PIPE-FLO Compressible

User's Manual & Method of Solution

Engineered Software, Inc.

Engineered Software, Inc. makes no claims or warranties with respect to the contents or accuracy of this publication, including any warranties of fitness or merchantability for a particular purpose. Information in this document is subject to change without notice.

The software described in this document is furnished under a license agreement or nondisclosure agreement. The software may be used or copied only in accordance with the terms of that agreement. It is against the law to copy the software except as specifically allowed in the license or nondisclosure agreement.

Companies, names, and data used in examples herein are fictitious.

PIPE-FLO Compressible includes some routines from the following source:
LAPACK Users' Guide, Third Edition, ISBN 0-89871-447-8
Authors: E. Anderson, Z. Bai, C. Bischof, S. Blackford, J. Demmel, J. Dongarra, J. Du Croz, A. Greenbaum, S. Hammarling, A. McKenney, and D. Sorensen

Engineered Software, Inc.
4529 Intelco Loop SE, Suite A
Lacey, WA 98503-5941
360-412-0702

If you need technical support for PIPE-FLO Compressible, call 360-292-4060.

Our e-mail address is solutions@eng-software.com

August 2008 Edition
for PIPE-FLO Compressible 2007 and above

CM-V0808

TABLE OF CONTENTS

INTRODUCTION

PIPE-FLO Compressible Overview

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

- Draw a piping system schematic on the FLO-Sheet showing all the compressors, components, pressure sources, control valves and interconnecting pipelines.
- Determine when a fluid is approaching sonic conditions.
- Size the individual pipelines using electronic pipe, valve, and fluid data tables.
- Calculate how the system operates including pressures and flow rates.
- Create FLO-Links to provide immediate access to supporting documents needed to design, build, and operate the piping system.

New this version

- Isometric drawings – Draw or display pipeline diagrams in the new isometric grid mode, complete with rotatable text and symbols.
- Customized shapes – Create your own symbols for pressure sources, compressors, control valves, and other components.
- Copy & paste improvements – Copy all or part of a system from one project file to another.
- Integrated flow meter calculations — Size an orifice and insert it into a pipeline in a single step.
- More flexible fluid assignment – Add fluid properties directly to system lineups to make changing fluids quick and easy.
- Customized results – Alter piping schematic and fly-by viewer text to show

only the items you want to see.

Getting Started

Installation

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

Learning to use the software

Learning new software can be a daunting experience. To speed up the process, PIPE-FLO Compressible includes a comprehensive tutorial accessible directly from the program's startup screen. Simply launch the software, click the Software Tutorial button and choose either the Metric or US unit-based tutorial.

The tutorial is designed to walk users through all aspects of the software as quickly as possible by demonstrating, step by step, the process of building a piping system.

Using Program Help

PIPE-FLO Compressible'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 its 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 Compressible software includes 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 Compressible'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 Compressible program. The calculations associated with system elements, such as pipes, compressors/ blowers, components, and controls, is discussed, along with the methodology applied in performing the piping system calculations.

Terms and Definitions

Adiabatic Process – A process with no heat transfer. If there is no heat transfer to the system, the total (or stagnation) temperature is a constant. PIPE-FLO Compressible assumes adiabatic flow.

Ideal (Thermally Perfect) Gas - An ideal gas is one that obeys the ideal gas law:

$$\rho = P(MW)/(RT)$$

equation 1

ρ = fluid density
P = pressure
R = universal gas constant
MW = molecular weight
T = temperature

PIPE-FLO Compressible assumes the ideal gas law for the equation of state. For the accuracy required in many engineering computations, air and many other common gases behave as ideal gases.

Compressiblity Factor (Z) - The compressibility factor is defined as follows:

$$Z = P(MW)/(\rho RT)$$

equation 2

Z = compressibility factor
P = pressure
ρ = density
R = universal gas constant
T = temperature

Note that for an ideal gas, Z = 1, and the deviation of Z from unity is a measure of the deviation of the actual relation from the ideal gas equation of state. PIPE-FLO Compressible always assumes an ideal gas (Z = 1).

Specific Heat Ratio - The ratio of the specific heat at constant pressure to specific heat at constant volume:

$$k = Cp/Cv$$

equation 3

k = specific heat ratio
Cp = specific heat at constant pressure
Cv = specific heat at constant volume

When performing calculations for a system, PIPE-FLO Compressible assumes that the specific heat ratio is constant. The user can directly enter the specific heat ratio, or if a fluid is selected from the fluid table, the program calculates the specific heat ratio using the following equation:

$$k = Cp/(Cp - R)$$

equation 4

Cp = specific heat at constant pressure
R = universal gas constant

The specific heat at constant pressure is calculated using the system total temperature and the formula stored in the selected fluid table.

Mach Number - The ratio of the fluid velocity to the speed of sound in the fluid at a particular point.

$$M = V/c$$

equation 5

M = Mach number
V = flow velocity
c = speed of sound

Subsonic Flow - A flow with Mach number less than one.

Critical or Choked Flow - A flow which occurs when the Mach number equals one. This is also referred to as choked, or sonic flow. When a subsonic flow becomes choked, the flow rate has reached its maximum possible value – it cannot be increased even if the downstream pressure is lowered. PIPE-FLO Compressible considers a flow choked when the Mach number at the outlet of a pipeline or a device is one or close to one. If choking occurs prior to the pipeline or device outlet, the status of the pipeline or device is marked as "Invalid," indicating that the program cannot converge to a valid solution.

Supersonic Flow - A flow with Mach number greater than one. PIPE-FLO Compressible does not handle supersonic flow.

Total (Stagnation) Property - A point in a flow field in which the flow is brought to rest is called a stagnation point, and properties at that point are called *total (stagnation)* properties. Thus in contrast to static properties, measurements of stagnation properties require that the flow be brought to rest with respect to the observer's instruments. Total properties include both static and dynamic effects. See Figure 1 on the following page.

Static Property - Static properties are those that would be measured if one could travel with the fluid at its exact velocity (in this case the relative velocity between the observer and the fluid would zero – hence the term “static”). Static properties do not contain dynamic effects. Figure 1 illustrates the difference between static and total pressure.

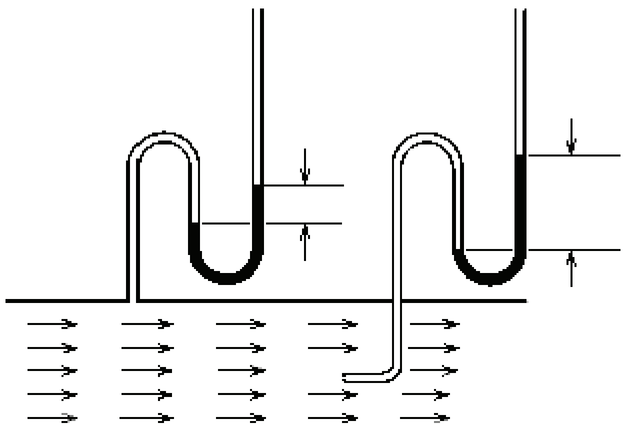


Figure 1

Isentropic – Term used to describe a constant entropy process. Isentropic processes involve no heat transfer; that is, they are adiabatic as well as reversible. Although real processes are never truly isentropic, irreversible effects are often negligible. An example of isentropic flow would be a simple area change. For a simple area change, both the stagnation temperature and stagnation pressure are constant.

Simple Flow - In an internal compressible flow, factors such as area change, friction, heat transfer, and mass flow rate change can affect the state of the flow at a location. If only one of these effects is occurring (for example, an area change in the absence of friction, heat transfer, and mass flow rate change), then the flow is termed simple.

Fanno Flow - Flow with friction. Fanno flow is adiabatic, but irreversible (not isentropic).

Compressible Flow Equations

For compressible flows through pipelines, the state of the flow can be affected by such things as friction, area change, heat transfer, and mass flow rate change. The flow is described as simple if only one of these effects is present, for example, an area change occurs without friction, heat transfer, or a mass flow rate change. PIPE-FLO Compressible makes the assumption that the flow is a simple flow. For many compressible systems this provides results well within the range of acceptable engineering accuracy. The equations for the simple flows can be obtained by first developing the generalized, steady, one-dimensional compressible flow equations. These generalized equations are developed by considering the conservation equations for mass, momentum, and energy, along with the equation of state for a perfect gas and various thermodynamic definitions. Once the generalized equations have been developed, the equations for the simple flows can be derived by considering each type of flow as a special case of the generalized flow. For brevity, the generalized, steady, one-dimensional compressible flow equations are not developed and listed here. Interested readers are directed to consult Reference 5 for a complete derivation.

PIPE-FLO Compressible assumes an adiabatic process (no heat transfer) with no mass addition or removal. Thus, the two simple flows handled by the program are Fanno flow (flow with friction) and simple area change.

Fanno Flow

Flow in an adiabatic, constant area duct with no mass addition is called Fanno flow. Since there is friction in the duct, Fanno flow is irreversible. For subsonic Fanno flows, the equation variables change along the pipe (or duct) as indicated below in Figure 2:

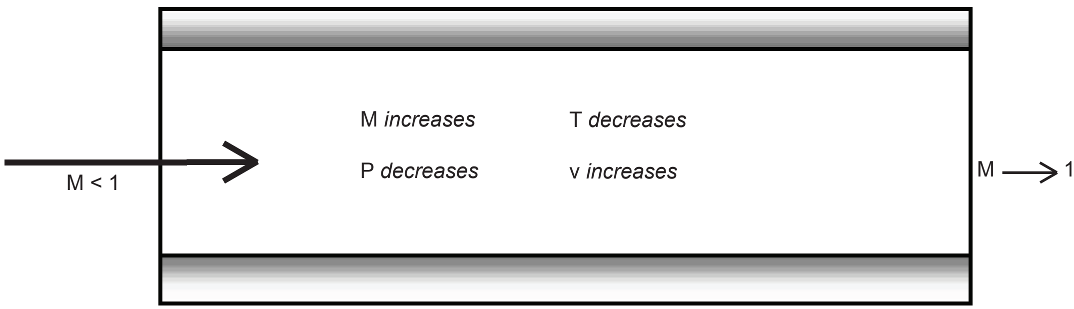


Figure 2

For Fanno flow, the following relations apply:

$$(P/P^*) = (1/M)[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-1/2}$$

equation 6

$$(\rho/\rho^*) = (1/M)[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{1/2}$$

equation 7

$$(T/T^*) = [(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-1}$$

equation 8

$$(V/V^*) = M[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-1/2}$$

equation 9

$$(P_0/P_0^*) = (1/M)[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{(k+1)/[2(k-1)]}$$

equation 10

$$A = A^* = \text{constant}$$

equation 11

$$T_0 = T_0^* = \text{constant}$$

equation 12

$$W = W^* = \text{constant}$$

equation 13

M = Mach number
k = specific heat ratio
P = pressure
ρ = density
T = Temperature
V = flow velocity
A = cross sectional area
W = mass flow rate
0 = subscript indicating total (or stagnation) state
* = superscript indicating critical state

Fanno flow moves toward the Mach 1 state as the compressible fluid propagates down a frictional pipe. Given an initial Mach number, M, the Sonic Length Formula can be used to calculate the length of pipe downstream through which the fluid must flow to reach sonic velocity:

$$S = (fL^*/D) = (1-M^2)/(kM^2) + [(k+1)/2k]\ln[(k+1)M^2/(2+(k-1)M^2)]$$

equation 14

L* = length of pipe the fluid must flow through to reach sonic velocity
D = pipe diameter
f = Darcy friction factor
M = initial mach number
k = specific heat ratio

A plot of sonic length vs. initial mach number is shown below in Figure 3:

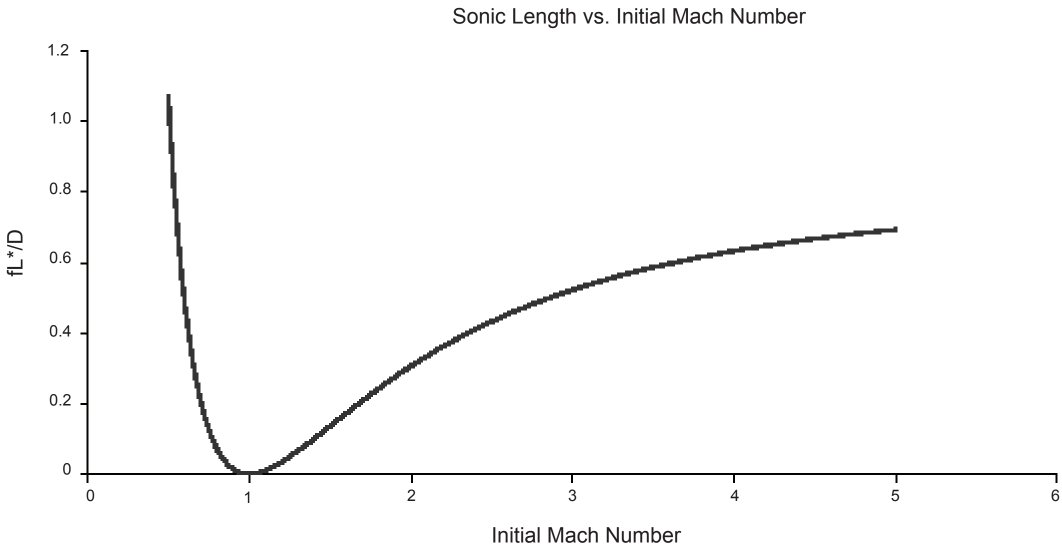
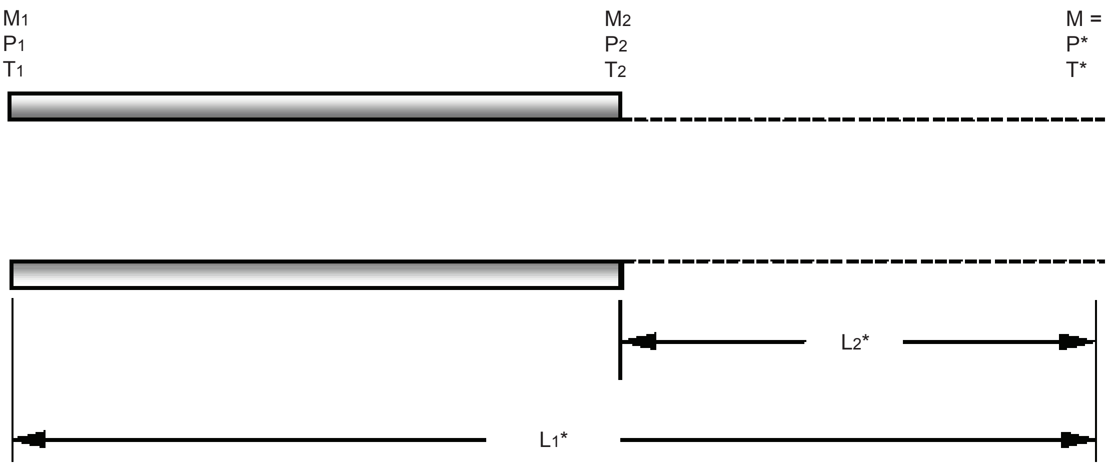


Figure 3

Duct Addition

Duct addition is a recurring circumstance that is involved in many Fanno flow situations. In duct addition, information on the initial state, the friction factor, and the duct diameter and length are given; the final state after the flow traverses the duct length is desired. Conditions at the duct inlet (M_1, P_1, T_1, f, D, and L) are known. The duct length required to achieve Mach 1 is L_1*, and for state 2 the duct length required to achieve Mach 1 is L_2*. The duct location of the sonic point (M = 1) and the conditions at the sonic point in a given Fanno flow are unique. For both the inlet and the outlet the sonic location and the conditions at the sonic location must be the same. Figure 4 on the following page illustrates the problem.



Schematic of Duct Addition Problem

Figure 4

L_1* can be computed directly by substituting the known inlet Mach number (M_1) into the Sonic Length Formula (equation 14). L_2* can then be computed by subtracting the actual duct length from L_1*:

$$(fL_2^*/D) = (fL_1^*/D) - (fL/D)_{actual}$$

equation 15

Since the Sonic Length Formula is non-analytic with respect to the Mach number, the outlet Mach number (M_2) cannot be directly calculated. A numerical method is required to determine this value.

Newton's Method

As noted above, the sonic length function is a non-analytic function with respect to the Mach number. Thus, to determine the outlet Mach number PIPE-FLO Compressible employs Newton's method. A brief description of this method follows.

NOTE: For convenience, S will be used to refer to the dimensionless quantity (fL/D).

1 Start with an initial guess for the outlet Mach number, M_2^0.

2 From this initial value of M_2^0, compute $(S_2^*)_0$.

3 Compute the value of the derivative dS*/dM evaluated at M_2^0.

4 Find the equation of the line that is tangent to the sonic length function at M_2^0 and $(S_2^*)_0$:

$$S^* = (M - M_2^0)(dS^*/dM)|_{M2}{}^0 + (S_2^*)_0$$

equation 16

5 Find the intersection of the tangent line with horizontal line $S^* = S_2^*$.

$$S_2^* = (M - M_2^0)(dS^*/dM)|_{M2}{}^0 + (S_2^*)_0$$

Solving for M,

$$M = [S_2^* - (S_2^*)_0]/(dS^*/dM)|_{M2}) + M_2^0$$

equation 17

The value of M calculated in equation 17 is the improved guess for M_2, M_2^1.

6 Repeat steps 2 through 5, substituting the improved guess M_2^1 for the initial guess. When $(S_2^*)_n$ is within an acceptable tolerance of the target S_2^* (calculated with equation 15), stop. At this point, $(M_2)_n$ is a good approximation to the theoretical M_2 we are trying to calculate.

Calculating the Friction Factor

The pipeline 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:

$$Re = W/(d\mu)$$

equation 18

W = flow rate
d = pipe diameter
μ = fluid dynamic viscosity

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.

For engineering calculations, the upper practical limit of laminar flow has been set at a Reynolds number of Re = 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. Because most compressible gas flow in pipes is turbulent, the program considers only turbulent flow.

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:

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

equation 19

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 Compressible uses an equation from Reference 4 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 20

Equation 20 gives accurate values of the friction factor and can be solved quickly without performing iterative calculations.

Simple Area Change Flow

For a simple area change, the friction can be neglected since it has a small effect compared to the effect of the area change. Therefore, flow through a simple area change is an isentropic process. For an adiabatic isentropic process, the total (or stagnation) temperature and pressure are constant. The following equations apply:

$$(A/A^*) = (1/M)[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{(k+1)/[2(k-1)]}$$

equation 21

$$(P/P^*) = [(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-k/(k-1)}$$

equation 22

$$(\rho/\rho^*) = [(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-1/(k-1)}$$

equation 23

$$(T/T^*) = [(2/(k+1))(1 + 0.5(k - 1)M^2)]^{-1}$$

equation 24

$$(V/V^*) = M[(2/(k+1))(1 + 0.5(k - 1)M^2)]^{½}$$

equation 25

$$(P/P_0) = [1 + 0.5(k - 1)M^2]^{-k/[(k-1)}$$

equation 26

$$(\rho/\rho_0) = [1 + 0.5(k - 1)M^2]^{-1/[(k-1)}$$

equation 27

$$(T/T_0) = [1 + 0.5(k - 1)M^2]^{-1}$$

equation 28

$$T_0 = \text{constant}$$

equation 29

$$P_0 = \text{constant}$$

equation 30

$$W = \text{constant}$$

equation 31

M = Mach number
k = specific heat ratio
A = cross sectional area
P = pressure
ρ = density
T = Temperature
V = flow velocity
A = cross sectional area
W = mass flow rate
0 = subscript indicating total (or stagnation) state
* = superscript indicating critical state

Pipeline Sizing

Users can size pipelines based on sizing criteria entered in the pipe specifications. The sizing criteria selected can either be an inlet Mach number or dP per 100 ft (or m). The following equation is used for pipeline sizing when the inlet Mach number is selected for the sizing criteria:

$$d = [(4W/\pi PM)/(RT/k)^{1/2}]^{1/2}$$

equation 32

d = pipe diameter
W = mass flow rate
P = static pressure
M = inlet Mach number
R = gas constant
T = static temperature
k = specific heat ratio

For preliminary pipeline sizing based on pressure drop, PIPE-FLO Compressible uses the Darcy-Weisbach equation. Even though the Darcy-Weisbach equation assumes that the density in the pipeline does not change significantly, its use can be extended to compressible applications if the pressure drop in the pipeline is less than 40% of the inlet pressure. Since most pipelines are not sized to achieve excessive pressure drops, the Darcy-Weisbach equation provides a quick and reasonably accurate tool for estimating pipe size.

The Darcy-Weisbach equation is as follows:

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

equation 33

If a fluid table is selected for the system fluid, the density used in the pipeline sizing equation is obtained using the equation of state stored in the fluid table. If the fluid is a custom fluid, the ideal gas law is used to calculate the density.

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

Valves and Fittings

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:

$$K = f_T L/d$$

equation 34

f_T = turbulent friction factor
L = length
d = pipe diameter

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 Compressible uses the following equation from Nikuradse:

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

equation 35

d = pipe diameter
e = absolute roughness of clean commercial steel pipe

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.

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 Compressible 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 5 below).

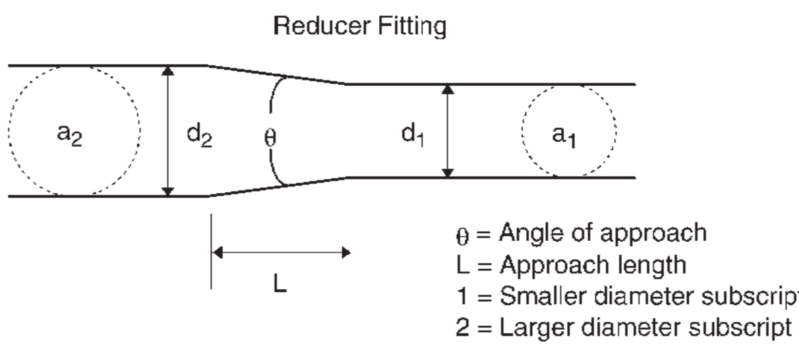


Figure 5

For an enlargement, the direction of flow in Figure 5 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

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

equation 36a

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

equation 36b

Reducer - Enlargement

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

equation 37a

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

equation 37b

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

Reduced Seat Valves and Fittings

Reduced seat valves are broken down into three sections: the reducer (contraction) section, the valve section, and the enlarger section (see Figure 6 below). A K value is determined for each section in relation to the size of the valve in the pipeline.

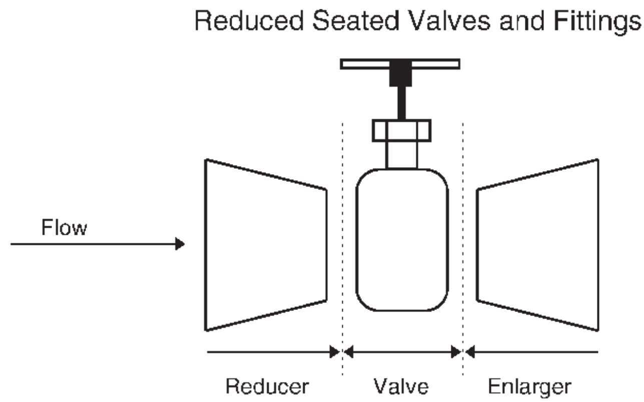


Figure 6

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:

$$K_2 = K_{Reducer} + K1/\beta^4 + K_{Enlarger}$$

equation 38

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
$\beta = d_{valve}/d_{pipe}$

The inside diameter is used when calculating a K value with equation 39. If you later change the pipe diameter, you should reinstall the K value.

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

Cv Values

Many valve manufacturers express the pressure drop characteristics of their valves using a flow coefficient (C_v) rather than a K value. 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.

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

$$K \;.\; 891 d^4 / C_v^2$$

equation 39

d = inside pipe diameter, inches

PIPE-FLO Compressible Valve Table Formulas

PIPE-FLO Compressible uses the method outlined in the Crane Technical Paper 410 (Reference 1) when calculating the K values for valves and fittings. PIPE-FLO Compressible'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 Compressible 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 Compressible 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 Compressible selects the appropriate equation and calculates the K value.

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

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

Formula 1

Formula 3 - Pipe Enlargement

Formula 3 is used to calculate the K value for pipe enlargements. PIPE-FLO Compressible 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 Compressible selects the appropriate equation and calculates the K value.

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

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

Formula 3

Formula 5 - Reduced Seat Valve, Gradual Change in Diameter

Formula 5 is used to calculate the K value for reduced seat 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 Compressible 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.

$$K2 = K_{Reducer} + K1/\beta^4 + K_{Enlarger}$$

Formula 5

Formula 7 - Reduced Seat Valve, Abrupt Change in Diameter

Formula 7 is used to calculate the K value for reduced seat 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.

$$K2 = K_{Reducer} + K1/\beta^4 + K_{Enlarger}$$

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 Compressible 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 Seat Valves and Fittings

Formula 10 is used to calculate K values for valves and fittings that do not support reduced seat 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 Compressible calculates the K value for fixed Cv valves using equation 39.

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 Compressible automatically selects the appropriate coefficient based on the pipe diameter the valve is installed in and calculates the K value using equation 39.

Modeling Reduced Seat (R/S) Valves

PIPE-FLO Compressible handles R/S valves differently than full seat valves. The following points should be noted:

- Only one R/S valve can be installed in a pipeline
- The program locates the R/S valve next to the inlet of the pipeline (the inlet is designated according to the direction of flow that is assumed when the pipeline is defined).

NOTE: Once the calculations are performed, if the direction of flow in the pipeline is reversed from the assumed direction, the location of the R/S valve does not change. For example, if a pipeline direction is initially assumed to be from node A to B, the program locates the valve next to node A. If the flow is actually calculated to be from node B to node A, the valve is still located next to node A.

The R/S valve is modeled by the program as an area change, connected to a pipe with a diameter equal to the reduced seat, connected to another area change. The length of the reduced seat diameter pipeline is equal to an equivalent length calculated from the valve K value. This model is illustrated in the example below.

Example of an R/S Valve Model

The following example illustrates how PIPE-FLO Compressible models an R/S valve: A 6” pipeline has an Angle 90° globe valve installed in it with a reduced seat of 4.” Figure 7 below shows the program model:

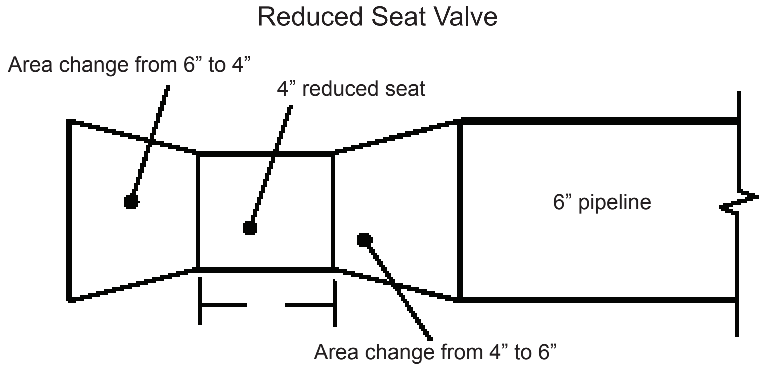


Figure 7

The reduced seat diameter pipeline equivalent length is calculated from the following equations:

$$K_{EQ} = K_{R/S}\beta^4$$

equation 40

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

equation 41

$$L_{EQ} = K_{EQ}D_{R/S}/ f_T$$

equation 42

$K_{R/S}$ = reduced seat valve K value calculated using Formula 5 or formula 7
K_{EQ} = reduced seat K value in terms of the reduced seat diameter
$d_{R/S}$ = reduced seat diameter, inches
$D_{R/S}$ = reduced seat diameter, ft
L_{EQ} = equivalent pipeline length, ft
$\beta = d_{R/S}/d_{Pipe}$
f_T = turbulent friction factor

Compressors

When a compressor (or other pressure gain device, such as a blower) is modeled with a curve and installed in the system, the energy or pressure rise due to the compressor must be factored into the calculations. This pressure increase is obtained from the user specified curve data.

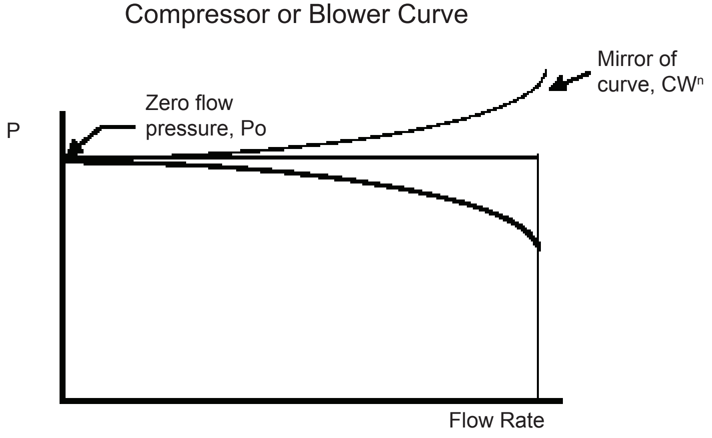


Figure 8

Figure 8 above shows the curve shape PIPE-FLO Compressible uses to model a compressor. Note that for this curve, the pressure decreases as the flow increases. Users can specify up to ten points to model a compressor. PIPE-FLO Compressible reduces these ten points to an exponential equation as shown in equation 43:

$$P = Po - CW^n$$

equation 43

The equation for the pressure curve has two parts, a constant value Po and a coefficient times the flow rate raised to the nth power, CW^n. The constant Po corresponds to the pressure at zero flow. C and n are determined using geometric regression.

As shown in Figure 8, the compressor is modeled as a step increase in pressure (Po) followed by a pressure loss due to flow through the compressor ($-CW^n$). If the solution indicates that the flow rate through the compressor falls outside the data point range entered for the curve, PIPE-FLO Compressible provides notification that the compressor is out of range. If the compressor can still be used with an increased flow rate, the model should be adjusted to use an extended flow rate range and the calculation should be performed again. If it is not possible to use the same compressor with a higher flow rate, another compressor should be selected and inserted into the system.

The curve shape shown in Figure 8 is a reasonable approximation for a centrifugal compressor. On the other hand, typical curves for pressure gain devices such as axial flow and centrifugal blowers have dips in them (see Figure 9 below).

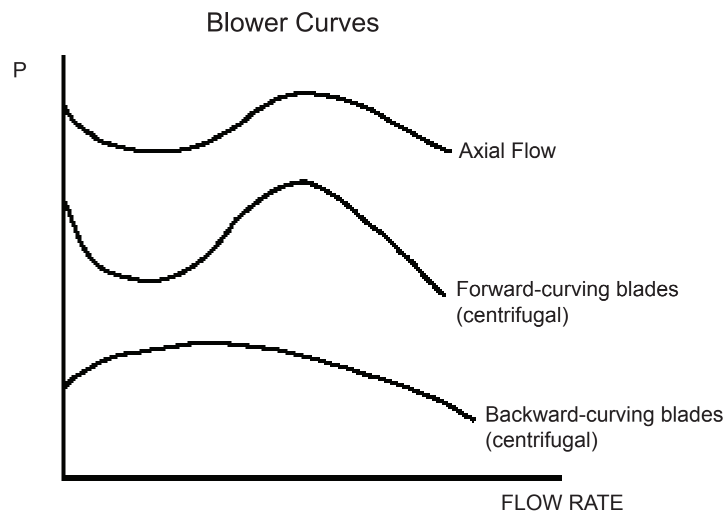


Figure 9

When modeling these curves in PIPE-FLO Compressible, it is not possible to accuately model the dips in the curves with an equation. For these curves, PIPE-FLO Compressible linearly interpolates between the specified curve data.

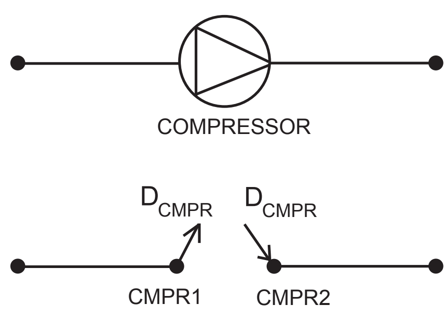
Running a Compressor at a Fixed Flow Rate

Instead of entering performance curve data, the user can also specify a fixed flow rate and have PIPE-FLO Compressible calculate the developed pressure.

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

- When a fixed flow rate is specified for a compressor, the performance curve data (if specified) is overridden.
- PIPE-FLO Compressible models a fixed flow compressor by creating two nodes in the system where the compressor is installed (refer to Figure 10). A flow demand value equal to the set flow rate is taken out of the first compressor node (the inlet node) and an equal flow demand is set entering the second node (the outlet node).
- If a compressor 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 compressor was not installed. The compressor 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 compressor.

Fixed Flow Compressor Model



$$dP_{CMPR} = P_{CMPR1} - P_{CMPR2}$$

Figure 10

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

Components

PIPE-FLO Compressible allows any component that can be described by a series of pressure vs. flow data points to be modeled and inserted into the network. These components include such items as filters and screens. From the specified data points, PIPE-FLO Compressible calculates an exponential equation to describe the pressure drop vs. flow characteristic for a component:

dPcomp = CWn

equation 44

The user inputs up to ten points for the pressure drop vs. flow relationship. The values of n and C are determined using geometric regression. This expression is valid for the range of flow rates between 0 and the last data point entered by the user. When the flow is outside this range of values, the program 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 Compressible 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.

Pressure Controlling Devices

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 11 below). A flow demand leaving the system is set at the first node (the inlet node), and a pressure source 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 pressure source set on the other end.

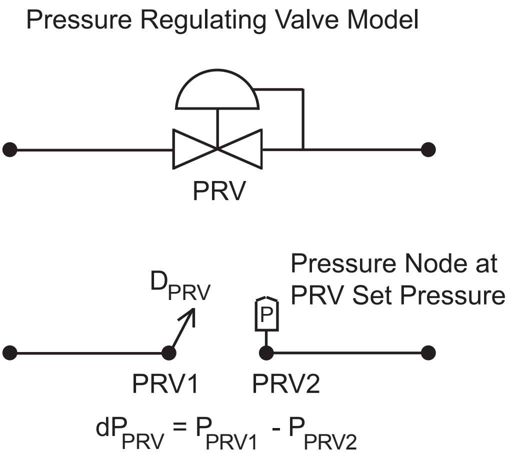


Figure 11

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 Compressible, 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 downstream 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 Compressible, 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 pressure gain device rather than as a pressure control.

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.

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

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 12 below). A pressure source 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 pressure source set on the other end.

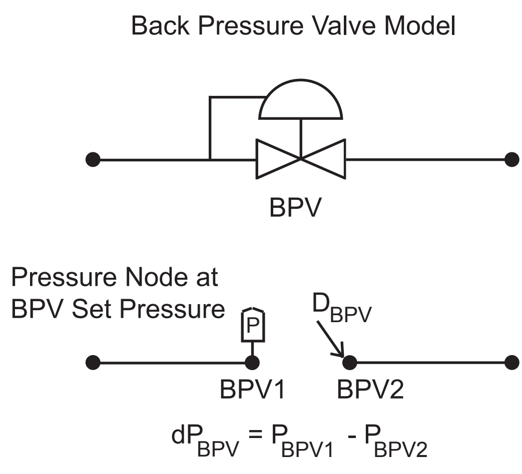


Figure 12

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 Compressible, 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 Compressible, 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 pressure gain device rather than as a pressure control.

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.

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

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.

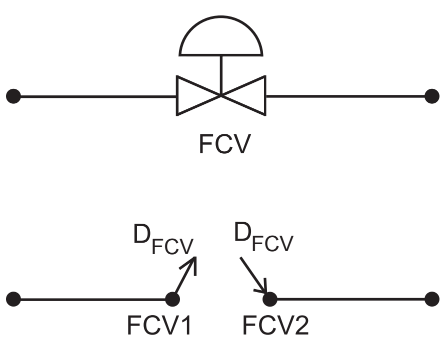
Flow Control Valves

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 Compressible models an FCV by creating two nodes in the system where the valve is installed (refer to Figure 13). 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 pressure gain device 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 Compressible calculate the flow rate.

Flow Control Valve Model



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

Figure 13

PIPE-FLO Compressible 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 each type of control (PRV, BPV, and FCV), you can import control valve catalog data. PIPE-FLO Compressible 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 Compressible program.

Total System Volume Calculation

The total system volume is listed in the Bill of Materials report and is calculated as follows:

$$V = \sum A_i * L_i$$

equation 45

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

The total volume includes both open and closed pipelines as well as pipelines that are disconnected from the system.

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 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's First Law: The Node Continuity Equation

In PIPE-FLO Compressible, a node is a point in the piping system where one or more pipelines are connected to the network. Figure 14 illustrates a network node, NN, with four connecting pipelines.

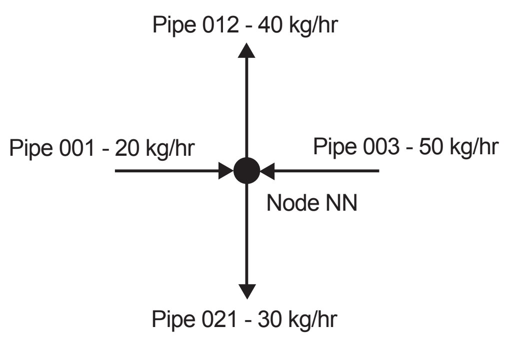


Figure 14

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 user's guide: flow into a node is assigned a negative value, and flow out of a node is assigned a positive value.

Kirchoff's First 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 46

W = 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 47

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 47) 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's Second Law: The Loop Energy Equation

The network nodes are interconnected by pipelines, generating closed circuits or loops within the system. Kirchoff's second 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 15 illustrates a network that will be used to demonstrate how the loop equations are developed.

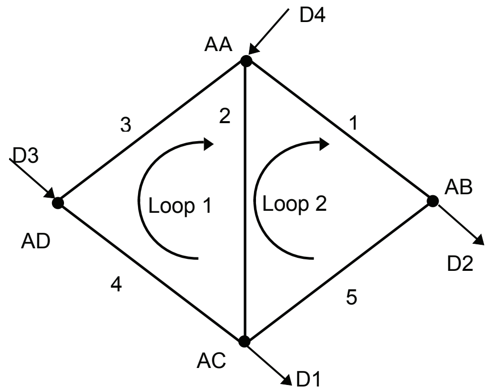


Figure 15

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 document, 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.

For the system shown in Figure 15, the following loop pressure loss equations are developed:

Loop 1

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

equation 48

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

equation 49

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 15 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 15:

$$\text{Node AA : } -W_{D4} + W_1 + W_3 - W_2 = 0$$

$$\text{Node AB : } -W_1 + W_{D2} + W_5 = 0$$

$$\text{Node AC : } -W_5 - W_4 + W_{D1} + W_2 = 0$$

equation 50

The loop pressure drop equations are:

Loop 1

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

equation 51

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

equation 52

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.

Solving Systems with Unknown Demands

If a network is supplied from two or more pressure sources, then the flow of the fluid into or out of the network is not fixed but is a function of the pressure driving the fluid into the network. In other words, the demands into or out of the network are unknown at these pressure sources, 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 Compressible, nodes with a specified constant pressure are referred to as Pressure Sources.

At a Pressure Source, the pressure is fixed at a specific value. This fixed pressure affects the flow rate in each network pipeline. A constraint in PIPEFLO Compressible is that each network must have at least one set Pressure Source. The Pressure Source is used to supply a starting pressure point for the system pressure calculations.

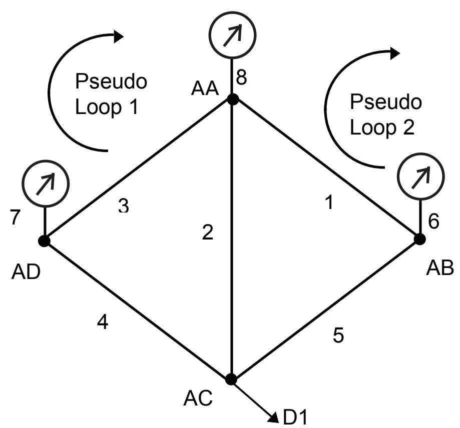


Figure 16

In Figure 16, there are three pressure nodes which are represented by a pressure gage symbol. Between any two Pressure Sources there exists a pressure difference. 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 Pressure Sources. A fictitious pipeline is installed outside the actual network to connect the Pressure Sources, completing the loop. The pressure drop in the pseudo loop is fixed at a value that accounts for the difference in the set pressures of the two Pressure Sources. Equations 53 and 54 represent the pseudo loops for the network in Figure 16.

Pseudo loop 1

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

equation 53

Pseudo loop 2

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

equation 54

Notice that with three Pressure Sources 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.

Solving the Network

The network solution can be represented as two vectors, the loop mass flow vector and the pressure vector. The mass flow rate in each independent loop in the network is represented by each coordinate of the loop mass flow vector (vectors are denoted with boldface type):

$$\mathbf{W} = (W_1, W_2, \ldots, W_{\#loops})$$

equation 55

The pressure at each network node is represented by each coordinate of the pressure vector. The size of the pressure vector will be equal to the number of nodes in the network for which the pressure is not fixed.

$$\mathbf{P} = (P_1, P_2, \ldots, P_{\#nodes})$$

equation 56

The loop mass flow vector and the pressure vector can be combined into one vector, **X**, that contains all of the system variables:

$$\mathbf{X} = (W_1, W_2, \ldots, W_{\#loops}, P_1, P_2, \ldots, P_{\#nodes})$$

equation 57

The flow rates and pressures in this vector by definition satisfy Kirchoff's first and second laws, respectively.

Penalty Function

Using the variable naming convention discussed above, Kirchoff's laws are satisfied by any values we assign to the pressure and loop flow vectors. In order to balance the network, we must find the pressure and loop flow vectors which simultaneously satisfy all of the elemental equations (Fanno flow and isentropic area change equations). Various techniques exist to search for the solution vectors (discussed below). All of these techniques depend on the concept of a penalty function.

A penalty function is a measure of how close a given vector **X** is to the solution vector denoted by **X***. The smaller the penalty, the closer the vector is to the solution. For a single element, the mass flow rate and the pressures at either end (W, P_{in}, and P_{out}) can be assigned from the coordinates of the vector **X**. A penalty for the element can then be computed, indicating how much the assigned values W, P_{in}, and P_{out} deviate from the values which would satisfy the elemental relationship. The total network penalty is the sum of the penalties produced from each element. For a balanced network, the total network penalty should equal zero. Thus, the problem of balancing the network becomes a problem of minimizing the total network penalty, which in turn is the sum of E elemental penalty functions.

Single Element Penalty

The following example illustrates the penalty function produced by a single element. From the equations of simple one dimensional compressible fluid dynamics, we know that we can compute P_{out} if P_{in} and W are known. If the values (W, P_{in}, and P_{out}) are assigned to an element, the penalty can be computed by finding the magnitude of the difference between the assigned P_{out} and the value for P_{out} calculated from the assigned P_{in} and W. For example, suppose that P_{in} is fixed at 10000 and W = 50. Using the equations of compressible fluid dynamics, we determine that P_{out} should equal 9500 – this satisfies the elemental relationship. However, if P_{out} was assigned a value of 8000, the value of the penalty for the element would be 9500 – 8000 = 1500.

Network Penalty

From the description above, we can now move onto the concept of a network penalty. Figure 17 below shows a simple network composed of two pipes connected in series with fixed pressures set at the network inlet and outlet.

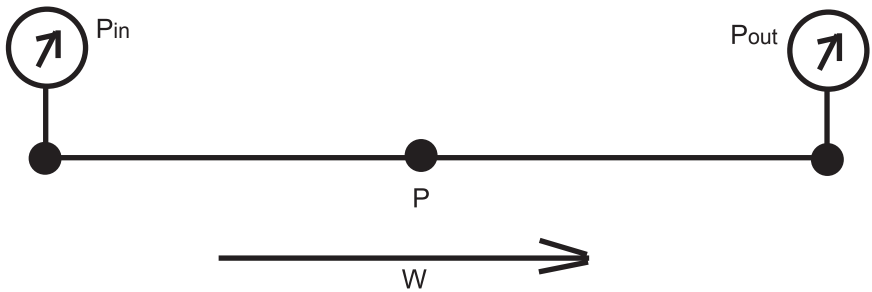


Figure 17

The network contains one pseudo-loop connecting the two pressure sources, and one internal node where the pressure is not fixed. Thus, both the loop flow vector and the pressure vector have one coordinate. For this example, we will denote the loop flow as W and the pressure at the intermediate node as P. The fixed pressures at the system inlet and outlet have values of P_{in} and P_{out}, respectively. If we assign values to W and P we can compute the total network penalty for the system as follows:

1 Compute the penalty for element A using the values W, Pin, and P.

2 Compute the penalty for element B using the values W, P, and Pout.

3 Add the elemental penalties found in steps 1 and 2.

Many penalty minimizing techniques produce an improved estimate for the assigned values (W, P_{in}, and P_{out}) by approximating the gradient of the penalty function. Therefore, we would like the gradient of the penalty function to point towards coordinates which satisfy the elemental relationship.

Convergence Methods

The network balancing calculations incorporate a number of advanced numerical solution techniques, including three penalty minimization techniques (steepest descent, Hardy Cross, and a genetic algorithm) as well as a flow averaging algorithm, a pressure averaging algorithm, and the Newton-Raphson method. The program performs the network balancing calculations using a combination of these methods, choosing the best method to drive the solution. This insures that the best method is used for each type of system during all phases of the calculation process.

The steepest descent, Hardy Cross, genetic algorithm, flow averaging, and pressure averaging methods are discussed below. The methodology of the Newton-Raphson method was discussed in a previous section.

Steepest Descent

Recall that a penalty function is a measure of how close a given vector **X** is to the solution vector denoted by **X***. The smaller the penalty, the closer the vector is to the solution. The steepest descent method is a minimization technique that depends on the gradient of the penalty function. Essentially the steepest descent method consists in the following:

1 Begin with an initial estimate for the system vector **X**.

2 Compute the gradient of the total penalty function at the current estimate.

3 Generate an improved estimate as follows:

$$\mathbf{X}_{i+1} = \mathbf{X}_i - \langle V[\text{penalty}(\mathbf{X}_i)]$$

equation 58

It should be noted that for sufficiently complex systems, it is difficult, if not impossible, to directly compute the gradient of the penalty function. In this case, one can approximate the gradient by perturbing each of the system variables in turn and recording the change in total penalty. The approximate gradient is given by the following equation:

$$\text{vpenalty}(\mathbf{X}) = \{\ [\text{penalty}(x_1 + .x,\ x_2,\ x_3,\ \ldots) - \text{penalty}(\mathbf{X})]/.x,$$
$$[\text{penalty}(x_1,\ x_2 + .x,\ x_3,\ \ldots) - \text{penalty}(\mathbf{X})]/.x,\ \ldots\}$$

equation 59

For each iteration of the steepest descent method, the penalty function must be evaluated (n+1) times in order to compute the approximate gradient, here n is equal to the number of system variables. This can be very time consuming for large systems. Also, the value of 〈 must be determined so as to be large enough that the penalty of the new estimate is significantly decreased, but not so large that the new estimate overshoots the minimum penalty along the gradient direction. One way to find the correct 〈 is to proceed along the gradient in small increments, evaluating the penalty function at each step, continuing until the penalty function starts to increase, at which point backtrack and zero in on the minimum along the gradient direction. This is also a time consuming process.

Another thing to note about the steepest descent method is that it can easily be caught in local minima. If the penalty function only had one local minimum (which then would also be the global minimum) the steepest descent method would be guaranteed to converge on this minimum, no matter what the initial estimate. However, even simple systems can have penalty functions that contain multiple local minima. An initial estimate close to a local minimum would be drawn into that local minimum, thereby missing the desired global minimum at which the solution to the network can be found. Thus, convergence of the steepest descent method can depend greatly on the initial estimate.

Hardy-Cross

The Hardy-Cross method, which is used to solve incompressible fluid networks, is a variant of the steepest descent method. In the Hardy-Cross method, only the loop flow system variables are perturbed and adjusted accordingly. This significantly reduces the number of penalty function evaluations which must be performed, while maintaining some of the favorable characteristics of the steepest descent method. After each iteration of the Hardy-Cross method, the pressure variables must be maintained by means of projecting them into the system from fixed pressure sources using the newly estimated flow rates. It should be noted that since the Hardy-Cross method is a minimization technique based on the gradient, it too can fall prey to local minima.

Genetic Algorithm

A genetic algorithm is a minimization technique which does not depend on the gradient, and thus is not hindered by local minima. The central concept of a genetic algorithm is to store a population of P data structures called chromosomes in computer memory. Each chromosome contains a record of one system vector estimate. The total penalty can be computed for each chromosome by applying its system vector to the network. Chromosomes which have lower associated penalties are more “fit” than chromosomes with greater penalties. By the principle of “survival of the fittest,” a percentage of the chromosome population with the lowest penalties survive to the next iteration of the algorithm, while the rest are deleted from memory. The resulting free space is then filled with chromosomes spawned from the lucky survivors using a combination algorithm which exploits the fact that minima tend to be centered between chromosomes with low penalties. The goal of the combination process is to produce a few children chromosomes with penalties lower than that of their parents’ chromosomes.

In order for the genetic algorithm to be robust, there must exist some mechanism by which chromosomes can diverge from one another to explore previously uncharted regions of the solution space. The process of mutation provides this mechanism. After the combination cycle of a generation of chromosomes is complete, a number of chromosomes are randomly selected from the population to be mutated. Mutation involves adding or subtracting a random amount from each of the original chromosome’s system variables.

During the early iterations of a genetic algorithm, the magnitude of mutation should be relatively high, which has the effect of enabling the chromosomes to be more mobile and to travel quickly into regions of low penalty. As the algorithm progresses, the chromosomes tend to cluster around regions of low penalty. At this stage, large mutations rarely have beneficial results, whereas genetic combination and small mutations in the correct direction may produce a significant decrease in penalty function.

Flow Averaging

The flow averaging method calculates the pipeline flow rates in the following way: For each pipeline, a flow rate is calculated from the pressures at its endpoints. Note that these calculated flow rates will not necessarily satisfy Kirchoff’s first law (continuity at the nodes) until the program is close to converging to a solution. The next step is to balance all of the flow rates at the nodes while still remaining as close as possible to the flow rates that were calculated from the endpoint pressure values. To do this, a weighted linear least squares method is used.

Pressure Averaging

The pressure averaging method calculates the nodal pressures in the following way: Suppose we have a node A that has three pipelines connected to it, with nodes B, C, and D at the opposite ends of these pipelines (refer to Figure 18). Each of these nodes has a pressure value associated with it. Using the pressure value at each node opposite of A and the flow through the corresponding pipeline, a pressure value at node A can be calculated. This will give three different pressure values which are then averaged to determine a new pressure value at node A. This process is done for each node in the system (except fixed pressure sources). Once the average pressure values are calculated for the entire network, the process is repeated and new average pressure values are obtained. These new average pressures are compared to the previously calculated average pressures. Until the change between the calculated average pressures is below a predefined threshold, the averaging process is repeated.

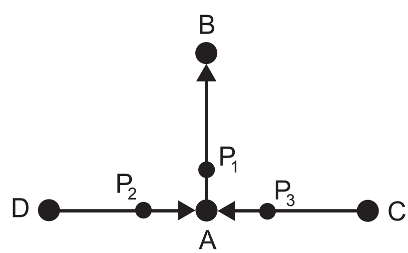


Figure 18

Standard Flow Rate Units

A standard flow rate is the gas flow rate expressed at standard conditions of temperature and pressure. The following standard US flow rate units are available in PIPE-FLO Compressible:

- 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 in PIPE-FLO Compressible:

- sm^3/min (standard cubic meters per minute)
- sm^3/hr (standard cubic meters per hour)
- sm^3/s (standard cubic meters per second)

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

References

1 Crane, Crane Technical Paper 410, Flow of Fluids through Valves, Fittings, and Pipe, 24th printing, Crane Company 1988.

2 Jeppson, Analysis of Flow in Pipe Networks, 4th edition, Ann Arbor Science Publishers, Inc. 1981.

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

4 Streeter - Wylie, Fluid Mechanics, 8th edition, McGraw Hill, Inc., New York, 1985.

5 Hodge and Koenig, Compressible Fluid Dynamics with Personal Computer Applications, Prentice Hall, Englewood cliffs, New Jersey, 1995.

6 Nichols, Compressible Fluid Flow in Networks of Frictional Pipes: Problem Solving through Graphical Analysis and Genetic Algorithms, Pacific Lutheran University, 1999.

7 Shapiro, The Dynamics and Thermodynamics of Compressible Fluid Flow, vol. 1, Ronald Press, New York, 1953.

8 Lindeburg, Mechanical Engineering Reference Manual, 8th edition, Professional Publications, Inc., Belmont CA, 1990.

CONTROL VALVE SELECTION

Introduction

The sizing and selection of valves is based upon many piping system variables. PIPE-FLO Compressibles'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 Compressible'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 Compressible 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 Compressible 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 Compressible'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 Stock's control valve selection module are as follows:

$$w = N_6 F_P F_R C_v (dP/\rho)^{½}$$

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 Compressible performs a linear interpolation to determine the Cv values for positions between the increments found in the valve catalog.

Fp Piping Geometry Factor

The Cv 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 Fp factor takes into account the effects of the inlet and outlet reducers.

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

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

equation 2

N2 = 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 Compressible 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 Compressible 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:

$$C_v\ (\text{search}) = C_v\ (\text{full size}) / F_p$$

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 and 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 Compressible 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 Cv value for laminar flow (equation 5)
Cvt = the Cv value for fully developed turbulent flow (equation 1, without Fp)

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

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

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

N6 = conversion coefficient
QMAX = maximum mass flow rate
Pvc = absolute pressure at vena contracta
FLP = liquid pressure recovery factor with reducers installed,

dimensionless

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

$$P_{vc} = F_F\ P_v$$

equation 9

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 Compressible'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:

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

equation 10

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:

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

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 Compressible's control valve selection module are as follows:

$$w = N_6 F_P C_v Y (X P_1 \rho)^{½}$$

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 Compressible'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 X_T is determined experimentally and supplied by the valve manufacturer. The valve catalog stores X_T 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 X_T, the valve is not suitable for use with compressible fluids. If a manufacturer only provides a partial data set for X_T, 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 X_T, resulting in a new factor designated as X_{TP}. The value of X_{TP} 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 = 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 Compressible'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 Compressible 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 Compressible 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 Compressible'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 Compressible'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 Compressible'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 = 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. Fa is the thermal expansion correction factor. The value of Fa depends on the meter material, the pipe material, and the temperature of the process fluid. The following equation is used to determine Fa:

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

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 Compressible'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 = 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 Compressible 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.

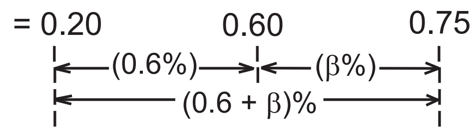
Equation Limits and Discharge Coefficient Uncertainties

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



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

$2000 \leq R_D \leq 10{,}000$

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.