

## Chapter 4 – Fluid Flow

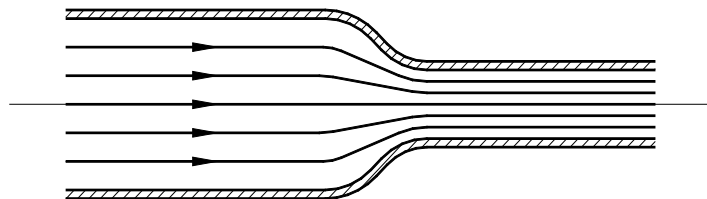
### 4.1 Introduction

The theory of fluid motion is highly complex. The development of the methods of solution of most practical fluid motion problems therefore combine theoretical considerations together with experimental results. Often simplifications will be made during the solution process. The solutions to problems thus determined tend to be somewhat approximate by nature. Also, in many situations the data used in the solution is not well defined. Most problems can be solved to within a few percent of the "true" solution, but with some problems a solution within twenty percent of the actual value would be considered acceptable. In problems involving a high expense or risk of loss of life, a greater attention to detail is required and solutions should be verified by experimental analysis whenever possible.

### 4.2 Characteristics of Fluid Motion

Fluid motion can be categorized in different ways. Before we characterize fluid motion, we need to consider some fundamental characteristics of the dynamics of fluids. In classical dynamics, we consider the motion of a particle. We could follow this approach and consider the fluid to be made up of a large number of particles. The problem with this approach is that an enormous effort would be required to trace the motion of each and every particle. A simpler approach is to consider a fixed point in space and consider the particles that pass by the point. Such an approach is known as the **Eulerian approach**.

As we examine the particles of fluid moving by a point, it is convenient to consider lines that represent the path of the fluid particles. Such lines are called **streamlines**. Streamlines are always parallel to the velocity vectors of the particles at the position under consideration. Figure 4.1 shows the streamlines for a pipe reducer. Streamlines can be considered to be analogous to the smeared light images that we see in a time exposure of a downtown street corner at night.



**Figure 4.1 Streamlines**

If the velocity vectors in a fluid flow do not change in either magnitude or direction from one position to another, the streamlines will be parallel and the flow is said to be **uniform**. On the other hand, if the velocity at one location on a streamline is different than that of a point further along the streamline, the flow is said to be **non-uniform**. In other words, for uniform flow, the rate of change of velocity with respect to position is equal to zero. In Figure 4.1, the streamlines in the sections of pipe before and after the reducer section are straight and parallel indicating that the flow is uniform. Within the reducer section, however, the streamlines are converging indicating non-uniform flow.

If the velocity vectors at a given position remain constant in both magnitude and direction over the period of time under consideration, then the flow is referred to as being **steady**. On the other hand, if the velocity at a given position varies with time, the flow is said to be **unsteady** or **non-steady**.

Examples of the various types of flow and their characteristics are summarized in Table 4.1.

**Table 4.1 Types of Flow**

Type of Flow	Example
Steady, uniform flow	Flow at a constant rate through a long straight pipe of constant cross-section.
Steady non-uniform flow	Flow at a constant rate through a tapering pipe.
Non-steady, uniform flow	Accelerating or decelerating flow through a long straight pipe of constant cross-section.
Non-steady, non-uniform flow	Accelerating or decelerating flow through a tapering pipe.

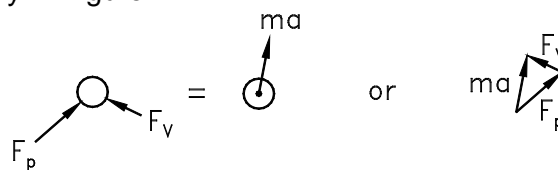
The majority of the problems addressed in this text will involve steady uniform flow. However, we will examine non-uniform flow in open channels, and will consider unsteady flow when examining a phenomenon known as 'water hammer'.

**4.2 Laminar and Turbulent Flow**

In addition to the classifications discussed in Section 4.1, fluid flow can be classified as either laminar or turbulent. In laminar flow, the streamlines move in smooth consistent paths. In turbulent flow, the particle motion is random. In general, if the flow is slow and/or the fluid is highly viscous, the flow will be laminar, whereas if the flow is rapid and/or the viscosity of the fluid is low, the flow will be turbulent.

Technically, turbulent flow will be non-steady. At a given location, the velocity will vary with time. However, if we average the velocity over a reasonable period of time, the time-averaged velocity will be constant and the flow can then be considered to be steady flow.

When studying the causes and conditions for turbulent flow, Reynolds determined that the ratio of the inertia forces to the viscous forces was an important parameter. We can consider Newton's second law,  $\Sigma F = ma$ , as being applied to a 'particle' of fluid. The right side of the equation represents the 'inertia forces'. The sum of the forces on the left side of the equation consists of the pressure force,  $F_p$ , which tends to drive the motion of the fluid and the viscous or friction force,  $F_v$ , which tends to retard the motion. In order to satisfy the equation the vector sum of the pressure force and the viscous force equals the 'inertia force'. This is shown graphically in Figure 4.2.



**Figure 4.2 Forces acting on a particle of fluid**

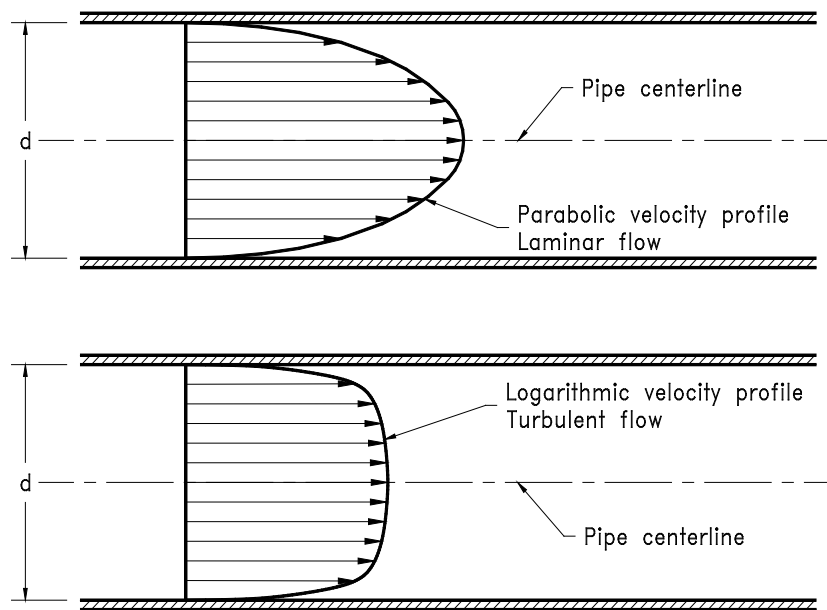
The ratio of the inertia forces to the viscous forces is known as **Reynolds number** and, for a closed conduit, can be represented as

$$N_R = \frac{vd}{\nu} \tag{4.1}$$

where  $v$  = velocity of the fluid  
 $d$  = a characteristic dimension (the diameter for a round pipe)  
 $\nu$  = kinematic viscosity =  $\frac{\mu}{\rho}$  .

Through experiment, Reynolds discovered that for pipe flow, if  $N_R$  is less than approximately 2000, the flow would remain laminar. This value of  $N_R$  is often referred to as the **critical Reynolds number**. In general, if  $N_R$  exceeds 4000, the flow will become turbulent. In between, is a **transition zone** with  $2000 < N_R < 4000$  where it is impossible to predict whether the flow would be laminar or turbulent. Today a better understanding of transitional flow has been achieved by applying a relatively new branch of mathematics known as chaos theory.

The flow velocity profile within a pipe varies for laminar vs. turbulent flow. For laminar flow, the fluid motion is entirely in one direction and an examination of the forces acting can give a mathematically exact representation of the flow. It can be shown that the velocity will vary from zero at the pipe wall to a maximum at the pipe centerline as a parabolic function. See figure 4.3. For turbulent flow the random motion of the particles of fluid precludes an exact mathematical solution, although experimental results indicate that the time-averaged velocity profile follows a logarithmic curve as shown in figure 4.3. The variation of velocity across the pipe cross-section has some significance when energy and momentum are discussed later on.



**Figure 4.3 Velocity Profiles for Laminar and Turbulent Flow in Pipes**

**Example 4.1**

Oil with a relative density of 0.925 and an absolute viscosity of 0.10 Pa·s flows through a 6.0 mm diameter pipe with a mean velocity of 2.0 m/s. Will the flow be laminar or turbulent?

**Solution:**

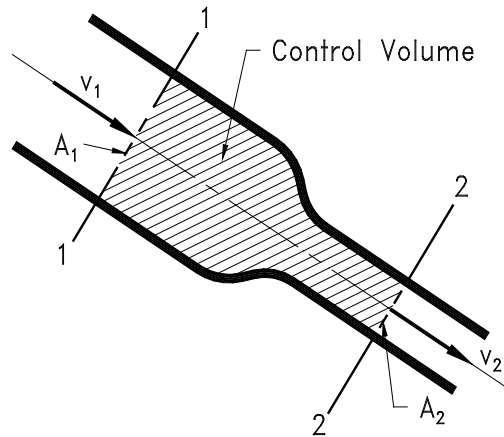
$$N_R = \frac{vd}{\nu} = \frac{\rho vd}{\mu} = \frac{(0.925)(1000 \text{ kg / m}^3)(2.0 \text{ m / s})(0.006 \text{ m})}{0.10 \text{ Pa} \cdot \text{s}} = 111.0 < 2000$$

Therefore the flow will be laminar.

**4.3 The Continuity Equation for Steady Incompressible Flows**

Many problems in engineering can be solved by considering the flow to be steady and incompressible. For most problems, liquids can be considered incompressible, although, as was noted in Section 2.7, they do compress to a certain degree.

Figure 4.4 shows a section of pipe with a liquid flowing through it. We will consider the body of fluid that initially lies between the between sections 1-1 and 2-2. Such a body of fluid is known as a control volume. In some ways it is analogous to the free-body diagram used in statics.



**Figure 4.4 Flow through a control volume**

The mass of the fluid entering the control volume in a period of time,  $\Delta t$ , can be given as  $\rho v_1 A_1 \Delta t$ , where  $v_1$  is the velocity of fluid entering the control volume and  $A_1$  is the cross-sectional area of the pipe at section 1-1. Similarly the mass leaving the control volume is  $\rho v_2 A_2 \Delta t$ . For incompressible steady flow, the principle of conservation of mass requires that the mass entering the control volume equals the mass leaving the control volume. We have

$$\rho v_1 A_1 \Delta t = \rho v_2 A_2 \Delta t$$

Dividing through by  $\rho$  and  $\Delta t$  gives  $v_1 A_1 = v_2 A_2 = Q$  (4.2)

where  $Q$  is the volume flow rate (usually in  $\text{m}^3/\text{s}$ ).

Equation 4.2 is known as the steady state **continuity equation** for incompressible flow. It should be noted that the velocities indicated are the mean velocities.

**Example 4.2**

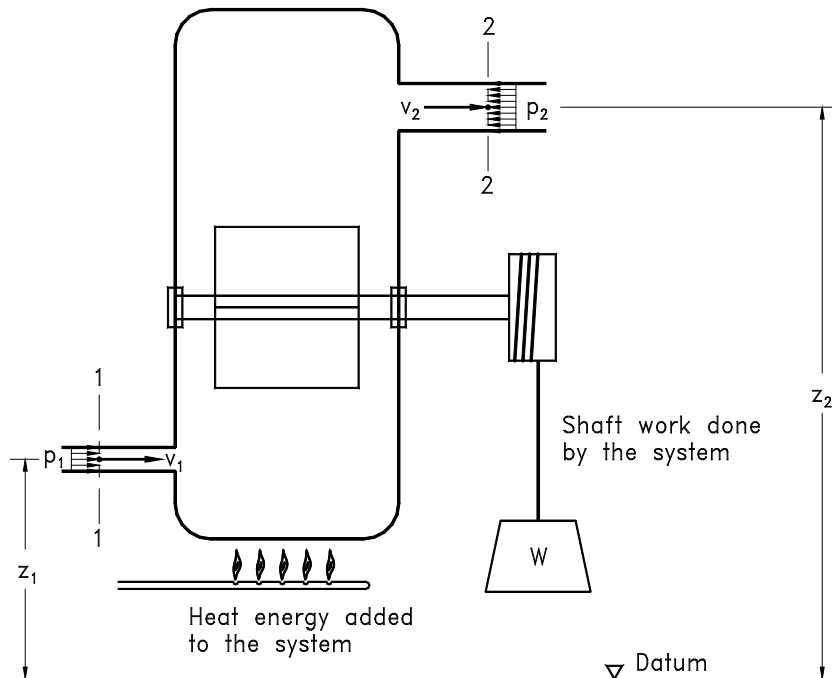
Water flows through a pipe reducer section at a rate of 30 L/s. The pipe diameter before the reducer is 200 mm and the diameter after the reducer is 150 mm. Determine the velocity in both sections of pipe.

**Solution:**

$$v_1 = \frac{Q}{A_1} = \frac{(0.03 \text{ m}^3 / \text{s})}{\pi(0.200 \text{ m})^2} = \underline{0.955 \text{ m/s}} \quad v_2 = \frac{Q}{A_2} = \frac{(0.030 \text{ m}^3 / \text{s})}{\pi(0.150 \text{ m})^2} = \underline{1.698 \text{ m/s}}$$

**4.4 The Energy Equation**

As well as satisfying the principle of conservation of mass, the principle of conservation of energy must also be satisfied. Energy can take many forms. In applying the principle of conservation of energy, we will consider potential energy due to position in the earth's gravitational field, kinetic energy, pressure energy, heat energy and shaft work. Consider the device shown in Figure 4.5. The fluid enters the device with potential, kinetic and pressure energy. Within the device, heat energy may be added to the fluid and/or heat energy will be lost due to friction (viscous drag). In addition, the device may do some shaft work as for example power a turbine.



**Figure 4.5 Energy in a fluid flow**

**Pressure energy:**

Pressure energy is another form of potential energy and is equal to the product of the pressure and a volume. For an incompressible fluid, the volume of fluid entering the control volume during the time period,  $\Delta t$ , is the same as that leaving and is equal to  $Q\Delta t$ . The pressure energy of the fluid entering the system is given as  $p_1Q\Delta t$  and the pressure energy of the fluid leaving the system is  $p_2Q\Delta t$ .

**Gravitational Potential Energy:**

The mass entering the system in a given period of time,  $\Delta t$ , is  $\rho v_1 A_1 \Delta t = \rho Q \Delta t$ . The potential energy,  $m \cdot g \cdot h$ , of this mass is  $P.E._1 = \rho Q \Delta t \cdot g \cdot z_1$ . Similarly the potential energy of the mass leaving the control volume is  $P.E._2 = \rho Q \Delta t \cdot g \cdot z_2$ .

**Kinetic Energy:**

The kinetic energy is given by  $\frac{1}{2}mv^2$ . Therefore, the mass entering the control volume during the time,  $\Delta t$ , has kinetic energy  $K.E._1 = \frac{1}{2} \rho Q \Delta t \cdot v_1^2$  and the mass leaving the system has kinetic energy  $K.E._2 = \frac{1}{2} \rho Q \Delta t \cdot v_2^2$ . In calculating the kinetic energy, we use the mean velocity,  $v = Q/A$ . As shown in Figure 4.3, the velocity varies across the pipe diameter. This introduces an error when the kinetic energy is calculated using the mean velocity. A correction factor,  $\alpha$ , called the kinetic energy correction factor may be introduced. For turbulent flow in pipes,  $\alpha \approx 1.05$ . Since this is close to unity, the correction factor is often ignored for turbulent flow. For laminar flow in pipes,  $\alpha = 2.0$ . The kinetic energy correction factor in open channels can also be significant, ranging from approximately 1.1 to 1.7.

**Heat energy:**

The heat energy of a substance is determined by taking the product of its mass with a property known as its **specific heat** and the absolute temperature. Specific heat is then a measure of how much heat energy a substance holds at a particular temperature. The specific heat of water at various temperatures is given in Table 4.2.

**Table 4.2 Specific Heat of Water**

Temperature (°C)	Specific Heat (Joules/kg/°C)
0	4218
10	4192
20	4182
30	4179
40	4179
50	4181
60	4184
70	4190
80	4196
90	4205
100	4216

The fluid entering the system has a heat energy given as  $\rho Q \Delta t \cdot e_1$ , where  $e_1$  is the energy per unit mass for the fluid entering the system. Similarly the heat energy of the fluid leaving the system is  $\rho Q \Delta t \cdot e_2$ . In addition heat may be added to the system. The heat added to the system per unit mass will be designated as  $q$ .

**Shaft Work:**

The shaft work done by the system per unit mass will be designated as  $w'$ .

**Energy Balance:**

The energy balance will be done on a per unit mass basis. The gravitational, kinetic, pressure, and heat energy terms will have to be divided by  $\rho Q \Delta t$  to obtain energy per unit mass.

The energy per unit mass being added to the system is:

$$\frac{p_1}{\rho} + \frac{v_1^2}{2} + g \cdot z_1 + e_1 + q \tag{4.3}$$

The energy per unit mass taken out of the system is:

$$\frac{p_2}{\rho} + \frac{v_2^2}{2} + g \cdot z_2 + e_2 + w' \tag{4.4}$$

The law of conservation of energy states that the energy must balance; that is that the total system energy must remain constant. It is customary to divide through by  $g$  and represent the energies as energy head with units of metres (or feet). This in effect gives the energy per unit weight.

Equating 4.3 and 4.4, and dividing through by  $g$  gives:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 + \frac{e_1 - e_2 + q}{g} = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 + \frac{w'}{g} \tag{4.5}$$

In most cases the only heat transfer that takes place is due to heat generated by friction. In these situations this heat is lost from the system and the  $q$  is negative ( $-q$ ). The total head lost to friction,  $h_f$ , can then given by

$$h_f = \frac{e_2 - e_1 - q}{g} \tag{4.6}$$

Rewriting equation 4.5 substituting in 4.6 gives

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 - h_f = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 + \frac{w'}{g} \tag{4.7}$$

Equation 4.7 is known as the **steady flow energy equation**. It should be noted that, in the form given, it is valid only for constant density fluids. The term  $w' / g$  represents the shaft work per unit weight done by the fluid. When shaft work is done on the fluid as for instance from the action of a pump,  $w' / g$  will be negative.

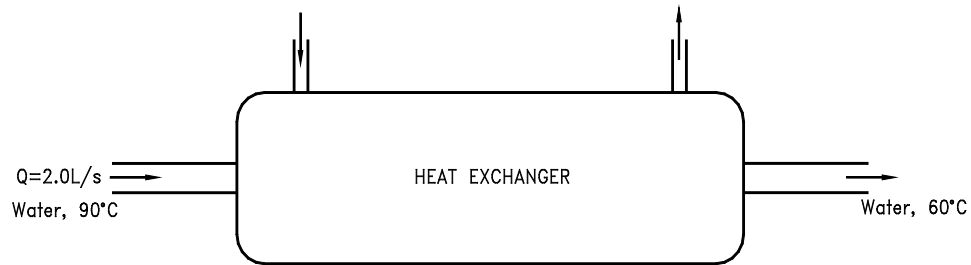
In some problems, friction may be neglected and if there is no shaft work done by or on the fluid, then equation 4.7 reduces to

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 \quad (4.8)$$

Equation 4.8 is known as **Bernoulli's equation**. In this text we will normally use the more general equation 4.7.

**Example 4.3**

Figure 4.6 shows a heat exchanger. Water at 90°C enters the heat exchanger with a flow rate of 2 L/s through a 100 mm diameter pipe. The water leaves the heat exchanger at a temperature of 60°C through a similar pipe. Determine the rate at which heat has been removed from the water, assuming that friction may be neglected and that the pressure remains constant.



**Figure 4.6**

**Solution:**

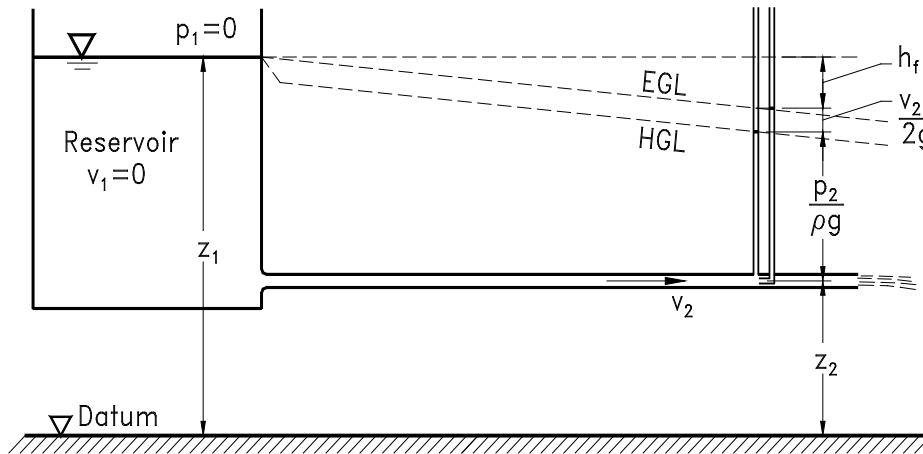
At an average temperature of 75°C, the specific heat of water, by interpolation from Table 4.2, is 4193 Joules/kg/°C, and the mass density from table 2.1 is 975 kg/m<sup>3</sup>. The heat transfer rate is:

$$\frac{4193 \text{ Joules}}{\text{kg} \cdot ^\circ\text{C}} \cdot \frac{975 \text{ kg}}{\text{m}^3} \cdot \frac{0.002 \text{ m}^3}{\text{s}} \cdot (90^\circ\text{C} - 60^\circ\text{C}) = \underline{\underline{245.3 \text{ kJ/s}}}$$



### 4.5 Hydraulic and Energy Grade Lines

As a further clarification of the energy head in a pipeline, consider the diagram in Figure 4.7.



**Figure 4.7 Energy and Hydraulic grade lines**

In the reservoir, the entire energy head consists of potential energy. As the liquid flows through the pipe, energy loss due to friction reduces the total head by an amount  $h_f$  that is proportional to the distance traveled in the pipe. A plot of the total available energy head is called the **energy grade line** (EGL). The EGL will have sudden drops at locations where minor losses occur, such as at fittings. As the water enters the pipe, it accelerates. A portion of the energy head in the pipe consists of kinetic energy, represented by  $v^2/2g$  (neglecting the energy correction factor). A line can be plotted which represents the subtraction of the kinetic energy from the EGL. This line, known as the **hydraulic grade line** (HGL), represents the sum of the potential and pressure energy.

### 4.6 Friction Head Losses

Friction head losses occur throughout the length of a pipe due to viscous drag. For turbulent flow, the head loss to friction,  $h_f$ , cannot be explicitly determined by direct application of mathematical and physical theory. The most popular formula based on experiment was derived by the French Engineer Henri Darcy (1803 – 1858) and the German Professor Julius Weisbach (1806 – 1871) in about 1845. This formula is therefore known as the **Darcy – Weisbach formula**, and is given as

$$h_f = \frac{fL}{d} \cdot \frac{v^2}{2g} \tag{4.9}$$

- where  $f$  = Darcy-Weisbach friction factor
- $L$  = length of pipe
- $d$  = pipe diameter
- $v$  = fluid velocity
- $g$  = gravitational acceleration

To accurately determine the head loss due to friction, it is necessary to determine the Darcy-Weisbach friction factor,  $f$ .

For laminar pipe flow, the friction factor,  $f$ , can be determined by applying force equilibrium to a short section of pipe. The resulting friction factor is given as

$$f = \frac{64}{N_R} \quad (4.10)$$

For turbulent flow, the friction factor is not easily determined. Much work has been done on this subject with the most well known formula developed by Colebrook and White in 1939. The formula known as the Colebrook formula is as follows

$$\frac{1}{\sqrt{f}} = -2.0 \log \left( \frac{e/d}{3.7} + \frac{2.51}{N_R \cdot \sqrt{f}} \right) \quad (4.11)$$

where

- $e$  = pipe roughness
- $d$  = pipe diameter
- $N_R$  = Reynold's number
- $f$  = Darcy-Weisbach friction factor

Note that this is not an explicit formula and the friction factor,  $f$ , appears on both sides of the equation. In 1944, Moody plotted the results in a diagram known as the Moody diagram, shown in Figure 4.8. For head loss calculations, the Moody diagram can be used to obtain a direct answer, but for determining flow rates for a given geometry or determining pipe diameters for a given flow rate, the use of the diagram involves a time consuming iterative approach. With the advent of programmable calculators with 'solver' capabilities, the Colebrook equation can be used to obtain direct solutions of these problems.

In 1976, Jain developed a formula that can be solved for  $f$  explicitly:

$$\frac{1}{\sqrt{f}} = -2 \log \left( \frac{e}{3.7d} + \frac{5.72}{N_R^{0.9}} \right) \quad (4.12)$$

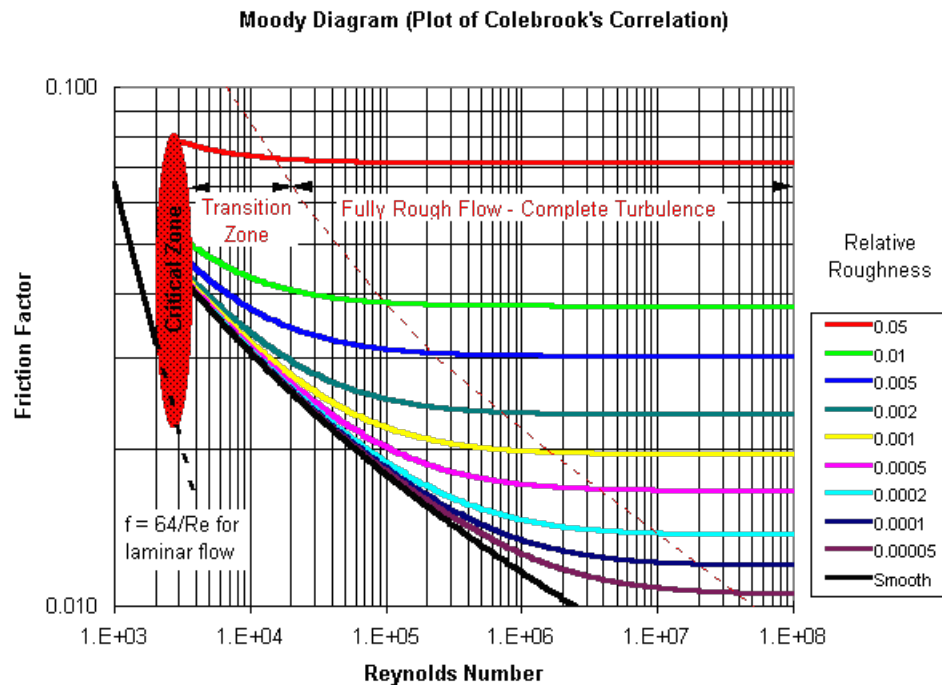
This formula gives results that are within 1% of the Colebrook equation.

The equations 4.11 and 4.12 are valid throughout the entire turbulent regime. Both equations indicate that the friction head losses depend upon the roughness of the pipe. Pipe roughness values are given in Table 4.3.

The 'PFLO' calculator program solves the energy equation for up to two pipes in series using the Darcy-Weisbach friction formula which is a built-in function on the HP 50g calculator. An example of the use of this program will be examined in the Section 4.7, which covers minor losses.

**Table 4.3 Pipe Roughness Values**

Pipe Material	e (mm)	e(ft)
Drawn Tubing, Plastic	0.0015	0.000005
Commercial Steel (enamel coated)	0.0048	0.000016
Commercial Steel (new)	0.045	0.00015
Cast Iron, Ductile Iron (new)	0.26	0.00085
Cast Iron (asphalt coated)	0.12	0.0004
Galvanized Iron	0.15	0.0005
Corrugated metal	45	0.15
Concrete (steel forms, smooth)	0.18	0.0006
Concrete (average)	0.36	0.0012
Concrete (rough, visible form marks)	0.60	0.002



**4.7 Hazen-Williams Formula for Friction Losses in Pipes**

The Hazen-Williams formula is an empirical formula, which was developed to determine losses due to friction for water flow in pipes. The formula gives reasonable results for pipe sizes greater than or equal to 50 mm in diameter and for velocities of less 3 m/s. This is typical of the ranges found in most municipal water supply systems, where the pipe sizes are usually 150 mm or larger and the velocities are usually restricted to a maximum of 2 m/s. Because the formula is empirical and is not dimensionally correct, there are two versions of the formula, one for metric and one for the fps or British system.

For British units, the formula is  $v = 1.318 C_{HW} R_H^{0.63} S^{0.54}$  (4.13)

and for metric units  $v = 0.85 C_{HW} R_H^{0.63} S^{0.54}$  (4.14)

where  $v$  = velocity (ft/s - equation 4.13, or m/s - equation 4.14)  
 $C_{HW}$  = Hazen-Williams coefficient - see Table 4.4  
 $R_H$  = hydraulic radius (ft - equation 4.13, or m - equation 4.14)  
 $S$  = slope of the energy gradient =  $\frac{h_f}{L}$

The hydraulic radius is defined as the ratio of the cross-sectional area of the flow to the wetted perimeter. For a round pipe running full, the hydraulic radius is given by

$$R_H = \frac{A}{P} = \frac{\pi d^2/4}{\pi d} = \frac{d}{4}$$
 (4.15)

**Table 4.4 Hazen-Williams Coefficient**

Pipe Material	Hazen-Williams Coefficient
Drawn Tubing, Plastic	150
Commercial Steel (enamel coated)	145-150
Commercial Steel (new)	120
Cast Iron, Ductile Iron (new)	130
Cast Iron (asphalt coated)	125
Galvanized Iron	120
Corrugated metal	65
Concrete (steel forms, smooth)	140
Concrete (average)	130
Concrete (rough, visible form marks)	120

It should be noted that the Hazen-Williams formula is not as accurate as the Darcy-Weisbach formula and only gives approximate results. The Hazen-Williams coefficient should, in reality, vary with the flow velocity, the viscosity and the pipe diameter. Jain et al showed that in some situations, the velocity calculated using the Hazen-Williams formula could be out by as much as 39%. Despite this, it is in common use, particularly for the design of municipal water systems, because of its simplicity.

**Example 4.4**

Water flows at a rate of 34 L/s through a horizontal  $\phi 150$  ductile iron pipe. Determine the head loss in 400 m of pipe using the Hazen-Williams formula.

**Solution:**

Continuity:  $v = \frac{Q}{A} = \frac{0.034 m^3/s}{\pi(0.15 m)^2/4} = 1.924 m/s$

Hazen-Williams:

$$S = \left( \frac{v}{0.85 C_{HW} R_H^{0.63}} \right)^{\frac{1}{0.54}} = \left( \frac{1.924 \text{ m/s}}{0.85 (130) (0.150 \text{ m} / 4)^{0.63}} \right)^{\frac{1}{0.54}} = 0.02546$$

$$h_f = S \cdot L = (0.02546) \cdot (400 \text{ m}) = \underline{\underline{10.19 \text{ m}}}$$

#### 4.8 Minor Losses

In addition to the friction head losses that occur throughout the length of the pipe, losses occur due to turbulence at elbows, bends, tees, valve fittings etc. as well as at entrances and exits and where contraction or expansion takes place. These losses are called **minor losses**, however, in short runs with many fixtures, they can be very significant.

In general minor losses are dependent upon the square of the velocity. One approach to including minor losses involves assigning a K factor to each minor loss. The losses are expressed in terms of  $\frac{v^2}{2g}$ . The total loss in the system is then given as

$$h_f = \left( \frac{fL}{d} + \Sigma K \right) \frac{v^2}{2g} \quad (4.16)$$

where  $\Sigma K$  represents the sum of the minor loss K factors. A summary of common K factors is given in table 4.5, but where possible K factors should be obtained from valve and fitting manufacturer's data as the tabulated values are approximate.

Another approach is to use what is known as the equivalent length method. With this method, an equivalent length of straight pipe is added to the length used in the Darcy-Weisbach equation. The equivalent length may be obtained from manufacturer's data or may be determined from a known K factor. The equivalent length,  $L_e$ , in terms of K is

$$L_e = \frac{K \cdot d}{f} \quad (4.17)$$

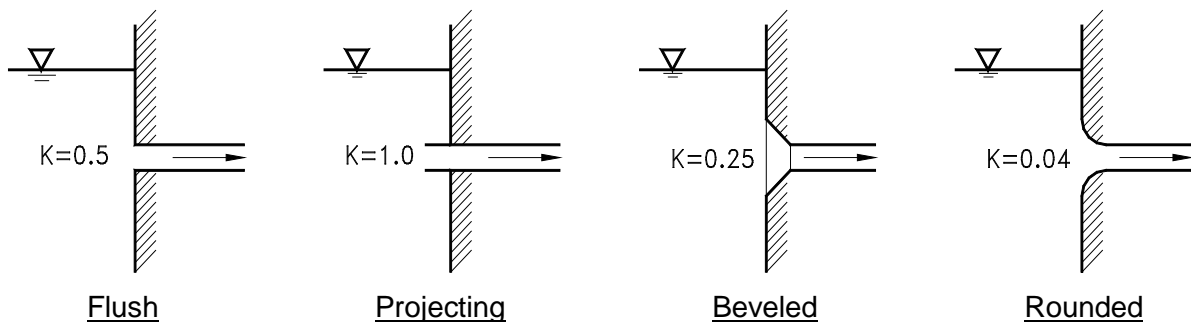
Note that equation 4.17 implies that the equivalent length depends upon the friction factor,  $f$ , which in turn implies that it is dependent upon the velocity and viscosity as well as the pipe roughness. Equivalent lengths from manufacturers are usually constant for a given diameter, and therefore results obtained using this method are approximate in nature. They are usually valid for a particular fluid and may reflect a flow velocity that is reasonable for a design situation where the fitting is likely to be used. In this text, we shall use the minor loss coefficient or K factor method.

**Table 4.5 Loss Coefficients for Valves and Fittings**

Item	K
Standard 45° Elbow	0.35
Standard 90° Elbow	0.75
Long Radius 90° Elbow	0.45
Coupling	0.04
Union	0.04
Gate Valve Open	0.2
¾ Open	0.9
½ Open	4.5
¼ Open	24.0
Globe Valve Open	6.4
½ Open	9.5
Tee (along run, line flow)	0.4
Tee (branch flow)	1.5
Entrance loss (flush end)	0.5
Entrance loss (projecting end)	1.0
Exit loss (pipe to tank)	1.0
Sudden Contraction (d/D = 0.50)	0.32
"    "    (d/D = 0.25)	0.39
"    "    (d/D = 0.10)	0.46
Sudden Expansion (d/D = 0.50)	0.56
"    "    (d/D = 0.25)	0.88
"    "    (d/D = 0.10)	0.99

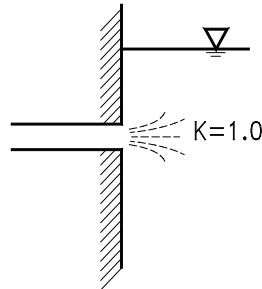
**Entrance and Exit Losses:**

When a fluid flows from a reservoir into a pipe, the losses at the entrance to the pipe depend upon the shape of the entrance. Figure 4.9 shows the entrance loss coefficients for various entrance shapes for pipe flow.



**Figure 4.9 Entrance Loss Coefficients**

When a pipe discharges to reservoir with a submerged outlet as shown in Figure 4.10, the kinetic energy of the flow is lost. Therefore,  $\frac{v^2}{2g}$  of energy head is lost and the exit loss coefficient is 1.0.



**Figure 4.10 Exit Loss Coefficient**

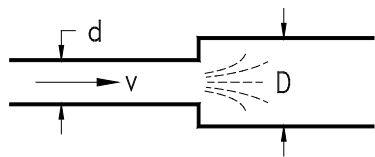
**Sudden Expansion and Contractions:**

The loss coefficient for a sudden expansion with turbulent flow in a pipe, as shown in Figure 4.11a, is given as

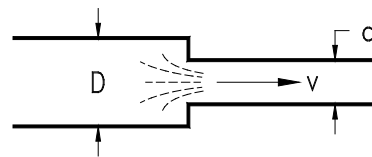
$$K_{SE} = \left[ 1 - \left( \frac{d}{D} \right)^2 \right]^2 \tag{4.18}$$

where d is the diameter of the smaller pipe and D is the diameter of the larger pipe.

This formula is based on theory. Experimental evidence shows that it is reasonably valid.



**Figure 4.11a Sudden Enlargement**



**Figure 4.11b Sudden Contraction**

For a sudden contraction, as shown in Figure 4.11b, the theory is not as well developed and a formula based on experiment is used. This empirical formula, approximate in nature, is given as

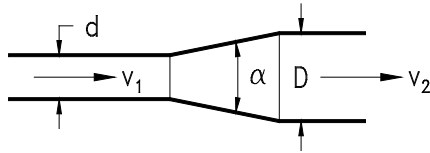
$$K_{SC} \cong 0.42 \left[ 1 - \left( \frac{d}{D} \right)^2 \right] \tag{4.19}$$

This formula gives reasonable results up to  $d/D = 0.76$ , after which equation 4.18 should be used.

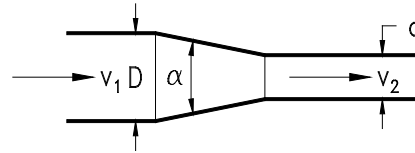
The head losses are  $h_f = K_{SE} \cdot \frac{v^2}{2g}$  and  $h_f = K_{SC} \cdot \frac{v^2}{2g}$  for a sudden expansion or for a sudden contraction, respectively. The velocity in both cases is the velocity in the smaller diameter pipe.

**Diffusers and Confusers:**

The head losses due to expansion and contractions can greatly reduced by providing a gently sloping conical transition from the one diameter to the other. If the conical section provides and expansion, it is known as a **diffuser** (see Figure 4.12a. If it provides a contraction, it is known as a **confuser** (see Figure4.12b).



**Figure 4.12a Conical Diffuser**



**Figure 4.12b Conical Confuser**

The head losses in diffusers and confusers depends upon the total included angle,  $\alpha$ , and on the ratio  $d/D$ . The losses are also somewhat dependent upon how well developed the flow is at the inlet to the fitting.

Table 4.6 gives the loss coefficients for a diffuser at two different angles and various  $d/D$  ratios. The loss coefficients in terms of the **entrance** velocity,  $v_1$ .

**Table 4.6 Head Loss Coefficients for a Conical Diffuser**

$d/D$	$K_d$ $\alpha = 10^\circ$	$K_d$ $\alpha = 180^\circ$
0.0		1.00
0.2	0.13	0.92
0.4	0.11	0.72
0.6	0.06	0.42
0.8	0.03	0.16

Table 4.7 gives the loss coefficients for a confuser at two different angles and various  $d/D$  ratios. The loss coefficients are in terms of the **exit** velocity,  $v_2$ .

**Table 4.7 Head Loss Coefficients for a Conical Confuser**

$d/D$	$K_c$ $\alpha = 60^\circ$	$K_c$ $\alpha = 180^\circ$
0.0	0.08	0.50
0.2	0.08	0.49
0.4	0.07	0.42
0.6	0.06	0.32
0.8	0.05	0.18
0.9	0.04	0.10

**Example 4.5**

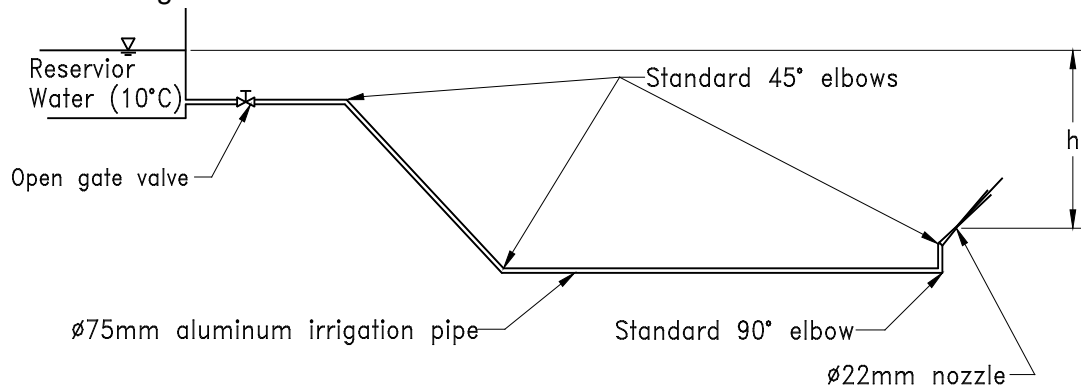
Figure 4.13 shows a reservoir that is to deliver water to a sprinkler head at a lower elevation. The  $\phi 22$  nozzle has a  $k$  factor of 1.2, based on the smaller diameter. The



water is to discharge from the nozzle with a velocity of 28m/s. The 1400m long aluminum pipe has an  $e = 0.00004\text{m}$ , and includes an open gate valve.

Using the HP 50g calculator program 'PFLO', determine:

- a.) The volume flow rate for this discharge.
- b.) The height  $h$  at which the water in the reservoir must be kept in order to get this discharge rate.



**Figure 4.13**

**Solution:**

- a.) From continuity, the required flow rate is

$$Q = v \cdot A = (28 \text{ m/s}) \cdot \frac{\pi (0.022 \text{ m})^2}{4} = 0.01064 \text{ m}^3 / \text{s} , \text{ or } \underline{\underline{10.64 \text{ L/s}}}$$

- b.) With the 'PFLO' program, the exit loss must be added to the sum of the K factors. Since the nozzle is a different diameter than the pipe, the pipeline will have to be modeled as two pipes in series. The loss in the nozzle is taken care of, so the length of  $\phi 22 \text{ mm}$  pipe can be taken as zero. The  $\Sigma K_2$  is then determined as follows

$$\begin{aligned} K_{\text{nozzle}} &= 1.2 \\ K_{\text{exit}} &= 1.0 \text{ to account for the exit loss } (v_2 \text{ present}). \\ \hline \Sigma K_2 &= 2.2 \end{aligned}$$

Summing up the minor loss coefficients in the pipe gives

$$\begin{aligned} K_{\text{entrance}} &= 0.50 \\ K_{\text{valve}} &= 0.20 \\ 3 \times K_{45} &= 1.05 \\ K_{90} &= 0.73 \\ \hline \Sigma K_1 &= 2.5 \end{aligned}$$

The input to the 'PFLO' program is summarized below:

$p_1 = 0$	$p_2 = 0$	$\rho = 999 \text{ kg/m}^3$	$g = 9.81 \text{ m/s}^2$
$z_1 = ?$	$z_2 = 0$	$e_1 = 0.00004 \text{ m}$	$d_1 = 0.075 \text{ m}$
$Q = 0.01064 \text{ m}^3/\text{s}$		$\nu = 1.306 \times 10^{-6} \text{ m}^2/\text{s}$	
$L_1 = 1400 \text{ m}$	$\Sigma K_1 = 2.5$	$e_2 = 0.00004 \text{ m}$	$d_2 = 0.022 \text{ m}$
$L_2 = 0$	$\Sigma K_2 = 2.2$	$w' = 0$	

Solving for  $h = z_1$  gives  $h = \underline{197.3 \text{ m}}$ .

Try other ways:

### 4.9 Pipelines and Pipe Networks

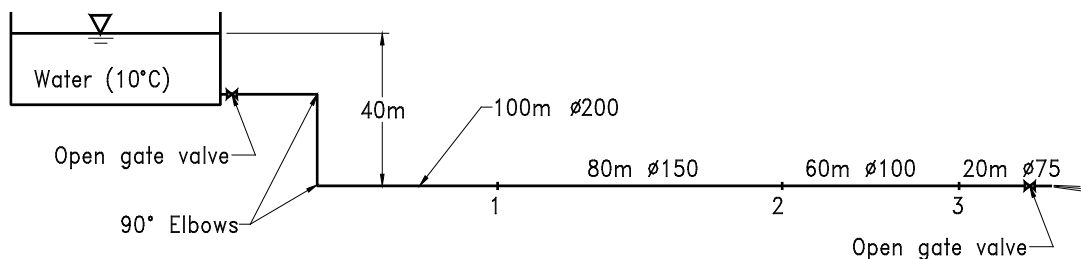
#### Pipelines:

If all the elements of a pipe system are connected in series, the system is known as a **pipeline**. The 'PFLO' program is designed to solve pipeline problems with up to two different diameters of pipe. As shown in the previous example, however, a nozzle with a given diameter must be treated like a pipe of different diameter to properly account for the exit loss.

The 'PFLO' program can be used to solve pipeline problems involving up to four different diameters of pipes on an iterative basis.

#### Example 4.6

Figure 4.14 shows a pipeline feeding from a reservoir. The pipe consists of four different diameters, all commercial steel pipe. Reducers are located at points 1, 2, and 3. Assume a K factor of 0.1 for each reducer. Determine the flow rate through the pipeline.



**Figure 4.14**

**Solution:**

The 'PFLO' program will be run in twice in each iteration; from the reservoir to the reducer at point 2, and from the reducer at point 2 to the outlet. An initial guess of 20 L/s will be used for the flow rate, and we will solve for the pressure,  $p_2$ , at point 2 in the first run. This pressure will then be used as  $p_1$  in the second run to solve for the flow rate,  $Q$ . The next iteration will use this flow rate in the first run. Usually a reasonable solution will be found in three iterations.

For the first run (reservoir to point 2), the minor loss coefficients are:

$K_{\text{entrance}} = 0.50$	$K_{\text{reducer}} = 0.10$
$K_{\text{valve}} = 0.20$	$v_2 \text{ present} = 1.00$
$2 \times K_{90} = 1.50$	
$\Sigma K_1 = 2.00$	$\Sigma K_2 = 1.10$

For the second run (point 2 to the exit), the minor loss coefficients are:

$V_1 \text{ present} = -1.00$	$K_{\text{reducer}} = 0.10$
$K_{\text{reducer}} = 0.10$	$K_{\text{valve}} = 0.20$
	$v_2 \text{ present} = 1.00$
$\Sigma K_1 = -0.90$	$\Sigma K_2 = 1.30$

For the first run of the first iteration, the input data is

$p_1 = 0$	$p_2 = ?$	$\rho = 999 \text{ kg/m}^3$	$g = 9.81 \text{ m/s}^2$
$z_1 = 49\text{m}$	$z_2 = 0$	$e_1 = 0.000045\text{m}$	$d_1 = 0.200 \text{ m}$
$Q = 0.020 \text{ m}^3/\text{s}$		$v = 1.306 \times 10^{-6} \text{ m}^2/\text{s}$	
$L_1 = 100\text{m}$	$\Sigma K_1 = 2.2$	$e_2 = 0.000045 \text{ m}$	$d_2 = 0.015 \text{ m}$
$L_2 = 80 \text{ m}$	$\Sigma K_2 = 1.1$	$w' = 0$	

Solving for  $p_2$  gives  $p_2 = 382523 \text{ Pa}$ . Using this pressure as  $p_1$  in the second run of the first iteration, the data for this run is:

$p_1 = 382523 \text{ Pa}$	$p_2 = 0$	$\rho = 999 \text{ kg/m}^3$	$g = 9.81 \text{ m/s}^2$
$z_1 = 0$	$z_2 = 0$	$e_1 = 0.000045 \text{ m}$	$d_1 = 0.1\text{m}$
$Q = ?$		$v = 1.306 \times 10^{-6} \text{ m}^2/\text{s}$	
$L_1 = 60 \text{ m}$	$\Sigma K_1 = -0.9$	$e_2 = 0.000045 \text{ m}$	$d_2 = 0.075 \text{ m}$

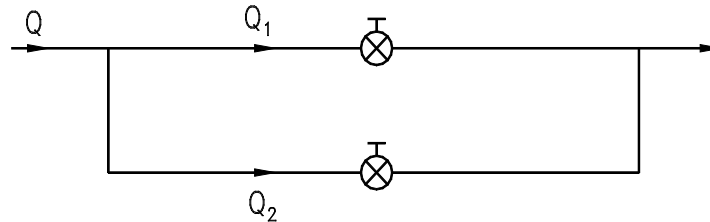
$$L_2 = 20 \text{ m} \quad \Sigma K_2 = 1.2 \quad w' = 0$$

Solving for Q gives  $Q = 0.0403 \text{ m}^3/\text{s}$ .

The second iteration starts the first run with this value for Q and yields a new  $Q = 0.03885 \text{ m}^3/\text{s}$ . A third iteration starting with a Q of  $0.03885 \text{ m}^3/\text{s}$  yields a new  $Q = 0.0390 \text{ m}^3/\text{s}$ . The change from the previous iteration is negligible so the final answer can be given as  $Q = 0.0390 \text{ m}^3/\text{s}$  or 39.0 L/s.

**Pipe Networks:**

A system in which the elements are not in series is known as a pipe network. One of the simplest forms of pipe networks consists of a single branch flow as shown in Figure 4.15.



**Figure 4.15**

Usually with such a system, the total flow rate, Q, is known and it is desired to determine the flow rate in each branch,  $Q_1$  and  $Q_2$ . The proportion of the flow that enters a given branch is dependent upon the frictional resistance in that branch. The solution is based on two principles that can be applied to larger, more complicated networks.

The first principle is that continuity or conservation of mass can be applied to each node. In other words, the sum of the flows entering a node must equal the sum of the flows leaving the node. In Figure 4.15, this principle yields the following equation:

$$Q = Q_1 + Q_2 \tag{4.20}$$

The second principle states that the energy head at any given point can only have a single unique value. This principle implies that the head loss to friction through branch 1 must be the same as the head loss through branch 2, or

$$h_{f1} = h_{f2} \tag{4.21}$$

In more complicated networks, a direction of flow is assumed for each line. This principle is then applied by stating that the sum of the head losses around any loop must be zero. Head losses are summed in one direction, usually counter-clockwise. For example, when moving counter-clockwise around the single loop in the network shown in Figure 4.15,  $h_{f2}$  would be positive because the direction of  $Q_2$  matches the counter-clockwise loop direction. On the other hand,  $h_{f1}$  would be taken as negative since  $Q_1$  opposes the counter-clockwise loop direction. In effect equation 4.21 becomes

$$h_{f2} - h_{f1} = 0 \tag{4.22}$$

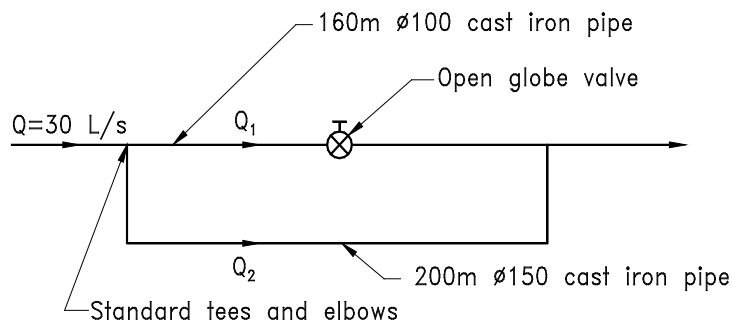
Equation 4.22 is seen to be identical to equation 4.21.

The two equations 4.20 and 4.22 (or 4.21) can be solved for the two unknown flow rates. The HP 50g calculator program 'PPIPE' solves equation 4.22 for the simple case of a single branch flow. The program only solves for  $Q_1$ .  $Q_2$  is then obtained by applying equation 4.20:

$$Q_2 = Q - Q_1 \tag{4.23}$$

**Example 4.7**

Figure 4.16 shows a branched pipe system. The total flow rate entering the system is 30 L/s. Use the 'PPIPE' program to determine the flow rates  $Q_1$  and  $Q_2$  in each branch of the system. The fluid is water at 10°C.



**Figure 4.16**

**Solution:**

The minor loss coefficients for branch 1 are:

$$\begin{array}{rcl} 2 \times K_{tee (line)} & = & 0.80 \\ K_{valve} & = & 6.40 \\ \hline \Sigma K_1 & = & 7.20 \end{array}$$

The minor loss coefficients for branch 2 are:

$$\begin{array}{rcl} 2 \times K_{tee (branch)} & = & 3.00 \\ 2 \times K_{90^\circ \text{ Elbow}} & = & 1.50 \\ \hline \Sigma K_2 & = & 4.50 \end{array}$$

The input to the 'PPIPE' program is summarized below:

$e_1 = 0.00026 \text{ m}$	$d_1 = 0.100 \text{ m}$	$Q_1 = ? \text{ (guess } 0.010 \text{ m}^3/\text{s)}$
$\nu = 1.306 \times 10^{-6} \text{ m}^2/\text{s}$	$L_1 = 160 \text{ m}$	$\Sigma K_1 = 7.2$
$e_2 = 0.00026 \text{ m}$	$d_2 = 0.15 \text{ m}$	$Q = 0.030 \text{ m}^3/\text{s}$

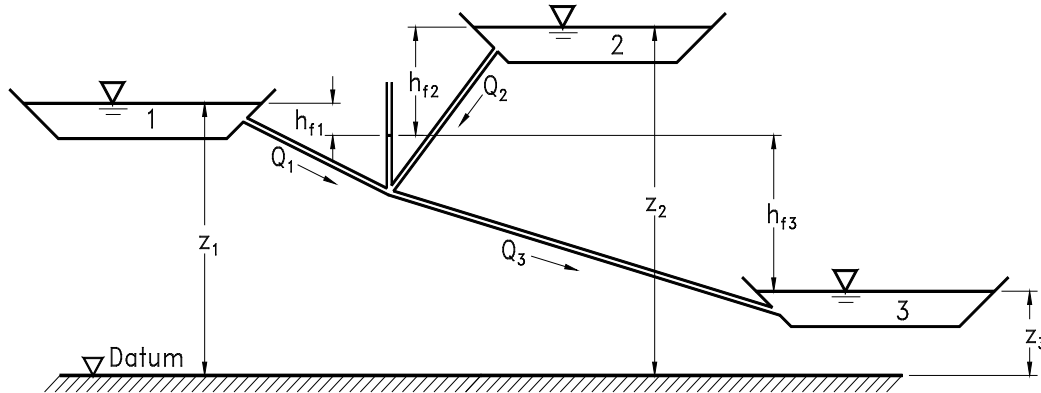
$$L_2 = 200\text{m}$$

$$\Sigma K_2 = 4.5$$

Solving for  $Q_1$  gives  $Q_1 = 0.008232 \text{ m}^3/\text{s}$ , or 8.23 L/s.

$$Q_2 = Q - Q_1 = 30 \text{ L/s} - 8.23 \text{ L/s} = \underline{21.77 \text{ L/s}}.$$

Branch systems or networks do not have to be closed systems. Figure 4.17 shows a branched system involving three reservoirs.



**Figure 4.17 Branched System with Reservoirs**

In this case the node equation is  $Q_1 + Q_2 = Q_3$  and the other equations involve the elevations of the free surfaces together with the head loss in each pipe section. Note that there can only be one unique head at the node. The equations become  $z_1 - h_{f1} = z_2 - h_{f2}$ , and  $z_2 - h_{f2} = z_3 + h_{f3}$ .

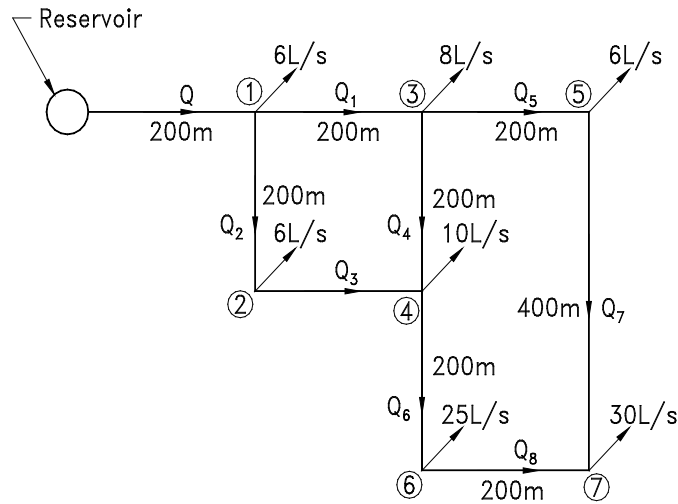
Water distribution systems usually involve networks that are much more complicated. Figure 4.18 shows a relatively simple water distribution system. In addition to the information shown, the elevation of each node would have to be known. Such complicated network problems are usually solved using a computer network analysis program. Many of these programs even allow the user to input the system with a graphical interface or a CAD program.

The shaft work,  $w'$ , discussed in Section 4.4 is measured in unit time. If a fluid is pumped by motors to the reservoir at higher elevation, the power of the motors is determined as:

$$P = \rho Q w'$$

- Where  $\rho$  = density of the fluid ( $\text{kg}/\text{m}^3$  or  $\text{lb}/\text{ft}^3$ );
- $Q$  = volume flow rate ( $\text{m}^3/\text{s}$  or  $\text{ft}^3/\text{s}$ );
- $w'$  = shaft work ( $\text{m}^2/\text{s}^2$  or  $\text{ft}^2/\text{s}^2$ ).

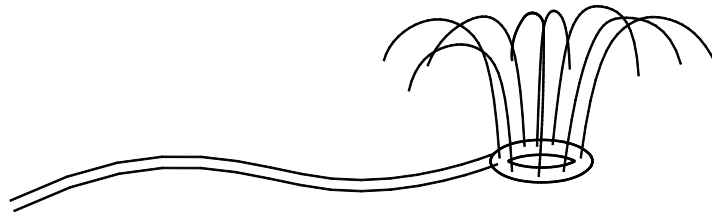
The SI unit for the power is Watt (W), which is equivalent to  $\text{kg}\cdot\text{m}^2/\text{s}^3$ . The US unit for the power is  $\text{ft}\cdot\text{lb}/\text{s}$ . A traditional common unit is horsepower (HP).  $1\text{HP} = 550 \text{ ft}\cdot\text{lb}/\text{s}$ ;  $1\text{HP} = 746 \text{ Watts}$ .



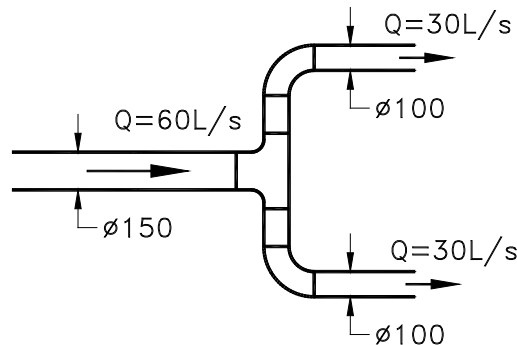
**Figure 4.18 Water Distribution Network**

### Problems – Chapter 4

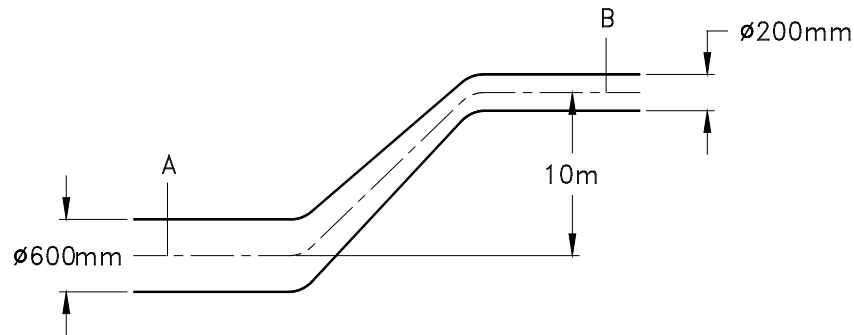
4.1) A garden hose has an inside diameter of 12mm and is connected to a lawn sprinkler that consists of an enclosure with 25 holes. Each hole has a diameter of 0.8 mm. The water in the hose moves at 2.0 m/s. Determine the velocity of the water as it leaves the sprinkler.



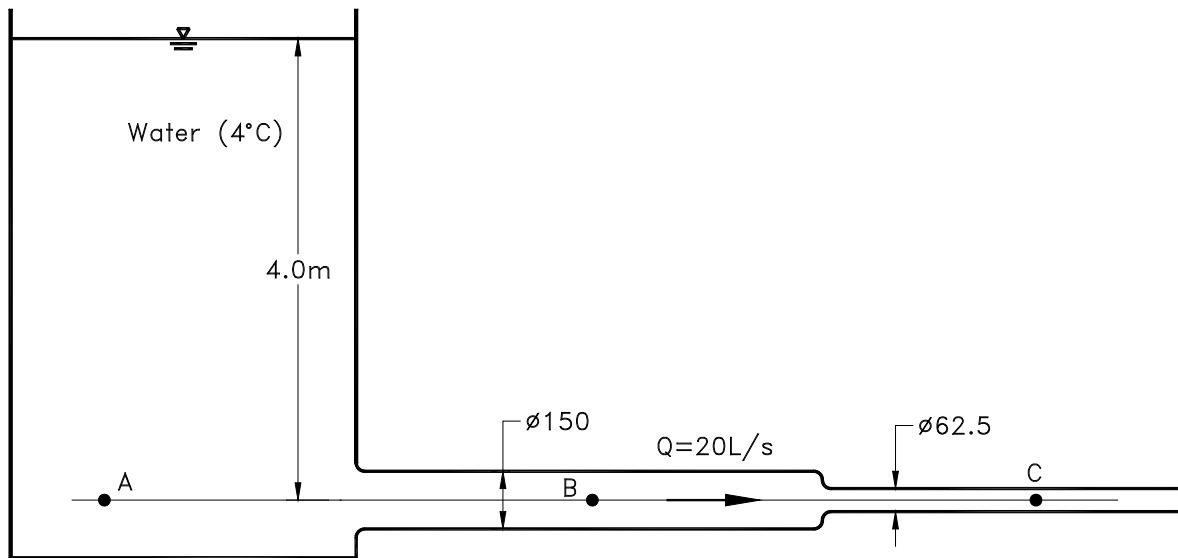
4.2) A  $\phi 150$  pipe, with a flow rate of 60L/s, branches with a reducing tee to two  $\phi 100$  pipes. Each branch pipe carries half the initial flow rate. Determine the velocity in both the initial  $\phi 150$  pipe and in each branch pipe.



- 4.3) A pipe carrying water tapers from a diameter of 600 mm at 'A' to a diameter of 200 mm at 'B'. At 'A' the mean velocity is 0.6 m/s and the gauge pressure is 450 kPa. Assuming friction effects are negligible, determine the pressure at 'B' which is 10.0 m above 'A'.



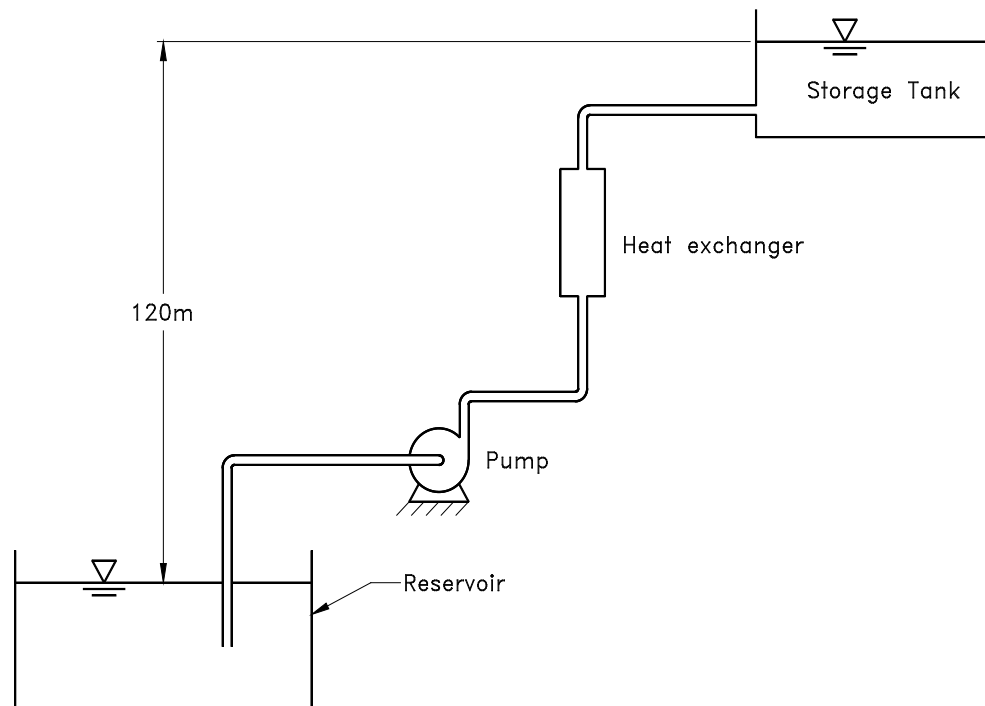
- 4.4) Water (4° C) flows from a reservoir at rate of 20L/s. Neglecting friction, determine the gauge pressure at:
- point A, 4 m below the water surface in the reservoir;
  - point B, inside the  $\phi 150$  pipe leading from the reservoir at a depth of 4.0 m;
  - point C, inside the  $\phi 62.5$  pipe after the reducer.



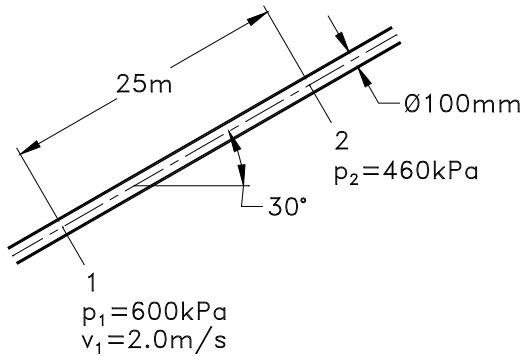
- 4.5) Calculate the Reynold's number for a fluid with a density of 750 kg/m<sup>3</sup> and a viscosity of 0.050 Pa-s flowing through a  $\phi 100$  mm pipe with a flow rate of 22 L/s. Estimate the mean *critical velocity* at turbulent flow.



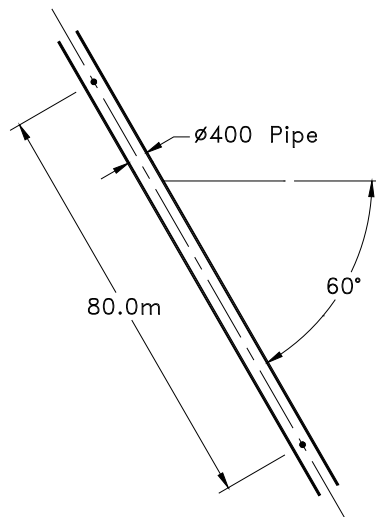
- 4.6) Municipal water supply lines are typically designed for a maximum flow velocity of 2 m/s. Determine the Reynolds Number for water (4° C) flowing at this velocity in a  $\phi 200$  water main. Is the flow laminar or turbulent?
- 4.7) A liquid, with a kinematic viscosity of  $450 \text{ mm}^2/\text{s}$ , flows through a  $\phi 75 \text{ mm}$  pipe at a rate of 8.0 L/s. What type of flow is to be expected?
- 4.8) What is the maximum diameter of pipe required to ensure turbulent flow for water (4°C) at a flow rate of 5L/s? What is the minimum diameter of pipe that will ensure laminar flow at the same rate?
- 4.9) Water, from a reservoir at 10°C, is pumped through a heat exchanger at a rate of 40 L/s.
- Neglecting friction, what is the power requirement of the pump?
  - If the water enters the storage tank at a temperature of 20°C, at what rate has the heat exchanger passed heat to the water?



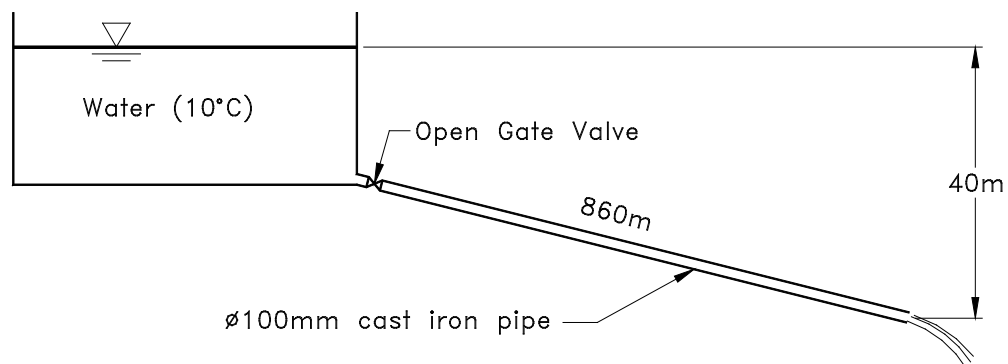
- 4.10) Water (4°C) enters an axial flow turbine at a velocity of 8 m/s and a gauge pressure of 1600 kPa. The water leaves the turbine at a velocity of 2 m/s and at atmospheric pressure. The turbine discharge is located 3 m below its intake. Neglecting friction losses, determine the power output for the turbine if the volume flow rate is  $1.8 \text{ m}^3/\text{s}$ .
- 4.11) Water at 20°C flows with a steady mean velocity of 2.0 m/s through a  $\phi 100\text{mm}$  pipe that slopes upward at 30° to the horizontal. At point 1, the pressure is 600 kPa, and at point 2, located 25 m further along the pipe, the pressure is 460 kPa. Determine the head loss to friction, the friction factor  $f$ , and the relative roughness  $e/D$ .



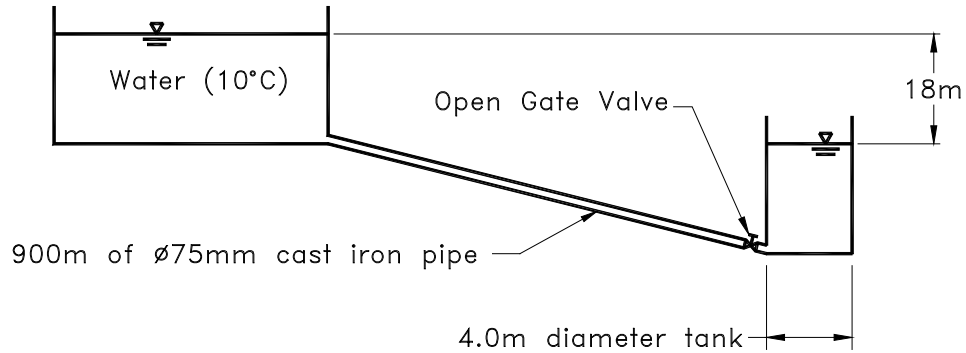
- 4.12) Water ( $4^\circ\text{C}$ ) flows with a steady mean velocity of  $4.0\text{ m/s}$  through a  $\phi 400$  penstock. The pipe slopes down at  $60^\circ$  to the horizontal. At point 1, the pressure is  $100\text{ kPa}$ , and at point 2 located  $80\text{ m}$  further along the pipe, the pressure is  $740\text{ kPa}$ . Determine the head loss to friction, the friction factor,  $f$  and the relative roughness  $e/d$ .



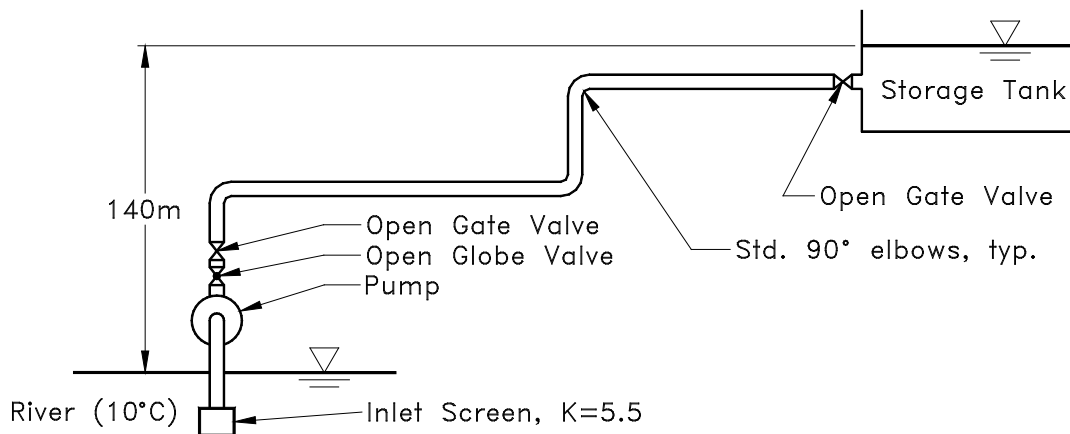
- 4.13) The diagram below shows a  $\phi 100$  cast iron pipe leading from a reservoir. Determine the flow rate through the pipe.



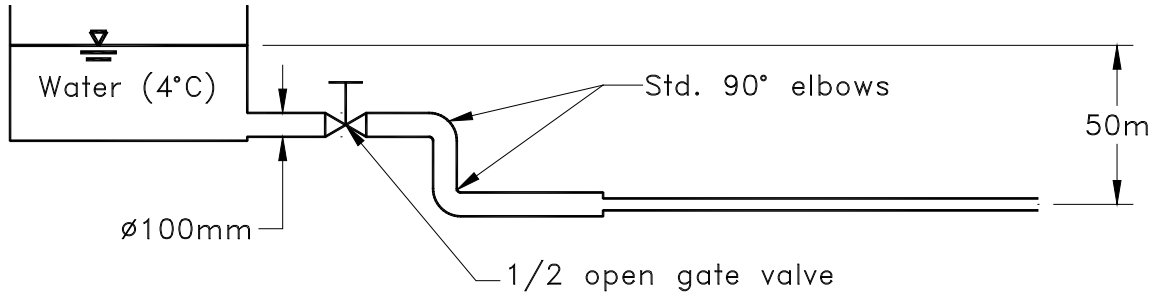
- 4.14) The diagram below shows a cast iron pipeline that feeds from a reservoir into to a  $4.0\text{ m}$  diameter tank. Determine the rate at which the water is rising in the tank when the level of water in the reservoir is  $18\text{ m}$  higher than the level in the tank.



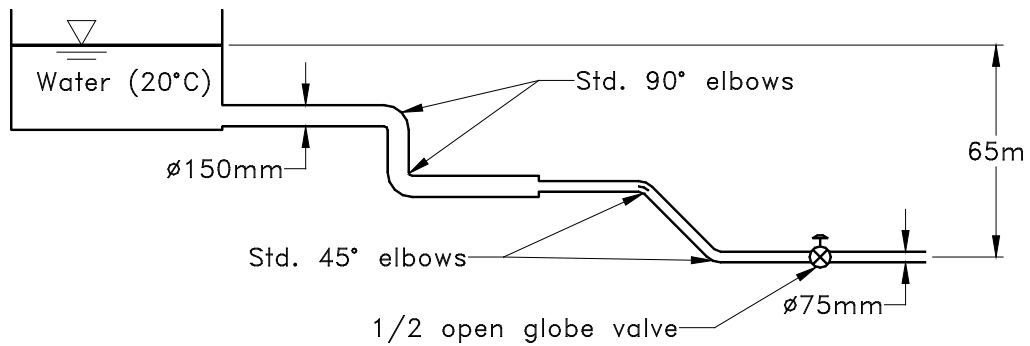
- 4.15) A new 1500 m long ductile iron pipe,  $e = 0.12$  mm,  $t$  is to have a discharge of 90 L/s of water at 10°C with a head loss not to exceed 12.0 m. Determine the required pipe diameter.
- 4.16) Sewage is to be pumped for 2300 m through a  $\phi 200$  horizontal plastic sewer pipe. What pressure must the pump develop if the flow rate is 100 L/s. The sewage can be treated as water at 10°C and the pipe discharges to the atmosphere.
- 4.17) A pump is used to lift 120 L/s of water at 10°C from a river to a storage tank that has a free surface 140 m above the level of the river surface. The pipeline is to be  $\phi 350$  plastic pipe, 1200 m long with a roughness  $e = 0.02$  mm. Determine the pump power requirement.



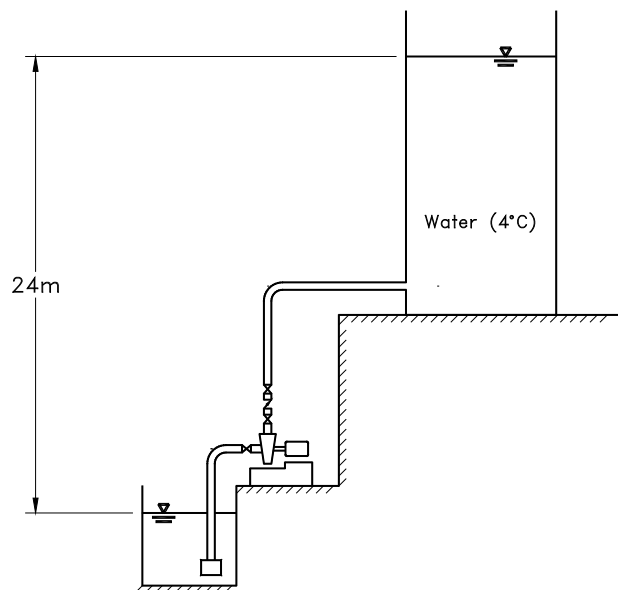
- 4.18) Water (4°C) flows from a reservoir through the pipe system shown. The 100 m long  $\phi 100$  mm pipe ends in a sudden contraction to a smaller diameter pipe that is 120 m long. Both pipes are commercial steel. Determine the smallest pipe size for the smaller pipe if a minimum desired flow rate of 20 L/s is to be met.



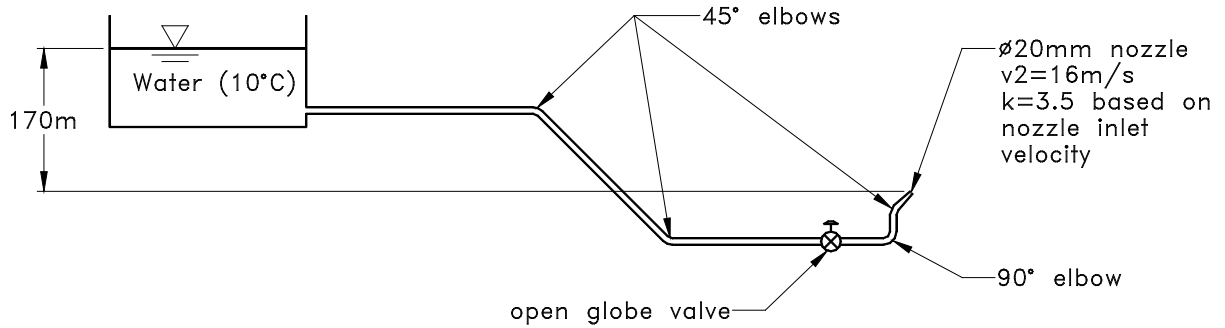
4.19) Water at 20°C flows from a reservoir through 200 m of  $\phi 150$  pipe and 80 m of  $\phi 75$  pipe as shown. The pipe is galvanized iron. Determine the flow rate.



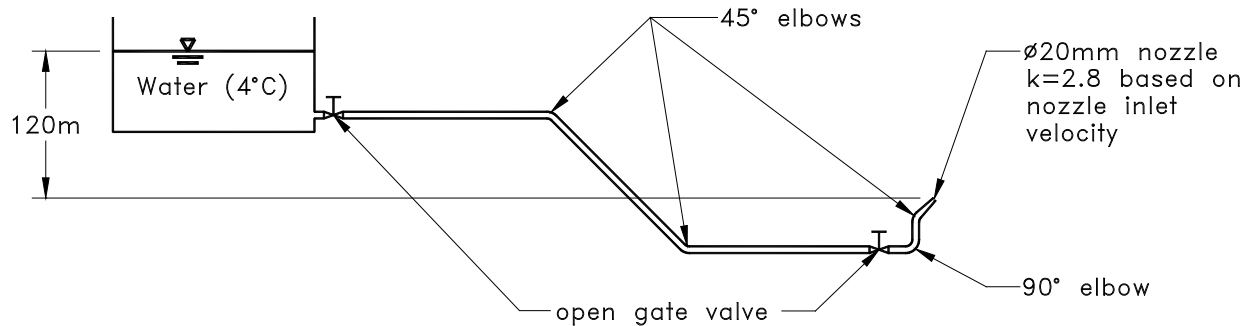
4.20) A pump is used to lift 66L/s of waste water at 4°C from a box culvert to a storage tank prior to treatment. The pipeline is galvanized steel. The  $\phi 100$  intake pipe is 8m long and has a screen,  $k = 5.0$ , a 90° elbow and an open gate valve. The pipe on the discharge side is also  $\phi 100$  and is 40 m long and has two 90° elbows, two open gate valves and a swing check valve with  $k = 4.6$ . Determine the electrical power requirement in kilowatts assuming the pump has an efficiency of 70% and the electric motor has an efficiency of 90%.



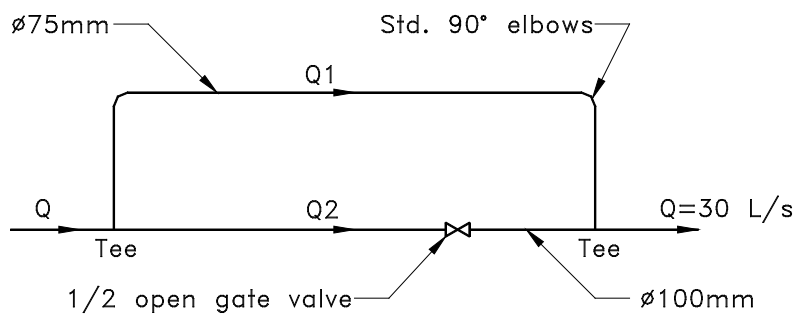
4.21) An irrigation system is to rely on gravity feed through 2300 m of aluminum pipe with a roughness  $e = 0.08$  mm as shown. Determine the required pipe diameter.



4.22) Determine the flow rate through the gravity feed irrigation system shown. The pipeline consists of  $1400\text{m}$  of  $\phi 100$  aluminum irrigation pipe with a roughness  $e = 0.06$  mm.



4.23) Water at  $10^\circ\text{C}$  flows through a branched pipe system as shown. Branch 1 is  $400\text{m}$  long and branch 2 is  $220\text{m}$  long. Determine the flow rates  $Q_1$  and  $Q_2$  in each branch. The pipe material is commercial steel pipe.



4.24) Water (80°C) flows through a branched galvanized steel pipe system as shown. Determine the flow rates  $Q_1$  and  $Q_2$  in each branch.

