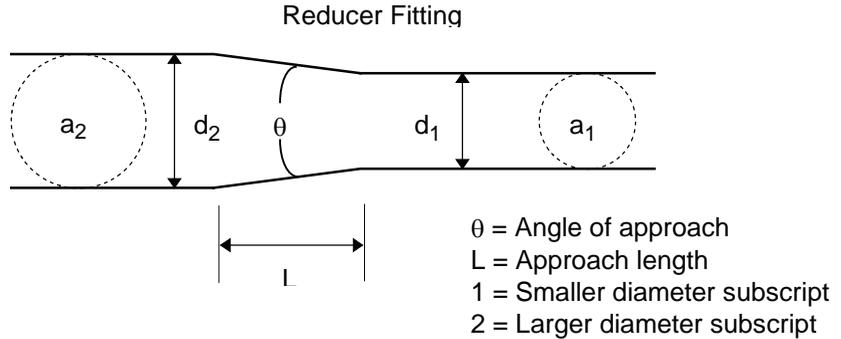
Figure 1

For an enlargement, the direction of flow in Figure 1 is from the smaller diameter (d_1) to the larger diameter (d_2). For a contraction, the direction of flow is from the larger diameter (d_2) to the smaller diameter (d_1).

Reducer Contraction

$$K_1 = 0.8\sin(\theta/2)(1-\beta^2) \ (\theta \leq 45°)$$

equation 12a

$$K_1 = \tfrac{1}{2}(1-\beta^2)(\sin(\theta/2))^{1/2} \ (45° < \theta \leq 180°)$$

equation 12b

Reducer Enlargement

$$K_1 = 2.6\sin(\theta/2)(1-\beta^2)^2 \ (\theta \leq 45°)$$

equation 13a

$$K_1 = (1-\beta^2)^2 \ (45° < \theta \leq 180°)$$

equation 13b

θ = Angle of approach
$\beta = d_{minor} / d_{major}$

Reduced Seated Valves and Fittings

Reduced seated valves are broken down into three sections: the reducer (contraction) section, the valve section, and the enlarger section (see Figure 2 below). A K value is determined for each section in relation to the size of the valve in the pipeline.

Reduced Seated Valves and Fittings
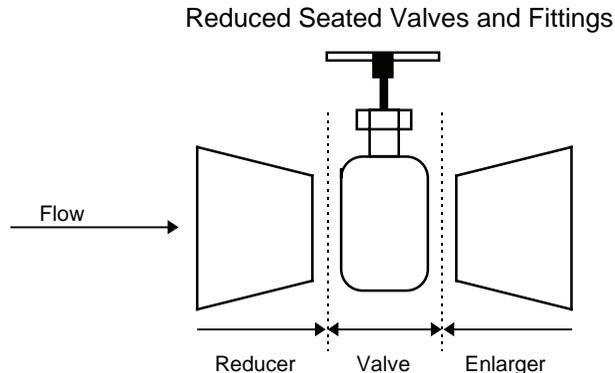
Figure 2

The total K value for the valve consists of K values for the reducer, the valve section, and the enlarger. Since the K value for the valve section is dependent on the valve seat diameter instead of the pipeline diameter, it must be adjusted by a correction factor to determine the correct K value for the diameter of the pipe. The valve correction factor is the ratio of valve seat diameter divided by pipeline diameter (often called beta) raised to the fourth power.

The total K for the reduced seat valve is:

$$K_2 = K_{Reducer} + K_1/\beta^4 + K_{Enlarger}$$

equation 14

K_2 = K value of the total valve
K_1 = K value of reduced seat
$K_{Reducer}$ = K value of reducer
$K_{Enlarger}$ = K value of enlarger
$\beta = d_{valve}/d_{pipe}$

PIPE-FLO Valve Table Formulas

PIPE-FLO uses the method outlined in the Crane Technical Paper 410 when calculating the K values for valves and fittings. PIPE-FLO's standard and specialty valve tables are set up such that each valve and fitting type references a predefined K value formula. Below is a listing of these formulas, the types of valves or fittings they support, and the input information PIPE-FLO requires from the user when defining the valve.

Formula 1 - Pipe Contraction

Formula 1 is used to calculate the K value for pipe contractions. PIPE-FLO determines which of the Formula 1 equations to use based on the reducer angle of approach (θ). For pipe contractions, the user must enter the reducer diameters and approach length in the Valve & Fitting dialog box. From this information, PIPE-FLO selects the appropriate equation and calculates the K value.

$$K_1 = 0.8\sin(\theta/2)(1-\beta^2)(\theta \leq 45°)$$

$$K_1 = \tfrac{1}{2}(1-\beta^2)(\sin(\theta/2))^{1/2} \ (45° < \theta \leq 180°)$$

Formula 1

Formula 3 - Pipe Enlargement

Formula 3 is used to calculate the K value for pipe enlargements. PIPE-FLO determines which of the Formula 3 equations to use based on the reducer angle of approach (θ). For pipe enlargements, the user must enter the reducer diameters and approach length in the Valve & Fitting dialog box. From this information, PIPE-FLO selects the appropriate equation and calculates the K value.

$$K_1 = 2.6\sin(\theta/2)(1-\beta^2)^2 \ (\theta \leq 45°)$$

$$K_1 = (1-\beta^2)^2 \ (45° < \theta \leq 180°)$$

Formula 3

Formula 5 - Reduced Seat Valve, Gradual Change in Diameter

Formula 5 is used to calculate the K value for reduced seated valves with a gradual change in diameter. Examples of valves which use Formula 5 are ball and gate valves. Formula 5 is a combination of Formulas 1 and 3 (discussed above) and the K value of the reduced seat valve type. For each supported valve or fitting, a full seat L/D coefficient is stored in the valve table. PIPE-FLO determines the angle of approach from the reduced seat diameter and the approach length specified by the user in the Valve & Fitting dialog box. If a reduced seat and approach length are not specified, the program assumes that the valve is full seated.

$$K_2 = K_{Reducer} + K_1/\beta^4 + K_{Enlarger}$$

Formula 5

Formula 7 - Reduced Seat Valve, Abrupt Change in Diameter

Formula 7 is used to calculate the K value for reduced seated valves with an abrupt change in diameter. Examples of valves using Formula 7 are globe, angle, lift check, and stop check valves. Formula 7 is a combination of Formulas 1 and 3 (with the angle of approach set to 180°) and the K value of the reduced seat valve type. A full seat L/D coefficient is stored in the table for each supported valve or fitting. The user must specify the reduced seat diameter in the Valve and Fitting dialog box. If a reduced seat is not specified, the program assumes that the valve is full seated.

$$K_2 = K_{Reducer} + K_1/\beta^4 + K_{Enlarger}$$

Formula 7

Formula 8 - Elbows and Bends

Formula 8 is used to calculate the K value for elbows and bends. A coefficient is stored in the valve and fitting table for each r/d ratio. In the Valve & Fitting dialog box, the user must specify the angle of the elbow.

$$K_b = (n-1)(0.25\pi f_T(r/d) + 0.5K) + K$$

Formula 8

n = number of 90° bends
K = resistance coefficient for one 90° bend

Formula 9 - L/D Varies with Pipe Diameter

A maximum of 100 size ranges can be specified for a Formula 9 valve.

Formula 9 is used to calculate the K value for valves that have an L/D coefficient that varies with pipe diameter. L/D coefficients for a range of pipe diameters are stored in the valve table. Valves that fall into this category are butterfly valves and tilting disk check valves. PIPE-FLO automatically selects the appropriate coefficient based on the pipe diameter the valve is installed in.

$$K = f_T \ (L/D)$$

Formula 9

Formula 10 - Full Seated Valves and Fittings

Formula 10 is used to calculate K values for valves and fittings that do not support reduced seated diameters. A full seat L/D coefficient is stored in the table for each supported valve or fitting. Valves and fittings which use Formula 10 include plug, foot, and swing check valves as well as tees and miter bends.

$$K = f_T \ (L/D)$$

Formula 10

Formula 11 - Fixed K Value

Formula 11 is used to enter a valve or fitting with a fixed K value. A K value is stored in the table for each supported valve or fitting. Examples of Formula 11 fittings are pipe entrances and exits.

Formula 12 - Fixed Cv Value

Formula 12 is used to enter a valve with a fixed Cv value. A Cv value is stored in the table for each supported valve. PIPE-FLO calculates the K value for fixed Cv valves using equation 11.

Formula 13 - Cv Value Varies with Pipe Diameter

A maximum of 100 size ranges can be specified for a Formula 13 valve.

Formula 13 is used for valves that have a Cv value that varies with pipe diameter. Cv values for a range of pipe diameters are stored in the valve table. PIPE-FLO automatically selects the appropriate coefficient based on the pipe diameter the valve is installed in and calculates the K value using equation 11.

The Hazen-Williams Method

As noted above, another method for calculating the friction head loss is the Hazen-Williams equation. This empirical equation is commonly used by civil engineers for piping systems that transport water. A form of this equation is listed below (Reference 8):

$$HL = 10.44LQ^{1.85}/(C^{1.85}d^{4.8665})$$

equation 15

HL = head loss in feet
L = pipe length in feet
Q = flow rate in gpm
d = pipe diameter in inches
C = Hazen-Williams roughness coefficient

The Hazen-Williams equation should only be used for turbulent flow. It gives good results for liquids with viscosities similar to that of 60°F water.

NOTE: The user has the option of selecting either the Darcy-Weisbach method or the Hazen-Williams method for the pressure drop equation. As noted previously, the Darcy-Weisbach method gives good results for a wide variety of Newtonian fluids, whereas the Hazen-Williams method has a more limited range of applicability. However, PIPE-FLO does not restrict the use of the Hazen-Williams method - users have the final responsibility for determining if the Hazen-Williams method should be used for their particular applications.

Valve/Fitting Losses

When the Hazen-Williams method is selected, a variation of the equivalent lengths method is used to determine the losses due to the pipeline valves/fittings. The equivalent length is calculated using the following equation:

$$Le = KD/f_T$$

equation 16

Le = equivalent length
K = total valve/fitting K value for the pipeline
f_T = turbulent friction factor

The equivalent length is then substituted into the Hazen-Williams equation (equation 15).

NOTE: Users who would prefer to use equivalent lengths obtained from another source or reference can do so by manually adding the equivalent length values to the pipeline length.

Mass and Volumetric Flow Rate Units

PIPE-FLO supports a wide variety of both mass and volumetric flow rates. All of the program's internal pressure drop calculations are performed in the mass flow rate unit of kg/hr.

When volumetric units are selected, the density of the fluid is used to convert from volumetric flow rate units to the mass flow rate units used internally by PIPE-FLO. After the calculations are performed, the results are then converted back to the volumetric units.

NOTE: As noted above, PIPE-FLO uses the fluid density in each node's connecting pipelines to convert the mass flows used internally by the program to volumetric flows. If the connecting pipelines have different fluid densities, the volumetric flows do not sum to zero at the node (the flows into the node do not equal the flows out of the node). However, this does not mean that the node is not balanced. If you were to convert the volumetric flows to mass flows, the flow rates would sum to zero. See *Network Flow Equations* below.

Standard Flow Rate Units

The following standard US flow rate units are available in PIPE-FLO:

- scfm (standard cubic feet per minute)
- scfh (standard cubic feet per hour)
- scfs (standard cubic feet per second)

These units are based on a reference condition of 60°F and 14.7 psia.

The following standard metric flow rate units are available:

- sm³/min (standard cubic meters per minute)
- sm³/hr (standard cubic meters per hour)
- sm³/s (standard cubic meters per second)

These units are based on a reference condition of 15°C and 1.01325 bar a.

Compressible Fluid Flow in Pipelines

The Darcy-Weisbach method assumes the density of the fluid flowing in the pipeline is constant. This is a valid assumption for non-compressible fluids, but it is not always the case for compressible fluids such as steam, air, and most industrial gases. The density of a compressible fluid is a function of the fluid pressure. If the fluid undergoes a pressure drop in a pipeline, the density of the fluid changes.

Extensive research has been conducted to determine when the Darcy-Weisbach formula can be applied to compressible fluids. The following restrictions should be observed when using the Darcy-Weisbach formula for compressible fluids:

1   If the calculated pressure drop in a pipeline (P_{in} - P_{out}) is less than 10% of the inlet pressure, acceptable accuracy is obtained if the density used for the fluid is based upon either the upstream or downstream conditions.

2   If the calculated pressure drop (P_{in} - P_{out}) is greater than 10% but less than 40% of the inlet pressure, the Darcy-Weisbach method provides acceptable accuracy if the fluid density is based upon the average of the inlet and outlet fluid conditions.

3   If the pressure drop in a pipeline (P_{in} - P_{out}) is greater than 40% of the inlet pressure, the Darcy-Weisbach method will not give valid results.

Most gas distribution systems in a process plant fall under either category 1 or 2. A gas distribution system is designed to provide a fluid at a given pressure. Any losses caused by the piping system are usually minimized to reduce the loss of energy (due to turbulence) of the compressible fluid. As a result, the formulas used in PIPE-FLO are valid for the majority of compressible fluid distribution applications, including air and steam distribution. Systems where the pressure drop exceeds the 40% range include long gas transmission pipelines and systems where sonic flow conditions exist.

PIPE-FLO does not automatically account for changes in fluid density due to the pressure drops calculated in the lineup. Once a fluid zone pressure has been selected during the pipeline sizing portion of the program, the density corresponding to that pressure is used throughout the analysis, regardless of the pressures calculated at the pipeline endpoints. If the pressure drops in the system pipelines exceed 10% of their inlet pressures, you may need to adjust the fluid zone pressures.

Two Phase Flow and Slurries

PIPE-FLO does not accurately model two phase flow in a pipeline such as gas/liquid mixtures or steam/saturated fluid mixtures.

Slurries that can be considered homogeneous mixtures, such as paper stock or limestone water slurries, can be used in PIPE-FLO. In slurries, the solids are held in suspension due to the turbulence of the fluid. If this is the case, a minimum flow rate is required in the system pipelines to keep the slurry homogeneous. The minimum fluid velocity in the pipe specification can be set so that if the fluid velocity goes below the value needed to keep the slurry solids in suspension, you are notified on the results screen and printed report.

System Fluid Pressure

The Bernoulli theorem states that the total energy of the fluid anywhere in the system is composed of three components: the pressure head, the velocity head, and the elevation head. This relationship is expressed in the following way:

$$H = P/\rho + v^2/2g + Z$$

equation 17

H = total head
P = pressure, gage units
ρ = fluid density
v = fluid velocity
Z = elevation
g = gravitational constant

Provided no work is done on the fluid, the energy of the fluid must remain the same throughout the piping system. The pressure at any point in the system can be found if the energy at one point is known and the velocity and elevation heads are known. Since all real piping systems have friction losses, a system energy loss will result in the form of heat released due to friction. This loss is the head loss calculated in the Darcy-Weisbach equation and must be factored into the Bernoulli equation when determining the system pressures. The fluid energy at any point in the system can be determined from the following equation:

$$P_1/\rho + v_1^2/2g + Z_1 = P_2/\rho + v_2^2/2g + Z_2 + H_L$$

equation 18

When performing the pressure calculation at a node in the system, the fluid velocity value is unknown at the node so it is not factored into the pressure calculation. This is a valid assumption because the fluid velocity in a process system is designed to be fairly constant.

Atmospheric Pressure Calculations

The atmospheric pressure calculator in PIPE-FLO can be used to calculate the atmospheric pressure at any elevation ranging from -10,000 ft to 15,000 ft. The atmospheric pressure calculation is based on a regression of the 1976 U.S. Standard Atmosphere data (Reference 11). All specified or calculated gage pressures are relative to the specified atmospheric pressure. Atmospheric pressure varies with elevation and it is important to use the correct value. The atmospheric pressure is used in calculating the NPSHa of a system, which determines whether a pump will cavitate. It is also used in identifying partial or absolute vacuum conditions, choked flow in control valves, flashing in flow meters, and changes in fluid state.

Lineup Calculations

To perform the calculations for a PIPE-FLO lineup, hydraulic network analysis methods are used. The network calculation determines the pressures and flow rates of piping systems with Newtonian fluids and allows fluids to enter and leave the system at various nodes with different pressures and elevations.

As outlined previously, PIPE-FLO uses the Darcy-Weisbach method to calculate the pressure drop in the balancing calculations. This method allows for the solution of piping systems with varying fluid properties when a mass flow rate is selected. The calculation of the balanced flow rates in the system is carried out using the Simultaneous Path Adjustment method developed by Don J. Wood of the University of Kentucky. This method uses the Hardy Cross method until the program is close to a solution, then it switches to the linear method to complete the calculations.

PIPE-FLO has no fixed program limit to the number of pipes, nodes, and loops in a system. This is due to Microsoft Windows advanced memory management techniques and PIPE-FLO's Dynamic Allocation of Memory.

Network Flow Equations

In order to solve for the flow rates in all of the network paths, a series of simultaneous equations must be developed that include every pipeline in the network. The Kirchoff laws form the basis for the development of the equations used to balance the flow in the network and are listed below:

1 The algebraic sum of the mass flow into and out of all nodes must equal zero.

2 The algebraic sum of the pipeline pressure drops around a loop must equal zero.

Kirchoff Law 1: The Node Continuity Equation

In PIPE-FLO, a node is a point in the piping system where one or more pipelines are connected to the network. A node must have a specified elevation. Figure 3 illustrates a network node, NN, with four connecting pipelines.

Pipe 012 - 40 kg/hr

Pipe 001 - 20 kg/hr   Pipe 003 - 50 kg/hr

Node NN

Pipe 021 - 30 kg/hr
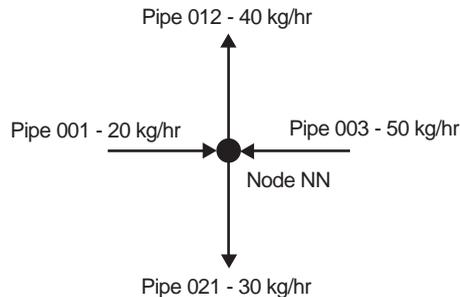
Figure 3

PIPE001 and PIPE003 have flows going into Node NN and PIPE012 and PIPE021 have flows going out of Node NN.

The following flow sign convention is used in this section of the reference book: flow into a node is assigned a negative value, and flow out of a node is assigned a positive value.

The first Kirchoff Law states that the algebraic sum of the flows at Node NN must be zero. In other words, the flow into and out of the node must balance.

This relationship gives the following continuity equation describing the flow at Node NN:

$$- W_{001} - W_{003} + W_{012} + W_{021} = 0$$

equation 19

$$W = \text{mass flow rate}$$

Flows leaving or entering the network by way of a node are referred to as demands on the network. All demands must enter and exit the network at nodes. The demands must also be accounted for when developing the Junction Node continuity equations. The sign convention for demands is the same as for pipelines, flow into the node is negative and flow out of the node is positive. The complete node flow continuity relationship is described below:

$$(- W_{001} - W_{003} + W_{012} + W_{021}) + W_{demands} = 0$$

equation 20

Each node in the piping network has a corresponding flow continuity equation. If the network consists of N nodes with all external flows or demands into and out of the network known, there exists N-1 independent equations describing the flows in the pipelines. Notice that the node flow continuity equation (equation 20) is linear.

In order to determine the flow rate in each pipeline, there must be as many independent equations as pipelines with unknown flow rates. The node equations only provide N-1 independent equations. The second Kirchoff law is used to develop the remaining equations necessary for the determination of the pipeline flow rates.

Kirchoff Law 2: The Loop Energy Equation

The network nodes are interconnected by pipelines, generating closed circuits or loops within the system. The second Kirchoff law states that the algebraic sum of the pressure drop (or energy loss) caused by the flow of fluid around a loop must equal zero. In other words, after completely tracing a loop, all of the pressure energy must be accounted for.

Figure 4 illustrates a network that will be used to demonstrate how the loop equations are developed.
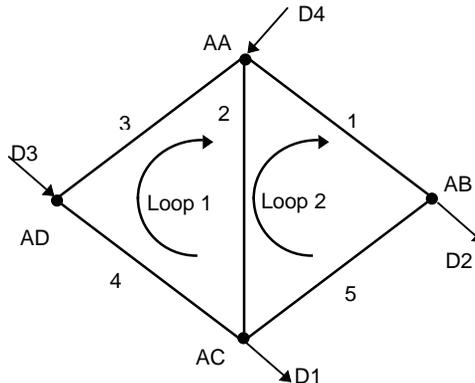
Figure 4

The network, which includes 5 pipelines and 4 nodes, consists of two loops. All of the nodes have flow demands. In developing the loop pressure drop equations, node AA will be used as the starting point for both loops. The first loop traces around pipelines 2, 4, and 3. The second loop traces around pipelines 1, 5, and 2.

In this user's guide, the sign convention for the pressure drop around the loop is positive for the clockwise direction and negative for the counterclockwise direction. If the flow in a pipeline is in a clockwise direction, the pressure drop in the pipeline is given a positive (+) value. If the flow in the pipeline is in a counterclockwise direction, the pressure drop in the pipeline is given a negative (-) value.

Using the Darcy-Weisbach equation plus the change in elevation in each pipeline ($dP = KW^2 + \Delta Z\rho$) to determine the pressure drop around the loop, the following loop pressure loss equations are developed:

Loop 1

$$(-dP_2) + (-dP_4) + (-dP_3) = 0$$

equation 21

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

<div align="right">*equation 22*</div>

Notice that in pipelines 2, 4, and 3 the flow directions are counter to the established standard clockwise loop direction. Therefore, the pressure drops in these pipelines are assigned a negative value. In a network, every non-overlapping loop provides one pressure drop equation. In the Figure 4 network, the loop with pipelines 1, 5, 4, and 3 is an overlapping loop so its pressure drop equation is not independent.

Now all of the necessary information is available for the development of the equations describing the network. With all demand flows known, the following equations can be used to solve the network in Figure 4:

$$\text{Node AA:} \quad -W_{D4} + W_1 + W_3 - W_2 = 0$$
$$\text{Node AB:} \quad -W_1 + W_{D2} + W_5 = 0$$
$$\text{Node AC:} \quad -W_5 - W_4 + W_{D1} + W_2 = 0$$

<div align="right">*equation 23*</div>

The loop pressure drop equations are:

Loop 1

$$(-dP_2) + (-dP_4) + (-dP_3) = 0$$

<div align="right">*equation 21*</div>

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

<div align="right">*equation 22*</div>

Notice that there are now five independent flow rate equations for the solution of the five-pipeline network.

In review, if all of the external demands are known in the network, there must be as many independent flow equations as pipelines in the network. The node equations provide N-1 independent flow equations, with N being the number of nodes in the network. The network loop pressure drop equations supply the remaining equations necessary to solve for the flow rate in each pipeline.

The relationship described above provides a very useful formula for determining the number and type of equations needed to solve a piping network. The following equation must be satisfied:

$$P = (N-1) + L$$

equation 24

> P = pipelines in the network
> N = nodes in the network
> L = non-overlapping loops in the network

Solving Systems with Unknown Demands

If a network is supplied from two or more tanks or pressure demands, then the flow of the fluid into or out of the network is not fixed but is a function of the head driving the fluid into the network. In other words, the demands into or out of the network are unknown at these tanks or pressure demands, and an equation must be developed to describe them. Since the pressure is fixed, it is not a function of the flow rate. In PIPE-FLO, a constant pressure is specified by installing a tank in the system or setting a pressure demand.

At a tank or pressure demand, the pressure is fixed at a specific value. This fixed pressure affects the flow rate in each network pipeline. A constraint in PIPE-FLO is that each network must have at least one tank or set pressure demand. This fixed pressure is used to supply a starting pressure point for the system pressure calculations.
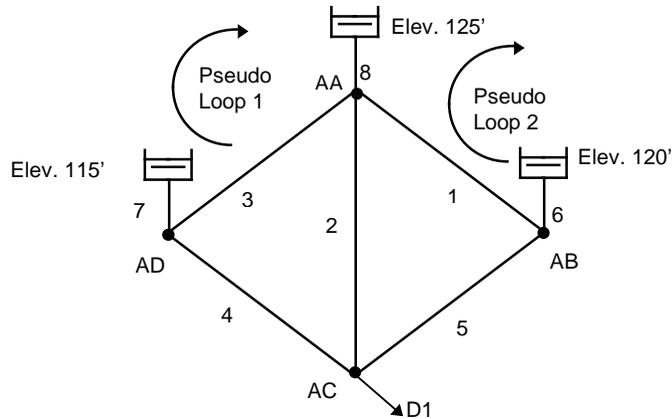
Figure 5

In Figure 5, there are three pressure nodes which are represented by a tank with an associated elevation. Between any two fixed pressures there exists a difference in elevation or hydraulic grade. This difference is accounted for in a "pseudo" or fictitious loop.

A pseudo loop consists of a path of pipelines in the network which connects two fixed pressure points (tanks or pressure demands). A fictitious pipeline is installed outside the actual network to connect the fixed pressure points, completing the loop. The pressure drop in the pseudo loop is fixed at a value that accounts for the difference in hydraulic grade between the two fixed pressure points. Equations 25 and 26 represent the pseudo loops for the network in Figure 5.

Pseudo loop 1

$$dP_8 + dP_3 - dP_7 = 125 - 115$$

equation 25

Pseudo loop 2

$$dP_6 - dP_1 - dP_8 = 120 - 125$$

equation 26

Notice that with three fixed pressure points there are two pseudo loop equations. This can be carried further to show that for P Pressure Nodes there are P-1 independent pseudo loop pressure drop equations.

Tanks

When installing a tank in PIPE-FLO, users have the following options:

- Specifying a tank geometry. There are ten different geometries to choose from.

- Defining a custom tank by specifying the tank volume as a function of liquid level.

- Specifying an infinite geometry/no geometry. This is used to model "tanks" whose geometry is effectively infinite, such as a large reservoir, and for which the addition or removal of fluid would not change the level or volume of the tank. You can also use this option if you are not concerned about the tank geometry.

Each tank can have up to six connecting pipelines, which can penetrate the tank at varying heights. The height is referenced to the bottom of the tank.

Volume Equations

When a specific tank geometry is selected (or a custom tank defined), PIPE-FLO calculates the tank volume for the specified liquid level. Conversely, using the Level Calculator feature, the program calculates a tank level given a specified volume. The equations below are obtained from References 9 and 10.

In the horizontal tank volume equations, the fluid cross-sectional area in the cylindrical section of the tank is calculated using equation 27:

$$A_f = R^2 \cos^{-1}[(R - h)/R] - (R - h)(2Rh - h^2)^{\frac{1}{2}}$$

equation 27

A_f = fluid cross-sectional area of fluid
R = radius of the tank cylindrical section
h = height of the fluid in the tank, measured from the lowest part of the tank to the fluid surface

The following variables are used in the horizontal tank equations below:

V_f = total fluid volume in the tank

A_f = fluid cross-sectional area of fluid

L = length of tank cylindrical section

a = distance tank heads extend beyond ($a > 0$) or into ($a < 0$) the tank cylindrical section

R = radius of the tank cylindrical section

h = height of the fluid in the tank, measured from the lowest part of the tank to the fluid surface

Horizontal Tank with Conical Heads

$0 \leq h < R$:

$$V_f = A_f L + (2aR^2/3)K$$

equation 28a

$h = R$:

$$V_f = A_f L + (2aR^2/3)\pi/2$$

equation 28b

$R < h \leq 2R$:

$$V_f = A_f L + (2aR^2/3)(\pi - K)$$

equation 28c

Where

$$K = \cos^{-1}(M) + M^3 + \cosh^{-1}(1/M) - 2M(1 - M^2)^{1/2}$$

$$M = |(R - h)/R|$$

Horizontal Tank with Ellipsoidal Heads

$$V_f = A_f L + \pi a h^2[1 - (h/3R)]$$

equation 29

Horizontal Tank with Spherical Heads

$h = R$, $|a| \leq R$:

$$V_f = A_fL + (\pi a/6)(3R^2 - a^2)$$

<div align="right">*equation 30a*</div>

$h = D$, $|a| \leq R$:

$$V_f = A_fL + (\pi a/3)(3R^2 - a^2)$$

<div align="right">*equation 30b*</div>

$h = 0$ or $a = 0$, R, $-R$:

$$V_f = A_fL + \pi ah^2[1 - (h/3R)]$$

<div align="right">*equation 30c*</div>

$h \neq R$, D; $a \neq 0$, R, $-R$; $|a| \geq 0.01D$:

$$V_f = A_fL + (a/|a|)\{(2r^3/3)[\cos^{-1}((R^2 - rw)/R(w - r)) +$$
$$\cos^{-1}((R^2 + rw)/R(w + r)) - (z/r)(2 + (R/r)^2)\cos^{-1}(w/R)]$$
$$-2(wr^2 - w^3/3)\tan^{-1}(y/z) + 4wyz/3\}$$

<div align="right">*equation 30d*</div>

$h \neq R$, D; $a \neq 0$, R, $-R$; $|a| < 0.01D$:

$$V_f = A_fL + (a/|a|)\{2 \int_w^R (r^2 - x^2)\tan^{-1}[(r^2 - x^2)/(r^2 - R^2)]dx - A_fz\}$$

<div align="right">*equation 30e*</div>

Where

$r = (a^2 + R^2)/(2|a|)$, $a \neq 0$

$a = \pm(r - [(r^2 - R^2)^{1/2}]$, $+(-)$ for convex (concave) heads

$w = R - h$

$y = (2Rh - h^2)^{1/2}$

$z = (r^2 - R^2)^{1/2}$

Horizontal Tank with Flat Heads

$$V_f = A_f L$$

equation 31

The following variables are used in the vertical tank equations below:

V_f = total fluid volume in the tank

a = distance tank heads extend beyond ($a > 0$) or into ($a < 0$) the tank cylindrical section

D = diameter of the tank cylindrical section

h = height of the fluid in the tank, measured from the lowest part of the tank to the fluid surface

Vertical Tank with Conical Bottom (Convex)

$h < a$:

$$V_f = (\pi/4)(Dh/a)^2(h/3)$$

equation 32a

$h \geq a$:

$$V_f = (\pi D^2/4)(h - 2a/3)$$

equation 32b

Vertical Tank with Conical Bottom (Concave)

$0 \leq h < |a|$:

$$V_f = (\pi D^2/12)[3h + a - (a + h)^3/a^2]$$

equation 33a

$h \geq |a|$:

$$V_f = (\pi D^2/12)(3h + a)$$

equation 33b

Vertical Tank with Ellipsoidal Bottom (Convex)

$h < a$:

$$V_f = (\pi/4)(Dh/a)^2(a - h/3)$$

equation 34a

$h \geq a$:

$$V_f = (\pi D^2/4)(h - a/3)$$

equation 34b

Vertical Tank with Ellipsoidal Bottom (Concave)

$0 \leq h < |a|$:

$$V_f = (\pi D^2/12)[3h + 2a - (a + h)^3(2a - h)/a^2]$$

equation 35a

$h \geq a$:

$$V_f = (\pi D^2/12)(3h + 2a)$$

equation 35b

Vertical Tank with Spherical Bottom (Convex)

$h < a$; $(a \leq D/2)$:

$$V_f = (\pi h^2/4)(2a + D^2/2a - 4h/3)$$

equation 36a

$h \geq a$; $(a \leq D/2)$:

$$V_f = (\pi h^2/4)(2a^3/3 - aD^2/2 + hD^2)$$

equation 36b

Vertical Tank with Spherical Bottom (Concave)

$0 \leq h < |a|$:

$$V_f = (\pi/12)\{3D^2h + (a/2)(3D^2 + 4a^2) + \\ (a + h)^3[4 - (3D^2 + 12a^2)/(2a(a + h))]\}$$

equation 37a

$h \geq a$:

$$V_f = (\pi/12)[3D^2h + (a/2)(3D^2 + 4a^2)]$$

equation 37b

Vertical Tank with Flat Bottom

$$V_f = \pi(D^2/4)h$$

equation 38

Rectangular Tank

$$V_f = XYh$$

equation 39

X, Y = cross-sectional dimensions
h = height of the fluid in the tank

Spherical Tank

$$V_f = (\pi/3)h^2(1.5D - h)$$

equation 40

D = tank diameter
h = height of the fluid in the tank

Custom Tank

To model a tank geometry that is not available in PIPE-FLO, users can specify a custom tank. A custom tank is defined by specifying the tank volume as a function of the liquid level. Up to five sets of data points can be entered. PIPE-FLO linearly interpolates between the specified level/volume data to calculate the volume at intermediate levels.

Pumps

When a pump is installed in the system, the energy or pressure rise due to the pump must be factored into the loop energy equations. A pump causes a head rise corresponding to the flow in the upstream pipeline. PIPE-FLO models pumps using specific pump head curve data. In this way, the actual head vs. flow relationship for that pump can be used in the network calculations. A typical head curve for a centrifugal pump is shown in Figure 6.

Pump Head Curve
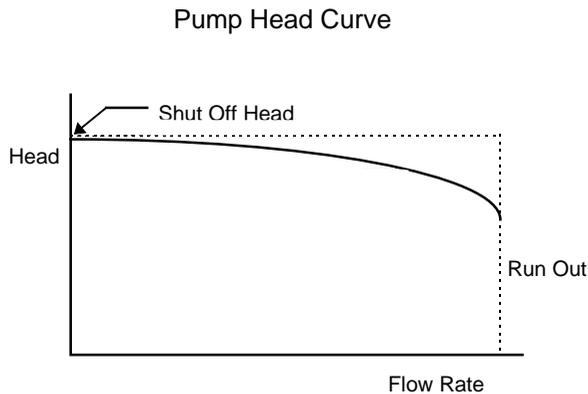
Figure 6

In addition to head data, users have the option of specifying efficiency and NSPHr (net positive suction head required) data. This data can also be imported into the piping model from a manufacturer's electronic catalog. When a system calculation is performed, PIPE-FLO linearly interpolates between the specified pump curve data.

Speed Adjustments

If either a minimum or a maximim speed is specified for a pump, the pump speed can be adjusted. PIPE-FLO performs this adjustment using the affinity laws. (See the Pump Selection section for a complete listing of these equations.) If variable speed operation is selected for a pump, PIPE-FLO automatically calculates the speed required to achieve a user specified flow rate.

Diameter Adjustments

Impeller diameters can also be changed in increments as specified by the pump manufacturer. The impeller can only be adjusted between the maximum and minimum diameters. For manually entered pump curves, impellers are trimmed by 1/16" or 1 mm.

Viscosity Corrections

If efficiency data is available, PIPE-FLO will automatically correct the curve for high viscosity when the pump is graphed and when a calculation is performed. See the Pump Selection section for complete information on the methodology PIPE-FLO uses to perform viscosity corrections.

Preferred Operating Region

For each pump, a preferred operating region (expressed as a percentage of the best efficiency point flow rate) is specified. The default values are 70% and 120%. Users can change or delete these values if desired. If the calculated pump flow rate is outside of the preferred operating region, a warning is issued.

Minimum Flow and Range Errors

If a pump that has been imported from a manufacturer's catalog has a minimum flow specified, PIPE-FLO compares the calculated flow rate to the minimum flow. If the calculated flow rate is below the minimum flow rate, a warning is issued.

If the calculated flow rate through the pump is beyond the data point range entered for the pump model, PIPE-FLO provides notification that the pump exceeds runout. If the pump can still be used with an increased flow rate, the pump should be modeled again with an extended flow rate range and the system calculation rerun. If it is not possible to use the same pump with a higher flow rate, another pump should be selected and installed in the system.

Running a Pump at a Fixed Flow Rate

Instead of entering performance curve data, the user can also specify a fixed flow rate and have PIPE-FLO calculate the total dynamic head required for the pump.

The following points about fixed flow rate pumps should be noted:

- When a fixed flow rate is specified for a pump, the performance curve data (if specified) is overridden.

- PIPE-FLO models a fixed flow pump by creating two nodes in the system where the pump is installed (refer to Figure 7). A flow demand value equal to the set flow rate is taken out of the first pump node (the suction node) and an equal flow demand is set entering the second node (the discharge node).

- If a pump status is listed as invalid, it means that you have set the flow rate lower than what would naturally occur through the line if the pump was not installed. The pump then acts as "valve" in order to limit the flow rate to the set value. If this occurs, you should either increase the set flow rate or remove the pump.

Fixed Flow Pump Model

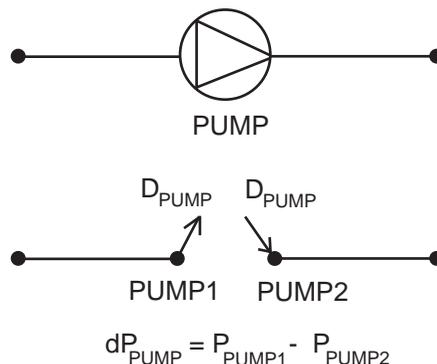

$$dP_{PUMP} = P_{PUMP1} - P_{PUMP2}$$

Figure 7

PIPE-FLO balances the network and calculates the pressure at each node. The program checks the calculated results to insure that the pump is operating properly (the suction pressure must be less than the discharge pressure).

Calculating NPSH Available

For each pump in the system, PIPE-FLO calculates the Net Positive Suction Head available (NPSHa) using the following equation:

$$NPSHa = (Pa - Pv + Pn)/\rho$$

equation 41

Pa = atmospheric pressure
Pv = vapor pressure (absolute)
Pn = gage pressure at the pump suction
ρ = fluid density

If NPSHr data is specified for a pump, PIPE-FLO checks that the available NPSH is greater than the pump's required NPSH multiplied by the margin ratio (NPSHa > NPSHr x margin ratio) when a calculation is performed. If the NPSHa is insufficient (NPSHa < NPSHr x margin ratio), a warning is issued.

Components

Any device with a head loss (pressure drop) vs. flow relationship can be modeled in PIPE-FLO, including filters, heat exchangers, air handlers, or similar devices. When a system calculation is performed, PIPE-FLO linearly interpolates between the specified curve data.

A constant pressure drop device can be modeled by specifying only one data point. When a calculation is performed, the pressure drop value specified will be the pressure drop across the component, regardless of the flow rate through the component.

If the calculated flow rate through the component is beyond the data point range entered for the component model, PIPE-FLO provides notification that the component is out of range. If this occurs, the range of flow rate data for the component should be extended or a different component should be used.

In addition to the generic component device, PIPE-FLO has a meter device available to model venturi, nozzle, and flat plate orifice differential pressure flow meters as well as balancing orifices. Please see the Flow Meters section.

Control Valves

Any time a network is analyzed with a PRV or BPV installed, it is always best to perform the initial analysis without the pressure controlling devices. Once a need for these devices has been identified, they should be installed one at a time so that their effect can be fully understood. Also, care must be taken not to disconnect the network due to the modeling of the pressure controlling devices.

Pressure Regulating Valves

The function of a pressure regulating valve is to lower the pressure in a section of the system to a specific downstream set pressure, regardless of the flow through the connecting piping.

The following points should be noted about PRVs:

- The set pressure at the outlet of the PRV is entered with the valve. For the PRV model, the program creates two nodes in the system where the PRV is installed (see Figure 8 below). A flow demand leaving the system is set at the first node (the inlet node), and a pressure demand equal to the PRV set pressure is set at the second node (the outlet node).

- The direction of flow through the PRV, from high pressure to low pressure, is determined from the direction of the connecting pipelines.

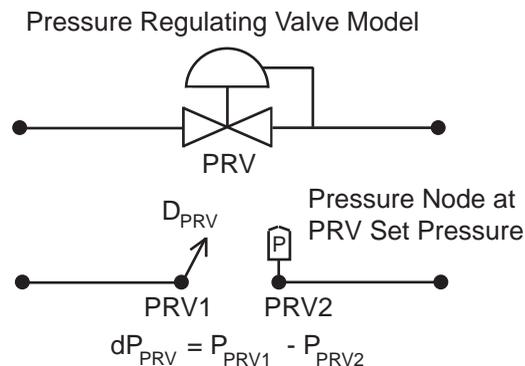- The PRV inlet and outlet pipelines cannot have a tank or pressure demand set on the other end.

Pressure Regulating Valve Model

Figure 8

When installing a PRV, there are three different conditions under which it can operate: (1) regulating, (2) fully closed, and (3) fully open. The manner in which the valve operates depends on the value entered for the set pressure. The fully closed and fully open positions represent the extreme valve positions. Each valve position is discussed below.

1 Regulating: The valve maintains the downstream pressure to the set value by throttling the flow rate through the PRV.

2 Fully Closed: This situation occurs if the valve set pressure is less than the pressure downstream of the valve for the case where the valve is closed. When this situation occurs in an actual piping system, the flow through the PRV reverses and the valve acts as a check valve, closing the pipeline. In PIPE-FLO, the flow also reverses, however the PRV does not act like a check valve. The pipeline remains open and the PRV results are invalid because the flow is going into the Pressure Node from the down-stream pipeline node in order to maintain the set pressure value.

3 Fully Open: This situation occurs if the valve set pressure is greater than the pressure at the valve inlet for the case where the valve is fully open. When this situation occurs in an actual piping system, the PRV maintains a fully open position and it has no effect on the pipeline flow conditions. In PIPE-FLO, the differential pressure across the valve will correspond to a pressure gain rather than a pressure drop. The PRV results are invalid because it is acting as a pump rather than as a pressure control.

You can close a valve by clicking on it with the Open/Close tool. This closes both the inlet and outlet pipelines.

If the valve status is listed as invalid, the flow direction should be checked to determine if condition 2 or 3 is occurring. If the flow is opposite the assumed direction, the valve should be closed and the calculations run again. If the flow is in the assumed direction, the PRV setting should be set to fully open and the calculations run again.

In most cases, PRV operation problems can be avoided by first determining the valve's pressure regulating range. This is done by running two lineup calculations which simulate the fully open and fully closed valve positions. These lineups provide the pressure range over which the valve will regulate the flow. For the fully open case, the PRV setting should be set to fully open. For the fully closed case, the valve should be closed. The maximum pressure the PRV can be set at is equal to the pressure at the valve inlet for the fully open case. The minimum pressure the PRV can be set at is equal to the pressure downstream of the valve for the fully closed case.

Back Pressure Valves

Back Pressure Valves are installed in systems to keep a section of the network above a minimum set pressure. A BPV maintains the set pressure on the high pressure or upstream side of the valve and only allows enough flow through the valve to maintain the upstream BPV set pressure.

The following points should be noted about BPVs:

- The set pressure at the upstream side of the BPV is entered with the valve. For the BPV model, the program creates two nodes in the system where the BPV is installed (see Figure 9 below). A pressure demand equal to the BPV set pressure is set at the first node (the inlet node), and a flow demand is set at the second node (the outlet node).

- The direction of flow through the BPV, from high pressure to low pressure, is determined from the direction of the connecting pipelines.

- The BPV inlet and outlet pipelines cannot have a tank or pressure demand set on the other end.
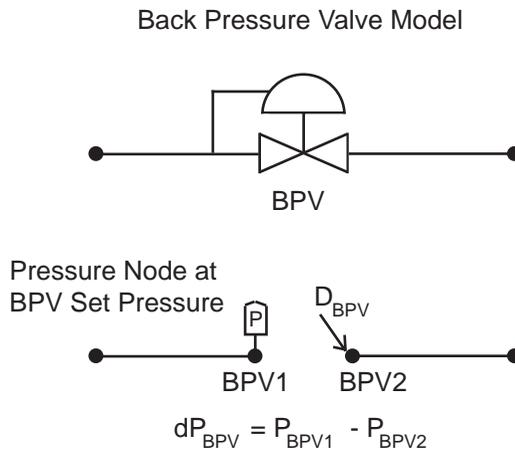
Back Pressure Valve Model

BPV

Pressure Node at
BPV Set Pressure $\quad$ D_{BPV}

BPV1 $\qquad$ BPV2

$$dP_{BPV} = P_{BPV1} - P_{BPV2}$$

Figure 9

When installing a BPV in a pipeline, there are three different conditions under which it can operate: (1) regulating, (2) fully closed, and (3) fully open. The manner in which the valve operates depends on the value entered for the set pressure. The fully closed and fully open positions represent the extreme valve positions. Each valve position is discussed below.

1 Regulating: The valve maintains the upstream pressure to the set value by regulating the flow rate through the BPV.

2 Fully Closed: This situation occurs if the valve set pressure is greater than the pressure upstream of the valve for the case where the valve is closed. When this situation occurs in an actual piping system, the flow through the BPV reverses and the valve acts as a check valve, closing the pipeline. In PIPE-FLO, the flow also reverses, however the BPV does not act like a check valve. The pipeline remains open and the BPV results are invalid because flow is going from the Pressure Node to the pipeline upstream node.

3 Fully Open: This situation occurs if the valve set pressure is less than the pressure at the valve outlet for the case where the valve is fully open. When this situation occurs in an actual piping system, the BPV maintains a fully open position and it has no effect on the pipeline flow conditions. In PIPE-FLO, the differential pressure across the valve will correspond to a pressure gain rather than a pressure drop. The BPV results are invalid because it is acting as a pump rather than as a pressure control.

You can close a valve by clicking on it with the Open/ Close tool. This closes both the inlet and outlet pipelines.

If the valve status is listed as invalid, the flow direction should be checked to determine if condition 2 or 3 is occurring. If the flow is opposite the assumed direction, the valve should be closed and the calculations run again. If the flow is in the assumed direction, the BPV setting should be set to fully open and the calculations run again.

In most cases, BPV operation problems can be avoided by first determining the valve's pressure regulating range. This is done by running two lineup calculations which simulate the fully open and fully closed valve positions. These lineups provide the pressure range over which the valve will regulate the flow. For the fully open case, the BPV setting should be set to fully open. For the fully closed case, the valve should be closed. The maximum pressure the BPV can be set at is equal to the pressure upstream of the valve for the fully closed case. The minimum pressure the BPV can be set at is equal to the pressure at the valve outlet for the fully open case.

Flow Control Valves

Flow control valves (FCVs) maintain the flow rate in a line to a fixed value and calculate the differential pressure across the control required to regulate the flow rate.

The following points about FCVs should be noted:

- The direction of flow through the FCV is determined from the direction of the connecting pipelines.

- PIPE-FLO models an FCV by creating two nodes in the system where the valve is installed (refer to Figure 10). A flow demand value equal to the set flow rate is taken out of the first node (the inlet node) and an equal flow demand is set entering the second node (the outlet node).

- If an FCV is status is Invalid, you should first check the direction of flow through the line. If the flow direction is correct, the invalid flag means that you have set the flow rate in the FCV to a value higher than would naturally occur in the line and the FCV is acting as a "pump" in order to achieve the set flow rate. In an actual piping system, the valve would be 100% open, so you should change valve setting to fully open and let PIPE-FLO calculate the flow rate.
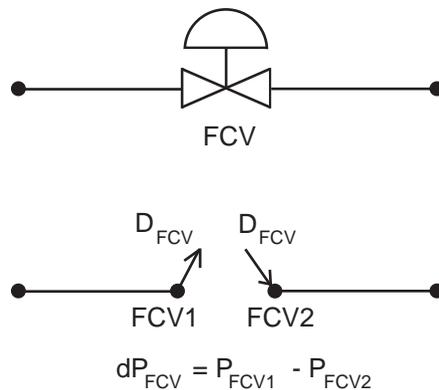
Flow Control Valve Model

FCV

D_{FCV} D_{FCV}

FCV1   FCV2

$$dP_{FCV} = P_{FCV1} - P_{FCV2}$$

Figure 10

PIPE-FLO balances the network and calculates the pressure at each node. The program checks the calculated results to insure the flow control is operating properly. For the FCV, the inlet pressure must be greater than the outlet pressure.

Using Control Valve Catalog Data

For valves set to a manual position, the valve inlet pressure is assumed to be equal to the pressure specifed in the upstream fluid zone.

For each type of control (PRV, BPV, and FCV), you can import control valve catalog data. PIPE-FLO can then calculate the required valve position for a variety of valve settings and system configurations. Manual control operation can also be modeled by fixing the valve to a set position. Calculations for control valves with catalog data are performed using the equations outlined in the Control Valve Selection section.

Valve data can also be manually entered or imported from manufacturers' valve selection programs when the programs have export capability. The calculations for these valves are performed using the same equations that are used for catalog valves.

NOTE: The required valve file format for manufacturers' exported files is documented in the cvalve.xls file located in the same folder as the PIPE-FLO program.

Calculation of Lineup Flow Rates and Pressures

PIPE-FLO uses the Simultaneous Path Adjustment method to calculate the balanced flow in each pipeline. This method uses the Hardy Cross method until the program is close to a solution. It then switches to the linear method to complete the solution.

First, PIPE-FLO calculates the flow rates in the pipelines. These are the flow rates that will balance the flow at each node and also balance the pressure drop around every loop in the network. PIPE-FLO has an algorithm that provides an initial set of flow rates that are balanced at the nodes. This initial set of flow rates does not necessarily provide the flows needed to balance the pressure drops around the loops. The initial guess is refined until a set of flow rates are arrived at that meet both the node continuity and loop energy balance requirements. This requires an iterative solution, one in which the flow rates are corrected until the desired accuracy is achieved.

Hardy Cross Method

In the Hardy Cross method, flow rates are assumed for each pipeline so that continuity is satisfied at every node in the network. A correction to the flow in each network loop is then computed in turn and applied to bring the loops into closer balance. The advantage to using the Hardy Cross method is that convergence problems are not experienced even if the initial assumed flow rates are a great deal off from the final solution. For further information on this method, refer to any standard fluid mechanics text book.

Linear Method

At the specified linear switch point, PIPE-FLO shifts to the linear method to calculate the flow rates and pressures. The program default for the linear switch point is 10%. The linear method is faster than the Hardy Cross method but the initial flow rates must be fairly close or convergence problems may be encountered. The continuity equations and the loop pressure drop equations provide the independent equations necessary to perform the network calculations for this method.

Since the pressure drop equations are non-linear equations they must be "linearized" before they can be inserted into the matrix to arrive at the final solution. To accomplish this, the Newton-Raphson technique is used. Simply stated, the function describing the loop pressure drop is divided by the first derivative of that function. The result is a first order or linear equation that can be used in the matrix calculations.

PIPE-FLO uses the head gradient corrective flow rate method to describe the loop pressure drop equations. The gradient, or pressure drop, around the loop is expressed in three parts: the pressure drop caused by the pipe, the pressure drop caused by the valves and fittings, and the pressure drop (gain) due to the components and/or pumps. The pressure drop relationship can be written as follows:

$$dP = K_P W^n + K_{V\&F} W^2 - (P_o + K_c W^c)$$

equation 42

Taking the first derivative of the loop pressure drop equation (equation 42) produces the following expression.

$$dP' = nK_P W^{n-1} + 2K_{V\&F}W - (cK_c W^{c-1})$$

equation 43

The result of dP / dP' is a linear equation that is used to solve for the flow rate in each pipeline in the network.

To arrive at the solution in the least number of iterations, a correction factor is needed for each loop in the network. The head gradient method is used to arrive at this correction factor. The correction factor for each loop is expressed as the pressure drop change that is required to balance the pressure drops around the loop. These pressure drops are in terms of the flow changes needed in the loop and the flow changes needed in adjacent loops. For more information on the linear method and the head gradient method please refer to References 2 and 3.

All of the simultaneous linear equations necessary to solve the flow rates in each pipeline in the network are now available. PIPE-FLO uses the Gaussian method with pivotal condensation to solve the simultaneous equations. All matrix operations are carried out using the Microsoft "C" language with double precision accuracy. This greatly reduces division rounding errors.

Calculational Controls

In the Lineup Calculation dialog box, the user can specify the percent tolerance, the maximum number of iterations, the laminar flow cutoff Reynolds number, as well as the linear switch point (the percent tolerance where the program will switch from the Hardy Cross method to the linear method). When this screen is first displayed, defaults are entered for all of these values. The default values are recommended for all lineup calculations unless convergence problems are encountered.

System Resistance Curve

Users can generate a system resistance curve for any piping system. The following limitations apply:

- There can only be one active pump in the system (all other pumps are shut off).

- The program sets all controls (FCVs, PRVs, and BPVs) in the system to fully open.

- The program sets all flow demands in the system to zero.

PIPE-FLO generates the system resistance curve by applying five different fixed flow rates to the selected pump and calculating the corresponding total head for these flow rates. The data is then plotted against the pump curve.

System Volume Calculations

The total system volume is calculated as follows:

$$V = \sum A_i L_i + \sum V f_i$$

equation 44

V = total system volume
A_i = cross-sectional area of a pipe
L_i = length of a pipe
Vf_i = tank fluid volume

The total volume includes both open and closed pipelines as well as pipelines that are disconnected from the system. The tank volumes are calculated using the liquid levels in the active lineup.

NOTE: The system volume printed on the list report includes the tank volumes calculated using the liquid levels for the active lineup. The bill of materials report lists the maximum total tank volume, which assumes the maximum liquid level for each tank.

References

1 Crane, Crane Technical Paper 410, "Flow of Fluids through Valves, Fittings, and Pipe" twenty fourth printing, Crane Company 1988.

2 Wood, "BASIC Computer Program for the Analysis of Pressure and Flow in Pipe Distribution Systems Including Extended Period Simulations", Office of Engineering Continuing Education, College of Engineering, University of Kentucky, 1982.

3 Jeppson, "Analysis of Flow in Pipe Networks", fourth printing, Ann Arbor Science Publishers, Inc. 1981.

4 Anton, "Elementary Linear Algebra", third edition, John Wiley & Sons, Inc. 1981.

5 Brater and King, "Handbook of Hydraulics" sixth edition, McGraw-Hill, New York 1976.

6 Vennard, "Elementary Fluid Mechanics" fourth edition, John Wiley & Sons, Inc., New York, London, 1961.

7 Streeter - Wylie, "Fluid Mechanics" eighth edition, McGraw Hill, Inc., New York, 1985.

8 Lindeburg, "Civil Engineering Reference Manual" seventh edition, Professional Publications, Inc., 1999.

9 Jones, D., "Computing Fluid Tank Volumes," Chemical Processing, November 2002.

10 Jones, D., "Computing Fluid Volumes in Vertical Tanks," Chemical Processing, December 2003.

11 U.S. Standard Atmosphere, 1976, U.S. Government Printing Office, Washington, D.C., 1976.

PUMP SELECTION

PIPE-FLO's pump selection module selects and evaluates centrifugal pumps. Centrifugal is a broad classification of pumps which use kinetic energy to move the fluid. They use the centrifugal force of a rotating impeller to impart kinetic energy to the fluid (as opposed to jet pumps and eductors).

The Hydraulic Institute Standards (Reference 1) is the basis for the pump selection module. The Hydraulic Institute is composed of organizations and individuals who manufacture and sell pumps in the open market. When there is a discrepancy between the program reference book and the current revision of the Hydraulic Institute Standards, the Standards take precedence.

PIPE-FLO selects pumps from a pump catalog and evaluates their operation in an application. Within the range of the manufacturer's recommendations, the program allows you to adjust the pump parameters and see the effect it has on the pump operation.

Definitions

The definitions that follow are found in Reference 1.

Head The quantity used to express a form (or combination of forms) of the energy content of the liquid, per unit weight of the liquid, referred to any arbitrary datum. All head quantities are in terms of foot-pounds of energy per pound of liquid, or feet of liquid.

Flow The unit of flow rate in the United States is expressed in units of gallons per minute (gpm). The standard fluid for all pump curves is water at 60 °F.

NPSH The net positive suction head is the total suction head in feet of liquid (absolute) determined at the suction nozzle and the referred datum less the vapor pressure of the liquid in feet (absolute). NPSHa is the net positive suction head available in the pumping system. NPSHr is the net positive suction head required by the pump.

Pump Input The horsepower delivered to the pump shaft (designated as brake horsepower).

Pump Efficiency The ratio of the energy delivered by the pump to the energy supplied to the pump shaft (the ratio of the liquid horsepower to the brake horsepower).

Pump Head Curve

Pump vendors perform pump tests to determine the operating characteristics of the pumps they manufacture. The pumps are tested as outlined in Reference 1. All pump data used by PIPE-FLO is supplied by the pump manufacturers who are solely responsible for the content.

Catalog Search Criteria

PIPE-FLO compiles a list of pumps that meet the criteria specified by the user. This criteria includes the manufacturer's catalog, the types and speeds, and the design point (head and flow rate). The catalog search can be further limited by specifying that the design point must be to the left of a pump's best efficiency point (BEP). Users can also specify a head tolerance, which is expressed as a percentage of the design point head. Specifying a tolerance places pumps on the selection list that do not meet the design point, or are outside of the selection window. This allows the consideration of pumps that are "near misses."

Advanced Search Criteria

Advanced criteria can be used to further refine the catalog search. Pumps that do not meet one or more of the specified criteria are listed in red on the selection list. A detailed warnings list is also provided for each pump. The available criteria is listed below:

Preferred Operating Region

The preferred operating region has a lower and upper limit and is expressed as a percentage of the best efficiency point (BEP) flow rate. If the design point is outside of the preferred operating region, a warning flag is issued.

Secondary Operating Point

The secondary operating point is a flow rate and head specified by the user. If the secondary operating point is outside of the selection window, a warning flag is issued.

Pump Limits

Max temperature: The maximum operating temperature is compared to the maximum temperature specified in the catalog by the manufacturer. If the maximum temperature value exceeds the catalog value, a warning flag is issued.

Max suction pressure: The maximum pressure at the pump suction is added to the shutoff dP and then this sum is compared to the maximum pressure specified in the catalog by the manufacturer. If the maximum suction pressure plus the shutoff dP is greater than the catalog maximum pressure, a warning flag is issued.

Max sphere size: The largest particle size in the fluid is compared to the maximum sphere size specified in the catalog by the manufacturer. If the maximum sphere size exceeds the catalog value, a warning flag is issued.

Max power: The maximum power for the pump assembly or magnetic drive is compared to the power limit specified in the catalog by the manufacturer. If the maximum power exceeds the catalog value, a warning flag is issued.

Max suction specific speed (Nss): The maximum suction specific speed is compared to the suction specific speed specified in the catalog by the manufacturer. If the catalog value exceeds the maximum suction specific speed, a warning flag is issued.

Curve Limits

Minimum trim: The design curve diameter, expressed as a percentage of the pump's maximum impeller diameter. If the selected design curve diameter is greater than this value, a warning flag is issued.

Min head rise: The minimum head rise from the design point to shutoff, expressed as a percentage from the design point head to the shutoff head. If a pump's head rise from design point to shutoff is less than this value, a warning flag is issued.

Pump Sizing

Each pump in the catalog can have up to ten impeller diameters or speed curves. Each curve can have up to twenty data points describing the pump operation. The data points for the curve consist of the flow rate and head, and optionally the pump's efficiency (or power) and NPSHr.

When the design point of the pump falls between a set of known curves, the program interpolates between them to arrive at a calculated curve. Often manufacturers allow impeller diameters to be adjusted only in fixed increments of their choosing. For example, a manufacturer can force the program to limit the impeller diameter increments to 0.125 inch. Or, they may not allow any trimming of the impellers.

The Affinity Rules

In hydraulically similar pumps, the head and capacity of a pump vary with the rotational speed of the impeller in such a way that the pump head curves retain their characteristic features. The variation of head, capacity, and brake horsepower follow a set of ratios that are known as the Affinity Rules. These rules are expressed in equations 1a, 1b and 1c.

$$(Q_1/Q_0) \ = \ (N_1/N_0)(D_1/D_0)$$

equation 1a

$$(H_1/H_0) \ = \ (N_1/N_0)^2(D1/D_0)^2$$

equation 1b

$$(P_1/P_0) \ = \ (N_1/N_0)^3(D_1/D_0)^3$$

equation 1c

Q = Capacity, US gpm
N = Impeller speed, rpm
D = Impeller diameter
H = Pump head, ft
P = Pump power, hp
Subscripts
    0 = Pump test speed or diameter
    1 = New pump speed or diameter

Multi-stage Pumps

Pumps which have multiple impeller stages are designated as multi-stage pumps. For these pump types, the single stage base impeller curves and the impeller trim increment are specified in the manufacturer's catalog along with the range of allowed impeller combinations.

During the pump selection process, PIPE-FLO determines the number of full diameter impeller stages necessary to achieve the design point. Once the number of stages has been determined, the program calculates the impeller diameter needed to go through the design point.

Adjustable Speed Pumps

Some pump manufacturers have pumps available in an Adjustable speed class. These pumps can be stored in a catalog under two different formats. Pumps that use the first format have one speed curve specified along with a maximum speed and a minimum speed. When adjustable speed pumps with this format are selected, PIPE-FLO uses the affinity rules to calculate the speed needed to pass through the specified design point. Pumps that use the second format have up to ten speed curves stored per pump. When the design point of the pump falls between a set of known speed curves, the program interpolates between them to arrive at a calculated curve.

Multiple Pump Configurations

Multiple pumps for both parallel and series configurations can be analyzed. To plot the performance curve for multiple pumps in series, PIPE-FLO multiplies the head values of the single pump curve by the number of pumps in series. The flow values for series configurations are the same as those for a single pump.

To plot the performance curve for multiple pumps in parallel, PIPE-FLO multiplies the flow rate values for a single pump by the number of pumps in parallel. The head values for parallel configurations are the same as those for a single pump.

Net Positive Suction Head

The Net Positive Suction Head (NPSH) is the value of the minimum suction head required to prevent cavitation in a pump. Cavitation is the rapid formation and collapse of vapor pockets in regions of very low pressure.

In a centrifugal pump, cavitation causes a decrease in a pump's efficiency and is capable of causing physical damage to the pump and impeller. Since cavitation has such a detrimental affect on a pump, it must be avoided at all costs. Cavitation can be avoided by keeping the NPSH available (NPSHa) greater than the NPSH required (NPSHr).

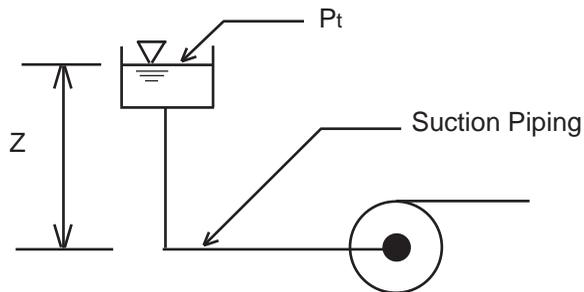The NPSH available can be entered directly or calculated using the NPSHa Calculator.

Figure 1

The formula used to calculate the NPSHa is as follows (refer to Figure 1 above):

$$NPSHa = ((Pt - Pvp)/\rho) + Z - HL$$

equation 2

P_t = absolute pressure on the free surface of the liquid in the tank connected to the pump suction

P_{vp} = pumping fluid vapor pressure in absolute pressure units at the operating temperature

ρ = fluid density

Z = static suction head (this value is negative if a suction lift condition exists)

HL = head loss due to friction in the pipeline between the tank and the pump suction.

The units of NPSH are in feet of fluid absolute.

An NPSH margin ratio can also be specified. This factor is applied to the pump's NPSHr value. PIPE-FLO checks that the NPSHa is greater than the pump's NPSHr multiplied by the margin ratio. If it is not, a warning flag is issued.

The NPSHr for a pump is determined by the pump manufacturer and is listed in their catalog. The NPSHr values are arrived at through actual tests as outlined in Reference 1.

If the fluid is a hydrocarbon, or high temperature water, then the required NPSH of the pump may be reduced as outlined in Reference 1. Using the *NPSH Reductions for Pumps Handling Hydrocarbon Liquids and High Temperature Water* chart found in Reference 1, it is possible to reduce the NPSHr values specified by the vendor without causing cavitation.

The PIPE-FLO program does not perform the NPSH reduction calculations. If based on your experience you can reduce the NPSH requirements of the pump, the reduction value should be subtracted from the value presented by PIPE-FLO. Always check with your pump supplier when adjusting the NPSH requirements.

Temperature Variations

Variations in the temperature of the fluid being pumped cause changes in the fluid density. Any reduction in the fluid density results in a reduction of the liquid horsepower, along with a proportional reduction to the input power. As a result, there is little or no change in the pump's efficiency.

Viscosity Variations with Hot Water

The viscosity of a fluid has the greatest impact on the pump curves. Variations in fluid viscosity also have an influence on the pump's efficiency. The changes in efficiency are due to:

- Internal leakage losses within the pump
- Disc friction losses
- Hydraulic skin friction losses

When pumping hot water in circulating pumps, Reference 1 allows vendors to adjust the performance data of their pump using an empirical formula. However, PIPE-FLO does not perform the efficiency variation corrections for circulating hot water.

Viscosity Variations with Viscous Fluids

The viscosity of oils and other viscous fluids (as compared to water) has a more pronounced impact on the operating conditions of the pump. Pumps that are tested with water but are used to transport viscous fluids must have their head, flow, and efficiency values corrected to approximate their performance with the viscous fluid.

The methodology outlined in Reference 6 is used by PIPE-FLO to correct the pump performance curves for viscous conditions. The correction equations are based on a pump performance Reynolds number adjusted for specific speed (parameter B), which has been statistically curve-fitted to a body of test data.

The parameter B is calculated as follows:

$$B = 26.6 * [(\nu^{1/2}) * (H_{BEP-W}^{0.0625})]/[(Q_{BEP-W}^{0.375}) * (N^{0.25})]$$

equation 3

ν = kinematic viscosity of the pump liquid, centistokes (cSt)

Q_{BEP-W} = the water flow rate at the best efficiency point, US gpm

H_{BEP-W} = the water head per stage at the best efficiency flow rate, ft

N = pump speed, rpm

The test data includes conventional single-stage and multi-stage pumps, and covers the following ranges:

1 cSt < ν < 3000 cSt

13 US gpm < Q (@ BEP) < 1140 US gpm

20 ft < H per stage (@ BEP) < 430 ft

B < 40

NOTE: PIPE-FLO uses 4.3 cSt for the lower limit of the viscosity range (this corresponds to the value listed in Reference 1 and was used in previous versions of the program).

The correction equations are not exact for any particular pump, but are rather a generalized method based on empirical data. This method may be applied to pump performance outside the ranges listed above, however the uncertainty of the performance prediction is increased. In such situations, PIPE-FLO still corrects the pump performance data, then issues a warning to indicate that there is increased uncertainty in the performance prediction.

For a complete listing of the correction equations, please see Reference 6.

Curve Corrections & Losses

In some cases, it may be necessary to apply hydraulic correction factors to a pump's performance data. For example, with vertical turbine pumps, the performance can vary depending on the material used for the impeller. Solids in suspension also affect the operation of a pump, depending on the both the percentage and nature of the solids. Reference 1 does not offer a recommendation for the modification of the pump data in these cases. However, through the use of hydraulic correction factors, PIPE-FLO allows for the modification of the pump data for specific pumping applications.

To adjust the pump data, the program multiplies the appropriate water pump data (head, flow rate, NPSHr and efficiency) by the corresponding correction factors specified by the user. Users also have the option of specifying additional power losses to account for mechanical seal losses.

The correction factors and losses are values that a user should have obtained from his or her own experience or preferably from the pump manufacturer. In the case of slurries, pump vendors should be consulted regarding the impact of solids in suspension on the operation of specific pumps.

NOTE: Speed adjustments on a pump are done prior to applying the hydraulic curve corrections. If viscous conditions exist (viscosity of 4.3 centistokes or greater), hydraulic curve corrections are first applied to the pump performance data, then the viscosity corrections. Power losses are applied after both the curve corrections and viscosity corrections are performed.

Motor Sizing

Motor size tables used by PIPE-FLO can contain up to four different standards. Each standard can have a maximum of four enclosure types. For each standard and enclosure type, the table contains speed, frame, and motor efficiency data.

When specifying the standard and enclosure type to use, the user also specifies the sizing criteria. There are three different criteria available: the power required at the design point flow rate, the maximum power required on the design curve, and the maximum power required for the maximum impeller diameter.

Based on the sizing criteria specified by the user, PIPE-FLO automatically sizes the motor for each pump that is put on the selection list. The smallest motor that meets the sizing criteria is selected. When performing operating cost analyses, the program uses the efficiency data stored in the motor size table.

Energy Cost

The energy cost is the cost of the power required to run a pump for one year. PIPE-FLO can calculate the annual energy cost for pumps running under both fixed and variable speed conditions. This cost information provides another parameter for consideration when comparing the advantages of using one pump over another. It also provides useful information for determining if the cost savings associated with operating a variable speed pump justifies the cost of the variable speed drive.

PIPE-FLO uses information from the operating load profile and the manufacturer's pump curve to calculate the energy cost. If the cost is being calculated for a variable speed drive pump, the resistance curve information is used as well. The pump and motor efficiencies are also factored into the calculation.

For a fixed speed pump, the sequence outlined below is followed for each load specified in the operating load profile.

The brake horsepower is calculated:

$$bhp = Q*TH* \rho /(247,000 * eff_p)$$

equation 4

bhp = brake horsepower
Q = flow rate, US gpm
TH = total head, ft
ρ = fluid density, lb/ft^3

$$eff_p = \text{pump efficiency}$$

The electrical horsepower is calculated:

$$ehp = bhp / eff_m$$

<div align="right">*equation 5*</div>

$$ehp = \text{motor electrical horsepower}$$
$$eff_m = \text{motor efficiency}$$

The cost for the load is calculated:

$$\text{Cost/Load} = ehp * (0.7457 \text{ kW/hp}) * T * COST$$

<div align="right">*equation 6*</div>

$$T = \text{duration of load, hrs/yr}$$
$$COST = \text{power cost, per kWh}$$

Once this process is completed, the total annual energy cost is determined by summing up the costs calculated for each specified load in the profile.

Calculating the energy cost for a variable speed pump requires two more steps for each load specified in the operating load profile. First, the speed of the pump is determined at which the pump curve intersects the selected resistance curve at the required flow rate. Next, the operating condition of the pump is determined for the required speed using the affinity rules. The procedure then follows that for the fixed speed drive as outlined above.

Life Cycle Cost

The life cycle cost (LCC) for a pump is the total "lifetime" cost to purchase, install, operate, maintain, and dispose of the pump. The LCC is calculated as follows:

$$LCC = C_i + C_{in} + df * \Sigma[C_e + C_m + C_o + C_s + C_{annual\ other}]$$

$$+ [C_d + C_{d\ other}]/[1 + (i - p)]^n$$

equation 7

C_i = initial cost
C_{in} = installation and commissioning cost
df = discount factor, a total sum factor over the n years for the present value
C_e = annual energy cost
C_m = annual maintenance and repair cost
C_o = annual operating cost
C_s = annual downtime cost
$C_{annual\ other}$ = annual miscellaneous costs
C_d = disposal and decommissioning cost
$C_{d\ other}$ = miscellaneous costs associated with decommissioning the equipment
i = interest rate
p = inflation rate
n = number of years

NOTE: The annual energy cost used in calculating the life cycle cost is the last type of energy cost calculated (fixed speed or variable speed).

For more information, refer to Reference 7.

References

1 Hydraulic Institute Standards for Centrifugal, Rotary & Reciprocating Pumps, 14th edition, Hydraulic Institute, 1983.

2 Pump Handbook, Igor J. Karassik, William C. Krutzsch, Warren H. Fraser, and Joseph P. Messina, editors; McGraw-Hill, Inc., 1976.

3 Science and Engineering Programs, edited by John Heilborn, McGraw-Hill, Inc., 1981.

4 Crane, Crane Technical Paper 410, "Flow of Fluids through Valves, Fittings, and Pipe" twenty fourth printing, Crane Company 1988.

5 Engineering Handbook AODD Pumps, Yamada America, Inc.

6 Effects of Liquid Viscosity on Rotodynamic (Centrifugal and Vertical) Pump Performance, ANSI/HI 9.6.7, 2004.

7 Pump Life Cycle Costs: A Guide to LCC Analysis for Pumping Systems, Europump and Hydraulic Institute, 2001.

CONTROL VALVE SELECTION

Introduction

The sizing and selection of valves is based upon many piping system variables. PIPE-FLO's control valve selection module analyzes the operation of the valve while taking into account the hydraulic characteristics of the system.

The source document for the sizing calculations used in PIPE-FLO's control valve selection module is the American National Standard Institute, Instrument Society of America standard ANSI/ISA-S75.01-1985 (R 1995) *Flow Equations for Sizing Control Valves*.

PIPE-FLO provides a quick and effective means of evaluating various valves using the ANSI/ISA standard. The specific formulas from this standard used are supplied in this reference book. When there is a discrepancy between the standard and the reference book, the standard takes precedence.

The majority of valve manufacturers have adopted the ANSI/ISA standard for the sizing of their valves and they provide the necessary factors in their valve catalogs. When there is a discrepancy between a valve manufacturer's data and the data presented in the supplied catalog disks, you should contact the valve manufacturer.

A few valve manufacturers have developed their own proprietary formulas and factors for sizing their valves. PIPE-FLO does not use any proprietary method for sizing valves. Using the ANSI/ISA standard allows the valves from various manufacturers to be compared with the same approach, thus insuring that the best valve is selected for the application.

PIPE-FLO's control valve selection module is not intended to be used to design or test valves.

Sizing Valves For Liquid Service

The equations for the flow of a non-compressible fluid through a valve as described by the standard and used in PIPE-FLO's control valve selection module are as follows:

$$w = N_6 \, F_P \, F_R \, C_v \, (dP/\rho)^{\frac{1}{2}}$$

equation 1

N_6 = conversion coefficient
C_v = valve flow coefficient, dimensionless
F_P = piping geometry factor, dimensionless
F_R = Reynolds number factor, dimensionless
    (F_R = 1 for turbulent flow)
dP = pressure drop across the valve (P_1-P_2), pressure units
ρ = fluid density, mass/unit volume

Cv Valve Flow Coefficient

The flow coefficient (C_v) describes the flow vs pressure relationship through a valve. By definition, C_v is the number of gallons per minute of 60°F water which will pass through a valve with a fixed pressure drop of 1 psi.

The valve manufacturer supplies the C_v value of the valve for various valve body types, sizes, trim characteristics, and valve positions. The C_v value stored in the valve catalog is a function of the valve travel at 5%, 10%, 20% and every subsequent 10% of rated travel up to and including 100%. PIPE-FLO performs a linear interpolation to determine the Cv values for positions between the increments found in the valve catalog.

Fp Piping Geometry Factor

The C_v values for valves are obtained experimentally by installing the valve in a straight run of pipe without any inlet or outlet reducers. Since many applications of valves do require reducers, the F_p factor takes into account the effects of the inlet and outlet reducers.

The F_p factor is defined as the ratio of the valve C_v installed with reducers to the rated C_v of the valve installed without reducers. The following equation can be used to determine F_p:

$$F_P = (((K_{SUM}\ C_v^2)/(N_2\ Dv^4))+1)^{-\frac{1}{2}}$$

equation 2

N_2 = conversion coefficient
Dv = nominal valve diameter

$$K_{SUM} = K_1 + K_2 + K_{B1} - K_{B2}$$

equation 3

K_1 = inlet reducer resistance coefficient
K_2 = outlet reducer resistance coefficient
K_{B1} = inlet Bernoulli coefficient
K_{B2} = outlet Bernoulli coefficient

PIPE-FLO calculates the values of the coefficients with the following equations as found in the standard:

$$K_1 = 0.5\ (1 - (D_v^2/D_1^2))^2$$

equation 4A

$$K_2 = 1.0\ (1 - (D_v^2/D_2^2))^2$$

equation 4B

$$K_{B1} = 1 - (D_v/D_1)^4$$

equation 4C

$$K_{B2} = 1 - (D_v/D_2)^4$$

equation 4D

D $\;$ = nominal pipe diameter
D_v = nominal valve diameter
1 = inlet
2 = outlet

PIPE-FLO also performs the F_P calculations during valve selection. The F_P factor is incorporated into the search value of C_v. This insures that the valve is selected based upon the installed piping arrangement. If a valve is evaluated that requires reducers, the following compensation is made to the "search" value for C_v:

Cv (search) = Cv (full size)/ Fp

If the flow through the valve is in the laminar and transition range, the standard states that pipe reducers are not to be installed around the valve. The value of F_P is therefore not factored into the sizing equation for the laminar and transition ranges. During sizing calculations, a turbulent flow check is performed. If the flow is found to be in the laminar or transitional range, the program does not consider reduced size valves as a valid option.

Non-turbulent Flow

If the flow through the valve is non-turbulent (due to a high fluid viscosity or low flow rate) a correction factor is added to the sizing equation to correct for the non-turbulent conditions. For fully turbulent flow, the correction factor (FR) is assumed to be 1. For flow in the laminar range, the Cv value can be calculated directly, eliminating the need to calculate the value of FR. When the flow is in the transition range, the FR value is calculated and used in the general sizing equation (equation 1).

Non-turbulent Flow & Valves with Close-coupled Reducers

The ANSI/ISA-S75.01-1985 (R 1995) standard states that for non-turbulent flow conditions, the effect of close-coupled reducers is not known. Thus, when the specified design condition is in the non-turbulent region, PIPE-FLO does not automatically select valves which are smaller than the pipeline size. However, users can manually select valves that are smaller than the pipeline size. In these cases, $F_p = 1$ is assumed.

The correction factors for laminar and transitional flow are described below:

Laminar Flow

The C_v calculation for the laminar flow range is as follows:

$$Cv = (1/F_S)(w\mu / N_S\, dP)^{2/3}$$

equation 5

N_S = conversion coefficient
F_S = laminar flow factor, dimensionless
w = mass flow rate
μ = fluid viscosity (absolute)

The value of F_S is determined by manufacturer testing and is stored in the valve catalog. Note that for each valve body type, the same value of F_S is used for flow to open, flow to close, and full and reduced seated trims.

Transitional Flow

When the flow is in the transitional range, the value of F_R varies depending on what type of calculation is being done.

In valve sizing calculations, the following formula is used for F_R:

$$FR = 1.044 - 0.358\,(C_{vs}/C_{vt})^{0.655}$$

equation 6

Cvs = the C_v value for laminar flow (equation 5)
Cvt = the C_v value for fully developed turbulent flow (equation 1, without Fp)

When calculating the flow rate, the following formula is used for F_R:

$$F_R = 1.004 - 0.358 (w_s/w_t)^{0.588}$$

equation 7

w_s = the mass flow rate for laminar flow (equation 5)

w_t = the mass flow for fully developed turbulent flow (equation 1)

Once F_R is calculated using either equation 6 or equation 7 above, it is inserted into the general sizing equation (equation 1). If the F_R value calculated is less than 0.48, the flow through the valve is laminar and equation 5 is used in all sizing calculations. When F_R is greater than 0.98, the flow is considered turbulent and equation 1 is used with F_R set equal to 1.

Whenever the flow through a valve is non-turbulent, a laminar line is displayed on the Flow vs dP and %Open vs Flow Graph Windows.

Choked Flow Conditions

As the inlet pressure to the valve is held constant and the outlet pressure is decreased, the flow rate through the valve will increase. This is true until the static pressure at the vena contracta (the point of lowest pressure in the valve) falls below the vapor pressure of the fluid. The maximum pressure drop and flow rate for the valve have been reached and choked flow occurs, resulting in either cavitation or flashing. If the outlet pressure is greater than the vapor pressure of the liquid, cavitation occurs. If the outlet pressure is equal to or less than the vapor pressure of the liquid, flashing occurs.

The calculation of the choked flow and pressure conditions are as follows:

$$Q_{MAX} = N_6 F_{LP} C_v ((P_1 - P_{vc}) \rho)^{1/2}$$

equation 8

N_6 = conversion coefficient

Q_{MAX} = maximum mass flow rate

P_{vc} = absolute pressure at vena contracta

F_{LP} = liquid pressure recovery factor with reducers installed, dimensionless

The value of Pvc can be calculated from the following formula:

$$P_{vc} = F_F P_v$$

<div align="right">*equation 9*</div>

F_F = liquid critical pressure ratio factor, dimensionless

P_v = vapor pressure at inlet temperature

The value of QMAX is displayed on the Flow vs dP and %Open vs Flow Graph Windows when choked flow conditions occur in the specified flow rate range.

F_L Liquid Pressure Recovery Factor

The liquid pressure recovery factor, FL, is a measure of the valve's ability to convert the kinetic energy of the fluid at the vena contracta back into pressure. The internal geometry of the valve determines the value of FL. It is a function of the direction of flow through the valve, the valve position, and whether the valve has a full or reduced seated trim.

The values of FL used in PIPE-FLO's control valve selection module are supplied in the manufacturer's catalog. FL values are stored for each valve body type for flow to open, flow to close, and full and reduced seated trims at 10% increments of valve position.

If a manufacturer does not provide a value for a particular condition, the program will display a warning message and allow the user to enter a value. It is strongly recommended that users consult with the manufacturer for the suitability of the valve to operate under such conditions.

If reducers are installed around the valve, their effects are factored into the value of FL. A new factor, called FLP, is calculated and used in the valve sizing equation for non-compressible fluids with reducers. FLP is calculated as follows:

$$F_{LP} = F_L [(F_L^2 (K_1 + K_{B1})/N_2)(C_v^2 / Dv^2)^2 + 1]^{-\frac{1}{2}}$$

<div align="right">*equation 10*</div>

Control Valve Selection   67

N_2 = conversion coefficient

F_F Liquid Critical Pressure Ratio Factor

The liquid critical pressure ratio factor, F_F, is the ratio of the apparent vena contracta pressure of the liquid under choked flow conditions to the vapor pressure of the liquid at the inlet temperature. The following equation (as found in the standard) is used to calculate the value of F_F used in the choked flow equations:

$$F_F = 0.96 - 0.28 \left(P_v / P_c \right)^{1/2}$$

equation 11

P_c = critical pressure of the liquid
P_v = vapor pressure of the liquid

The above equation is based on the assumption that the fluid is always in thermodynamic equilibrium. Because this is usually not the case for a liquid as it flashes across a valve, the flow rate predicted using equation 11 will be less than the actual flow rate.

Sizing Valves for Compressible Service

The equations for the flow of a gas or vapor through a valve as described by the standard and used in PIPE-FLO's control valve selection module are as follows:

$$w = N_6 F_P C_v Y (X P_1 \rho)^{\frac{1}{2}}$$

equation 12

N_6 = conversion coefficients
w = mass flow rate
X = ratio of pressure drop to absolute inlet static pressure dP/P_1, dimensionless
Y = expansion factor, dimensionless
ρ = density of the fluid, mass per unit volume

The piping geometry factor (F_P) is identical to the one used in the calculations for non-compressible fluids. The Reynolds factor (F_R) is not used in the gas sizing equation because for a gas it can be assumed that the flow through a valve is always turbulent. The values of Y and X are unique to the gas sizing equation and are explained below.

Y Expansion Factor

The expansion factor (Y) accounts for the change in the fluid density as it passes from the valve inlet to the vena contracta. The value of Y is affected by the following factors:

1   Ratio of the valve trim area to the inlet area

2   Shape of the flow path

3   Pressure drop ratio (X)

4   Ratio of specific heats (k) of the fluid.

The effect of items 1, 2, and 3 are accounted for in the pressure drop ratio factor, XT. The value of XT is determined experimentally for each valve. XT factors are supplied in the manufacturer's catalog.

The effect of item 4 is accounted for by using the ratio of specific heat factor, F_k.

The calculated value of Y is determined by the following equation:

$$Y = 1 - X /(3 F_k X_T)$$

equation 13

(limits $0.67 \leq Y \leq 1.0$)

F_k = ratio of specific heat factor, dimensionless
X = ratio of pressure drop to absolute inlet static pressure dP/P1, dimensionless
X_T = pressure drop ratio factor, dimensionless

Ratio of Pressure Drop to Inlet Pressure (X)

The value of X is the ratio of the differential pressure to the inlet static pressure. X is defined in the standard as follows:

$$X = (P_1 - P_2)/P_1$$

equation 14

(limit $X = X_T F_k$)

As the differential pressure increases and the inlet pressure is held constant, the value of X increases. This results in a higher mass flow rate through the valve. The value of X continues to increase until it equals $X_T F_k$. This corresponds to a minimum value of 0.67 for Y. When this condition occurs, the flow through the valve is sonic. Once sonic flow is achieved, the reduction of outlet pressure has no further effect on the mass flow rate through the valve.

PIPE-FLO's control valve selection module program checks for sonic or choked flow and indicates when these conditions exist.

Fk Ratio of Specific Heats Factor

The flow rate through a valve is affected by the ratio of specific heats for the compressible fluid. The factor F_k accounts for this effect. The standard uses the following formula to determine F_k.

$$F_k = k / 1.40$$

equation 15

k = ratio of specific heats

The ratio of specific heats (k) can be found in fluid tables for most common gases.

Rated Pressure Drop Ratio Factor (XT)

The value of XT is determined experimentally and supplied by the valve manufacturer. The valve catalog stores XT values for each valve body type for flow to open, flow to close, and full and reduced seated trims at 10% increments of valve position. If a manufacturer does not supply a value for XT, the valve is not suitable for use with compressible fluids. If a manufacturer only provides a partial data set for XT, the user is given the opportunity to manually enter a value. Users are strongly urged to contact the manufacturer to insure that the valve is suitable for a particular application.

If there is a reducer on the valve inlet, its effect must be factored into XT, resulting in a new factor designated as XTP. The value of XTP is calculated by the following equation:

$$x_{TP} = (x_T / F_P^2)[(x_T K_{IN} C_v^2/(N_5 Dv^4)) + 1]^{-1}$$

equation 16

$$N_5 = \text{conversion coefficient}$$
$$K_{IN} = (K_1 + K_{B1})$$

References

1 American National Standard Institute, Instrument Society of America Standard, Flow Equations for Sizing Control Valves, ANSI/ISA-75.01-1985 (R 1995), sponsor Instrument Society of America, Research Triangle, 1995.

2 Les Driskell, "Control-Valve Selection and Sizing," first edition, Instrument Society of America, Research Triangle, 1983.

3 J.W. Hutchison, "ISA Handbook of Control Valves," second edition, Instrument Society of America, 1976.

4 Crane, Crane Technical Paper 410, "Flow of Fluids through Valves, Fittings, and Pipe" twenty-fifth printing, Crane Company 1988.

5 Vennard, "Elementary Fluid Mechanics" fourth edition, John Wiley & Sons Inc., New York, 1961.

FLOW METERS

Introduction

The flow meter provides a differential pressure which is related to a known rate of flow through the meter. PIPE-FLO's flow meter sizing feature calculates the size of an opening in a flow meter to achieve the desired pressure drop for the design flow conditions. Venturi, nozzle, and flat plate orifice differential pressure flow meters are supported. PIPE-FLO can also size an orifice to supply a fixed non-recoverable pressure drop for use in balancing flow rates.

When sizing flow meters PIPE-FLO assumes that the mass flow rate through the meter is constant with respect to time. In other words, the flow is not subject to pulsation. Reference 1 provides a discussion of pulsating flow effects on fluid meters.

The source documents for the PIPE-FLO's meter device are the American Society of Mechanical Engineers standards ASME MFC-3M-1989 (Reference 1) and ASME MFC-14M-1995 (Reference 2).

PIPE-FLO's flow meter sizing feature is designed to provide a quick and effective means of sizing differential pressure flow meters using the referenced ASME standards. Specific formulas from these references are supplied in this reference book. When there is a discrepancy between the standards and this book, the standard takes precedence.

Sizing Flow Meters

The general sizing equation for the flow of a fluid through a differential pressure flow meter as described by the standard and used in PIPE-FLO's flow meter sizing feature is:

$$q_m = 0.09970190 F_a C Y_1 \beta^2 D^2 (h_w \rho_{f1}/(1-\beta^4))^{1/2}$$

equation 1

q_m = mass rate of flow (lb/sec)
F_a = thermal expansion correction factor
C = discharge coefficient
Y_1 = expansion factor based on upstream pressure
D = upstream internal pipe diameter in inches at measured temperature (68°F)
β = diameter ratio (orifice diameter/pipe diameter, d/D)
h_w = differential pressure (inches H2O)
ρ_{f1} = density of flowing fluid at upstream conditions (lb/ft3)

Discharge Coefficient

The actual flow rate through the meter is seldom equal to the theoretical flow and usually turns out to be less than the theoretical flow. In order to account for this difference, a discharge coefficient C is introduced into the flow equation.

$$C = \text{actual flow rate/theoretical flow rate}$$

equation 2

Thermal Expansion Factor

If the meter is used at a temperature that differs from the temperature at which it was manufactured, the thermal expansion of the meter must be taken into account. F_a is the thermal expansion correction factor. The value of F_a depends on the meter material, the pipe material, and the temperature of the process fluid. The following equation is used to determine F_a:

$$F_a = 1 + [2/(1 - \beta^4_{meas})][\alpha_{PE} - \beta^4_{meas} \, \alpha_P][t - t_{meas}]$$

equation 3

α_P = thermal expansion factor of the pipe   (in/in/°F)
α_{PE} = thermal expansion factor of the flow meter (in/in/°F)
t = temperature of the flowing fluid (°F)
t_{meas} = reference temperature for measured bore (68°F)
β_{meas} = diameter ratio at the reference temperature (68°F)

Fluid Compressibility

When a fluid flows through a meter, there is a pressure drop as it passes through the constriction. When a compressible fluid (a gas or vapor) flows through a meter, the resulting pressure drop causes a change in fluid density at the constriction. As a result, the fluid densities at the meter inlet and within the meter are different.

The expansion factor corrects for density differences between pressure taps due to expansion to the lower pressure. It is equal to one for incompressible fluids and less than one for compressible fluids. The expansion factor equation for each flow meter is given below.

Orifices

$$Y_1 = 1 - (0.41 + 0.35\beta^4)(h_w/27.73kp_1)$$

equation 4

Y_1 = expansion factor based on upstream pressure

k = isentropic exponent (specific heat ratio)

p_1 = static upstream pressure of the fluid (psi)

Nozzles and Venturi Tubes

$$Y_1 = \{[k\tau^{2/k}/(k - 1)][(1 - \beta^4)/(1 - \beta^4\tau^{2/k})][(1 - \tau^{(k - 1)/k})/(1 - \tau)]\}^{1/2}$$

equation 5

τ = pressure ratio, p_2/p_1

Calculating Discharge Coefficients

In order to solve the meter equation, a value for the discharge coefficient, C will have to be found. One means of arriving at the meter flow coefficient is to install the meter in a test stand and generate a calibration curve. This method is expensive and time consuming, and fortunately it is not necessary in most cases. A considerable number of tests have been performed on various meter combinations and geometries, and a set of equations have been developed to calculate the value of the discharge coefficient. These coefficients fall within the range of experimental accuracy.

Except for small bore orifices, the coefficients used in PIPE-FLO's flow meter sizing feature are those listed in Reference 1. Small bore orifices are covered by Reference 2.

The formula used to calculate the value of the coefficient depends on the meter type and manufacturing method or pressure tap arrangement used to measure the differential pressure. The most common meter types are supported by the program. These types are: orifices (corner, flange, 1D - ½D, and balancing), small bore orifices (corner groove and flange), flow nozzles (1D - ½D), venturi tubes (rough-cast, rough-welded, and machined convergent). The discharge coefficient for each type of meter is given below.

Orifices

Corner Taps

Corner taps are located such that the tap holes break through the wall flush with the faces of the meter plate. The discharge coefficient for corner taps is as follows:

$$C = 0.5959 + 0.0312\beta^{2.1} - 0.1840\beta^8 + 91.71\beta^{2.5}R_D^{-0.75}$$

equation 6

$$R_D = \text{Reynolds number}$$
$$= 22738\ q_m\ /\ \mu D$$

Flange Taps

Flange pressure taps are located so that the inlet tap is 1 inch from the upstream face of the meter plate and the outlet tap is 1 inch from the downstream face of the meter plate. The discharge coefficient for flange taps when $D \geq 2.3$ in. is as follows:

$$C = 0.5959 + 0.0312\beta^{2.1} - 0.1840\beta^8 + 0.0900D^{-1}\beta^4(1 - \beta^4)^{-1}$$
$$- 0.0337D^{-1}\beta^3 + 91.71\beta^{2.5}R_D^{-0.75}$$

equation 7

For flange taps when $2 < D < 2.3$ in.:

$$C = 0.5959 + 0.0312\beta^{2.1} - 0.1840\beta^8 + 0.0390D^{-1}\beta^4(1 - \beta^4)^{-1}$$
$$- 0.0337D^{-1}\beta^3 + 91.71\beta^{2.5}R_D^{-0.75}$$

equation 8

1D - ½D Taps

1D - ½D pressure taps are located so that the inlet tap is a distance of one pipe diameter upstream of the meter plate inlet face and the outlet tap is located a distance of one half pipe diameter downstream of the meter plate inlet face. The discharge coefficient for 1D - ½D taps is as follows:

$$C = 0.5959 + 0.0312\beta^{2.1} - 0.1840\beta^8 + 0.0390\beta^4(1 - \beta^4)^{-1}$$
$$- 0.01584\beta^3 + 91.71\beta^{2.5}R_D^{-0.75}$$

equation 9

Balancing

For the balancing orifice, the discharge coefficient is the same as that for the 1D - ½D tap arrangement. For small bore balancing orifices, the discharge coefficient is the same as that for the corner tap arrangement.

Small Bore - Corner Taps

Corner pressure taps are located in annular grooves on each side of the plate. The discharge coefficient for small bore corner taps is as follows:

$$C = [0.5991 + 0.0044/D + (0.3155 + 0.0175/D)(\beta^4 + 2\beta^{16})](1 - \beta^4)^{1/2}$$
$$+ [0.52/D - 0.192 + (16.48 - 1.16/D)(\beta^4 + 4\beta^{16})][(1 - \beta^4)/R_D]^{1/2}$$

equation 10

Small Bore - Flange Taps

Flange pressure taps are located so that the inlet tap is 1 inch from the upstream face of the meter plate and the outlet tap is 1 inch from the downstream face of the meter plate. The discharge coefficient for small bore flange taps is as follows:

$$C = [0.5980 + 0.468(\beta^4 + 10\beta^{12})] (1 - \beta^4)^{1/2}$$
$$+ (0.87 + 8.1\beta^4)[(1 - \beta^4)/R_D]^{1/2}$$

equation 11

Flow Nozzles

The upstream tap is located a distance of one pipe diameter upstream from the plane of the nozzle inlet face. The downstream tap is located either in the nozzle throat or one half pipe diameter from the nozzle inlet face. Location of the downstream tap beyond the plane of the nozzle exit end is not permitted. The discharge coefficient for flow nozzles is as follows:

$$C = 0.9975 - 0.00653(10^6\beta/R_D)^{1/2}$$

equation 12

Venturi Tubes

The upstream pressure taps are located a distance of one half pipe diameter upstream from the inlet of the convergent entrance. The throat taps are located a distance of one half the venturi throat diameter from the end of the convergent entrance. The venturi discharge coefficient is dependent on the method of manufacture. The discharge coefficients for each type of venturi is as follows:

Rough-cast and Rough-welded Convergent

$$C = 0.984$$

Machined Convergent

$$C = 0.995$$

Pressure Loss

For orifices and nozzles, the pressure loss is the difference in static pressure between a wall pressure measured on the upstream side of the meter where the influence of the approach impact pressure adjacent to the plate becomes negligible (approximately 1D upstream of the meter) and the pressure measured on the downstream side of the meter where the static pressure recovery by expansion of the jet may be considered as just completed (approximately 6D downstream of the meter).

For venturi tubes, the pressure loss is the difference in pressure measured between a point 1D upstream from the upstream end of the venturi and a point 6D downstream from the downstream end of the venturi minus the difference in pressure at those same locations prior to installation of the venturi.

The equations for the pressure loss are given below.

Orifices

$$h = \{[(1 - \beta^4)^{1/2} - C\beta^2]/[(1 - \beta^4)^{1/2} + C\beta^2]\}h_w$$

equation 13

h = pressure loss (in H_2O)
C = discharge coefficient

Nozzles

$$h = (1 + 0.014\beta - 2.06\beta^2 + 1.18\beta^3)hw$$

equation 14

Venturi Tubes

15° Divergent Angle

$$h = (0.436 - 0.86\beta + 0.59\beta^2)h_w$$

equation 15

7° Divergent Angle

$$h = (0.218 - 0.42\beta + 0.38\beta^2)h_w$$

equation 16

Sonic Flow

Sonic flow is the maximum flow that can be attained through a meter. It is primarily a function of the pressure ratio, beta ratio, and isentropic exponent. As an approximation, PIPE-FLO considers the flow to approach the sonic velocity when the value of the expansion factor (Y) reaches 2/3. This approximation is intended as a guideline to aid in identifying design conditions which are potentially sonic.

Flashing Condition

For non-compressible fluids, a flashing condition will occur if the pressure drop specified causes the downstream fluid pressure to equal or fall below the fluid vapor pressure. When a meter design is close to the flashing condition, notification is provided in the graph window.

Equation Limits and Discharge Coefficient Uncertainties

In the standard, discharge coefficient uncertainty values are given for each type of flow meter. Limits of use for the flow meter equations are also provided.

Orifices

Uncertainty

β

$10,000 < R_D \leq 10^8$

$2000 \leq R_D \leq 10,000$

```
= 0.20          0.60              0.75
|—(0.6%)—→|←—(β%)—→|
|————— (0.6 + β)% —————|
```

Limits

$2" \leq D \leq 36"$

$0.20 \leq \beta \leq 0.75$

$2000 \leq R_D \leq 10^8$

$p_2/p_1 \geq 0.75$

Small Bore Orifices

Uncertainty

±0.75%

Limits, Corner Taps

$0.5" \leq D < 2.0"$

$0.10 \leq \beta \leq 0.80$

$R_D > 1000$

Limits, Flange Taps

$1" \leq D < 2.0"$

$0.15 \leq \beta \leq 0.70$

$R_D > 1000$

Nozzles

Uncertainty
2%

Limits
$4" \leq D \leq 30"$
$0.20 \leq \beta \leq 0.80$
$10^4 \leq R_D \leq 6 \times 10^6$
$p_2/p_1 \geq 0.75$

Venturi Tubes

Uncertainty
1%

Limits, Rough-cast and Rough-welded Convergent
$4" \leq D \leq 48"$
$0.30 \leq \beta \leq 0.75$
$2 \times 10^5 \leq R_D \leq 6 \times 10^6$
$p_2/p_1 \geq 0.75$

Limits, Machined Convergent
$2" \leq D \leq 10"$
$0.30 \leq \beta \leq 0.75$
$2 \times 10^5 \leq R_D \leq 2 \times 10^6$
$p_2/p_1 \geq 0.75$

Thermal Expansion Factor Limits

The Standard has thermal expansion data for the following materials over the temperature ranges shown:

Material	Temperature Limits, °C	Temperature Limits, °F
Carbon Steel	-198 to 760	-325 to 1400
Alloy Steel	-198 to 760	-325 to 1400
Series 300 SS	-198 to 760	-325 to 1400
Series 400 SS	-198 to 760	-325 to 1400
CrNi 25/20	-198 to 760	-325 to 1400
Monel 67/30	-198 to 760	-325 to 1400
Monel 66/29	-198 to 760	-325 to 1400
Aluminum	-198 to 316	-325 to 600
Bronze	-198 to 649	-325 to 1200
Brass	-198 to 649	-325 to 1200
Wrought Iron	-198 to 538	-325 to 1000
CuNi 70/30	-198 to 204	-325 to 400

References

1 American Society of Mechanical Engineers Standard, Measurement of Fluid Flow in Pipes Using Orifice, Nozzle, and Venturi (ASME MFC-3M-1989); issued January 1990.

2 American Society of Mechanical Engineers Standard, Measurement of Fluid Flow Using Small Bore Precision Orifice Meters (ASME MFC-14M-1995); issued June 1995.

3 Crane Technical Paper No. 410, Flow of Fluids Through Valves, Fittings, and Pipe; twenty fourth printing, 1988.