

# **PIPE-FLO® PROFESSIONAL METHOD OF SOLUTION**

## **PIPE-FLO Overview**

PIPE-FLO calculates the balanced flow rates and pressures within fluid piping systems. Systems are modeled by drawing the piping system and then designing and sizing the individual pipelines to form the complete network. Components such as pumps, filters, and heat exchangers can be modeled and inserted directly into the system. Various controls for limiting the flow and pressure within the system are available for use. The program performs all internal flow calculations using mass flow rates. This method of solution describes the engineering methods used and the program assumptions made with PIPE-FLO

## **Calculating the Pressure Drop**

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

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

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

PIPE-FLO uses the Darcy-Weisbach equation to calculate the pressure drop in a pipeline. This method takes into account fluid viscosity and pipe roughness, providing valid results for incompressible Newtonian fluids flowing in any round fully charged pipe. This formula can also be extended to compressible fluids with some restrictions.

The Darcy-Weisbach equation is as follows:

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

*equation 1*

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

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

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

*equation 2*

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

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

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

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

*equation 3*

The Darcy friction factor is referred to throughout the PIPE-FLO User’s Guide and program.

## Determining the Pipe Friction Factor

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

Reynolds developed the following relationship:

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

*equation 4*

W = flow rate, kg/hr

d = pipe diameter, mm

$\mu$  = fluid dynamic viscosity, centipoise

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

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

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

$$f = 64/R_e$$

*equation 5*

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

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

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

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

- 1 At high Reynolds numbers the pipe friction factor becomes constant.
- 2 For rough pipes the ratio of surface material roughness to pipe diameter, or relative roughness, is more important than the Reynolds number for determining the friction factor.

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

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

*equation 6*

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

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

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

*equation 7*

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

## Valves and Fittings

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

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

*equation 8*

$f_T$  = turbulent friction factor

L = length, m

d = pipe diameter, mm

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

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

*equation 9*

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

## Differential Pressure Calculator

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

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

*equation 10*

$\Delta P$  = Change in pressure, bars

d = Pipe diameter, mm

$\rho$  = fluid density  $\text{kg/m}^3$

W = Mass flow rate, kg/hr

## 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.

The K value and  $C_v$  flow coefficient have the following correlation:

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

*equation 11*

d = inside pipe diameter, inches

If you know the  $C_v$  flow coefficient for a valve, you can use the dP Calculator feature described above to quickly determine its corresponding K value. Simply specify the  $C_v$  value for the flow rate and a pressure drop of 1 psi in the dP Calculator, and the program calculates the corresponding K value. Note that you may have to make adjustments to the  $C_v$  value if the properties of the fluid you are using vary significantly from that of 60°F water. Also, if you are not using gpm and psi for your flow rate and pressure units, you must make the necessary conversions prior to calculating the K value.

## Reducers

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

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

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

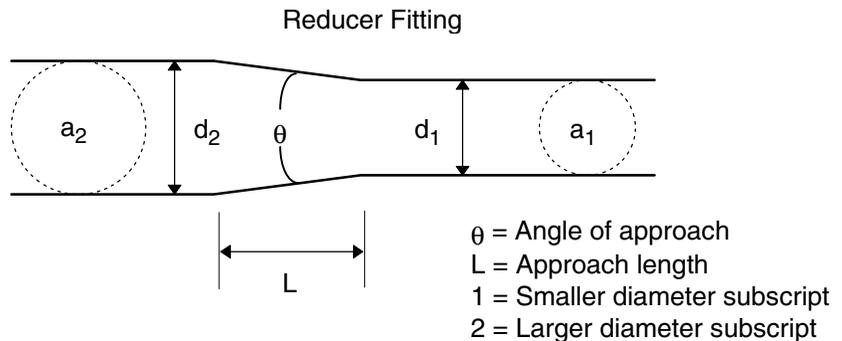


Figure 1

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

### Reducer Contraction

$$K_1 = 0.8 \sin(\theta/2) (1 - \beta^2) \quad (\theta \leq 45^\circ) \quad \text{equation 12a}$$

$$K_1 = \frac{1}{2} (1 - \beta^2) (\sin(\theta/2))^{1/2} \quad (45^\circ < \theta \leq 180^\circ) \quad \text{equation 12b}$$

**Reducer Enlargement**

$$K_1 = 2.6\sin(\theta/2)(1-\beta^2)^2 \quad (\theta \leq 45^\circ) \quad \text{equation 13a}$$

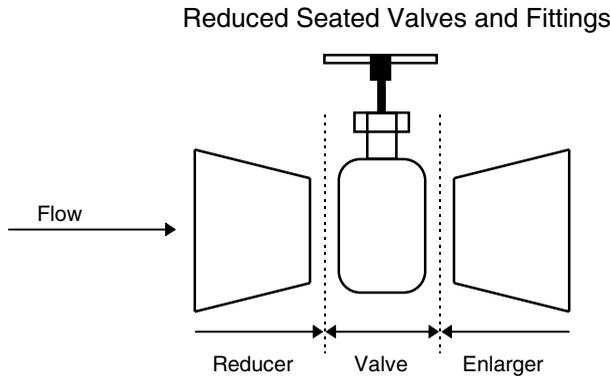
$$K_1 = (1-\beta^2)^2 \quad (45^\circ < \theta \leq 180^\circ) \quad \text{equation 13b}$$

$\theta$  = Angle of approach

$\beta$  =  $d_{\text{minor}} / d_{\text{major}}$

**Reduced Seated Valves and Fittings**

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



*Figure 2*

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

The total K for the reduced seat valve is:

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

equation 14

$K_2$  = K value of the total valve

$K_1$  = K value of reduced seat

$K_{\text{Reducer}}$  = K value of reducer

$K_{\text{Enlarger}}$  = K value of enlarger

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

## PIPE-FLO Valve Table Formulas

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

### Formula 1 - Pipe Contraction

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

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

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

Formula 1

**Formula 3 - Pipe Enlargement**

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

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

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

*Formula 3*

**Formula 5 - Reduced Seat Valve, Gradual Change in Diameter**

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

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

*Formula 5*

**Formula 7 - Reduced Seat Valve, Abrupt Change in Diameter**

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

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

*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**

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

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

*Formula 9*

**Formula 10 - Full Seated Valves and Fittings**

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

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

*Formula 10*

**Formula 11 - Fixed K Value**

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

## Mass and Volumetric Flow Rate Units

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

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

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

## Compressible Fluid Flow in Pipelines

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

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

- 1 If the calculated pressure drop in a pipeline ( $P_{in} - P_{out}$ ) is less than 10% of the inlet pressure, acceptable accuracy is obtained if the density used for the fluid is based upon either the upstream or downstream conditions.
- 2 If the calculated pressure drop ( $P_{in} - P_{out}$ ) is greater than 10% but less than 40% of the inlet pressure, the Darcy-Weisbach method provides acceptable accuracy if the fluid density is based upon the average of the inlet and outlet fluid conditions.

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

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

PIPE-FLO does not automatically account for changes in fluid density due to the pressure drops calculated in the lineup. Once a fluid pressure has been selected during the pipeline sizing portion of the program, the density corresponding to that pressure is used throughout the analysis, regardless of the pressures calculated at the pipeline endpoints.

Systems where the pressure drop exceeds the 40% range include long gas transmission pipelines and systems where sonic flow conditions exist. Also, if work is being done on the fluid by a device, such as a compressor or a turbine, the pressure drop will fall outside the 40% range as well.

The pressure design limits in the PIPE-FLO pipe specifications can be set up to inform you when the 40% pressure drop restriction is exceeded.

When analyzing a compressible fluid system, it is recommended that mass flow rate units be used instead of volumetric flow units.

## **Two Phase Flow and Slurries**

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

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

## System Fluid Pressure

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

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

*equation 15*

H = total head

P = pressure

$\rho$  = fluid density

v = fluid velocity

Z = elevation

g = gravitational constant

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

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

*equation 16*

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

In certain applications, such as pump suction and control valve sizing, the velocity component of the pressure must be taken into account. If required, the user can manually factor the velocity component into the final system pressure. PIPE-FLO's pump selection and control valve selection modules take the fluid velocity information into account when looking at these special cases.

## **Lineup Calculations**

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

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

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

## **Network Flow Equations**

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

- 1 The algebraic sum of the mass flow into and out of all nodes must equal zero.
- 2 The algebraic sum of the pipeline pressure drops around a loop must equal zero.

## Kirchoff Law 1: The Node Continuity Equation

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

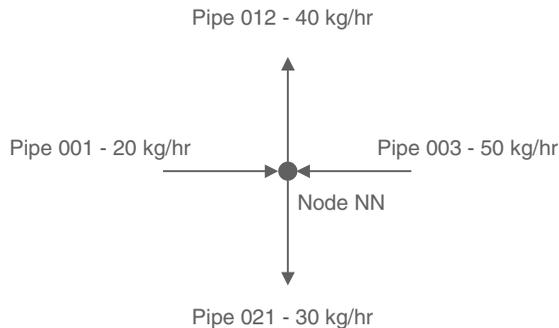


Figure 3

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

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

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

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

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

*equation 17*

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:

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

*equation 18*

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 18) is linear.

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

## **Kirchoff Law 2: The Loop Energy Equation**

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

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

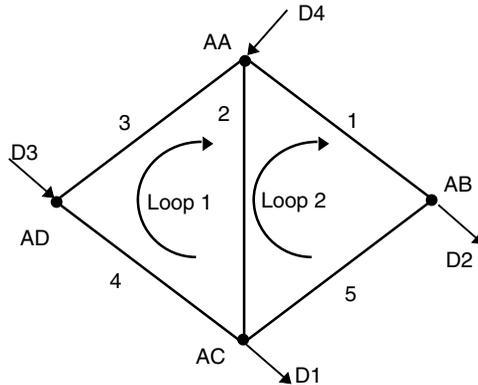


Figure 4

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

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

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

Loop 1

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

*equation 19*

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

*equation 20*

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

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

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

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

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

*equation 21*

The loop pressure drop equations are:

Loop 1

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

*equation 19*

Loop 2

$$dP_1 + dP_5 + dP_2 = 0$$

*equation 20*

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

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

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

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

*equation 22*

P = pipelines in the network

N = nodes in the network

L = non-overlapping loops in the network

## **Solving Systems with Unknown Demands**

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

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

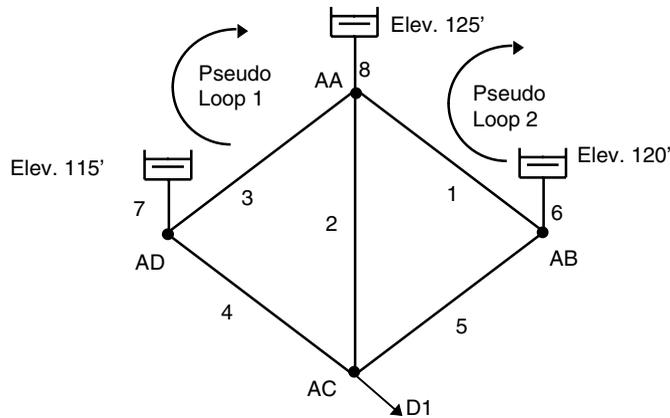


Figure 5

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

A pseudo loop consists of a path of pipelines in the network which connects two fixed pressure points (tanks or pressure demands). A fictitious pipeline is installed outside the actual network to connect the fixed pressure points, completing the loop. The pressure drop in the pseudo loop is fixed at a value that accounts for the difference in hydraulic grade between the two fixed pressure points. Equations 23 and 24 represent the pseudo loops for the network in Figure 5.

Pseudo loop 1

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

equation 23

Pseudo loop 2

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

equation 24

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

## Pumps

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

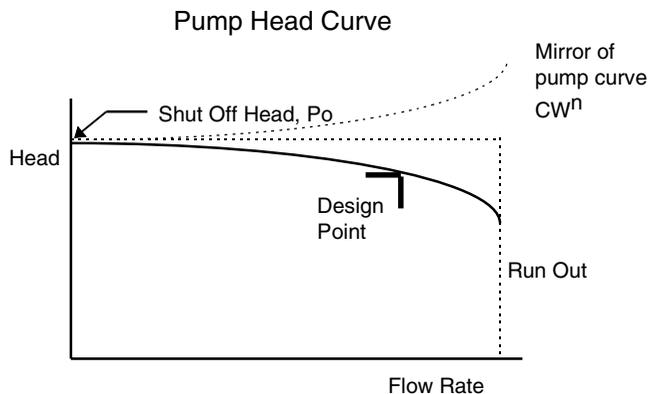


Figure 6

A characteristic head curve for a centrifugal pump is shown in Figure 6. Notice on the curve that the pressure head is the highest at a zero flow rate. This is called the shut off head and it is the maximum head the pump can deliver. As the flow rate through the pump increases, the pump's head decreases. The maximum flow which the pump can safely pass is called the run out flow.

PIPE-FLO reduces five points on the curve to an exponential equation as shown in equation 25.

$$P_{\text{pump}} = P_0 - CW^n$$

*equation 25*

The equation for the head curve has two components, a constant value  $P_0$  and a coefficient times the flow rate raised to the  $n$ th power,  $CW^n$ . The constant  $P_0$  corresponds to the shut off head of the pump.  $C$  and  $n$  are determined using geometric regression.

As shown in Figure 6, the pump is being modeled as a step increase in head ( $P_0$ ) followed by a head loss due to flow through the pump ( $-CW^n$ ). This is similar to the way a component is modeled (see equation 26 in the next section). By using similar methods to model both pumps and components, all of the energy equation exponents are approximately second order. This improves the convergence capability of the solution.

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

## Running a Pump at a Fixed Flow Rate

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

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

- When a fixed flow rate is specified for a pump, the performance curve data (if specified) is overridden.
- PIPE-FLO models a fixed flow pump by creating two nodes in the system where the pump is installed (refer to Figure 7). A flow demand value equal to the set flow rate is taken out of the first pump node (the suction node) and an equal flow demand is set entering the second node (the discharge node).
- If a pump status is listed as invalid, it means that you have set the flow rate lower than what would naturally occur through the line if the pump was not installed. The pump then acts as “valve” in order to limit the flow rate to the set value. If this occurs, you should either increase the set flow rate or remove the pump.

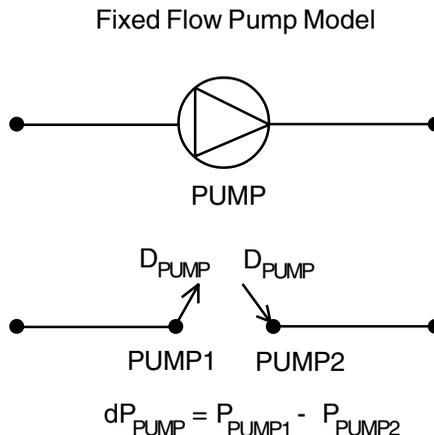


Figure 7

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

## Calculating NPSH Available

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

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

*equation 25a*

Pa = atmospheric pressure

Pv = vapor pressure (absolute)

Pn = gage pressure at the pump suction

## Components

PIPE-FLO allows any component that can be described by a series of head vs. flow data points to be modeled and inserted into the network. These components include such items as special valves, heat exchangers, and filters. From the specified data points, PIPE-FLO calculates an exponential equation to describe the head vs. flow characteristic for a component. The form of the component equation is similar to that of the pipeline pressure drop equation:

$$dP_{\text{comp}} = CW^n$$

*equation 26*

The user inputs five points on the head 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.

## 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 8 below). A flow demand leaving the system is set at the first node (the inlet node), and a pressure demand equal to the PRV set pressure is set at the second node (the outlet node).
- The direction of flow through the PRV, from high pressure to low pressure, is determined from the direction of the connecting pipelines.
- The PRV inlet and outlet pipelines cannot have a tank or pressure demand set on the other end.

Pressure Regulating Valve Model

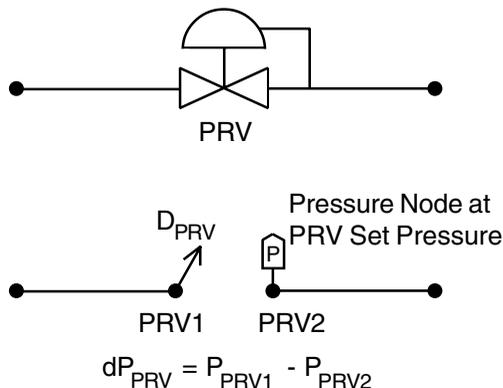


Figure 8

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

- 1 **Regulating:** The valve maintains the downstream pressure to the set value by throttling the flow rate through the PRV.
- 2 **Fully Closed:** This situation occurs if the valve set pressure is less than the pressure downstream of the valve for the case where the valve is closed. When this situation occurs in an actual piping system, the flow through the PRV reverses and the valve acts as a check valve, closing the pipeline. In PIPE-FLO, the flow also reverses, however the PRV does not act like a check valve. The pipeline remains open and the PRV results are invalid because the flow is going into the Pressure Node from the 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, the differential pressure across the valve will correspond to a pressure gain rather than a pressure drop. The PRV results are invalid because it is acting as a pump rather than as a pressure control.

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

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

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

## Back Pressure Valves

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

The following points should be noted about BPVs:

- The set pressure at the upstream side of the BPV is entered with the valve. For the BPV model, the program creates two nodes in the system where the BPV is installed (see Figure 9 below). A pressure demand equal to the BPV set pressure is set at the first node (the inlet node), and a flow demand is set at the second node (the outlet node).
- The direction of flow through the BPV, from high pressure to low pressure, is determined from the direction of the connecting pipelines.
- The BPV inlet and outlet pipelines cannot have a tank or pressure demand set on the other end.

Back Pressure Valve Model

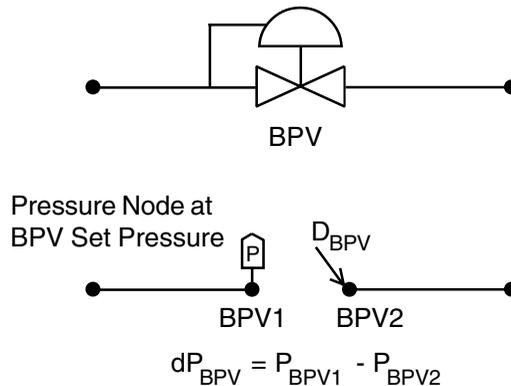


Figure 9

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

- 1 **Regulating:** The valve maintains the upstream pressure to the set value by regulating the flow rate through the BPV.
- 2 **Fully Closed:** This situation occurs if the valve set pressure is greater than the pressure upstream of the valve for the case where the valve is closed. When this situation occurs in an actual piping system, the flow through the BPV reverses and the valve acts as a check valve, closing the pipeline. In PIPE-FLO, the flow also reverses, however the BPV does not act like a check valve. The pipeline remains open and the BPV results are invalid because flow is going from the Pressure Node to the pipeline upstream node.
- 3 **Fully Open:** This situation occurs if the valve set pressure is less than the pressure at the valve outlet for the case where the valve is fully open. When this situation occurs in an actual piping system, the BPV maintains a fully open position and it has no effect on the pipeline flow conditions. In PIPE-FLO, the differential pressure across the valve will correspond to a pressure gain rather than a pressure drop. The BPV results are invalid because it is acting as a pump rather than as a pressure control.

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

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

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

## Flow Control Valves

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

The following points about FCVs should be noted:

- The direction of flow through the FCV is determined from the direction of the connecting pipelines.
- PIPE-FLO models an FCV by creating two nodes in the system where the valve is installed (refer to Figure 10). A flow demand value equal to the set flow rate is taken out of the first node (the inlet node) and an equal flow demand is set entering the second node (the outlet node).
- If an FCV is status is Invalid, you should first check the direction of flow through the line. If the flow direction is correct, the invalid flag means that you have set the flow rate in the FCV to a value higher than would naturally occur in the line and the FCV is acting as a “pump” in order to achieve the set flow rate. In an actual piping system, the valve would be 100% open, so you should change valve setting to fully open and let PIPE-FLO calculate the flow rate.

Flow Control Valve Model

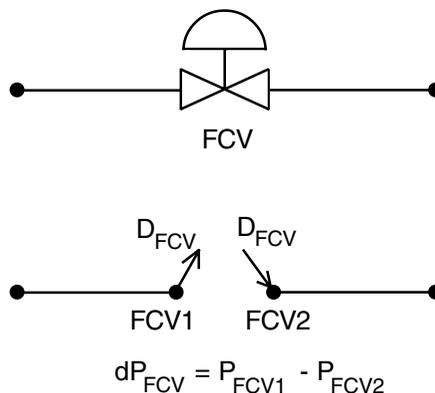


Figure 10

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

## **Calculation of Lineup Flow Rates and Pressures**

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

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

### **Hardy Cross Method**

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

## Linear Method

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

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

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

$$dP = K_P W^n + K_{V\&F} W^2 - (P_O + K_C W^C)$$

*equation 27*

Taking the first derivative of the loop pressure drop equation (equation 27) produces the following expression.

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

*equation 28*

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

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

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

## **Calculational Controls**

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

## **System Resistance Curve**

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

- There can only be one active pump in the system (all other pumps are shut off).
- The program removes all controls (FCVs, PRVs, and BPVs) in the system.
- The program removes all flow demands in the system.
- The resistance curve can only be calculated for one system loop.

PIPE-FLO generates the system resistance curve by applying five different fixed flow rates to the selected pump and calculating the corresponding TDH for these flow rates. The data is then plotted against the pump curve (if one is entered for the pump).

## Total System Volume Calculation

The total system volume is calculated as follows:

$$V = \sum A_i * L_i$$

*equation 28*

V = total system volume

A<sub>i</sub> = cross-sectional area of a pipe

L<sub>i</sub> = length of a pipe

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

## References

- 1 Crane, Crane Technical Paper 410, "Flow of Fluids through Valves, Fittings, and Pipe" twenty fourth printing, Crane Company 1988.
- 2 Wood, "BASIC Computer Program for the Analysis of Pressure and Flow in Pipe Distribution Systems Including Extended Period Simulations", Office of Engineering Continuing Education, College of Engineering, University of Kentucky, 1982.
- 3 Jeppson, "Analysis of Flow in Pipe Networks", fourth printing, Ann Arbor Science Publishers, Inc. 1981.
- 4 Anton, "Elementary Linear Algebra", third edition, John Wiley & Sons, Inc. 1981.
- 5 Brater and King, "Handbook of Hydraulics" sixth edition, McGraw-Hill, New York 1976.
- 6 Vennard, "Elementary Fluid Mechanics" fourth edition, John Wiley & Sons, Inc., New York, London, 1961.
- 7 Streeter - Wylie, "Fluid Mechanics" eighth edition, McGraw Hill, Inc., New York, 1985.