University of Anbar
College of Engineering
Mechanical Engineering Dept.

Fluid Mechanics-II
(ME 2305)

Handout Lectures for Year Two
Chapter Five/ Flow over bodies: drag and lift

Course Tutor

Assist. Prof. Dr. Waleed M. Abed

Ramadi 2021

Chapter Five
Flow over bodies: drag and lift

5.1. Introduction

Fluid flow over solid bodies frequently occurs in practice, and it is responsible for numerous physical phenomena such as the drag force acting on automobiles, power lines, trees, and underwater pipelines; the lift developed by airplane wings; upward draft of rain, snow, hail, and dust particles in high winds; the transportation of red blood cells by blood flow; the entrainment and disbursement of liquid droplets by sprays; the vibration and noise generated by bodies moving in a fluid; and the power generated by wind turbines (Figure 5.1). Therefore, developing a good understanding of external flow is important in the design of many engineering systems such as *aircraft, automobiles, buildings, ships, submarines, and all kinds of turbines*. Late-model cars, for example, have been designed with particular emphasis on aerodynamics. This has resulted in significant reductions in fuel consumption and noise, and considerable improvement in handling.

Figure 5.1: Flow over bodies is commonly encountered in practice.

Sometimes a fluid moves over a stationary body (such as the wind blowing over a building), and other times a body moves through a quiescent fluid (such as a car moving through air). These two seemingly different processes are equivalent to each other; what matters is the relative motion between the fluid and the body. Such motions are conveniently analyzed by fixing the coordinate system on the body and are referred to as ***flow over bodies or external flow***. The aerodynamic aspects of different airplane wing designs, for example, are studied conveniently in a lab by placing the wings in a wind tunnel and blowing air over them by large fans. Also, a flow can be classified as being steady or unsteady, depending on the reference frame selected. Flow around an airplane, for example, is always unsteady with respect to the ground, but it is steady with respect to a frame of reference moving with the airplane at cruise conditions.

5.2. Drag and Lift Forces

Drag is usually an undesirable effect, like friction, and we do our best to minimize it. Reduction of drag is closely associated with the reduction of fuel consumption in automobiles, submarines, and aircraft; improved safety and durability of structures subjected to high winds; and reduction of noise and vibration. But in some cases drag produces a very beneficial effect and we try to maximize it. Friction, for example, is a "*life saver*" in the brakes of automobiles. Likewise, it is the drag that makes it possible for people to parachute, for pollens to fly to distant locations, and for us all to enjoy the waves of the oceans and the relaxing movements of the leaves of trees.

For two-dimensional flows, the resultant of the pressure and shear forces can be split into two components: one in the direction of flow, which is the drag force, and another in the direction normal to flow, which is the lift, as shown in Figure 5.2.

For three-dimensional flows, there is also a side force component in the direction normal to the page that tends to move the body in that direction.

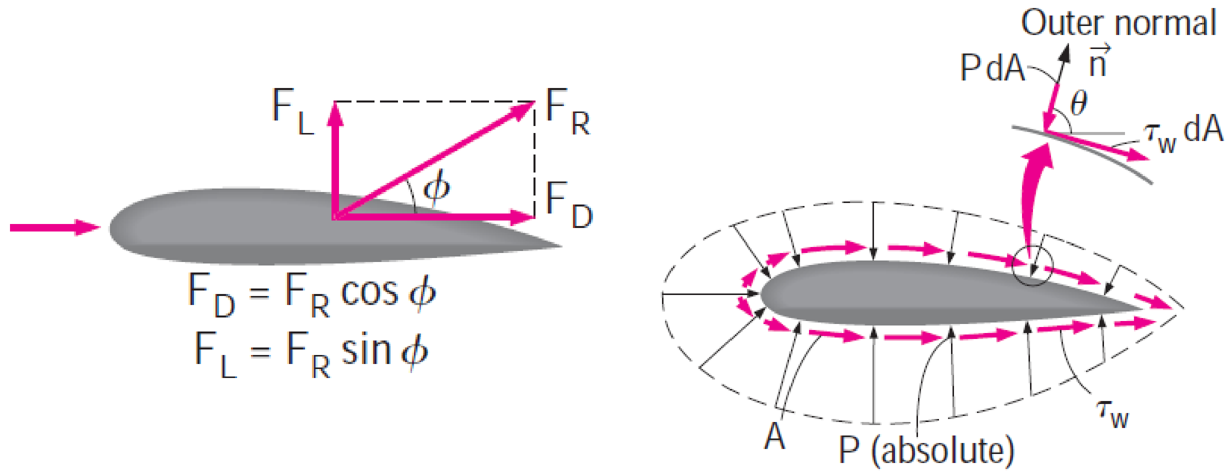


Figure 5.2: The pressure and viscous forces acting on a two-dimensional body and the resultant lift and drag forces.

The pressure and shear forces acting on a differential area *dA* on the surface are *PdA* and $\tau_w\ dA$, respectively. The differential drag force and the lift force acting on *dA* in two-dimensional flow are (Figure 5.2).

$$dF_D = -P\,dA\cos\theta + \tau_w\,dA\sin\theta \qquad (5.1)$$

$$dF_L = -P\,dA\sin\theta - \tau_w\,dA\cos\theta \qquad (5.2)$$

where θ is the angle that the outer normal of *dA* makes with the positive flow direction. The total drag and lift forces acting on the body are determined by integrating Eqs. 5.1 and 5.2 over the entire surface of the body,

Drag force: $$F_D = \int_A dF_D = \int_A (-P\cos\theta + \tau_w\sin\theta)\,dA \qquad (5.3)$$

Lift force: $$F_L = \int_A dF_L = -\int_A (P\sin\theta + \tau_w\cos\theta)\,dA \qquad (5.4)$$

The wings of airplanes are shaped and positioned specifically to generate lift with minimal drag. This is done by maintaining an angle of attack during cruising, as shown in Figure 5.3. Both lift and drag are strong functions of the angle of attack, as we discuss later in this chapter. The pressure difference between the top and bottom surfaces of the wing generates an upward force that tends to lift the wing and thus the airplane to which it is connected. For slender bodies such as wings, the shear force acts nearly parallel to the flow direction, and thus its contribution to the lift is small. The drag force for such slender bodies is mostly due to shear forces (the skin friction).

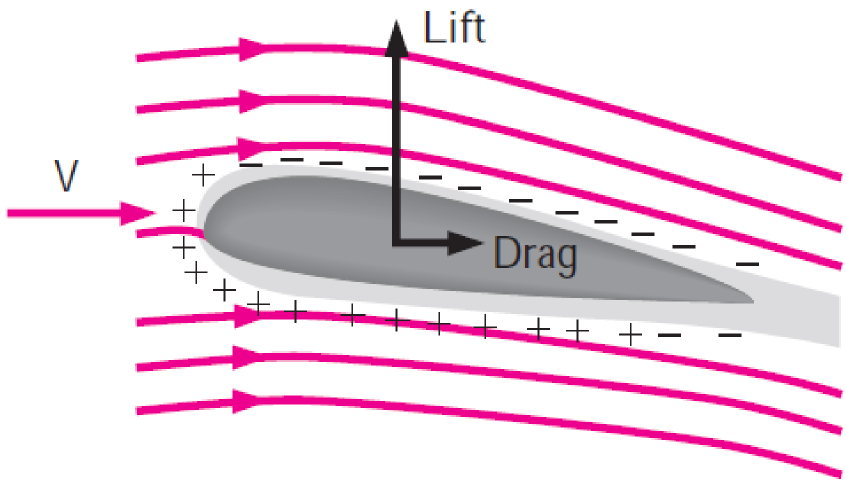


Figure 5.3: Airplane wings are shaped and positioned to generate sufficient lift during flight while keeping drag at a minimum. Pressures above and below atmospheric pressure are indicated by plus and minus signs, respectively.

The drag and lift forces depend on the density ρ of the fluid, the upstream velocity V, and the size, shape, and orientation of the body, among other things, and it is not practical to list these forces for a variety of situations. Instead, it is found convenient to work with appropriate dimensionless numbers that represent the drag and lift characteristics of the body. These numbers are the ***drag coefficient*** C_D, and the ***lift coefficient*** C_L, and they are defined as

Drag coefficient: $$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \qquad (5.5)$$

Lift coefficient: $$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} \qquad (5.6)$$

where *A* is ordinarily the ***frontal area*** (the area projected on a plane normal to the direction of flow) of the body. In other words, *A* is the area that would be seen by a person looking at the body from the direction of the approaching fluid. The frontal area of a cylinder of diameter *D* and length *L*, for example, is $A = LD$. In lift calculations of some thin bodies, such as airfoils, *A* is taken to be the ***planform area***, which is the area seen by a person looking at the body from above in a direction normal to the body. The ***drag*** and ***lift*** coefficients are primarily functions of the shape of the body. However, in some cases they also depend on the ***Reynolds number*** and the ***surface roughness***. The term ($\frac{1}{2}\rho V^2$) in Equations 5.5 and 5.6 is the ***dynamic pressure***.

Example 5.1: The drag coefficient of a car at the design conditions of 1 atm, 70°F (ρ = 0.07489 lbm/ft^3), and 60 mi/h is to be determined experimentally in a large wind tunnel in a full-scale test (Figure 5.4). The frontal area of the car is 22.26 ft^2. If the force acting on the car in the flow direction is measured to be 68 lbf, ***determine*** the drag coefficient of this car.

Solution:

The drag force acting on a body and the drag coefficient are given by

$$F_D = C_D A \frac{\rho V^2}{2}$$

$$C_D = \frac{2F_D}{\rho A V^2}$$

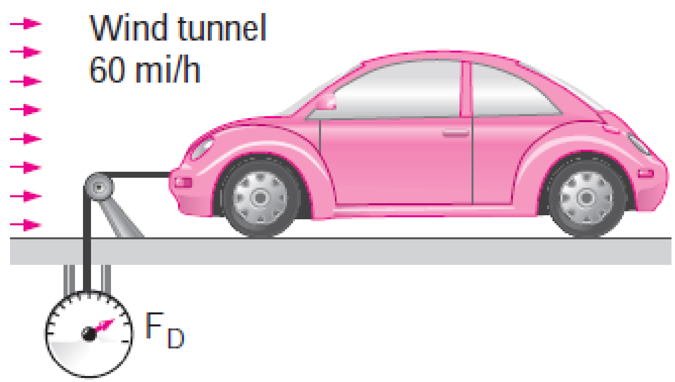


Figure 5.4: Schematic for Example 5.1.

where A is the frontal area. Substituting and noting that 1 ml/h = 1.467 ft/s, the drag coefficient of the car is determined to be

$$C_D = \frac{2 \times (68\ \text{lbf})}{(0.07489\ \text{lbm/ft}^3)(22.26\ \text{ft}^2)(60 \times 1.467\ \text{ft/s})^2}\left(\frac{32.2\ \text{lbm} \cdot \text{ft/s}^2}{1\ \text{lbf}}\right) = \mathbf{0.34}$$

5.3. Drag coefficients of common geometries

The drag coefficient exhibits different behavior in the low (***creeping***), moderate (***laminar***), and high (***turbulent***) regions of the Reynolds number. The inertia effects are negligible in low Reynolds number flows ($Re < 1$), called ***creeping flows***, and the fluid wraps around the body smoothly. The drag coefficient in this case is inversely proportional to the Reynolds number, and for a ***sphere*** it is determined to be

For sphere: $$C_D = \frac{24}{Re} \qquad (Re \lesssim 1) \qquad (5.7)$$

Then the drag force acting on a spherical object at low Reynolds numbers becomes

$$F_D = C_D A \frac{\rho V^2}{2} = \frac{24}{Re} A \frac{\rho V^2}{2} = \frac{24}{\rho V D/\mu} \frac{\pi D^2}{4} \frac{\rho V^2}{2} = 3\pi\mu V D$$

This relation shows that at very low Reynolds numbers, the drag force acting on spherical objects is proportional to the diameter, the velocity, and the viscosity of the fluid. This relation is often applicable to dust particles in the air and suspended solid particles in water. The drag coefficients for low Reynolds number flows past some other geometries are given in Figure 5.5. Note that at low Reynolds numbers, the shape of the body does not have a major influence on the drag coefficient.

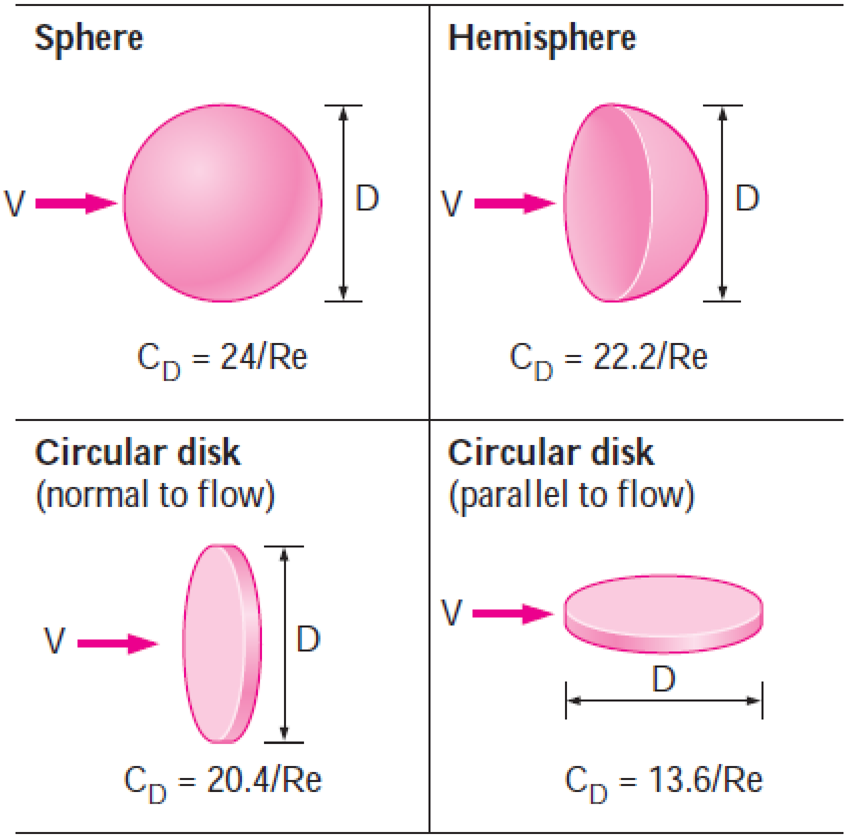
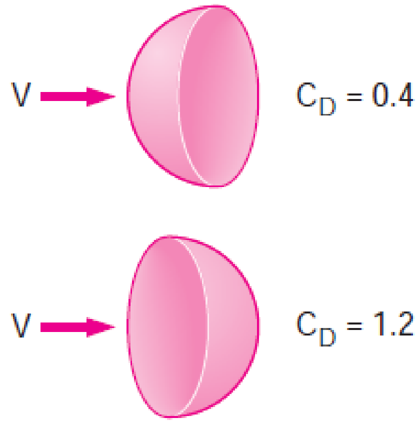


Figure 5.5: Drag coefficients C_D at low velocities ($Re < 1$ where $Re = VD/\nu$ and $A = \pi D^2/4$).

The drag coefficients for various two- and three-dimensional bodies are given in Tables 5.1 and 5.2 for large Reynolds numbers. We can make several observations from these tables about the drag coefficient at high Reynolds numbers. First of all, the orientation of the body relative to the direction of flow has a major influence on the drag coefficient. For example, the drag coefficient for flow over a hemisphere is 0.4 when the spherical side faces the flow, but it increases three-fold to 1.2 when the flat side faces the flow (Figure 5.6). This shows that the rounded nose of a bullet serves another purpose in addition to piercing: reducing drag and thus increasing the range of the gun.

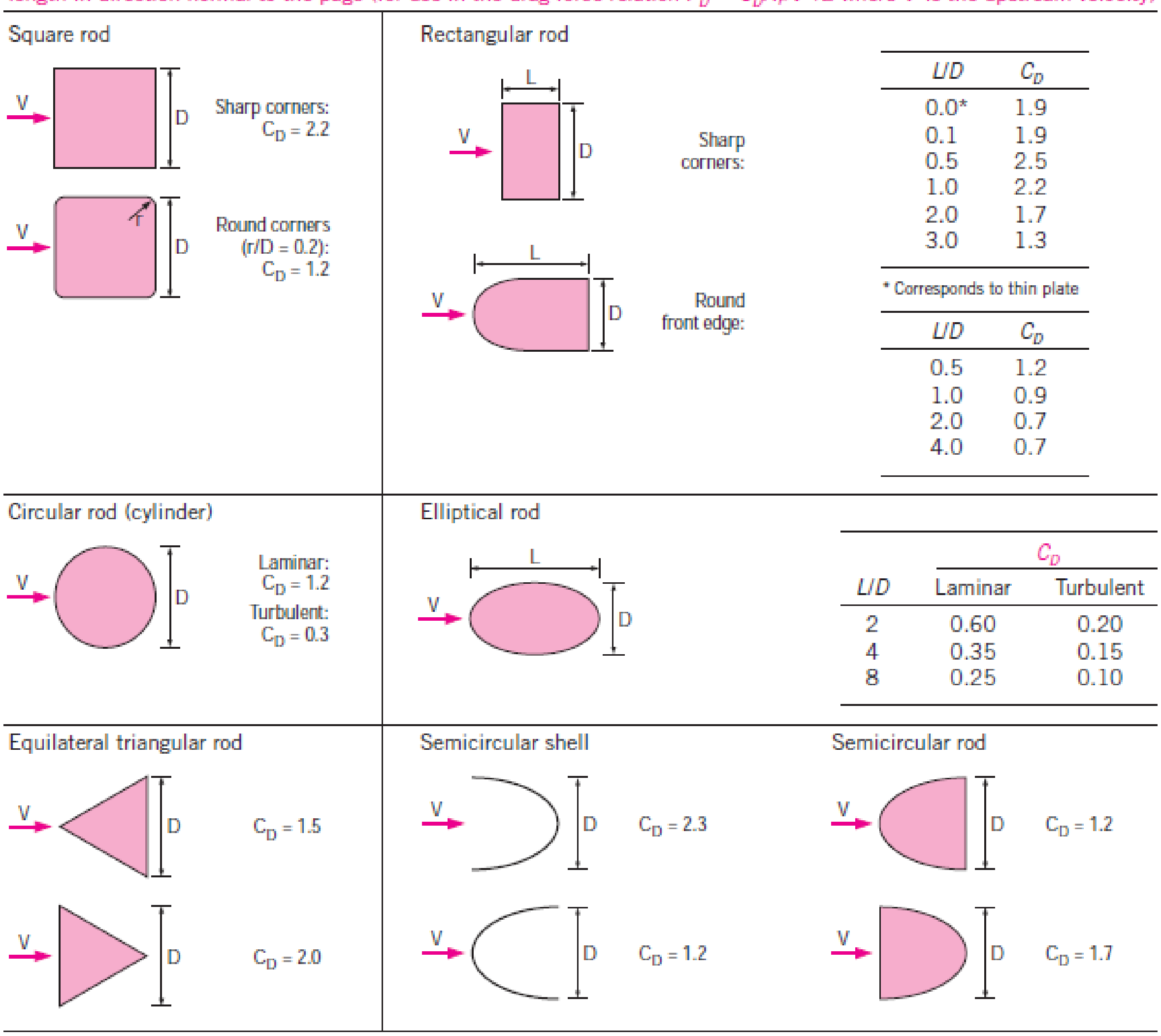


A hemisphere at two different orientations for $Re > 10^4$

Figure 5.6: The drag coefficient of a body may change drastically by changing the body's orientation (and thus shape) relative to the direction of flow.

Table: 5.1

Drag coefficients C_D of various two-dimensional bodies for Re > 10^4 based on the frontal area $A = bD$, where b is the length in direction normal to the page (for use in the drag force relation $F_D = C_D A \rho V^2/2$ where V is the upstream velocity)



Rectangular rod, sharp corners:

L/D	C_D
0.0*	1.9
0.1	1.9
0.5	2.5
1.0	2.2
2.0	1.7
3.0	1.3

* Corresponds to thin plate

Rectangular rod, round front edge:

L/D	C_D
0.5	1.2
1.0	0.9
2.0	0.7
4.0	0.7

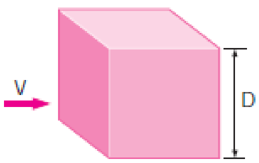
Elliptical rod:

	C_D	
L/D	Laminar	Turbulent
2	0.60	0.20
4	0.35	0.15
8	0.25	0.10

Table: 5.2 (Continued)

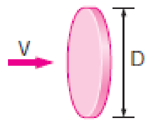
Representative drag coefficients C_D for various three-dimensional bodies for Re $> 10^4$ based on the frontal area (for use in the drag force relation $F_D = C_D A \rho V^2/2$ where V is the upstream velocity)

Cube, $A = D^2$



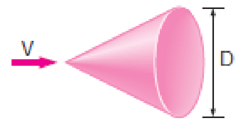
$C_D = 1.05$

Thin circular disk, $A = \pi D^2/4$



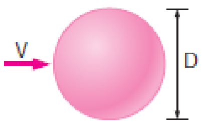
$C_D = 1.1$

Cone (for $\theta = 30°$), $A = \pi D^2/4$



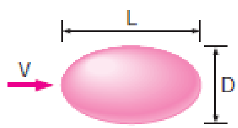
$C_D = 0.5$

Sphere, $A = \pi D^2/4$



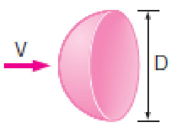
Laminar: $C_D = 0.5$
Turbulent: $C_D = 0.2$

Ellipsoid, $A = \pi D^2/4$

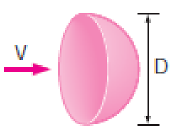


L/D	C_D Laminar	C_D Turbulent
0.75	0.5	0.2
1	0.5	0.2
2	0.3	0.1
4	0.3	0.1
8	0.2	0.1

Hemisphere, $A = \pi D^2/4$

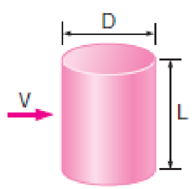


$C_D = 0.4$



$C_D = 1.2$

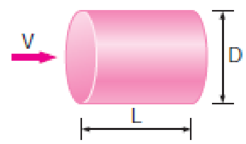
Short cylinder, vertical, $A = LD$



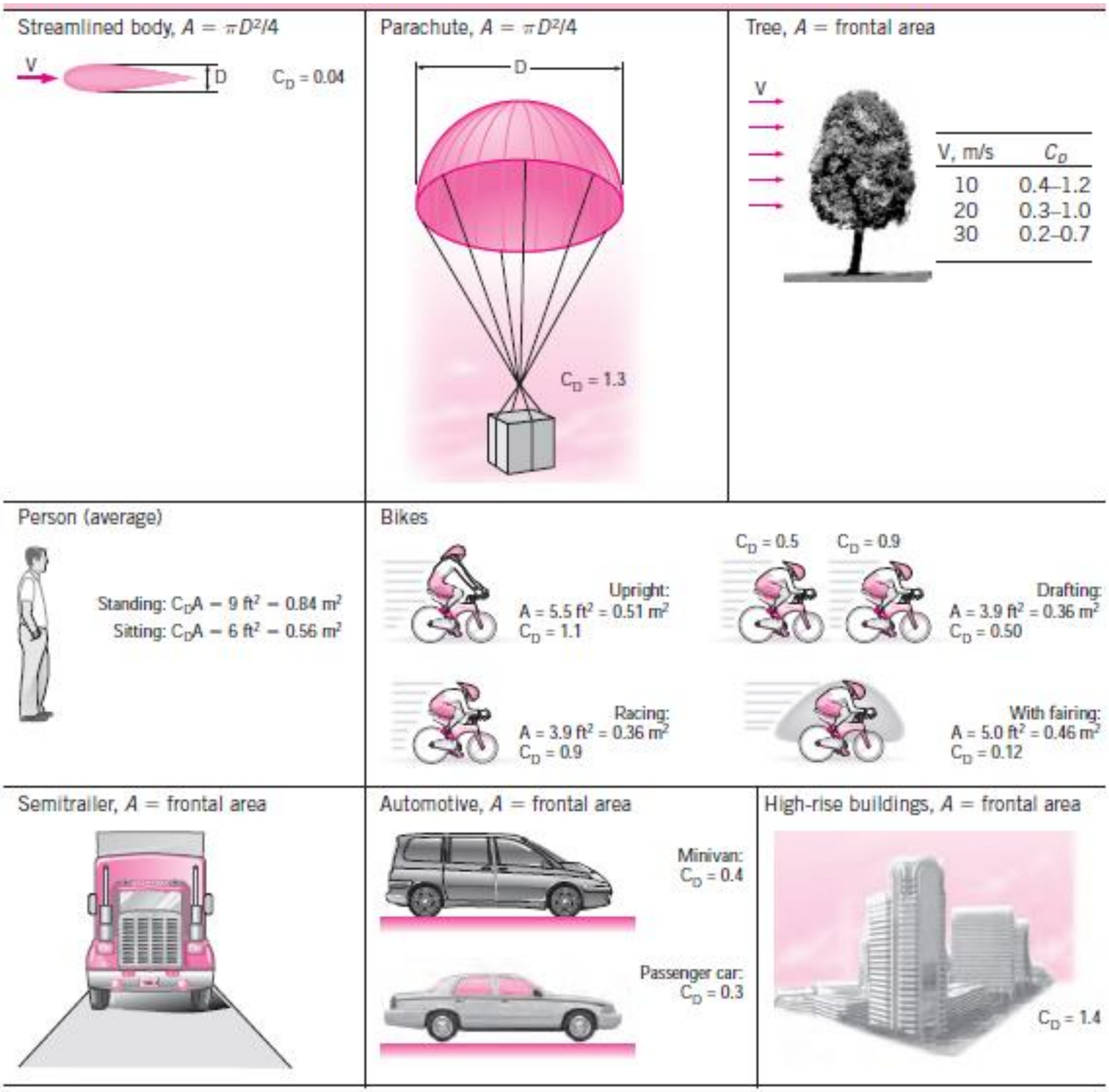
L/D	C_D
1	0.6
2	0.7
5	0.8
10	0.9
40	1.0
∞	1.2

Values are for laminar flow

Short cylinder, horizontal, $A = \pi D^2/4$



L/D	C_D
0.5	1.1
1	0.9
2	0.9
4	0.9
8	1.0


Streamlined body, $A = \pi D^2/4$
V
D
$C_D = 0.04$
Parachute, $A = \pi D^2/4$
D
$C_D = 1.3$
Tree, A = frontal area
V
V, m/s | C_D
10 | 0.4–1.2
20 | 0.3–1.0
30 | 0.2–0.7
Person (average)
Standing: $C_DA = 9\ ft^2 = 0.84\ m^2$
Sitting: $C_DA = 6\ ft^2 = 0.56\ m^2$
Bikes
Upright: $A = 5.5\ ft^2 = 0.51\ m^2$ $C_D = 1.1$
$C_D = 0.5$
$C_D = 0.9$
Drafting: $A = 3.9\ ft^2 = 0.36\ m^2$ $C_D = 0.50$
Racing: $A = 3.9\ ft^2 = 0.36\ m^2$ $C_D = 0.9$
With fairing: $A = 5.0\ ft^2 = 0.46\ m^2$ $C_D = 0.12$
Semitrailer, A = frontal area
Automotive, A = frontal area
Minivan: $C_D = 0.4$
Passenger car: $C_D = 0.3$
High-rise buildings, A = frontal area
$C_D = 1.4$

Example 5.2:

As part of the continuing efforts to reduce the drag coefficient and thus to improve the fuel efficiency of cars, the design of side rearview mirrors has changed drastically from a simple circular plate to a streamlined shape. Determine the amount of fuel and money saved per year as a result of replacing a 13-cm-diameter flat mirror by one with a hemispherical back (Figure 5.7). Assume the car is driven 24,000 km a year at an average speed of 95 km/h. Take the density and price of gasoline to be 0.8 kg/L and $0.60/L, respectively; the heating value of gasoline to be 44,000 kJ/kg; and the overall efficiency of the engine to be 30 percent.

Solution:

Properties: The densities of air and gasoline are taken to be 1.20 kg/m^3 and 800 kg/m^3, respectively. The heating value of gasoline is given to be 44,000 kJ/kg. The drag coefficients C_D are 1.1 for a circular disk and 0.4 for a hemispherical body.

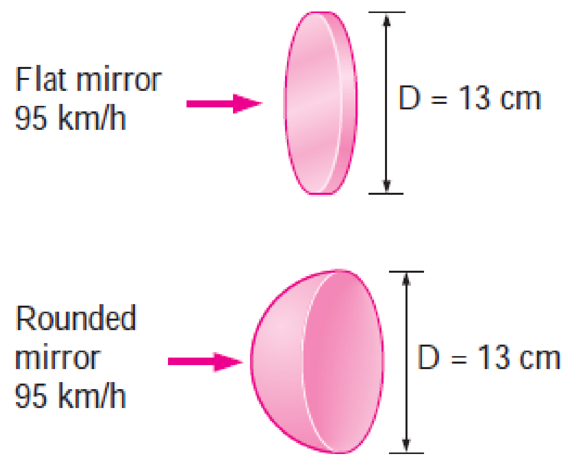


Figure 5.7: Schematic for Example 5.2.

The drag force acting on a body is determined from $F_D = C_D A \dfrac{\rho V^2}{2}$

where *A* is the frontal area of the body, which is A = $\pi D^2/4$ for both the flat and rounded mirrors. The drag force acting on the flat mirror is

$$F_D = 1.1\frac{\pi(0.13\text{ m})^2}{4}\frac{(1.20\text{ kg/m}^3)(95\text{ km/h})^2}{2}\left(\frac{1\text{ m/s}}{3.6\text{ km/h}}\right)^2\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 6.10\text{ N}$$

Noting that work is force times distance, the amount of work done to overcome this drag force and the required energy input for a distance of 24,000 km are

$$W_{drag} = F_D \times L = (6.10\text{ N})(24{,}000\text{ km/year}) = 146{,}400\text{ kJ/year}$$

$$E_{in} = \frac{W_{drag}}{\eta_{car}} = \frac{146{,}400\text{ kJ/year}}{0.3} = 488{,}000\text{ kJ/year}$$

Then the amount and costs of the fuel that supplies this much energy are

$$\text{Amount of fuel} = \frac{m_{fuel}}{\rho_{fuel}} = \frac{E_{in}/HV}{\rho_{fuel}} = \frac{(488{,}000\text{ kJ/year})/(44{,}000\text{ kJ/kg})}{0.8\text{ kg/L}}$$

$$= 13.9\text{ L/year}$$

$$\text{Cost} = (\text{Amount of fuel})(\text{Unit cost}) = (13.9\text{ L/year})(\$0.60/\text{L}) = \$8.32/\text{year}$$

That is, the car uses 13.9 L of gasoline at a cost of $8.32 per year to overcome the drag generated by a flat mirror extending out from the side of a car.

The drag force and the work done to overcome it are directly proportional to the drag coefficient. Then the percent reduction in the fuel consumption due to replacing the mirror is equal to the percent reduction in the drag coefficient:

$$\text{Reduction ratio} = \frac{C_{D,\text{ flat}} - C_{D,\text{ hemisp}}}{C_{D,\text{ flat}}} = \frac{1.1 - 0.4}{1.1} = 0.636$$

$$\text{Fuel reduction} = (\text{Reduction ratio})(\text{Amount of fuel})$$

$$= 0.636(13.9\text{ L/year}) = \mathbf{8.84\text{ L/year}}$$

$$\text{Cost reduction} = (\text{Reduction ratio})(\text{Cost}) = 0.636(\$8.32/\text{year}) = \mathbf{\$5.29/\text{year}}$$

Since a typical car has two side rearview mirrors, the driver saves more than $10 per year in gasoline by replacing the flat mirrors with hemispherical ones.

5.4. Lift

Lift was defined earlier as the component of the net force (due to viscous and pressure forces) that is perpendicular to the flow direction, and the lift coefficient was expressed as,

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} \quad (5.8)$$

where *A* in this case is normally the planform area, which is the area that would be seen by a person looking at the body from above in a direction normal to the body, and *V* is the upstream velocity of the fluid (or, equivalently, the velocity of a flying body in a quiescent fluid). For an airfoil of width (or span) b and chord length c (the length between the leading and trailing edges), the planform area is *A* = bc. The distance between the two ends of a wing or airfoil is called the ***wingspan*** or just the ***span***. For an aircraft, the wingspan is taken to be the total distance between the tips of the two wings, which includes the width of the fuselage between the wings (Figure 5.8). The average lift per unit planform area F_L/A is called the ***wing loading***, which is simply the ratio of the weight of the aircraft to the planform area of the wings (since lift equals the weight during flying at constant altitude).

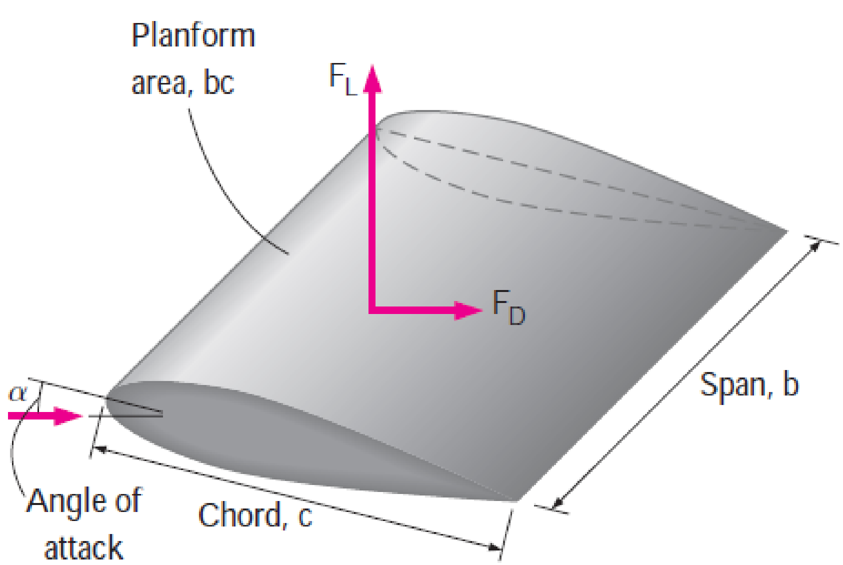


Figure 5.8: Definition of various terms associated with an airfoil.

The flow fields obtained from such calculations are sketched in Figure 5.9 for both symmetrical and nonsymmetrical airfoils by ignoring the thin boundary layer. At zero angle of attack, the lift produced by the symmetrical airfoil is zero, as expected because of symmetry, and the stagnation points are at the leading and trailing edges. For the nonsymmetrical airfoil, which is at a small angle of attack, the front stagnation point has moved down below the leading edge, and the rear stagnation point has moved up to the upper surface close to the trailing edge. To our surprise, the lift produced is calculated again to be zero-a clear contradiction of experimental observations and measurements. Obviously, the theory needs to be modified to bring it in line with the observed phenomenon.

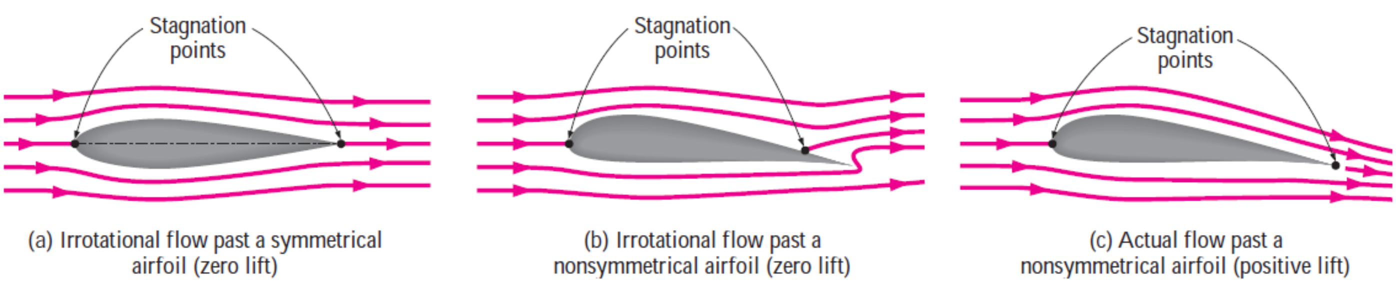


Figure 5.9: Irrotational and actual flow past symmetrical and nonsymmetrical two-dimensional airfoils.

The minimum flight velocity can be determined from the requirement that the total weight *W* of the aircraft be equal to lift and $C_L = C_{L,\,max}$. That is,

$$W = F_L = \frac{1}{2} C_{L,\,max}\,\rho V_{min}^2 A \qquad \rightarrow \qquad V_{min} = \sqrt{\frac{2W}{\rho C_{L,\,max} A}} \quad (5.9)$$

For a given weight, the landing or takeoff speed can be minimized by maximizing the product of the lift coefficient and the wing area, $C_{L,\,max}A$. One way of doing that is to use flaps, as already discussed. Another way is to control the boundary layer, which can be accomplished simply by leaving flow sections (slots) between the flaps, as shown in Figure 5.10. Slots are used to prevent the separation of the boundary layer from the upper surface of the wings and the flaps. This is done by allowing air to move from the high-pressure region under the wing into the low-pressure region at the top surface. Note that the lift coefficient reaches its maximum value $C_L = C_{L,\ max}$, and thus the flight velocity reaches its minimum, at stall conditions, which is a region of unstable operation and must be avoided. The ***Federal Aviation Administration*** (FAA) does not allow operation below 1.2 times the stall speed for safety.

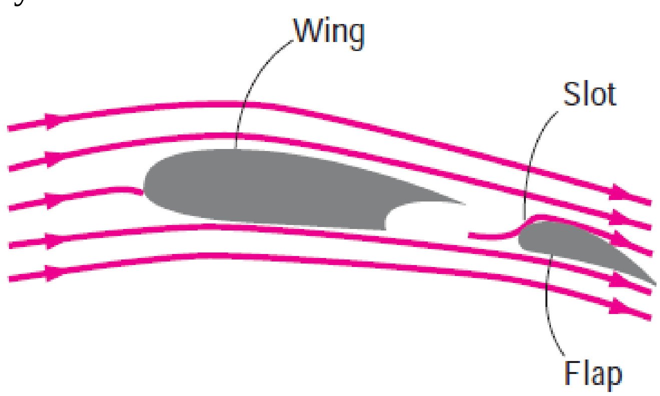


Figure 5.10: A flapped airfoil with a slot to prevent the separation of the boundary layer from the upper surface and to increase the lift coefficient.

5.5. Lift Generated by Spinning

The phenomenon of producing lift by the rotation of a solid body is called the ***Magnus effect*** after the German scientist Heinrich Magnus (1802–1870), who was the first to study the lift of rotating bodies, which is illustrated in Figure 5.11 for the simplified case of irrotational (potential) flow. When the ball is not spinning, the lift is zero because of top–bottom symmetry. But when the cylinder is rotated about its axis, the cylinder drags some fluid around because of the no-slip condition and the flow field reflects the superposition of the spinning and nonspinning flows. The ***stagnation points*** shift down, and the flow is no longer symmetric about the horizontal plane that passes through the center of the cylinder. The average pressure on the upper half is less than the average pressure at the lower half because of the ***Bernoulli effect***, and thus there is a net upward force (lift) acting on the cylinder.

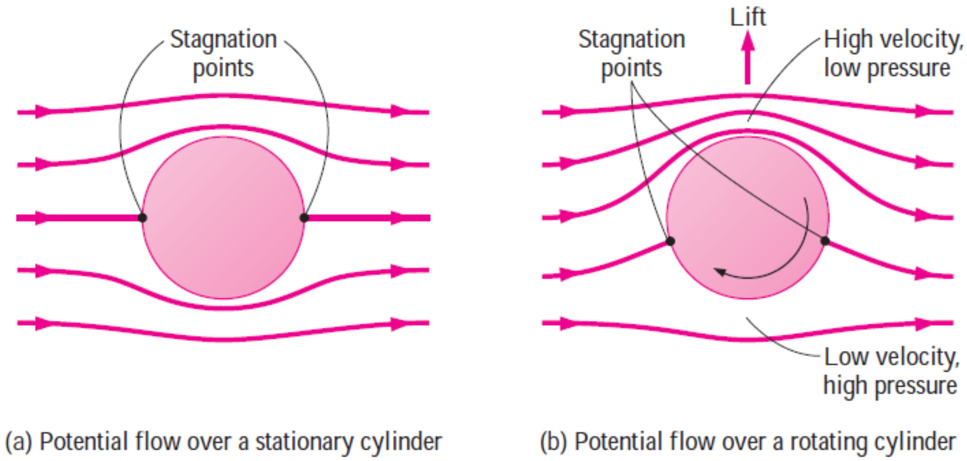


Figure 5.11: Generation of lift on a rotating circular cylinder for the case of "idealized" potential flow (the actual flow involves flow separation in the wake region.

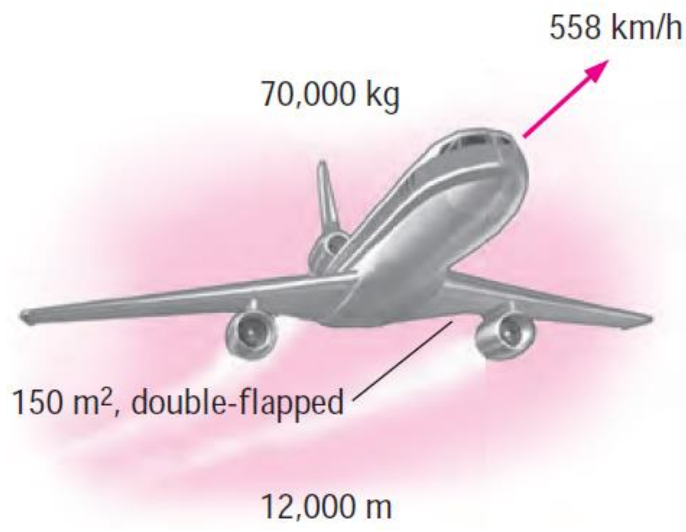
Example 5.3:

A commercial airplane has a total mass of 70,000 kg and a wing planform area of 150 m^2 (see Figure 5.12). The plane has a cruising speed of 558 km/h and a cruising altitude of 12,000 m, where the air density is 0.312 kg/m^3. The plane has double-slotted flaps for use during takeoff and landing, but it cruises with all flaps retracted. Assuming the lift and the drag characteristics of the wings can be approximated by NACA 23012, ***determine*** (**a**) the minimum safe speed for takeoff and landing with and without extending the flaps, (**b**) the angle of attack to cruise steadily at the cruising altitude, and (**c**) the power that needs to be supplied to provide enough thrust to overcome wing drag. The maximum lift coefficients $C_{L,max}$ of the wings are 3.48 and 1.52 with and without flaps, respectively

Solution: (**a**) The weight and cruising speed of the airplane are

$$W = mg = (70{,}000 \text{ kg})(9.81 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 686{,}700 \text{ N}$$

$$V = (558 \text{ km/h})\left(\frac{1 \text{ m/s}}{3.6 \text{ km/h}}\right) = 155 \text{ m/s}$$



The minimum velocities corresponding to the stall conditions without and with flaps, respectively, are obtained from Equation 5.9,

$$V_{min\,1} = \sqrt{\frac{2W}{\rho C_{L,\,max\,1} A}} = \sqrt{\frac{2(686{,}700\ N)}{(1.2\ kg/m^3)(1.52)(150\ m^2)}\left(\frac{1\ kg\cdot m/s^2}{1\ N}\right)} = 70.9\ m/s$$

$$V_{min\,2} = \sqrt{\frac{2W}{\rho C_{L,\,max\,2} A}} = \sqrt{\frac{2(686{,}700\ N)}{(1.2\ kg/m^3)(3.48)(150\ m^2)}\left(\frac{1\ kg\cdot m/s^2}{1\ N}\right)} = 46.8\ m/s$$

Then the "*safe*" minimum velocities to avoid the stall region are obtained by multiplying the values above by 1.2:

Without flaps: $V_{min\,1,\,safe} = 1.2V_{min\,1} = 1.2(70.9\ m/s) = 85.1\ m/s = \mathbf{306\ km/h}$

With flaps: $V_{min\,2,\,safe} = 1.2V_{min\,2} = 1.2(46.8\ m/s) = 56.2\ m/s = \mathbf{202\ km/h}$

since 1 m/s = 3.6 km/h. Note that the use of flaps allows the plane to take off and land at considerably lower velocities, and thus on a shorter runway.

(**b**) When an aircraft is cruising steadily at a constant altitude, the lift must be equal to the weight of the aircraft, $F_L = W$. Then the lift coefficient is determined to be

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} = \frac{686{,}700\ N}{\frac{1}{2}(0.312\ kg/m^3)(155\ m/s)^2(150\ m^2)}\left(\frac{1\ kg\cdot m/s^2}{1\ N}\right) = 1.22$$

For the case with no flaps, the angle of attack corresponding to this value of C_L is determined from following Figure to be **α ≈10°**.

(**c**) When the aircraft is cruising steadily at a constant altitude, the net force acting on the aircraft is zero, and thus thrust provided by the engines must be equal to the drag force. The drag coefficient corresponding to the cruising lift coefficient of 1.22 is determined from the following Figure to be $C_D \approx 0.03$ for the case with no flaps. Then the drag force acting on the wings becomes

$$F_D = C_D A \frac{\rho V^2}{2} = (0.03)(150\ \text{m}^2)\frac{(0.312\ \text{kg/m}^3)(155\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$= 16.9\ \text{kN}$$

Noting that power is force times velocity (distance per unit time), the power required to overcome this drag is equal to the thrust times the cruising velocity:

$$\text{Power} = \text{Thrust} \times \text{Velocity} = F_D V = (16.9\ \text{kN})(155\ \text{m/s})\left(\frac{1\ \text{kW}}{1\ \text{kN}\cdot\text{m/s}}\right)$$
$$= \mathbf{2620\ kW}$$

Therefore, the engines must supply 2620 kW of power to overcome the drag on the wings during cruising. For a propulsion efficiency of 30 percent (i.e., 30 percent of the energy of the fuel is utilized to propel the aircraft), the plane requires energy input at a rate of 8733 kJ/s.

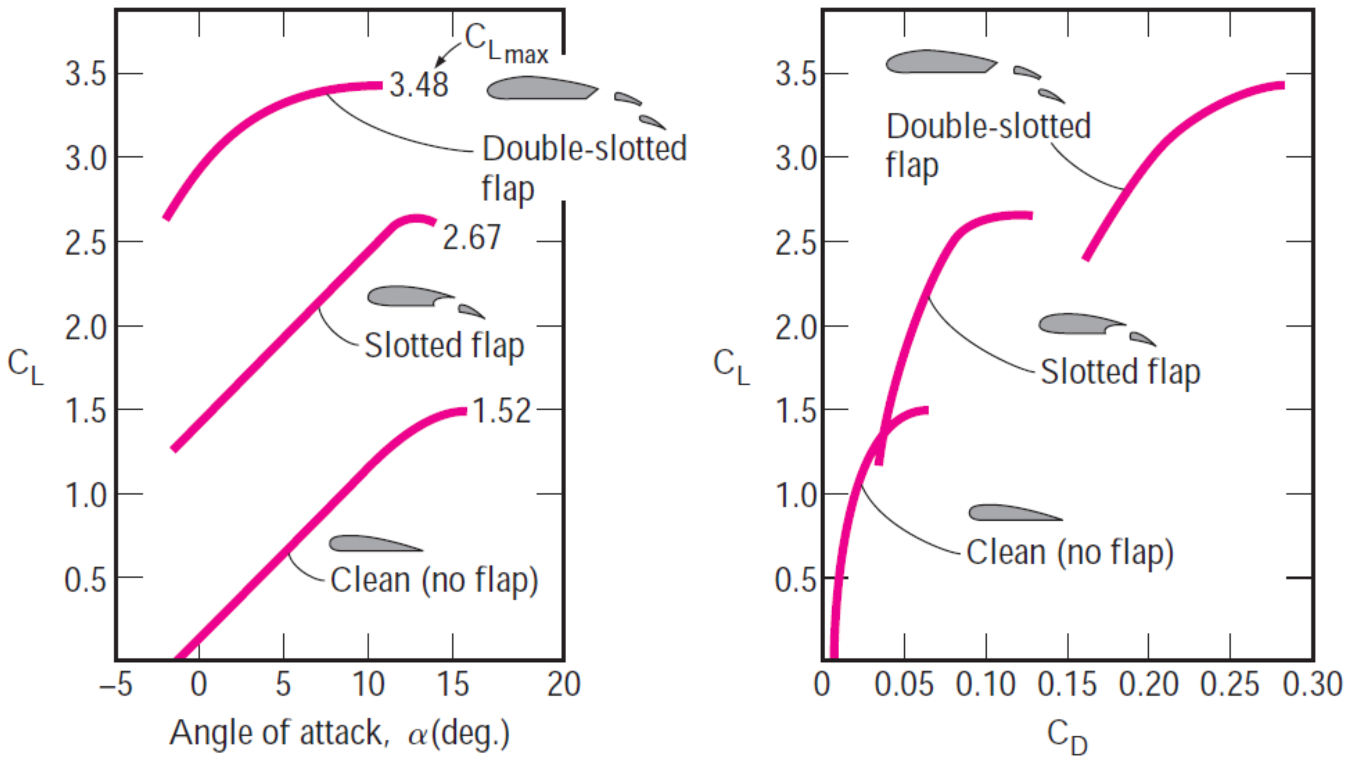
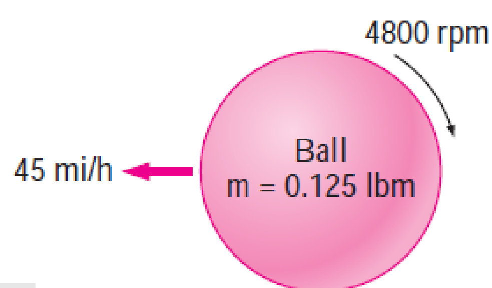


Figure Example 5.3: Effect of flaps on the lift and drag coefficients of an airfoil.
From Abbott and von Doenhoff, for NACA 23012 (1959).

Example 5.4:

A tennis ball with a mass of 0.125 lbm and a diameter of 2.52 in is hit at 45 mi/h with a backspin of 4800 rpm (see Figure below). Determine if the ball will fall or rise under the combined effect of gravity and lift due to spinning shortly after being hit in air at 1 atm and 80°F.



Solution:

The density and kinematic viscosity of air at 1 atm and 80°F are ρ = 0.07350 lbm/ft^3 and ν = 1.697×10^{-4} ft^2/s.

The lift can be determined from $$F_L = C_L A \frac{\rho V^2}{2}$$

where A is the frontal area of the ball, which is $A = \pi D^2/4$. The translational and angular velocities of the ball are

$$V = (45\ \text{mi/h})\left(\frac{5280\ \text{ft}}{1\ \text{mi}}\right)\left(\frac{1\ \text{h}}{3600\ \text{s}}\right) = 66\ \text{ft/s}$$

$$\omega = (4800\ \text{rev/min})\left(\frac{2\pi\ \text{rad}}{1\ \text{rev}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 502\ \text{rad/s}$$

$$\frac{\omega D}{2V} = \frac{(502\ \text{rad/s})(2.52/12\ \text{ft})}{2(66\ \text{ft/s})} = 0.80\ \text{rad}$$

From Figure below, the lift coefficient corresponding to this value is C_L= 0.21. Then the lift force acting on the ball is

$$F_L = (0.21)\frac{\pi(2.52/12\ \text{ft})^2}{4}\frac{(0.0735\ \text{lbm/ft}^3)(66\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right)$$

$$= 0.036\ \text{lbf}$$

The weight of the ball is

$$W = mg = (0.125\ \text{lbm})(32.2\ \text{ft/s}^2)\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = 0.125\ \text{lbf}$$

which is more than the lift. Therefore, the ball will ***drop*** under the combined effect of gravity and lift due to spinning with a net force of 0.125 - 0.036 = 0.089 lbf.

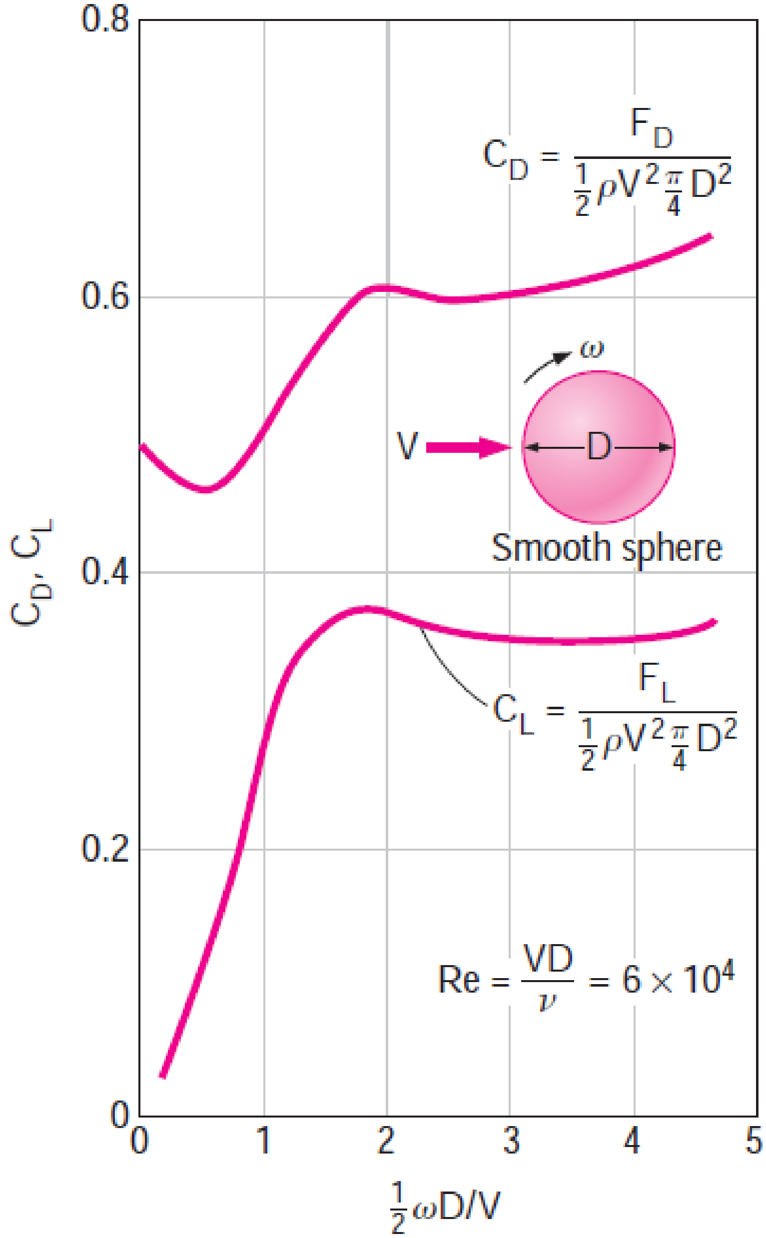


Figure Example 5.4: The variation of lift and drag coefficients of a smooth sphere with the nondimensional rate of rotation for $Re = VD/\nu = 6 \times 10^4$.

University of Anbar
College of Engineering
Mechanical Engineering Dept.

Fluid Mechanics-II
(ME 2305)

Handout Lectures for Year Two
Chapter Four/ Flow Rate and Velocity Measurement

Course Tutor

Assist. Prof. Dr. Waleed M. Abed

Chapter Four
Flow Rate and Velocity Measurement

4.1. Introduction

A major application area of fluid mechanics is the determination of the flow rate of fluids, and numerous devices have been developed over the years for the purpose of flow metering. Flowmeters range widely in their level of sophistication, size, cost, accuracy, versatility, capacity, pressure drop, and the operating principle. We give an overview of the meters commonly used to measure the flow rate of liquids and gases flowing through pipes or ducts. We limit our consideration to incompressible flow.

Some flowmeters measure the flow rate directly by discharging and recharging a measuring chamber of known volume continuously and keeping track of the number of discharges per unit time. But most flowmeters measure the flow rate indirectly-they measure the average velocity V or a quantity that is related to average velocity such as pressure and drag, and determine the volume flow rate $\dot{V}$ from

$$\dot{V} = V \times A_c \qquad (4.1)$$

where A_c is the cross-sectional area of flow. Therefore, measuring the flow rate is usually done by measuring flow velocity, and most flowmeters are simply velocimeters used for the purpose of metering flow.

The velocity in a pipe varies from zero at the wall to a maximum at the center, and it is important to keep this in mind when taking velocity measurements. For laminar flow, for example, the average velocity is half the centerline velocity. But this is not the case in turbulent flow, and it may be necessary to take the weighted average of several local velocity measurements to determine the average velocity.

4.2. Pitot and Pitot-Static Probes

Pitot probes (also called *Pitot tubes*) and **Pitot-static probes,** named after the French engineer Henri de Pitot (1695–1771), are widely used for flow rate measurement. A Pitot probe is just a tube with a pressure tap at the stagnation point that measures stagnation pressure, while a Pitot-static probe has both a stagnation pressure tap and several circumferential static pressure taps and it measures both stagnation and static pressures (Figures 4.1 and 4.2). Pitot was the first person to measure velocity with the upstream pointed tube, while French engineer Henry Darcy (1803–1858) developed most of the features of the instruments we use today, including the use of small openings and the placement of the static tube on the same assembly. Therefore, it is more appropriate to call the Pitot-static probes **Pitot–Darcy probes.**

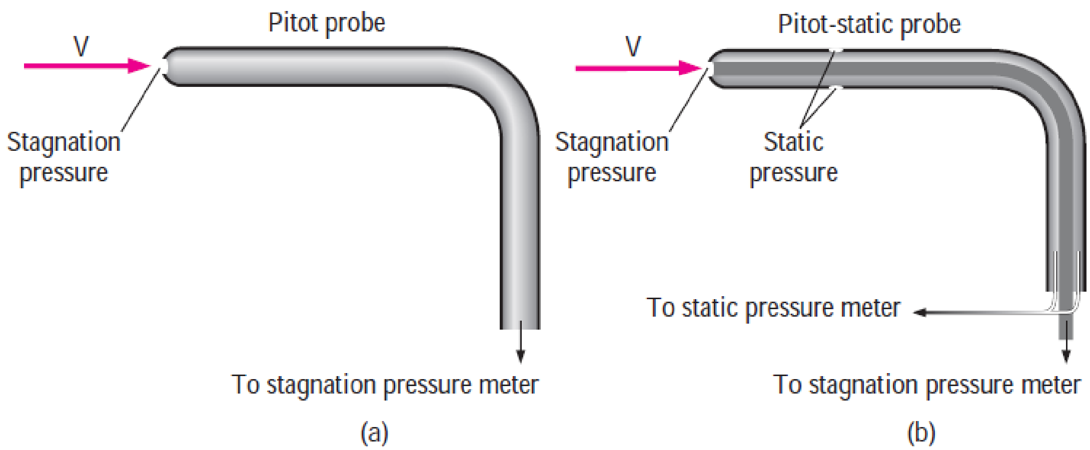


Figure 4.1: (**a**) A Pitot probe measures stagnation pressure at the nose of the probe, while (**b**) a Pitot-static probe measures both stagnation pressure and static pressure, from which the flow speed can be calculated.

The Pitot-static probe measures local velocity by measuring the pressure difference in conjunction with the Bernoulli equation. It consists of a slender double-tube aligned with the flow and connected to a differential pressure meter. The inner tube is fully open to flow at the nose, and thus it measures the stagnation pressure at that location (point 1). The outer tube is sealed at the nose, but it has holes on the side of the outer wall (point 2) and thus it measures the static pressure. For incompressible flow with sufficiently high velocities (so that the frictional effects between points 1 and 2 are negligible), the Bernoulli equation is applicable and can be expressed as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad (4.2)$$

Noting that $z_1 \approx z_2$ since the static pressure holes of the Pitot-static probe are arranged circumferentially around the tube and $V_1 = 0$ because of the stagnation conditions, the flow velocity $V = V_2$ becomes

Pitot formula:

$$V = \sqrt{\frac{2(P_1 - P_2)}{\rho}} \quad (4.3)$$

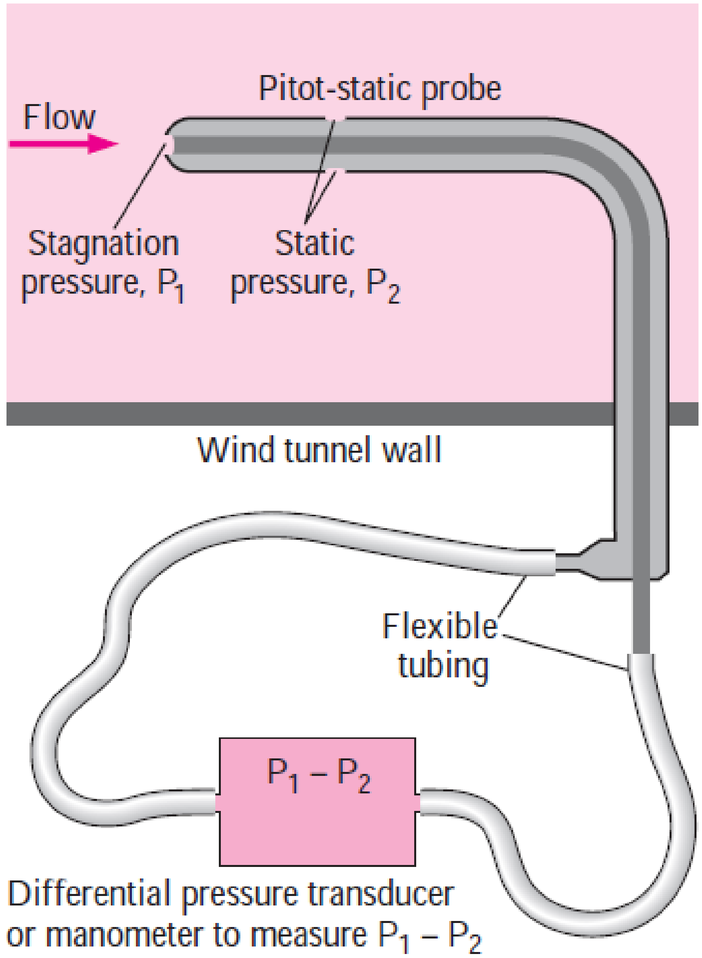


Figure 4.2: Measuring flow velocity with a Pitotstatic probe.

which is known as the Pitot formula. If the velocity is measured at a location where the local velocity is equal to the average flow velocity, the volume flow rate can be determined from ($\dot{V} = V \times A_c$).

4.3. Obstruction Flowmeters: Orifice, Venturi, and Nozzle Meters

Consider incompressible steady flow of a fluid in a horizontal pipe of diameter *D* that is constricted to a flow area of diameter *d*, as shown in Figure 4.3. The mass balance and the *Bernoulli equations* between a location before the constriction (point ①) and the location where constriction occurs (point ②) can be written as,

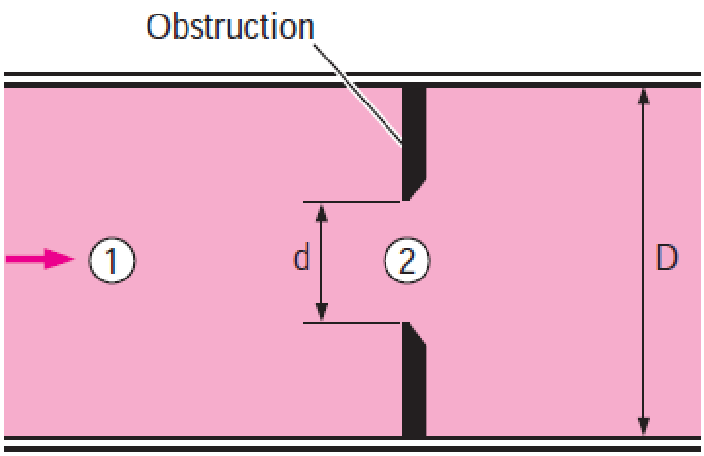


Figure 4.3: Flow through a constriction in a pipe.

Mass balance: $\dot{V} = A_1V_1 = A_2V_2 \quad \rightarrow \quad V_1 = (A_2/A_1)V_2 = (d/D)^2V_2$ (4.4)

Bernoulli equation ($z_1 = z_2$): $\dfrac{P_1}{\rho g} + \dfrac{V_1^2}{2g} = \dfrac{P_2}{\rho g} + \dfrac{V_2^2}{2g}$ (4.5)

Combining Equations 4.4 and 4.5 and solving for velocity V_2 gives

Obstruction (with no loss): $V_2 = \sqrt{\dfrac{2(P_1 - P_2)}{\rho(1 - \beta^4)}}$ (4.6)

where $\beta = d/D$ is the diameter ratio. Once V_2 is known, the flow rate can be determined from $\dot{V} = A_2V_2 = (\pi\ d^2/4)V_2$.

This simple analysis shows that the flow rate through a pipe can be determined by constricting the flow and measuring the decrease in pressure due to the increase in velocity at the constriction site. Noting that the pressure drop between two points along the flow can be measured easily by a differential pressure transducer or manometer, it appears that a simple flow rate measurement device can be built by obstructing the flow. Flowmeters based on this principle are called ***obstruction flowmeters*** and are widely used to measure flow rates of gases and liquids.

Both losses can be accounted for by incorporating a correction factor called the ***discharge coefficient*** C_d whose value (which is less than 1) is determined experimentally. Then the flow rate for obstruction flowmeters can be expressed as,

Obstruction flowmeters: $$\dot{V} = A_0 C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}} \qquad (4.7)$$

where $A_o = A_2 = \pi d^2/4$ is the cross-sectional area of the hole and $\beta = d/D$ is the ratio of hole diameter to pipe diameter. The value of C_d depends on both β and the Reynolds number $Re = V_1 D/\nu$, and charts and curve-fit correlations for C_d are available for various types of obstruction meters.

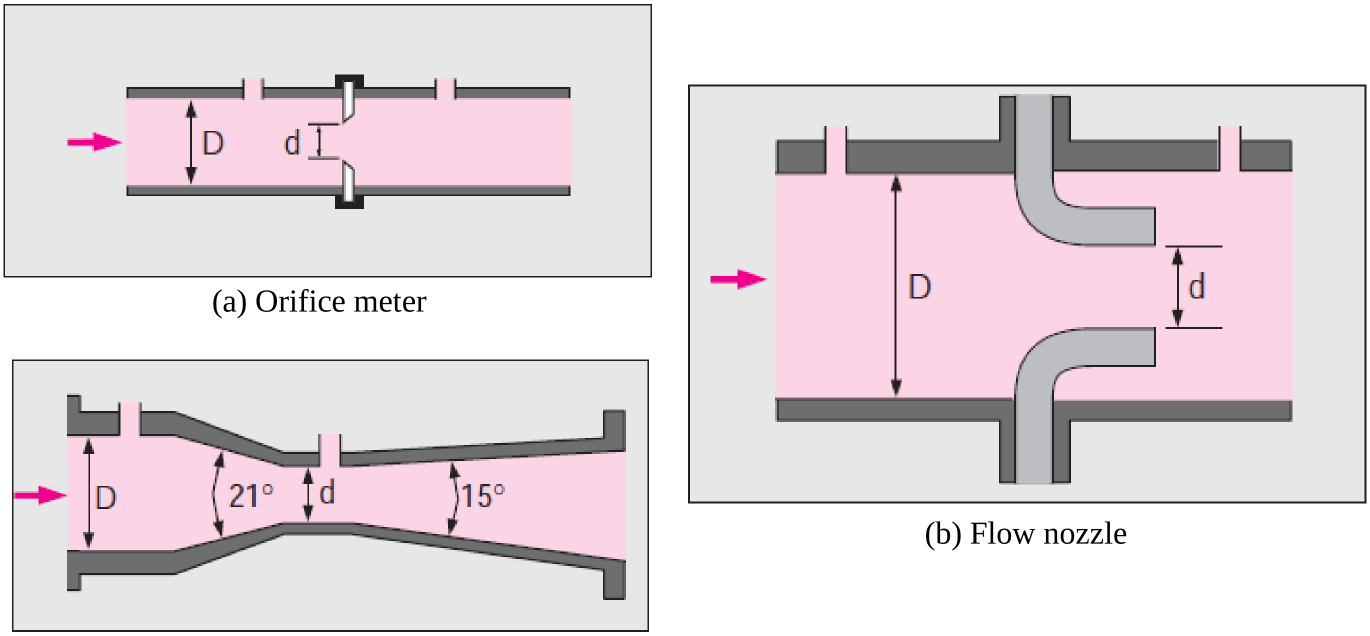
Of the numerous types of obstruction meters available, those most widely used are orifice meters, flow nozzles, and Venturi meters (Figure 4.4). The experimentally determined data for discharge coefficients are expressed as (Miller, 1997)

Orifice meters: $$C_d = 0.5959 + 0.0312\beta^{2.1} - 0.184\beta^8 + \frac{91.71\beta^{2.5}}{Re^{0.75}} \qquad (4.8)$$

Nozzle meters: $$C_d = 0.9975 - \frac{6.53\beta^{0.5}}{Re^{0.5}} \qquad (4.9)$$

These relations are valid for $0.25 < \beta < 0.75$ and $10^4 < Re < 10^7$. Precise values of C_d depend on the particular design of the obstruction, and thus the manufacturer's data should be consulted when available. For flows with high Reynolds numbers

(Re > 30,000), the value of C_d can be taken to be 0.96 for flow nozzles and 0.61 for orifices.



(a) Orifice meter

(b) Flow nozzle

(c) Venturi meter

Figure 4.4: Common types of obstruction meters.

Example 4.1:

The flow rate of methanol at 20°C (ρ 788.4 kg/m^3 and μ = 5.857×10^{-4} kg/ms) through a 4 cm diameter pipe is to be measured with a 3 cm diameter orifice meter equipped with a mercury manometer across the orifice place, as shown in Figure 4.5. If the differential height of the manometer is read to be 11 cm, ***determine*** the flow rate of methanol through the pipe and the average flow velocity. The discharge coefficient of the orifice meter is C_d = 0.61 and take the density of mercury to be 13600 kg/m^3.

Solution: The diameter ratio and the throat area of the orifice are

$$\beta = \frac{d}{D} = \frac{3}{4} = 0.75$$

$$A_0 = \frac{\pi d^2}{4} = \frac{\pi (0.03\ \text{m})^2}{4} = 7.069 \times 10^{-4}\ \text{m}^2$$

The pressure drop across the orifice plate can be expressed as

$$\Delta P = P_1 - P_2 = (\rho_{Hg} - \rho_{met})gh$$

Then the flow rate relation for obstruction meters becomes

$$\dot{V} = A_0 C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= A_0 C_d \sqrt{\frac{2(\rho_{Hg} - \rho_{met})gh}{\rho_{met}(1-\beta^4)}}$$

$$= A_0 C_d \sqrt{\frac{2(\rho_{Hg}/\rho_{met} - 1)gh}{1-\beta^4}}$$

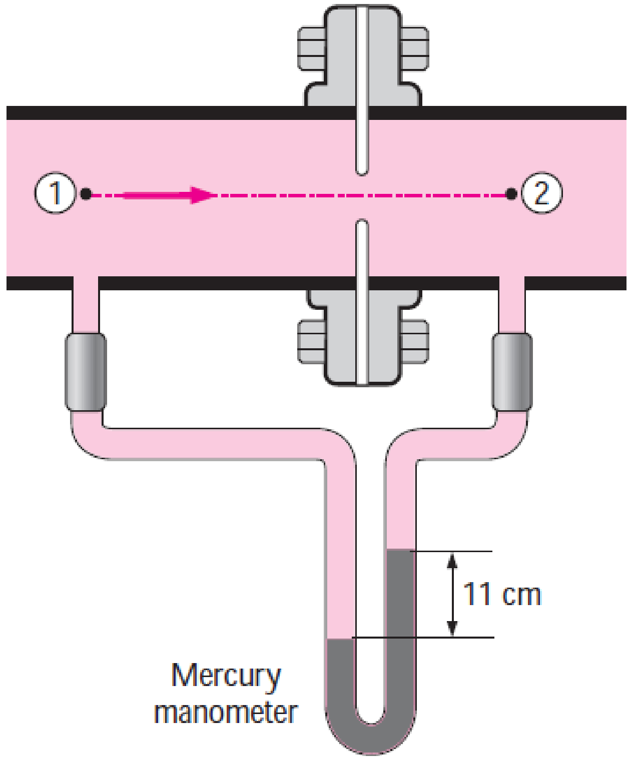


Figure 4.5: Schematic for the orifice meter considered in Example 4.1.

Substituting, the flow rate is determined to be

$$\dot{V} = (7.069 \times 10^{-4}\ m^2)(0.61)\sqrt{\frac{2(13{,}600/788.4 - 1)(9.81\ m/s^2)(0.11\ m)}{1 - 0.75^4}}$$

$$= \mathbf{3.09 \times 10^{-3}\ m^3/s}$$

which is equivalent to 3.09 L/s. The average flow velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{3.09 \times 10^{-3}\ m^3/s}{\pi(0.04\ m)^2/4} = \mathbf{2.46\ m/s}$$

Example 4.2:

A vertical Venturi meter equipped with a differential pressure gage shown in Figure 4.6 is used to measure the flow rate of liquid propane at 10°C (ρ= 514.7 kg/m^3) through an 8 cm diameter vertical pipe. For a discharge coefficient of 0.98, ***determine*** the volume flow rate of propane through the pipe.

Solution:

The diameter ratio and the throat area of the meter are

$$\beta = d/D = 5/8 = 0.625$$

$$A_0 = \pi d^2/4 = \pi(0.05\,\text{m})^2/4 = 0.001963\,\text{m}^2$$

Noting that ΔP = 7 kPa = 7000 N/m^2, the flow rate becomes

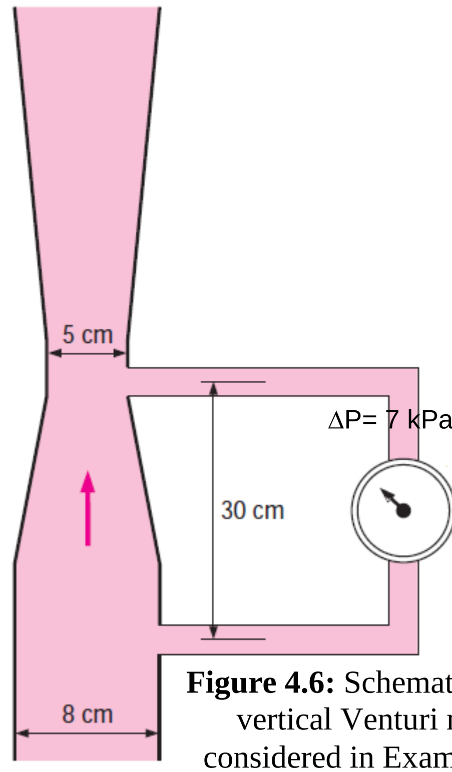


Figure 4.6: Schematic for the vertical Venturi meter considered in Example 4.2.

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (0.001963\,\text{m}^2)(0.98)\sqrt{\frac{2\times 7000\,\text{N/m}^2}{(514.7\,\text{kg/m}^3)((1-0.625^4)}\left(\frac{1\,\text{kg}\cdot\text{m/s}^2}{1\,\text{N}}\right)}$$

$$= \mathbf{0.0109\ m^3/s}$$

which is equivalent to 10.9 L/s. Also, the average flow velocity in the pipe is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.0109\,\text{m}^3/\text{s}}{\pi(0.08\,\text{m})^2/4} = 2.17\,\text{m/s}$$

Example 4.3:

The flow rate of water at 20°C (ρ = 998 kg/m^3 and μ = 1.002×10^{-3} kg/ms) through a 4 cm diameter pipe is measured with a 2 cm diameter nozzle meter equipped with an inverted air–water manometer as shown in Figure 4.7. If the manometer indicates a differential water height of 32 cm, ***determine*** the volume flow rate of water and the head loss caused by the nozzle meter. The percent pressure loss for nozzle meters is given for β = 0.5 to be 62%.

Solution:

The diameter ratio and the throat area of the meter are

$\beta = d / D = 2/4 = 0.50$

$A_0 = \pi d^2 / 4 = \pi(0.02\text{ m})^2 / 4 = 3.142\times 10^{-4}\text{ m}^2$

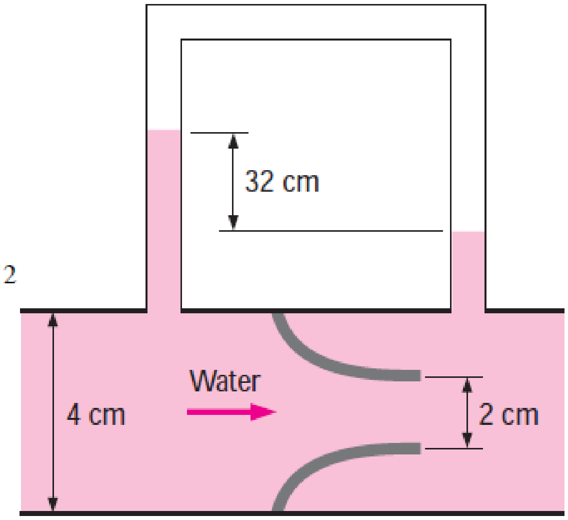


Figure 4.7: Schematic for the air–water manometer considered in Example 4.3.

the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= A_o C_d \sqrt{\frac{2\rho_w g h}{\rho_w(1-\beta^4)}} = A_o C_d \sqrt{\frac{2gh}{1-\beta^4}}$$

$$= (3.142\times 10^{-4}\text{ m}^2)(0.96)\sqrt{\frac{2(9.81\text{ m/s}^2)(0.32\text{ m})}{1-0.50^4}}$$

$$= \mathbf{0.781\times 10^{-3}\ m^3/s}$$

which is equivalent to 0.781 L/s. The average flow velocity in the pipe is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.781\times 10^{-3}\text{ m}^3/\text{s}}{\pi(0.04\text{ m})^2/4} = 0.621\text{ m/s}$$

The percent pressure (or head) loss for nozzle meters is given for β = 0.5 to be 62%. Therefore, h_L = (Permanent loss fraction), (Total head loss) = 0.62 (0.32m) = 0.20 mH_2O. The head loss between the two measurement sections can be determined from the energy equation, which simplifies to ($z_1 = z_2$)

$$h_L = \frac{P_1 - P_2}{\rho_f g} - \frac{V_2^2 - V_1^2}{2g} = h_w - \frac{\left[(D/d)^4 - 1\right]V_1^2}{2g}$$

$$= 0.32\text{ m} - \frac{\left[(4/2)^4 - 1\right](0.621\text{ m/s})^2}{2\left(9.81\text{ m/s}^2\right)} = 0.025\text{ m H}_2\text{O}$$

University of Anbar
College of Engineering
Mechanical Engineering Dept.

Fluid Mechanics-II (ME 2305)

Handout Lectures for Year Two
Chapter One/ Turbulent Flow in pipes

Course Tutor

Assist. Prof. Dr. Waleed M. Abed

Ramadi 2021

Chapter One
Turbulent Flow in Pipes

1.1. TURBULENT FLOW IN PIPES

Most flows encountered in engineering practice are turbulent, and thus it is important to understand how turbulence affects wall shear stress. However, turbulent flow is a complex mechanism dominated by fluctuations, and despite tremendous amounts of work done in this area by researchers, the theory of turbulent flow remains largely undeveloped. Therefore, we must rely on experiments and the empirical or semi-empirical correlations developed for various situations.

Turbulent flow is characterized by random and a rapid fluctuation of swirling regions of fluid, called eddies**,** throughout the flow. These fluctuations provide an additional mechanism for momentum and energy transfer. In laminar flow, fluid particles flow in an orderly manner along pathlines, and momentum and energy are transferred across streamlines by molecular diffusion. In turbulent flow, the swirling eddies transport mass, momentum, and energy to other regions of flow much more rapidly than molecular diffusion, greatly enhancing mass, momentum, and heat transfer. As a result, turbulent flow is associated with much higher values of friction, heat transfer, and mass transfer coefficients (see Figure 1.1).

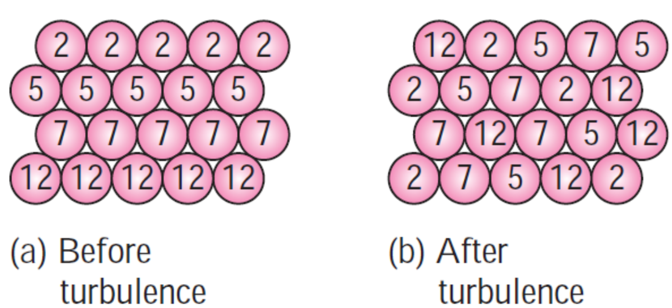


Figure 1.1: The intense mixing in turbulent flow brings fluid particles at different momentums into close contact and thus enhances momentum transfer.

Even when the average flow is steady, the eddy motion in turbulent flow causes significant fluctuations in the values of velocity, temperature, pressure, and even density (in compressible flow). Figure 1.2 shows the variation of the instantaneous velocity component u with time at a specified location, as can be measured with a hot-wire anemometer probe or other sensitive device. We observe that the instantaneous values of the velocity fluctuate about an average value, which suggests that the velocity can be expressed as the sum of an average value $\bar{u}$ and a fluctuating component u',

$$u = \bar{u} + u' \qquad \text{.......1.1}$$

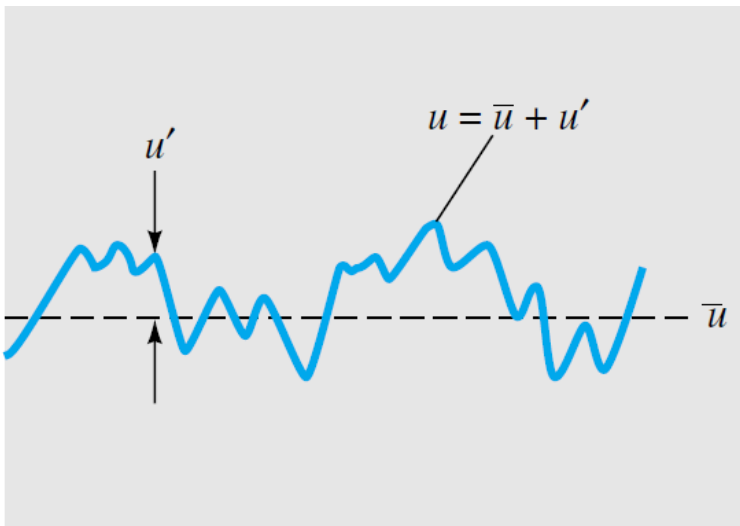


Figure 1.2: Fluctuations of the velocity component u with time at a specified location in turbulent flow.

1.2. Turbulent Shear Stress

It is convenient to think of the turbulent shear stress as consisting of two parts: the ***laminar component***, which accounts for the friction between layers in the flow direction (expressed as $\tau_{lam} = -\mu \frac{d\bar{u}}{dr}$), and the ***turbulent component***, which accounts for the friction between the fluctuating fluid particles and the fluid body (denoted as τ_{turb}) and is related to the fluctuation components of velocity). Then the total shear stress in turbulent flow can be expressed as

$$\tau_{total} = \tau_{lam} + \tau_{turb} \quad \ldots\ldots 1.2$$

The typical average velocity profile and relative magnitudes of laminar and turbulent components of shear stress for turbulent flow in a pipe are given in Figure 3.1.

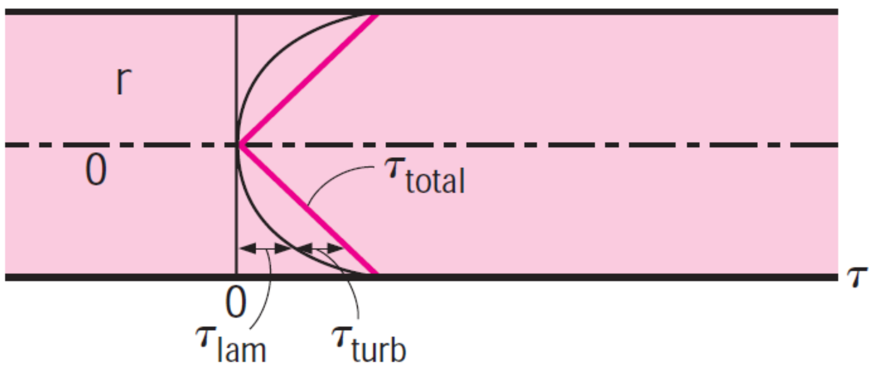


Figure 1.3: The velocity profile and the variation of shear stress with radial distance for turbulent flow in a pipe.

In many of the simpler turbulence models, turbulent shear stress is expressed in an analogous manner as suggested by the French mathematician *Joseph Boussinesq* (1842–1929) in 1877 as

$$\tau_{turb} = -\mu_t \frac{d\bar{u}}{dr} \quad \text{or} \quad \tau_{turb} = -\mu_t \frac{d\bar{u}}{dy} \quad \ldots\ldots 1.3$$

where μ_t is the eddy viscosity or turbulent viscosity, which accounts for momentum transport by turbulent eddies. Then the total shear stress can be expressed conveniently as

$$\tau_{total} = \tau_{lam} + \tau_{turb} = \mu \frac{d\bar{u}}{dy} + \mu_t \frac{d\bar{u}}{dy} = (\mu + \mu_t) \frac{d\bar{u}}{dy} \quad \ldots\ldots 1.4$$

$$\tau_{total} = \rho(\nu + \nu_t) \frac{d\bar{u}}{dy} \quad \ldots\ldots 1.5$$

where $\nu_t = {}^{\mu_t}/_{\rho}$ is the ***kinematic eddy viscosity*** or ***kinematic turbulent viscosity***

(also called the ***eddy diffusivity of momentum***). The concept of eddy viscosity is very appealing, but it is of no practical use unless its value can be determined. In other words, eddy viscosity must be modeled as a function of the average flow variables; we call this *eddy viscosity closure*. For example, in the early 1900s, the German engineer *L. Prandtl* introduced the concept of ***mixing length*** (l_m), which is related to the average size of the eddies that are primarily responsible for mixing, and expressed the turbulent shear stress as

$$\tau_{turb} = -\mu_t \frac{d\bar{u}}{dy} = \rho {l_m}^2 \left(\frac{d\bar{u}}{dy}\right)^2 \quad \text{.........1.6}$$

1.3. Turbulent Velocity Profile

Unlike laminar flow, the expressions for the velocity profile in a turbulent flow are based on both analysis and measurements, and thus they are semi-empirical in nature with constants determined from experimental data. Consider fully-developed turbulent flow in a pipe, and let *u* denote the time-averaged velocity in the axial direction.

Typical velocity profiles for fully developed laminar and turbulent flows are given in Figure 1.4. Note that the velocity profile is parabolic in laminar flow but is much fuller in turbulent flow, with a sharp drop near the pipe wall. Turbulent flow along a wall can be considered to consist of four regions, characterized by the distance from the wall. The very thin layer next to the wall where viscous effects are dominant is the ***viscous*** (or ***laminar*** or ***linear*** or ***wall***) sublayer. The velocity profile in this layer is very nearly *linear*, and the flow is streamlined. Next to the viscous sublayer is the ***buffer layer,*** in which turbulent effects are becoming significant, but the flow is still dominated by viscous effects. Above the buffer layer is the ***overlap*** (or ***transition***) **l*ayer,*** also called the ***inertial sublayer,*** in which the turbulent effects are much more significant, but still not dominant. Above that is the ***outer*** (or ***turbulent***) ***layer*** in the remaining part of the flow in which turbulent effects dominate over molecular diffusion (viscous) effects.

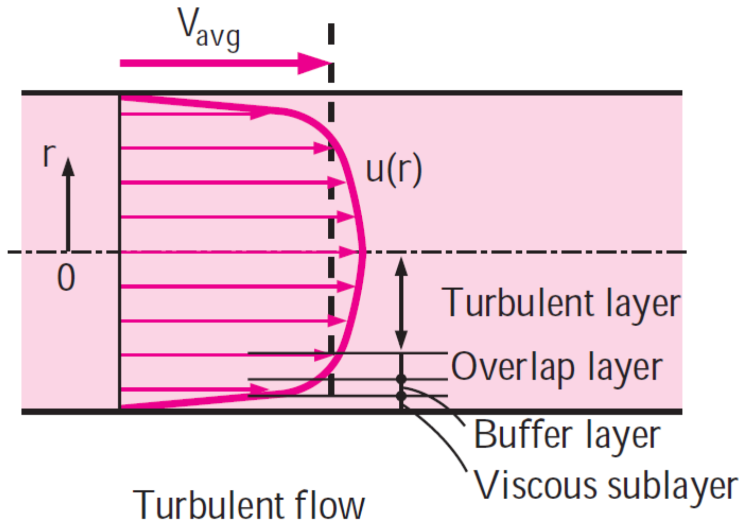


Figure 1.4: The velocity profile in fully developed pipe flow is parabolic in laminar flow, but much fuller in turbulent flow.

Then the velocity gradient in the viscous sublayer remains nearly constant at du/dy= u/y, and the wall shear stress can be expressed as

$$\tau_w = \mu \frac{u}{y} = \rho\nu \frac{u}{y} \qquad \text{or} \qquad \frac{\tau_w}{\rho} = \frac{\nu u}{y} \qquad \text{.....1.7}$$

where y is the distance from the wall (note that $y = R - r$ for a circular pipe). The quantity τ_w/ρ is frequently encountered in the analysis of turbulent velocity profiles. The square root of τ_w/ρ has the dimensions of velocity, and thus it is convenient to view it as a fictitious velocity called the ***friction velocity*** expressed as $u^* = \sqrt{\tau_w/\rho}$. Substituting this into Eq. 1.7, the velocity profile in the viscous sublayer can be expressed in dimensionless form as

Viscous sublayer: $\frac{u}{u^*} = \frac{yu^*}{\nu}$

This equation is known as the law of the wall, and it is found to satisfactorily correlate with experimental data for smooth surfaces for $0 \leq \frac{yu^*}{\nu} \leq 5$. Therefore, the thickness of the viscous sublayer is roughly

Thickness of viscous sublayer: $$y = \delta_{sublayer} = \frac{5\nu}{u_*} = \frac{25\nu}{u_\delta}$$

where u_σ is the flow velocity at the edge of the viscous sublayer, which is closely related to the average velocity in a pipe. The quantity $\frac{\nu}{u^*}$ has dimensions of length and is called the viscous length; it is used to nondimensionalize the distance y from the surface. In boundary layer analysis, it is convenient to work with nondimensionalized distance and nondimensionalized velocity defined as

Nondimensionalized variables: $$y^+ = \frac{yu_*}{\nu} \quad \text{and} \quad u^+ = \frac{u}{u_*}$$

Note that the friction velocity u^* is used to nondimensionalize both y and u, and y^+ resembles the Reynolds number expression.

Dimensional analysis indicates and the experiments confirm that the velocity in the overlap layer is proportional to the logarithm of distance, and the velocity profile can be expressed as

The logarithmic law: $$\frac{u}{u_*} = \frac{1}{\kappa} \ln \frac{yu_*}{\nu} + B \quad \text{……1.8}$$

where k and B are constants whose values are determined experimentally to be about 0.40 and 5.0, respectively. Equation 1.8 is known as the logarithmic law. Substituting the values of the constants, the velocity profile is determined to be

Overlap layer: $$\frac{u}{u_*} = 2.5 \ln \frac{yu_*}{\nu} + 5.0 \quad \text{or} \quad u^+ = 2.5 \ln y^+ + 5.0$$

A good approximation for the outer turbulent layer of pipe flow can be obtained by evaluating the constant *B* in Eq. 1.8 from the requirement that maximum velocity in a pipe occurs at the centerline where *r*= 0. Solving for *B* from Eq. 1.8 by setting $y = R$ & $r = R$ and $u = u_{max}$, and substituting it back into Eq. 1.8 together with $k = 0.4$ gives

Outer turbulent layer: $$\frac{u_{max} - u}{u_*} = 2.5 \ln \frac{R}{R - r}$$

The deviation of velocity from the centerline value $u_{max} - u$ is called the ***velocity defect***, and the above equation is called the ***velocity defect law***.

1.4. The Moody Chart

The friction factor in fully developed turbulent pipe flow depends on the Reynolds number and the ***relative roughness*** (ε/*D*), which is the ratio of the mean height of roughness of the pipe, to the pipe diameter. The functional form of this dependence cannot be obtained from a theoretical analysis, and all available results are obtained from painstaking experiments using artificially roughened surfaces (usually by gluing sand grains of a known size on the inner surfaces of the pipes). Most such experiments were conducted by *Prandtl's student J. Nikuradse* in 1933, followed by the works of others. The friction factor was calculated from the measurements of the flow rate and the pressure drop.

The experimental results obtained are presented in tabular, graphical, and functional forms obtained by curve-fitting experimental data. In 1939, *Cyril F. Colebrook* (1910–1997) combined the available data for transition and turbulent flow in smooth as well as rough pipes into the following implicit relation known as the ***Colebrook equation***:

Turbulent flow: $$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \quad .....1.9$$

We note that the logarithm in Eq. 1.9 is a base 10 rather than a natural logarithm. In 1942, the American engineer *Hunter Rouse* (1906–1996) verified *Colebrook's equation* and produced a graphical plot of f as a function of *Re* and the product $Re\sqrt{f}$. He also presented the laminar flow relation and a table of commercial pipe roughness. Two years later, ***Lewis F. Moody*** (1880–1953) redrew Rouse's diagram into the form commonly used today. The now famous ***Moody chart*** is given in the appendix as Figure 1.5. It presents the Darcy friction factor for pipe flow as a function of the Reynolds number and ε/*D* over a wide range. It is probably one of the most widely accepted and used charts in engineering. Although it is developed for circular pipes, it can also be used for noncircular pipes by replacing the diameter by the hydraulic diameter. An approximate explicit relation for f was given by *S. E. Haaland* in 1983 as

$$\frac{1}{\sqrt{f}} \cong -1.8 \log\left[\frac{6.9}{Re} + \left(\frac{\varepsilon/D}{3.7}\right)^{1.11}\right] \qquad \text{…….. 1.10}$$

The results obtained from this relation are within 2% of those obtained from the ***Colebrook equation***. Equivalent roughness values for some commercial pipes are given in Table 1.1 as well as on the Moody chart.

Table 1.1: Equivalent roughness values for new commercial pipes.

Material	Roughness, ε ft	Roughness, ε mm
Glass, plastic	0 (smooth)	
Concrete	0.003–0.03	0.9–9
Wood stave	0.0016	0.5
Rubber, smoothed	0.000033	0.01
Copper or brass tubing	0.000005	0.0015
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Wrought iron	0.00015	0.046
Stainless steel	0.000007	0.002
Commercial steel	0.00015	0.045

We make the following observations from the Moody chart:

- ✓ For laminar flow, the friction factor decreases with increasing Reynolds number, and it is independent of surface roughness.
- ✓ The friction factor is a minimum for a smooth pipe (but still not zero because of the no-slip condition) and increases with roughness. The *Colebrook equation* in this case ($\varepsilon = 0$) reduces to the *Prandtl equation* expressed as $1/\sqrt{f} = 2.0 \log(Re\sqrt{f}) - 0.8$
- ✓ The transition region from the laminar to turbulent regime (2300 < *Re* < 4000) is indicated by the shaded area in the Moody chart. The flow in this region may be laminar or turbulent, depending on flow disturbances, or it may alternate between laminar and turbulent, and thus the friction factor may also alternate between the values for laminar and turbulent flow. The data in this range are the least reliable. At small relative roughnesses, the friction factor increases in the transition region and approaches the value for smooth pipes.
- ✓ At very large Reynolds numbers (to the right of the dashed line on the chart) the friction factor curves corresponding to specified relative roughness curves are nearly horizontal, and thus the friction factors are independent of the Reynolds number. The flow in that region is called *fully rough turbulent flow* or just *fully rough flow* because the thickness of the viscous sublayer decreases with increasing Reynolds number, and it becomes so thin that it is negligibly small compared to the surface roughness height. The viscous effects in this case are produced in the main flow primarily by the protruding roughness elements, and the contribution of the laminar sublayer is negligible. The Colebrook equation in the *fully rough* zone ($Re \rightarrow \infty$) reduces to the ***von Kármán equation*** expressed as which is explicit in *f*.

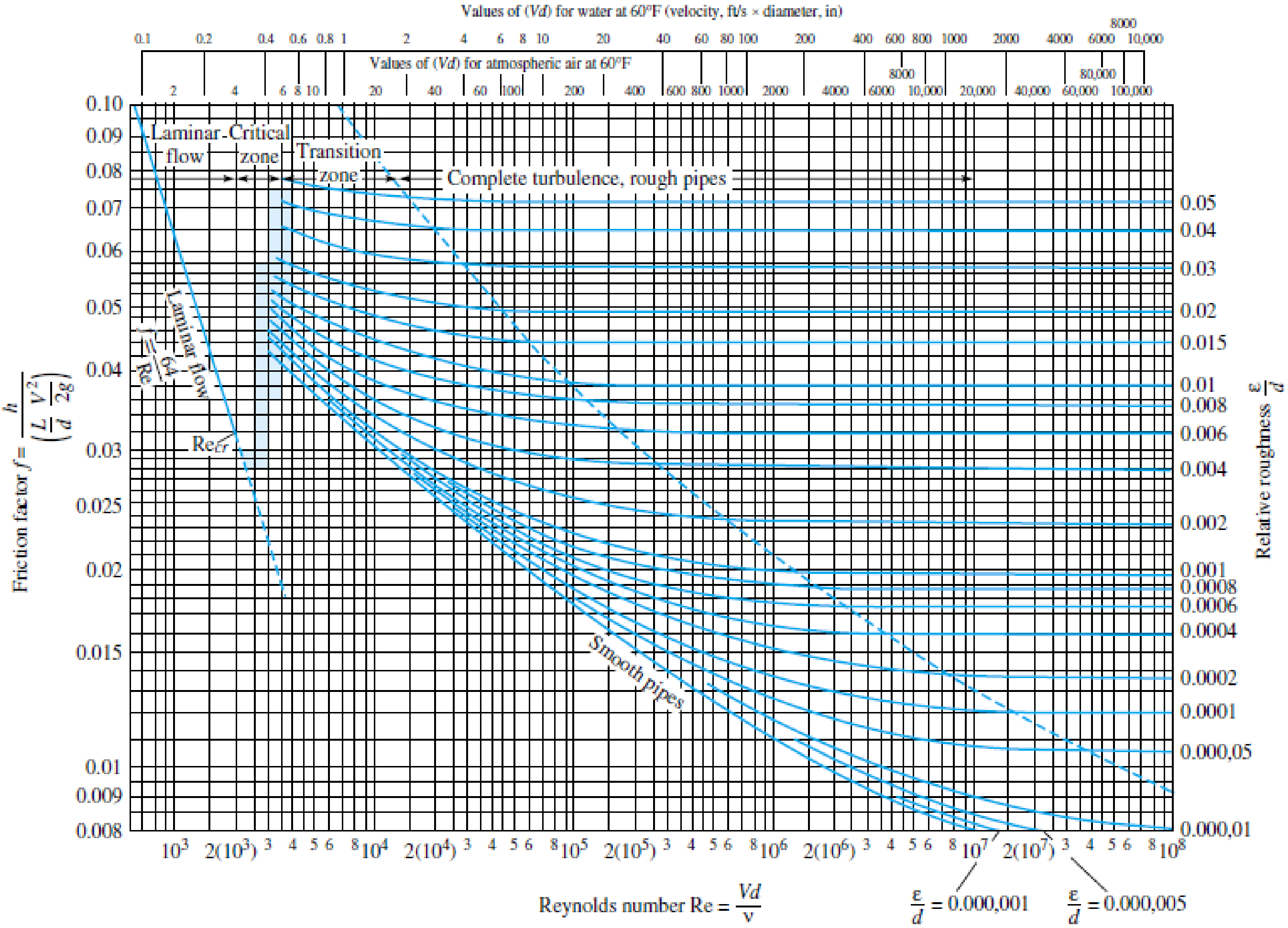


Figure 1.5: The Moody chart for the friction factor for fully developed flow.

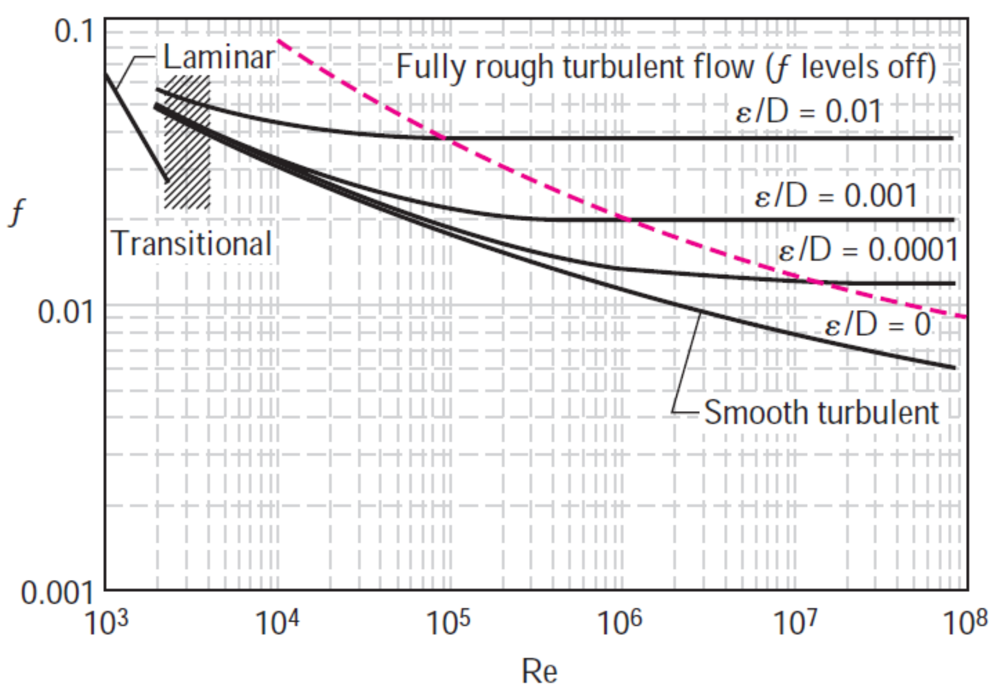


Figure 1.6: At very large Reynolds numbers, the friction factor curves on the Moody chart are nearly horizontal, and thus the friction factors are independent of the Reynolds number.

1.5. Types of Fluid Flow Problems

In the design and analysis of piping systems that involve the use of the Moody chart (or the *Colebrook equation*), we usually encounter three types of problems (the fluid and the roughness of the pipe are assumed to be specified in all cases).

1. Determining the **pressure drop** (or head loss) when the pipe length and diameter are given for a specified flow rate (or velocity)
2. Determining the **flow rate** when the pipe length and diameter are given for a specified pressure drop (or head loss)
3. Determining the **pipe diameter** when the pipe length and flow rate are given for a specified pressure drop (or head loss)

Problems of the *first type* are straightforward and can be solved directly by using the Moody chart. Problems of the *second type* and *third type* are commonly encountered in engineering design (in the selection of pipe diameter, for example, that minimizes the sum of the construction and pumping costs), but the use of the Moody chart with such problems requires an iterative approach unless an equation solver is used.

In problems of the *second type*, the diameter is given but the flow rate is unknown. A good guess for the friction factor in that case is obtained from the completely turbulent flow region for the given roughness. This is true for large Reynolds numbers, which is often the case in practice. Once the flow rate is obtained, the friction factor can be corrected using the Moody chart or the Colebrook equation, and the process is repeated until the solution converges. (Typically only a few iterations are required for convergence to three or four digits of precision.)

In problems of the *third type*, the diameter is not known and thus the Reynolds number and the relative roughness cannot be calculated. Therefore, we start calculations by assuming a pipe diameter. The pressure drop calculated for the assumed diameter is then compared to the specified pressure drop, and calculations are repeated with another pipe diameter in an iterative fashion until convergence.

To avoid tedious iterations in head loss, flow rate, and diameter calculations, *Swamee* and *Jain* proposed the following explicit relations in 1976 that are accurate to within 2% of the Moody chart:

$$h_L = 1.07\frac{\dot{V}^2 L}{gD^5}\left\{\ln\left[\frac{\varepsilon}{3.7D} + 4.62\left(\frac{\nu D}{\dot{V}}\right)^{0.9}\right]\right\}^{-2} \qquad \begin{matrix} 10^{-6} < \varepsilon/D < 10^{-2} \\ 3000 < Re < 3\times10^{8}\end{matrix}$$

$$\dot{V} = -0.965\left(\frac{gD^5 h_L}{L}\right)^{0.5}\ln\left[\frac{\varepsilon}{3.7D} + \left(\frac{3.17\nu^2 L}{gD^3 h_L}\right)^{0.5}\right] \qquad Re > 2000$$

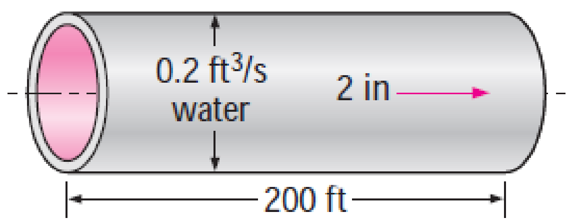
$$D = 0.66\left[\varepsilon^{1.25}\left(\frac{L\dot{V}^2}{gh_L}\right)^{4.75} + \nu\dot{V}^{9.4}\left(\frac{L}{gh_L}\right)^{5.2}\right]^{0.04} \qquad \begin{matrix} 10^{-6} < \varepsilon/D < 10^{-2} \\ 5000 < Re < 3\times10^{8}\end{matrix}$$

Examples:

Example 1:

Water (ρ= 62.36 lbm/ft^3 and μ= 7.536×10^{-4} lbm/ft·s) is flowing steadily in a 2 in diameter horizontal pipe made of stainless steel at a rate of 0.2 ft^3/s (see Figure below). Determine the pressure drop, the head loss, and the required pumping power input for flow over a 200 ft long section of the pipe.

Solution: We recognize this as a problem of the first type, since flow rate, pipe length, and pipe diameter are known. First we calculate the average velocity and the Reynolds number to determine the flow regime:



$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.2\ \text{ft}^3/\text{s}}{\pi(2/12\ \text{ft})^2/4} = 9.17\ \text{ft/s}$$

$$Re = \frac{\rho V D}{\mu} = \frac{(62.36\ \text{lbm/ft}^3)(9.17\ \text{ft/s})(2/12\ \text{ft})}{7.536\times10^{-4}\ \text{lbm/ft}\cdot\text{s}} = 126{,}400$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is calculated using Table1.1.

$$\varepsilon/D = \frac{0.000007\ \text{ft}}{2/12\ \text{ft}} = 0.000042$$

The friction factor corresponding to this relative roughness and the Reynolds number can simply be determined from the *Moody chart*. To avoid any reading error, we determine *f* from the *Colebrook equation*:

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{0.000042}{3.7} + \frac{2.51}{126{,}400\sqrt{f}}\right)$$

Using an equation solver or an iterative scheme, the friction factor is determined to be f = 0.0174. Then the pressure drop (which is equivalent to pressure loss in this case), head loss, and the required power input become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0174\frac{200\text{ ft}}{2/12\text{ ft}}\frac{(62.36\text{ lbm/ft}^3)(9.17\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 1700\text{ lbf/ft}^2 = 11.8\text{ psi}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.0174\frac{200\text{ ft}}{2/12\text{ ft}}\frac{(9.17\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)} = 27.3\text{ ft}$$

$$\dot{W}_{pump} = \dot{V}\,\Delta P = (0.2\text{ ft}^3\text{/s})(1700\text{ lbf/ft}^2)\left(\frac{1\text{ W}}{0.737\text{ lbf}\cdot\text{ft/s}}\right) = 461\text{ W}$$

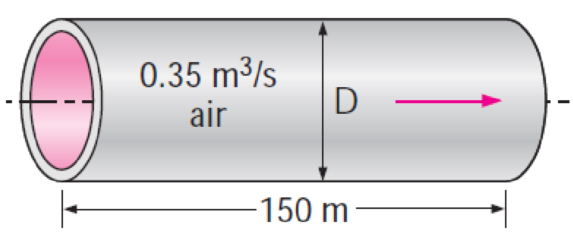
Example 2:

Heated air at 35°C is to be transported in a 150 m long circular plastic duct at a rate of 0.35 m^3/s (see Figure below). If the head loss in the pipe is not to exceed 20 m, determine the minimum diameter of the duct.

Solution:

The density, dynamic viscosity and kinematic viscosity of air at 35°C are ρ = 1.145 kg/m^3, μ= 1.895×10^{-5} kg/m·s, and ν= 1.655×10^{-5} m^2/s.

the friction factor, and the head loss relations can be expressed as (*D* is in m, *V* is in m/s, and *Re* and *f* are dimensionless)



$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.35\ m^3/s}{\pi D^2/4}$$

$$Re = \frac{VD}{\nu} = \frac{VD}{1.655 \times 10^{-5}\ m^2/s}$$

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) = -2.0\log\left(\frac{2.51}{Re\sqrt{f}}\right)$$

$$h_L = f\frac{L}{D}\frac{V^2}{2g} \quad \rightarrow \quad 20 = f\frac{150\ m}{D}\frac{V^2}{2(9.81\ m/s^2)}$$

The roughness is approximately zero for a plastic pipe (Table 1.1). Therefore, this is a set of four equations in four unknowns, and solving them with an equation solver such as EES gives

$D = \mathbf{0.267\ m}$, $f = 0.0180$, $V = 6.24\ m/s$, and $Re = 100{,}800$

Therefore, the diameter of the duct should be more than 26.7 cm if the head loss is not to exceed 20 m. Note that $Re > 4000$, and thus the turbulent flow assumption is verified.

The diameter can also be determined directly from the third *Swamee–Jain* formula to be

$$D = 0.66\left[\varepsilon^{1.25}\left(\frac{L\dot{V}^2}{gh_L}\right)^{4.75} + \nu\dot{V}^{9.4}\left(\frac{L}{gh_L}\right)^{5.2}\right]^{0.04}$$

$$= 0.66\left[0 + (1.655 \times 10^{-5}\ m^2/s)(0.35\ m^3/s)^{9.4}\left(\frac{150\ m}{(9.81\ m/s^2)(20\ m)}\right)^{5.2}\right]^{0.04}$$

$$= \mathbf{0.271\ m}$$

Example:

Liquid ammonia at -20°C is flowing through a 30 m long section of a 5 mm diameter copper tube at a rate of 0.15 kg/s. Determine the pressure drop, the head loss, and the pumping power required to overcome the frictional losses in the tube.

Solution:

The density and dynamic viscosity of liquid ammonia at -20°C are ρ= 665.1 kg/m^3 and μ= 2.361×10^{-4} kg/m.s. The roughness of copper tubing is 1.5×10^{-6} m.

First we calculate the average velocity and the Reynolds number to determine the flow regime:

$$V = \frac{\dot{m}}{\rho A_c} = \frac{\dot{m}}{\rho(\pi D^2/4)} = \frac{0.15\ \text{kg/s}}{(665.1\ \text{kg/m}^3)[\pi(0.005\ \text{m})^2/4]} = 11.49\ \text{m/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(665.1\ \text{kg/m}^3)(11.49\ \text{m/s})(0.005\ \text{m})}{2.361\times10^{-4}\ \text{kg/m}\cdot\text{s}} = 1.618\times10^5$$

which is greater than *Re* > 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon / D = \frac{1.5\times10^{-6}\ \text{m}}{0.005\ \text{m}} = 3\times10^{-4}$$

The friction factor can be determined from the *Moody chart*, but to avoid the reading error, we determine it from the *Colebrook equation* using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{3\times10^{-4}}{3.7} + \frac{2.51}{1.618\times10^5\sqrt{f}}\right)$$

It gives f = 0.01819. Then the pressure drop, the head loss, and the useful pumping power required become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2}$$

$$= 0.01819\frac{30\ \text{m}}{0.005\ \text{m}}\frac{(665.1\ \text{kg/m}^3)(11.49\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = 4792\ \text{kPa}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.01819\frac{30\ \text{m}}{0.005\ \text{m}}\frac{(11.49\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = \mathbf{734\ m}$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = \frac{\dot{m}\Delta P}{\rho} = \frac{(0.15\ \text{kg/s})(4792\ \text{kPa})}{665.1\ \text{kg/m}^3}\left(\frac{1\ \text{kW}}{1\ \text{kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{1.08\ kW}$$

University of Anbar
College of Engineering
Mechanical Engineering Dept.

Fluid Mechanics-II
(ME 2305)

Handout Lectures for Year Two
Chapter Three/ Piping networks and pump selection

Course Tutor

Assist. Prof. Dr. Waleed M. Abed

Ramadi 2021

Chapter Three
Piping Networks and Pump Selection

3.1. Introduction

Most piping systems encountered in practice such as the water distribution systems in cities or commercial or residential establishments involve numerous parallel and series connections as well as several sources (supply of fluid into the system) and loads (discharges of fluid from the system) (see Figure 3-1). A piping project may involve the design of a new system or the expansion of an existing system. The engineering objective in such projects is to design a piping system that will deliver the specified flow rates at specified pressures reliably at minimum total (initial plus operating and maintenance) cost. Once the layout of the system is prepared, the determination of the pipe diameters and the pressures throughout the system, while remaining within the budget constraints, typically requires solving the system repeatedly until the optimal solution is reached. Computer modeling and analysis of such systems make this tedious task a simple chore.

Figure 3.1: A piping network in an industrial facility.

Piping systems typically involve several pipes connected to each other in series and/or in parallel, as shown in Figure 3.2. When the pipes are connected ***in series***, the flow rate through the entire system remains constant regardless of the diameters of the individual pipes in the system.

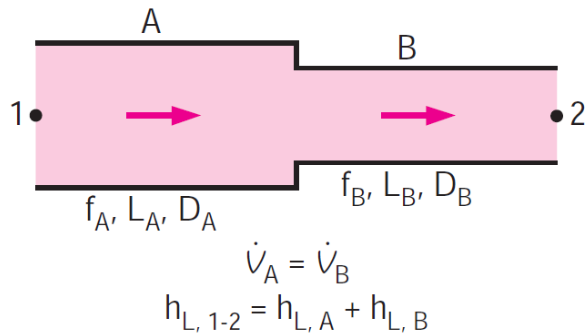


Figure 3.2: For pipes ***in series***, the flow rate is the same in each pipe, and the total head loss is the sum of the head losses in individual pipes.

For a pipe that branches out into two (or more) ***parallel pipes*** and then rejoins at a junction downstream, the total flow rate is the sum of the flow rates in the individual pipes, as shown in Figure 3.3. The pressure drop (or head loss) in each individual pipe connected in parallel must be the same since $\Delta P = P_A - P_B$ and the junction pressures P_A and P_B are the same for all the individual pipes. For a system of two parallel pipes 1 and 2 between junctions A and B with negligible minor losses, this can be expressed as

$$h_{L,1} = h_{L,2} \quad \rightarrow \quad f_1 \frac{L_1}{D_1}\frac{V_1^2}{2g} = f_2 \frac{L_2}{D_2}\frac{V_2^2}{2g} \qquad (3.1)$$

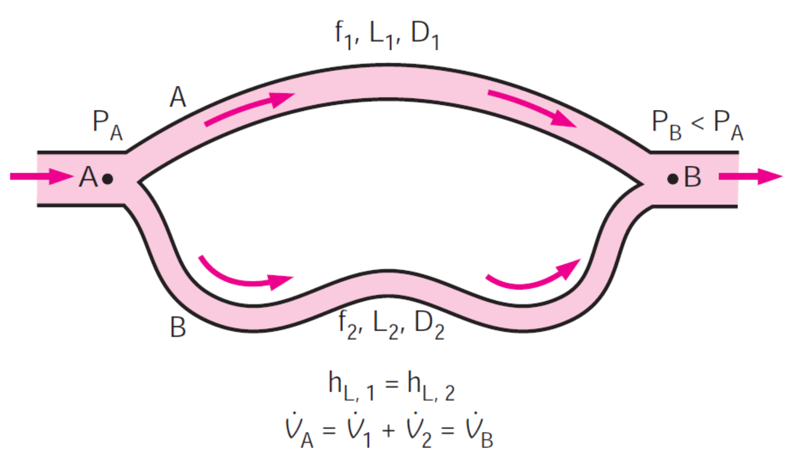


Figure 3.3: For pipes ***in parallel***, the head loss is the same in each pipe, and the total flow rate is the sum of the flow rates in individual pipes.

Then the ratio of the average velocities and the flow rates in the two parallel pipes become

$$\frac{V_1}{V_2} = \left(\frac{f_2}{f_1}\frac{L_2}{L_1}\frac{D_1}{D_2}\right)^{1/2} \quad \text{and} \quad \frac{\dot{V}_1}{\dot{V}_2} = \frac{A_{c,1}V_1}{A_{c,2}V_2} = \frac{D_1^2}{D_2^2}\left(\frac{f_2}{f_1}\frac{L_2}{L_1}\frac{D_1}{D_2}\right)^{1/2} \tag{3.2}$$

Therefore, the relative flow rates in parallel pipes are established from the requirement that the head loss in each pipe be the same. This result can be extended to any number of pipes connected in parallel.

The analysis of piping networks, no matter how complex they are, is based on two simple principles:

1. Conservation of mass throughout the system must be satisfied. This is done by requiring the total flow into a junction to be equal to the total flow out of the junction for all junctions in the system. Also, the flow rate must remain constant in pipes connected in series regardless of the changes in diameters.

2. Pressure drop (and thus head loss) between two junctions must be the same for all paths between the two junctions. This is because pressure is a point function and it cannot have two values at a specified point. In practice this rule is used by requiring that the algebraic sum of head losses in a loop (for all loops) be equal to zero. (A head loss is taken to be positive for flow in the clockwise direction and negative for flow in the counterclockwise direction.)

3.2. Piping Systems with Pumps and Turbines

When a piping system involves a pump and/or turbine, the steady-flow energy equation (*Bernoulli equation*) on a unit-mass basis can be expressed as

$$\frac{P_1}{\rho} + \alpha_1\frac{V_1^2}{2} + gz_1 + w_{pump,\,u} = \frac{P_2}{\rho} + \alpha_2\frac{V_2^2}{2} + gz_2 + w_{turbine,\,e} + gh_L \tag{3.3}$$

It can also be expressed in terms of ***heads*** as

$$\frac{P_1}{\rho g} + \alpha_1\frac{V_1^2}{2g} + z_1 + h_{pump,\,u} = \frac{P_2}{\rho g} + \alpha_2\frac{V_2^2}{2g} + z_2 + h_{turbine,\,e} + h_L \tag{3.4}$$

where $h_{pump,u} = W_{pump,u} /g$ is the useful pump head delivered to the fluid, $h_{turbine,e} = W_{pump,e} /g$ is the turbine head extracted from the fluid, α is the kinetic energy correction factor whose value is nearly 1 for most (turbulent) flows encountered in practice, and h_L is the total head loss in piping (including the minor losses if they are significant) between points 1 and 2. The pump head is zero if the piping system does not involve a pump or a fan, the turbine head is zero if the system does not involve a turbine, and both are zero if the system does not involve any mechanical work-producing or work-consuming devices.

Many practical piping systems involve a pump to move a fluid from one reservoir to another. Taking points 1 and 2 to be at the free surfaces of the reservoirs, the energy equation in this case reduces for the *useful pump* head required to (Figure 3.4). Since the velocities at free surfaces are negligible and the pressures are at atmospheric pressure.

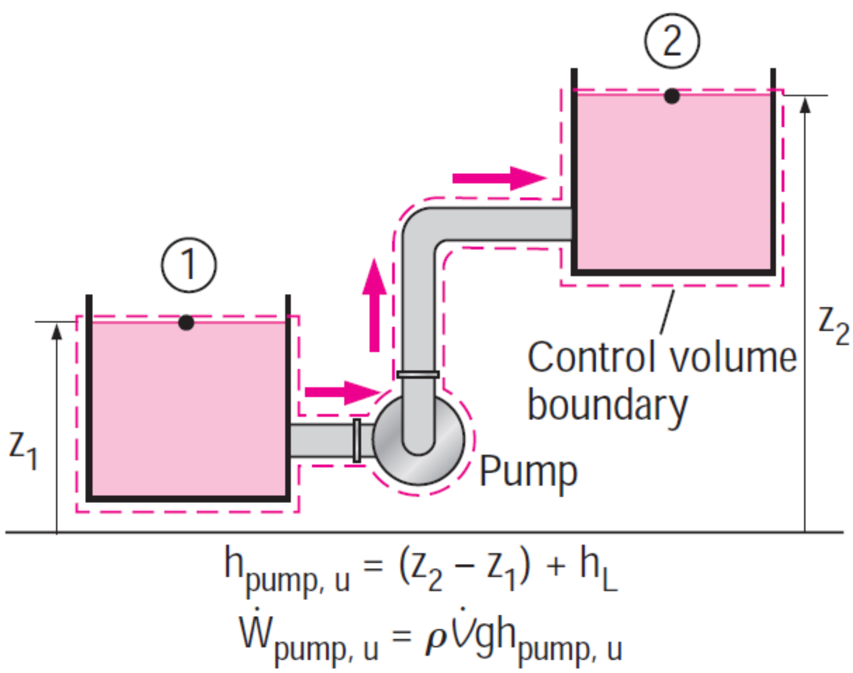


Figure 3.4: When a pump moves a fluid from one reservoir to another, the useful pump head requirement is equal to the elevation difference between the two reservoirs plus the head loss.

$$h_{pump,u} = (z_2 - z_1) + h_L \quad (3.5)$$

A similar argument can be given for the turbine head for a hydroelectric power plant by replacing $h_{pump,u}$ in Eq. 3.5 by $-h_{turbine,e}$.

3.3. The efficiency of the pump–motor combination

Once the useful pump head is known, the mechanical power that needs to be delivered by the pump to the fluid and the electric power consumed by the motor of the pump for a specified flow rate are determined from

$$\dot{W}_{pump,\ shaft} = \frac{\rho \dot{V} g h_{pump,\ u}}{\eta_{pump}} \quad \text{and} \quad \dot{W}_{elect} = \frac{\rho \dot{V} g h_{pump,\ u}}{\eta_{pump-motor}} \qquad (3.6)$$

where $\eta_{pump-motor}$ is the efficiency of the pump–motor combination, which is the product of the pump and the motor efficiencies. The pump–motor efficiency is defined as the ratio of the net mechanical energy delivered to the fluid by the pump to the electric energy consumed by the motor of the pump, and it usually ranges between 50 and 85 percent.

The head loss of a piping system increases (usually quadratically) with the flow rate. A plot of required useful pump head $h_{pump,u}$ as a function of flow rate is called the **system (**or **demand) curve.** The head produced by a pump is not a constant either. Both the pump head and the pump efficiency vary with the flow rate, and pump manufacturers supply this variation in tabular or graphical form, as shown in Figure 3.5. These experimentally determined $h_{pump,u}$ and $h_{pump,u}$ versus *V*.curves are called **characteristic (**or **supply** or **performance) curves.** Note that the flow rate of a pump increases as the required head decreases. The intersection point of the pump head curve with the vertical axis typically represents the *maximum head* the pump can provide, while the intersection point with the horizontal axis indicates the *maximum flow rate* (called the **free delivery**) that the pump can supply.

The *efficiency* of a pump is sufficiently high for a certain range of head and flow rate combination. Therefore, a pump that can supply the required head and flow rate is not necessarily a good choice for a piping system unless the efficiency of the pump at those conditions is sufficiently high. The pump installed in a piping system will operate at the point where the *system curve* and the *characteristic curve* intersect. This point of intersection is called the **operating point,** as shown in Figure 3.5. The useful head produced by the pump at this point matches the head requirements of the system at that flow rate. Also, the efficiency of the pump during operation is the value corresponding to that flow rate.

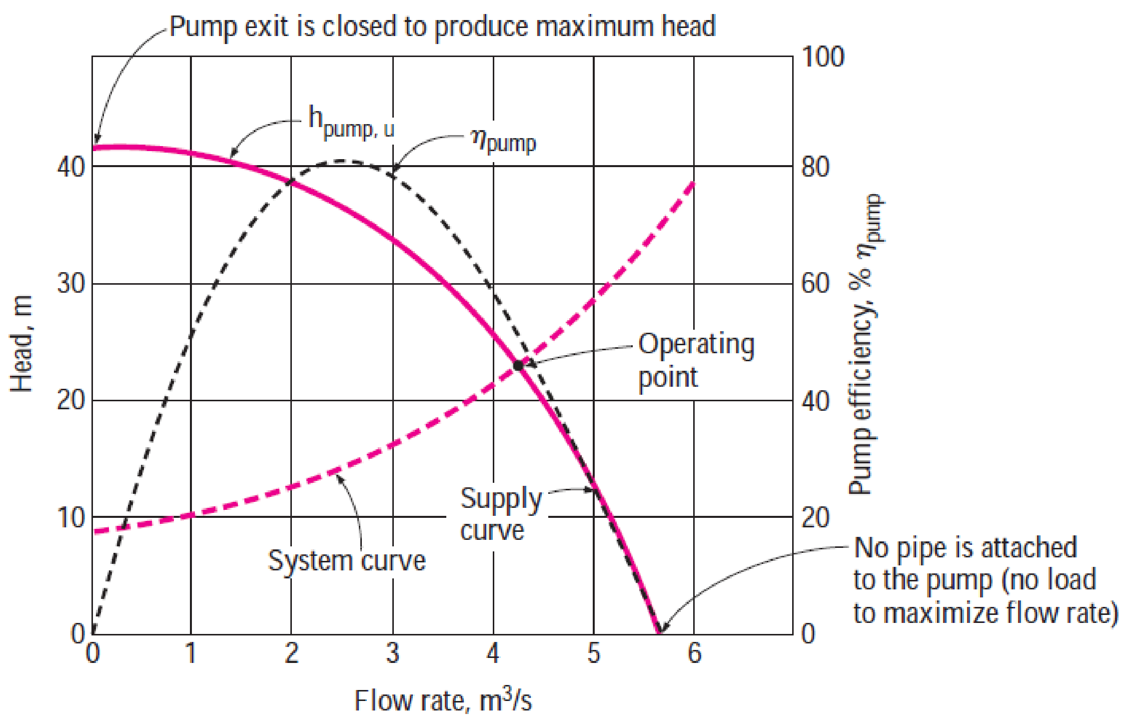


Figure 3.5: Characteristic pump curves for centrifugal pumps, the system curve for a piping system, and the operating point.

Example 3.1:

Water at 20°C is to be pumped from a reservoir (z_A = 5 m) to another reservoir at a higher elevation (z_B = 13 m) through two 36-m-long pipes connected in parallel, as shown in Figure 3.6. The pipes are made of commercial steel, and the diameters of the two pipes are 4 and 8 cm. Water is to be pumped by a 70% efficient motor–pump combination that draws 8 kW of electric power during operation. The minor losses and the head loss in pipes that connect the parallel pipes to the two reservoirs are considered to be negligible. Determine the total flow rate between the reservoirs and the flow rate through each of the parallel pipes.

<u>**Properties**</u> The density and dynamic viscosity of water at 20°C are ρ = 998 kg/m^3 and μ = 1.002×10^{-3} kg/m.s. The roughness of commercial steel pipe is ε = 0.000045 m.

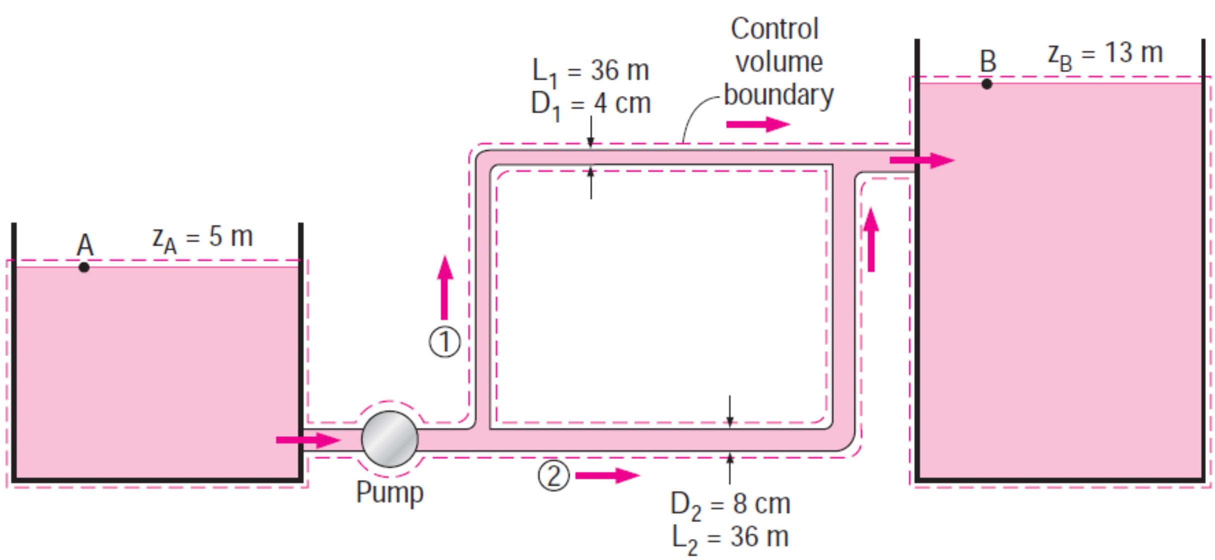


Figure 3.6: The piping system discussed in Example 3.1.

The useful head supplied by the pump to the fluid is determined from

$$\dot{W}_{elect} = \frac{\rho \dot{V} g h_{pump,\,u}}{\eta_{pump-motor}} \quad \rightarrow \quad 8000\text{ W} = \frac{(998\text{ kg/m}^3)\dot{V}(9.81\text{ m/s}^2)h_{pump,\,u}}{0.70}$$

We choose points A and B at the free surfaces of the two reservoirs. Noting that the fluid at both points is open to the atmosphere (and thus $P_A = P_B = P_{atm}$) and that the fluid velocities at both points are zero ($V_A = V_B = 0$), the energy equation for a control volume between these two points simplifies to

$$\frac{P_A}{\rho g} + \alpha_A \frac{V_A^2}{2g} + z_A + h_{pump,\,u} = \frac{P_B}{\rho g} + \alpha_B \frac{V_B^2}{2g} + z_B + h_L \rightarrow h_{pump,\,u} = (z_B - z_A) + h_L$$

$$h_{pump,\,u} = (13 - 5) + h_L$$

$$h_L = h_{L,\,1} = h_{L,\,2}$$

We designate the 4-cm-diameter pipe by 1 and the 8-cm-diameter pipe by 2. The average velocity, the Reynolds number, the friction factor, and the head loss in each pipe are expressed as

$$V_1 = \frac{\dot{V}_1}{A_{c,\,1}} = \frac{\dot{V}_1}{\pi D_1^2/4} \quad \rightarrow \quad V_1 = \frac{\dot{V}_1}{\pi (0.04\ \text{m})^2/4}$$

$$V_2 = \frac{\dot{V}_2}{A_{c,\,2}} = \frac{\dot{V}_2}{\pi D_2^2/4} \quad \rightarrow \quad V_2 = \frac{\dot{V}_2}{\pi (0.08\ \text{m})^2/4}$$

$$Re_1 = \frac{\rho V_1 D_1}{\mu} \quad \rightarrow \quad Re_1 = \frac{(998\ \text{kg/m}^3) V_1 (0.04\ \text{m})}{1.002 \times 10^{-3}\ \text{kg/m}\cdot\text{s}}$$

$$Re_2 = \frac{\rho V_2 D_2}{\mu} \quad \rightarrow \quad Re_2 = \frac{(998\ \text{kg/m}^3) V_2 (0.08\ \text{m})}{1.002 \times 10^{-3}\ \text{kg/m}\cdot\text{s}}$$

$$\frac{1}{\sqrt{f_1}} = -2.0 \log\left(\frac{\varepsilon/D_1}{3.7} + \frac{2.51}{Re_1 \sqrt{f_1}}\right)$$

$$\rightarrow \quad \frac{1}{\sqrt{f_1}} = -2.0 \log\left(\frac{0.000045}{3.7 \times 0.04} + \frac{2.51}{Re_1 \sqrt{f_1}}\right)$$

$$\frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{\varepsilon/D_2}{3.7} + \frac{2.51}{Re_2\sqrt{f_2}}\right)$$

$$\rightarrow \quad \frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{0.000045}{3.7 \times 0.08} + \frac{2.51}{Re_2\sqrt{f_2}}\right)$$

$$h_{L,1} = f_1\frac{L_1}{D_1}\frac{V_1^2}{2g} \quad \rightarrow \quad h_{L,1} = f_1\frac{36\text{ m}}{0.04\text{ m}}\frac{V_1^2}{2(9.81\text{ m/s}^2)}$$

$$h_{L,2} = f_2\frac{L_2}{D_2}\frac{V_2^2}{2g} \quad \rightarrow \quad h_{L,2} = f_2\frac{36\text{ m}}{0.08\text{ m}}\frac{V_2^2}{2(9.81\text{ m/s}^2)}$$

$$\dot{V} = \dot{V}_1 + \dot{V}_2$$

$\dot{V} = \mathbf{0.0300\ m^3/s}$, $\dot{V}_1 = \mathbf{0.00415\ m^3/s}$, $\dot{V}_2 = \mathbf{0.0259\ m^3/s}$

$V_1 = 3.30$ m/s, $V_2 = 5.15$ m/s, $h_L = h_{L,1} = h_{L,2} = 11.1$ m, $h_{pump} = 19.1$ m

$Re_1 = 131{,}600$, $Re_2 = 410{,}000$, $f_1 = 0.0221$, $f_2 = 0.0182$

Note that Re > 4000 for both pipes, and thus the assumption of turbulent flow is verified.

Example 3.2:

Water at 10°C flows from a large reservoir to a smaller one through a 5 cm diameter cast iron piping system, as shown in Figure 3.7. Determine the elevation z_1 for a flow rate of 6 L/s.

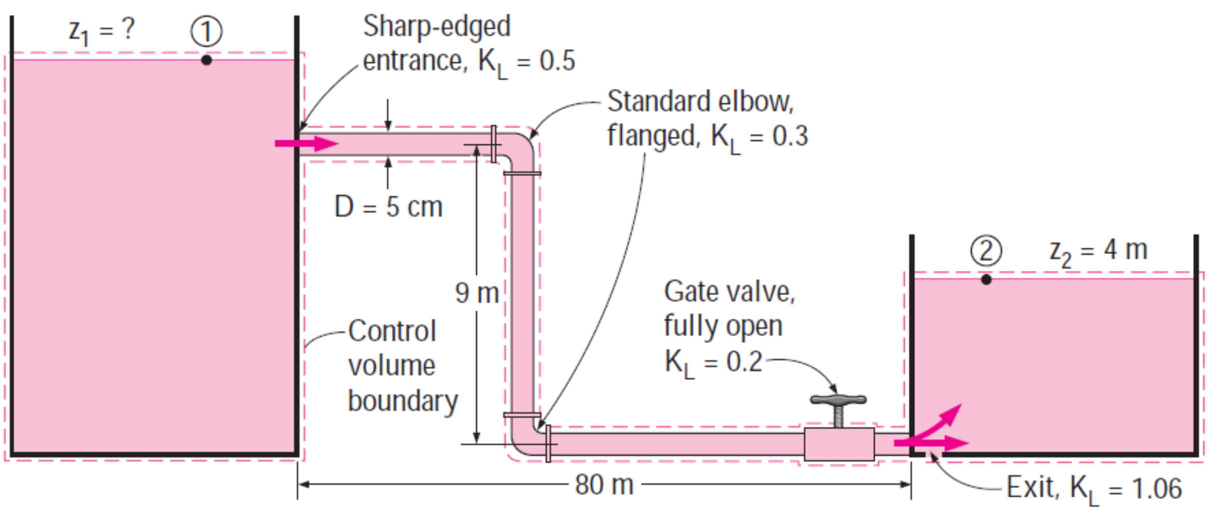


Figure 3.7: The piping system discussed in Example 3.2.

Properties The density and dynamic viscosity of water at 10°C are ρ = 999.7 kg/m^3 and μ = 1.307×10^{-3} kg/m.s. The roughness of cast iron pipe is ε = 0.00026m.

Solution:

The piping system involves 89 m of piping, a sharp-edged entrance (K_L = 0.5), two standard flanged elbows (K_L = 0.3 each), a fully open gate valve (K_L = 0.2), and a submerged exit (K_L = 1.06). We choose points 1 and 2 at the free surfaces of the two reservoirs. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocities at both points are zero ($V_1 = V_2 = 0$), the energy equation for a control volume between these two points simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g}^{\nearrow 0} + z_1 = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g}^{\nearrow 0} + z_2 + h_L \quad \rightarrow \quad z_1 = z_2 + h_L$$

where

$$h_L = h_{L,\,total} = h_{L,\,major} + h_{L,\,minor} = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g}$$

since the diameter of the piping system is constant. The average velocity in the pipe and the Reynolds number are

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.006\ \text{m}^3/\text{s}}{\pi(0.05\ \text{m})^2/4} = 3.06\ \text{m/s}$$

$$Re = \frac{\rho V D}{\mu} = \frac{(999.7\ \text{kg/m}^3)(3.06\ \text{m/s})(0.05\ \text{m})}{1.307 \times 10^{-3}\ \text{kg/m}\cdot\text{s}} = 117{,}000$$

The flow is turbulent since Re > 4000. Noting that e/D = 0.00026/0.05 = 0.0052, the friction factor can be determined from the Colebrook equation (or the ***Moody*** chart),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \quad \rightarrow \quad \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{0.0052}{3.7} + \frac{2.51}{117{,}000\sqrt{f}}\right)$$

It gives f = 0.0315. The sum of the loss coefficients is

$$\sum K_L = K_{L,\,\text{entrance}} + 2K_{L,\,\text{elbow}} + K_{L,\,\text{valve}} + K_{L,\,\text{exit}}$$
$$= 0.5 + 2 \times 0.3 + 0.2 + 1.06 = 2.36$$

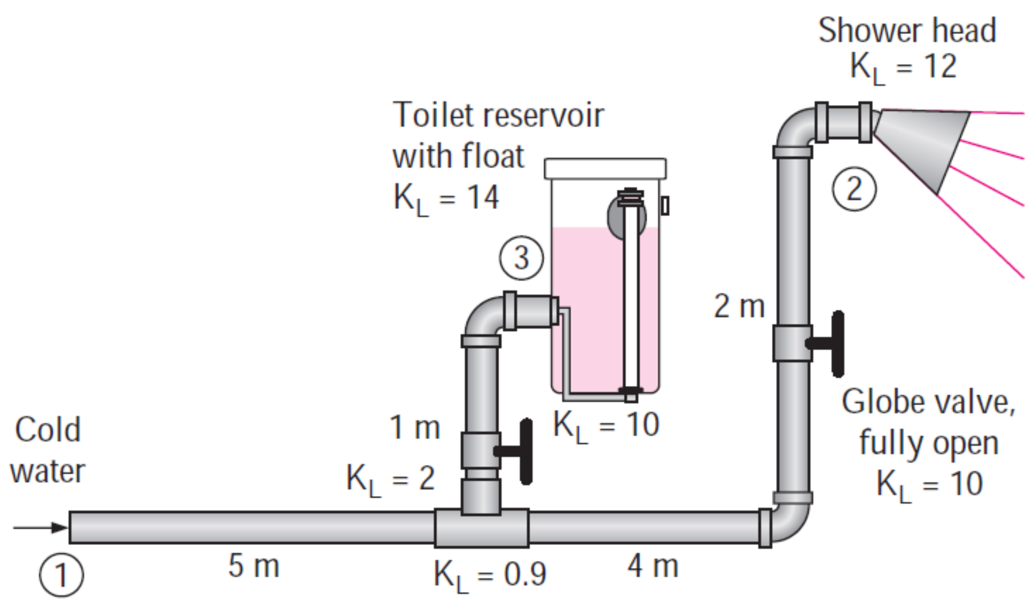
Then the total head loss and the elevation of the source become

$$h_L = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} = \left(0.0315\,\frac{89\text{ m}}{0.05\text{ m}} + 2.36\right)\frac{(3.06\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 27.9\text{ m}$$
$$z_1 = z_2 + h_L = 4 + 27.9 = \mathbf{31.9\text{ m}}$$

Therefore, the free surface of the first reservoir must be 31.9 m above the ground level to ensure water flow between the two reservoirs at the specified rate.

Example 3.3:

The bathroom plumbing of a building consists of 1.5-cm-diameter copper pipes with threaded connectors, as shown in Figure 3.8. (**a**) If the gage pressure at the inlet of the system is 200 kPa during a shower and the toilet reservoir is full (no flow in that branch), determine the flow rate of water through the shower head. (**b**) Determine the effect of flushing of the toilet on the flow rate through the shower head. Take the loss coefficients of the shower head and the reservoir to be 12 and 14, respectively.



Properties The density and dynamic viscosity of water at 20°C are $\rho = 998\ kg/m^3$ and $\mu = 1.002\times10^{-3}$ kg/m.s. The roughness of copper pipes is $\varepsilon = 1.5\times10^{-6}$ m.

Solution

(***a***) The piping system of the shower alone involves 11 m of piping, a tee with line flow ($K_L = 0.9$), two standard elbows ($K_L = 0.9$ each), a fully open globe valve ($K_L = 10$), and a shower head ($K_L = 12$). Therefore, $\Sigma K_L = 0.9 + 2 \times 0.9 + 10 + 12 = 24.7$. Noting that the shower head is open to the atmosphere, and the velocity heads are negligible, the energy equation for a control volume between points 1 and 2 simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{pump,\,u} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{turbine,\,e} + h_L$$

$$\rightarrow \quad \frac{P_{1,\,gage}}{\rho g} = (z_2 - z_1) + h_L$$

Therefore, the head loss is

$$h_L = \frac{200{,}000\ N/m^2}{(998\ kg/m^3)(9.81\ m/s^2)} - 2\ m = 18.4\ m$$

Also,

$$h_L = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} \quad \rightarrow \quad 18.4 = \left(f\frac{11\ m}{0.015\ m} + 24.7\right)\frac{V^2}{2(9.81\ m/s^2)}$$

since the diameter of the piping system is constant. The average velocity in the pipe, the Reynolds number, and the friction factor are

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} \quad \rightarrow \quad V = \frac{\dot{V}}{\pi(0.015\ m)^2/4}$$

$$Re = \frac{VD}{\nu} \quad \rightarrow \quad Re = \frac{V(0.015\ m)}{1.004 \times 10^{-6}\ m^2/s}$$

$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right)$$

$$\rightarrow \quad \frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{1.5 \times 10^{-6}\ m}{3.7(0.015\ m)} + \frac{2.51}{Re\sqrt{f}}\right)$$

This is a set of four equations with four unknowns, and solving them with an equation solver gives

$\dot{V} = 0.00053\ m^3/s, \quad f = 0.0218, \quad V = 2.98\ m/s, \quad \text{and} \quad Re = 44{,}550$

Therefore, the flow rate of water through the shower head is 0.53 L/s.

(**b**) When the toilet is flushed, the float moves and opens the valve. The discharged water starts to refill the reservoir, resulting in parallel flow after the tee connection. The head loss and minor loss coefficients for the shower branch were determined in (**a**) to be $h_{L,2}$ = 18.4 m and $\Sigma K_{L,2}$ = 24.7, respectively. The corresponding quantities for the reservoir branch can be determined similarly to be

$$h_{L,3} = \frac{200{,}000\ N/m^2}{(998\ kg/m^3)(9.81\ m/s^2)} - 1\ m = 19.4\ m$$

$$\Sigma K_{L,3} = 2 + 10 + 0.9 + 14 = 26.9$$

The relevant equations in this case are

$$\dot{V}_1 = \dot{V}_2 + \dot{V}_3$$

$$h_{L,2} = f_1 \frac{5\ m}{0.015\ m} \frac{V_1^2}{2(9.81\ m/s^2)} + \left(f_2 \frac{6\ m}{0.015\ m} + 24.7\right) \frac{V_2^2}{2(9.81\ m/s^2)} = 18.4$$

$$h_{L,3} = f_1 \frac{5\ m}{0.015\ m} \frac{V_1^2}{2(9.81\ m/s^2)} + \left(f_3 \frac{1\ m}{0.015\ m} + 26.9\right) \frac{V_3^2}{2(9.81\ m/s^2)} = 19.4$$

$$V_1 = \frac{\dot{V}_1}{\pi(0.015\ m)^2/4}, \quad V_2 = \frac{\dot{V}_2}{\pi(0.015\ m)^2/4}, \quad V_3 = \frac{\dot{V}_3}{\pi(0.015\ m)^2/4}$$

$$Re_1 = \frac{V_1(0.015\ m)}{1.004 \times 10^{-6} m^2/s}, \quad Re_2 = \frac{V_2(0.015\ m)}{1.004 \times 10^{-6} m^2/s}, \quad Re_3 = \frac{V_3(0.015\ m)}{1.004 \times 10^{-6} m^2/s}$$

$$\frac{1}{\sqrt{f_1}} = -2.0 \log\left(\frac{1.5 \times 10^{-6}\ m}{3.7(0.015\ m)} + \frac{2.51}{Re_1\sqrt{f_1}}\right)$$

$$\frac{1}{\sqrt{f_2}} = -2.0 \log\left(\frac{1.5 \times 10^{-6}\ m}{3.7(0.015\ m)} + \frac{2.51}{Re_2\sqrt{f_2}}\right)$$

$$\frac{1}{\sqrt{f_3}} = -2.0 \log\left(\frac{1.5 \times 10^{-6}\ m}{3.7(0.015\ m)} + \frac{2.51}{Re_3\sqrt{f_3}}\right)$$

Solving these 12 equations in 12 unknowns simultaneously using an equation solver, the flow rates are determined to be

$$\dot{V}_1 = 0.00090\ \text{m}^3/\text{s}, \quad \dot{V}_2 = 0.00042\ \text{m}^3/\text{s}, \quad \text{and} \quad \dot{V}_3 = 0.00048\ \text{m}^3/\text{s}$$

University of Anbar
College of Engineering
Mechanical Engineering Dept.

Fluid Mechanics-II
(ME 2305)

Handout Lectures for Year Two
Chapter Two/ Minor Losses in Pipe Systems

Course Tutor

Assist. Prof. Dr. Waleed M. Abed

Ramadi 2021

Chapter Two
Minor Losses in Pipe Systems

2.1. TURBULENT FLOW IN PIPES

The fluid in a typical piping system passes through various fittings, valves, bends, elbows, tees, inlets, exits, enlargements, and contractions in addition to the pipes. These components interrupt the smooth flow of the fluid and cause additional losses because of the flow separation and mixing they induce. In a typical system with long pipes, these losses are minor compared to the total head loss in the pipes (***the major losses***) and are called ***minor losses***. Although this is generally true, in some cases the minor losses may be greater than the major losses. This is the case, for example, in systems with several turns and valves in a short distance. The head loss introduced by a completely open valve, for example, may be negligible. But a partially closed valve may cause the largest head loss in the system, as evidenced by the drop in the flow rate. Flow through valves and fittings is very complex, and a theoretical analysis is generally not plausible. Therefore, minor losses are determined experimentally, usually by the manufacturers of the components.

Minor losses are usually expressed in terms of ***the loss coefficient*** K_L (also called ***the resistance coefficient***), defined as (Fig. 2.1)

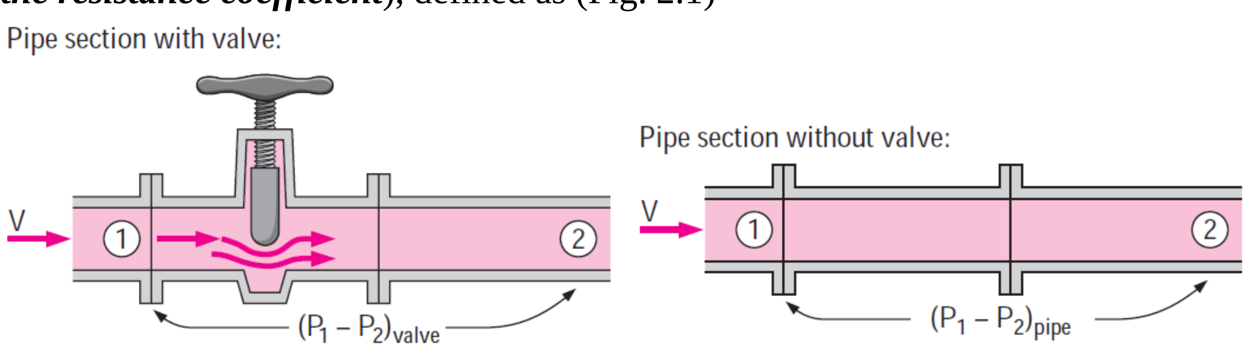


Figure 2.1: For a constant-diameter section of a pipe with a minor loss component, the loss coefficient of the component (such as the gate valve shown) is determined by measuring the additional pressure loss it causes and dividing it by the dynamic pressure in the pipe.

Loss coefficient: $$K_L = \frac{h_L}{V^2/(2g)} \quad (2.1)$$

where h_L is the additional irreversible head loss in the piping system caused by insertion of the component, and is defined as [$h_L = \Delta P_L / \rho g$]. For example, imagine replacing the valve in Figure 2.1 with a section of constant diameter pipe from location 1 to location 2. ΔP_L is defined as the pressure drop from 1 to 2 for the case with the valve, $(P_1 - P_2)_{valve}$, minus the pressure drop that would occur in the imaginary straight pipe section from 1 to 2 without the valve, $(P_1 - P_2)_{pipe}$ at the same flow rate. While the majority of the irreversible head loss occurs locally near the valve, some of it occurs downstream of the valve due to induced swirling turbulent eddies that are produced in the valve and continue downstream. These eddies "*waste*" mechanical energy because they are ultimately dissipated into heat while the flow in the downstream section of pipe eventually returns to fully developed conditions. When measuring minor losses in some minor loss components, such as elbows, for example, location 2 must be considerably far downstream (tens of pipe diameters) in order to fully account for the additional irreversible losses due to these decaying eddies.

When the inlet diameter equals outlet diameter, the loss coefficient of a component can also be determined by measuring the pressure loss across the component and dividing it by the dynamic pressure, $K_L = \Delta P_L/(0.5\ \rho V^2)$. When the loss coefficient for a component is available, the head loss for that component is determined from

Minor loss: $$h_L = K_L \frac{V^2}{2g} \quad (2.2)$$

The loss coefficient, in general, depends on the geometry of the component and the *Reynolds number*, just like the friction factor. However, it is usually assumed to be independent of the Reynolds number. This is a reasonable approximation since most flows in practice have large Reynolds numbers and the loss coefficients (including ***the friction factor***) tend to be independent of the Reynolds number at large Reynolds numbers. Minor losses are also expressed in terms of ***the equivalent length*** L_{equiv}, defined as (Figure 2.2)

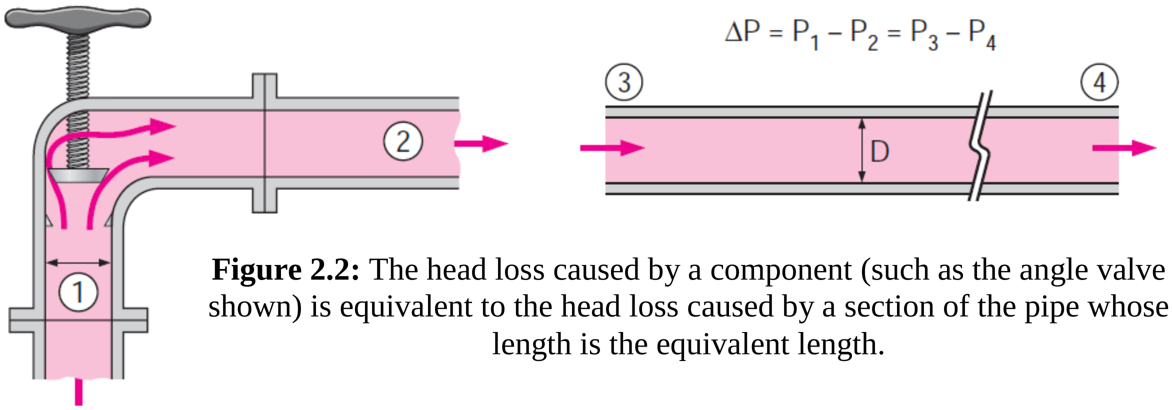


Figure 2.2: The head loss caused by a component (such as the angle valve shown) is equivalent to the head loss caused by a section of the pipe whose length is the equivalent length.

Equivalent length: $h_L = K_L \frac{V^2}{2g} = f \frac{L_{equiv}}{D} \frac{V^2}{2g} \rightarrow L_{equiv} = \frac{D}{f} K_L$)

where f is the friction factor and D is the diameter of the pipe that contains the component. The head loss caused by the component is equivalent to the head loss caused by a section of the pipe whose length is L_{equiv}. Therefore, the contribution of a component to the head loss can be accounted for by simply adding L_{equiv} to the total pipe length. Once all the loss coefficients are available, the total head loss in a piping system is determined from

Total head loss (general):

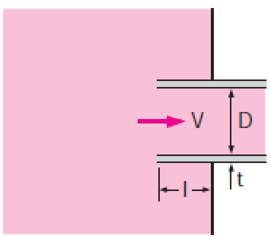
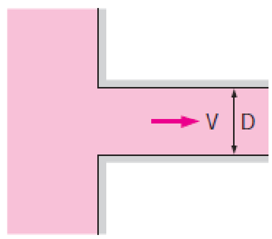
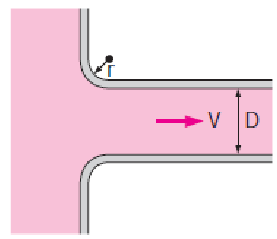
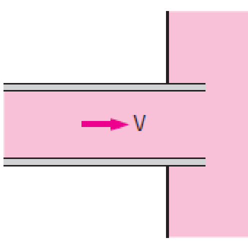
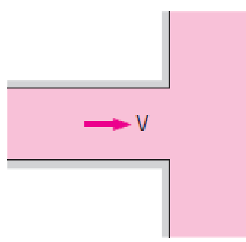
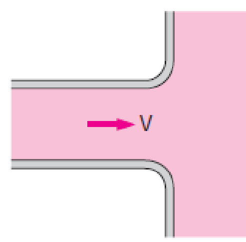
$$h_{L,\,total} = h_{L,\,major} + h_{L,\,minor} = \sum_i f_i \frac{L_i}{D_i} \frac{V_i^2}{2g} + \sum_j K_{L,j} \frac{V_j^2}{2g} \qquad (2.4)$$

where i represents each pipe section with constant diameter and j represents each component that causes a minor loss. If the entire piping system being analyzed has a constant diameter, Eq. 2.4 reduces to

Total head loss (D = constant): $h_{L,\,total} = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g}$ (2.5)

where V is the average flow velocity through the entire system (note that V = constant since D = constant). Representative loss coefficients K_L are given in Table 2.1 for inlets, exits, bends, sudden and gradual area changes, and valves.

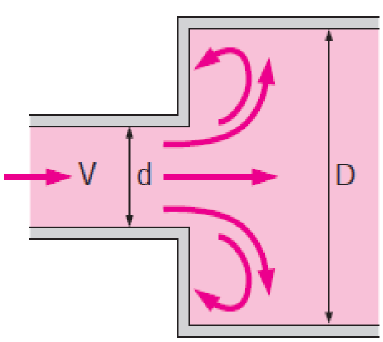
TABLE 2.1: Loss coefficients K_L of various pipe components for turbulent flow.

Pipe Inlet *Reentrant:* $K_L = 0.80$ ($t \ll D$ and $l \approx 0.1D$) 	*Sharp-edged:* $K_L = 0.50$ 	*Well-rounded* ($r/D > 0.2$): $K_L = 0.03$ *Slightly rounded* ($r/D = 0.1$): $K_L = 0.12$ (see Fig. 8–36) 
Pipe Exit *Reentrant:* $K_L = \alpha$ 	*Sharp-edged:* $K_L = \alpha$ 	*Rounded:* $K_L = \alpha$ 

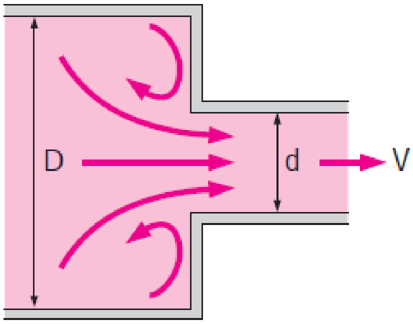
Note: The kinetic energy correction factor is $\alpha = 2$ for fully developed laminar flow, and $\alpha \approx 1$ for fully developed turbulent flow.

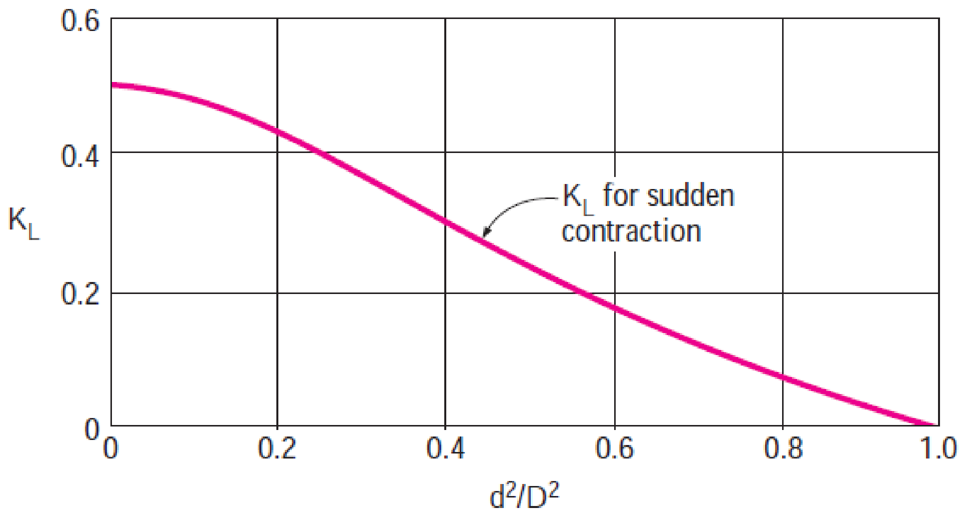
Sudden Expansion and Contraction (based on the velocity in the smaller-diameter pipe)

Sudden expansion: $K_L = \left(1 - \frac{d^2}{D^2}\right)^2$ (2.6)



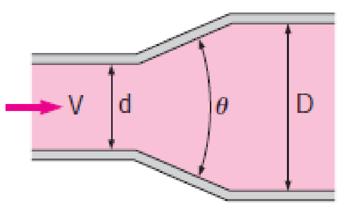
Sudden contraction: See chart.



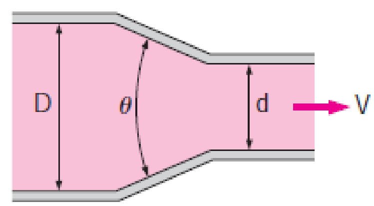


Gradual Expansion and Contraction (based on the velocity in the smaller-diameter pipe)

Expansion:
$K_L = 0.02$ for $\theta = 20°$
$K_L = 0.04$ for $\theta = 45°$
$K_L = 0.07$ for $\theta = 60°$

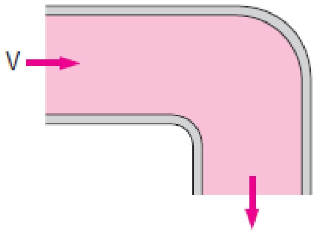


Contraction (for $\theta = 20°$):
$K_L = 0.30$ for $d/D = 0.2$
$K_L = 0.25$ for $d/D = 0.4$
$K_L = 0.15$ for $d/D = 0.6$
$K_L = 0.10$ for $d/D = 0.8$

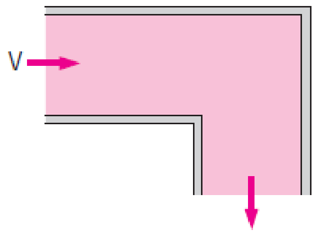


Bends and Branches

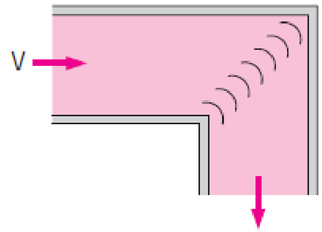
90° smooth bend:
Flanged: $K_L = 0.3$
Threaded: $K_L = 0.9$



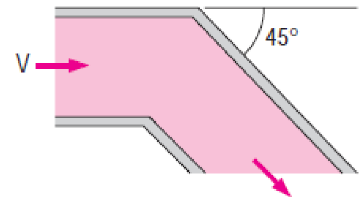
90° miter bend
(without vanes): $K_L = 1.1$



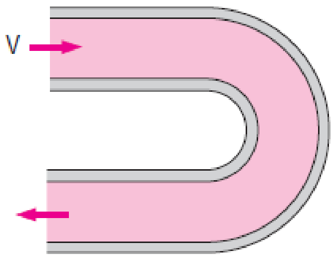
90° miter bend
(with vanes): $K_L = 0.2$



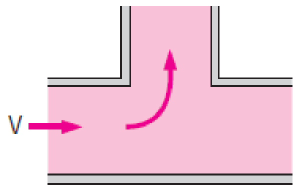
45° threaded elbow:
$K_L = 0.4$



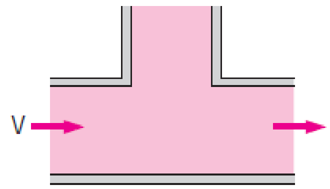
180° return bend:
Flanged: $K_L = 0.2$
Threaded: $K_L = 1.5$



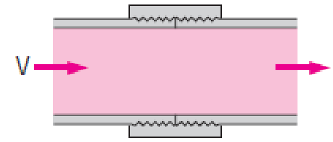
Tee (branch flow):
Flanged: $K_L = 1.0$
Threaded: $K_L = 2.0$



Tee (line flow):
Flanged: $K_L = 0.2$
Threaded: $K_L = 0.9$



Threaded union:
$K_L = 0.08$



Valves

Globe valve, fully open: $K_L = 10$
Angle valve, fully open: $K_L = 5$
Ball valve, fully open: $K_L = 0.05$
Swing check valve: $K_L = 2$

Gate valve, fully open: $K_L = 0.2$
$\frac{1}{4}$ closed: $K_L = 0.3$
$\frac{1}{2}$ closed: $K_L = 2.1$
$\frac{3}{4}$ closed: $K_L = 17$

Notice: These are representative values for loss coefficients. Actual values strongly depend on the design and manufacture of the components and may differ from the given values considerably (especially for valves). Actual manufacturer's data should be used in the final design.

Flow contraction and the associated head loss at a sharp-edged pipe inlet:

A sharp-edged inlet acts like a flow constriction. The velocity increases in the vena contracta region (and the pressure decreases) because of the reduced effective flow area and then decreases as the flow fills the entire cross section of the pipe. There would be negligible loss if the pressure were increased in accordance with Bernoulli's equation (the velocity head would simply be converted into pressure head). However, this deceleration process is far from ideal and the viscous dissipation caused by intense mixing and the turbulent eddies convert part of the kinetic energy into frictional heating, as evidenced by a slight rise in fluid temperature. The end result is a drop in velocity without much pressure recovery, and the inlet loss is a measure of this irreversible pressure drop.

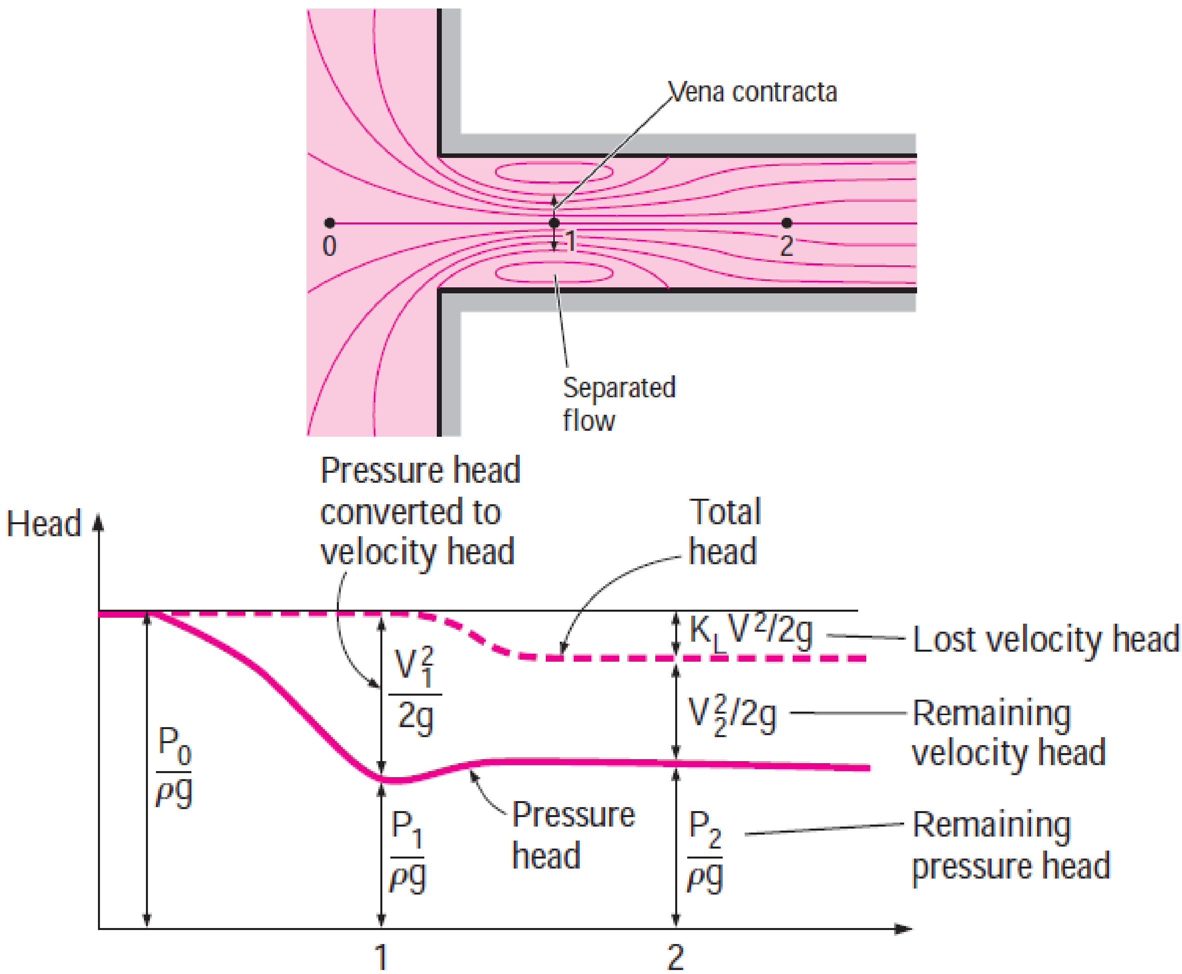


Figure 2.3: Graphical representation of flow contraction and the associated head loss at a sharp-edged pipe inlet.

Even slight rounding of the edges can result in significant reduction of K_L, as shown in Figure 2.4. The loss coefficient rises sharply (to about $K_L = 0.8$) when the pipe protrudes into the reservoir since some fluid near the edge in this case is forced to make a 180° turn.

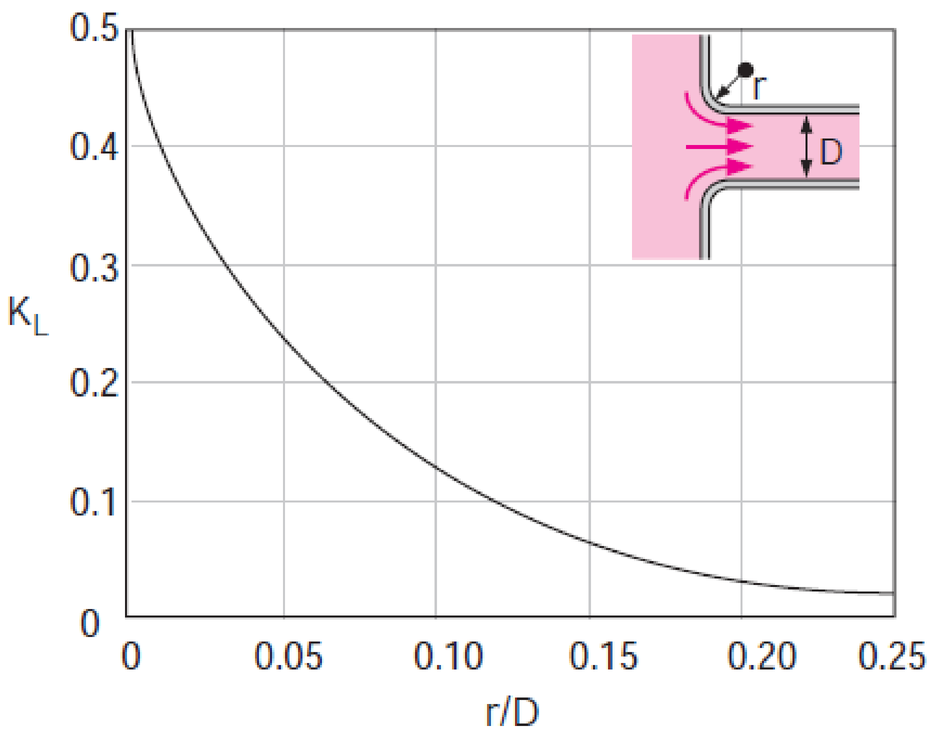


Figure 2.4: The effect of rounding of a pipe inlet on the loss coefficient.

Sudden or gradual expansion or contraction sections:

Piping systems often involve sudden or gradual expansion or contraction sections to accommodate changes in flow rates or properties such as density and velocity. The losses are usually much greater in the case of sudden expansion and contraction (or wide-angle expansion) because of flow separation. By combining the conservation of mass, momentum, and energy equations, the loss coefficient for the case of sudden expansion is approximated as

Sudden expansion: $$K_L = \left(1 - \frac{A_{small}}{A_{large}}\right)^2 \qquad (2.7)$$

where A_{small} and A_{large} are the cross-sectional areas of the small and large pipes, respectively.

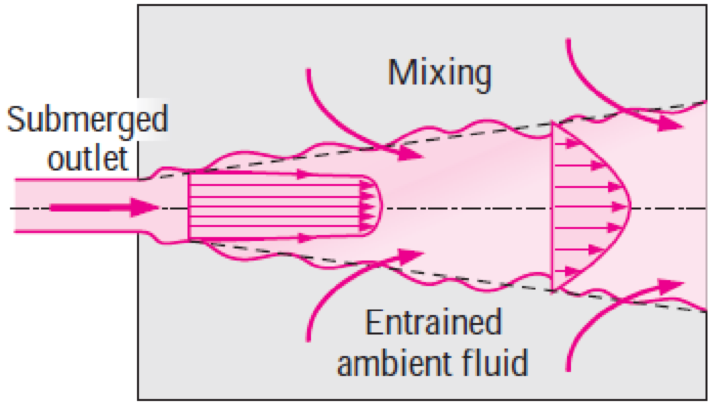


Figure 2.5: All the kinetic energy of the flow is "lost" (turned into thermal energy) through friction as the jet decelerates and mixes with ambient fluid downstream of a submerged outlet.

At any such exit, whether laminar or turbulent, the fluid leaving the pipe loses all of its kinetic energy as it mixes with the reservoir fluid and eventually comes to rest through the irreversible action of viscosity. This is true, regardless of the shape of the exit (Table 2.1 and Figure 2.5). Therefore, there is no need to round the pipe exits.

The losses during changes of direction can be minimized by making the turn "*easy*" on the fluid by using circular arcs (like the 90° elbow) instead of sharp turns (like miter bends) (Figure 2.6). But the use of sharp turns (and thus suffering a penalty in loss coefficient) may be necessary when the turning space is limited.

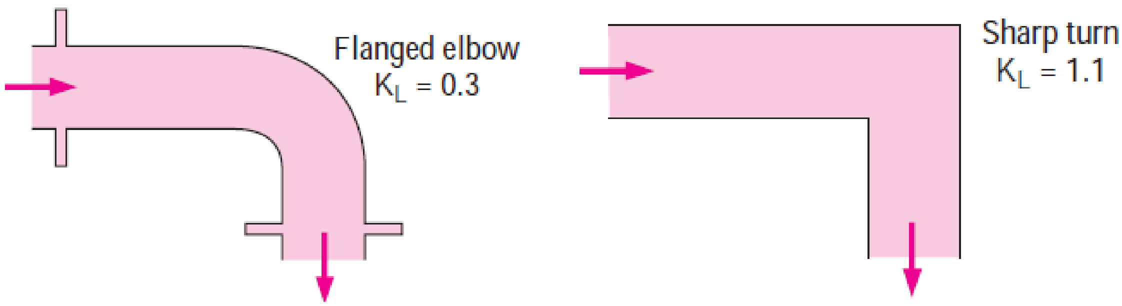


Figure 2.6: The losses during changes of direction can be minimized by making the turn "easy" on the fluid by using circular arcs instead of sharp turns.

Example 2.1:

A 6-cm-diameter horizontal water pipe expands gradually to a 9-cm-diameter pipe (Figure 2.7). The walls of the expansion section are angled 30° from the horizontal. The average velocity and pressure of water before the expansion section are 7 m/s and 150 kPa, respectively. Determine the head loss in the expansion section and the pressure in the larger-diameter pipe.

Solution:

We take the density of water to be ρ= 1000 kg/m^3. The loss coefficient for gradual expansion of θ = 60° total included angle is K_L= 0.07.

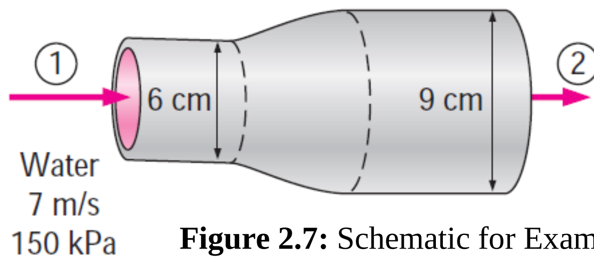


Figure 2.7: Schematic for Example 2.1.

$$\dot{m}_1 = \dot{m}_2 \rightarrow \rho V_1 A_1 = \rho V_2 A_2 \rightarrow V_2 = \frac{A_1}{A_2} V_1 = \frac{D_1^2}{D_2^2} V_1$$

$$V_2 = \frac{(0.06\text{ m})^2}{(0.09\text{ m})^2}(7\text{ m/s}) = 3.11\text{ m/s}$$

Then the irreversible head loss in the expansion section becomes

$$h_L = K_L \frac{V_1^2}{2g} = (0.07)\frac{(7\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = \mathbf{0.175\text{ m}}$$

Noting that $z_1 = z_2$ and there are no pumps or turbines involved, the energy equation for the expansion section can be expressed in terms of heads as

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + \cancel{z_1} + \overset{0}{h_{pump,\,u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + \cancel{z_2} + \overset{0}{h_{turbine,\,e}} + h_L$$

$$\rightarrow \frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + h_L$$

Solving for P_2 and substituting,

$$P_2 = P_1 + \rho\left\{\frac{\alpha_1 V_1^2 - \alpha_2 V_2^2}{2} - g h_L\right\} = (150\text{ kPa}) + (1000\text{ kg/m}^3)$$

$$\times \left\{\frac{1.06(7\text{ m/s})^2 - 1.06(3.11\text{ m/s})^2}{2} - (9.81\text{ m/s}^2)(0.175\text{ m})\right\}$$

$$\times \left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$

$$= \mathbf{169\text{ kPa}}$$

Example 2.2:

Consider flow from a water reservoir through a circular hole of diameter *D* at the side wall at a vertical distance *H* from the free surface. The flow rate through an actual hole with a sharp-edged entrance (K_L = 0.5) will be considerably less than the flow rate calculated assuming "frictionless" flow and thus zero loss for the hole. Disregarding the effect of the kinetic energy correction factor, obtain a relation for the "equivalent diameter" of the sharp-edged hole for use in frictionless flow relations.

Solution:

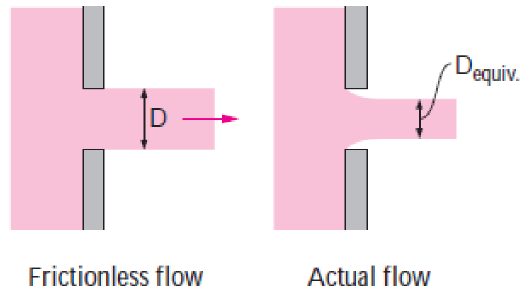


Figure 2.8: Schematic for Example 2.2.

The loss coefficient is K_L = 0.5 for the sharp-edged entrance, and K_L = 0 for the "frictionless" flow. We take point 1 at the free surface of the reservoir and point 2 at the exit of the hole, which is also taken to be the reference level (z_2 = 0). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is zero (V_1 = 0), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump,u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine,e}}+h_L \quad\rightarrow\quad H=\alpha_2\frac{V_2^2}{2g}+h_L$$

where the head loss is expressed as $h_L = K_L\dfrac{V^2}{2g}$. Substituting and solving for V_2 gives

$$H=\alpha_2\frac{V_2^2}{2g}+K_L\frac{V_2^2}{2g} \quad\rightarrow\quad 2gH=V_2^2(\alpha_2+K_L) \quad\rightarrow\quad V_2=\sqrt{\frac{2gH}{\alpha_2+K_L}}=\sqrt{\frac{2gH}{1+K_L}}$$

since α_2 = 1. Then the volume flow rate becomes

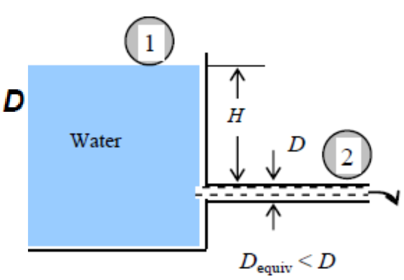
$$\dot{V}=A_cV_2=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+K_L}} \qquad (1)$$

Note that in the special case of K_L= 0 (frictionless flow), the velocity relation reduces to the ***Toricelli equation***, $V_{2,\text{frictionless}}$= 2gH. The flow rate in this case through a hole of D_e (equivalent diameter) is

$$\dot{V}=A_{c,\text{equiv}}V_{2,\text{frictionless}}=\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH} \qquad (2)$$

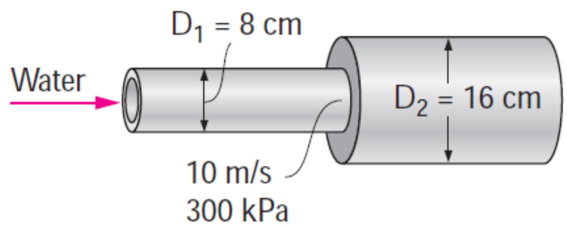
Setting Eqs. (1) and (2) equal to each other gives the desired relation for the equivalent diameter,

$$\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH}=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+K_L}} \qquad D_{\text{equiv}}=\frac{D}{(1+K_L)^{1/4}}=\frac{D}{(1+0.5)^{1/4}}=\mathbf{0.904\,D}$$



Example 2.3:

A horizontal pipe has an abrupt expansion from D_1 = 8 cm to D_2 = 16 cm. The water velocity in the smaller section is 10 m/s and the flow is turbulent. The pressure in the smaller section is P_1 = 300 kPa. Taking the kinetic energy correction factor to be 1.06 at both the inlet and the outlet, determine the downstream pressure P_2, and estimate the error that would have occurred if *Bernoulli's equation* had been used.



Solution:

Properties: the density of water to be ρ = 1000 kg/m^3.

The downstream velocity of water is,

$$\dot{m}_1 = \dot{m}_2 \rightarrow \rho V_1 A_1 = \rho V_2 A_2 \rightarrow V_2 = \frac{A_1}{A_2} V_1 = \frac{\pi D_1^2/4}{\pi D_2^2/4} V_1 = \frac{D_1^2}{D_2^2} V_1 = \frac{(0.08\text{ m})^2}{(0.16\text{ m})^2}(10\text{ m/s}) = 2.5\text{ m/s}$$

The loss coefficient for sudden expansion and the head loss can be calculated from

$$K_L = \left(1 - \frac{A_{\text{small}}}{A_{\text{large}}}\right)^2 = \left(1 - \frac{D_1^2}{D_2^2}\right)^2 = \left(1 - \frac{0.08^2}{0.16^2}\right)^2 = 0.5625$$

$$h_L = K_L \frac{V_1^2}{2g} = (0.5625)\frac{(10\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 2.87\text{ m}$$

Noting that $z_1 = z_2$ and there are no pumps or turbines involved, the energy equation for the expansion section can be expressed in terms of heads as

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump,u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L$$

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + h_L$$

Solving for P_2 and substituting,

$$P_2 = P_1 + \rho\left\{\frac{\alpha_1 V_1^2 - \alpha_2 V_2^2}{2} - g h_L\right\}$$

$$= (300\text{ kPa}) + (1000\text{ kg/m}^3)\left\{\frac{1.06(10\text{ m/s})^2 - 1.06(2.5\text{ m/s})^2}{2} - (9.81\text{ m/s}^2)(2.87\text{ m})\right\}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$

$= \mathbf{322\ kPa}$

When the head loss is disregarded, the downstream pressure is determined from the Bernoulli equation to be

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g}=\frac{P_2}{\rho g}+\frac{V_2^2}{2g} \quad\rightarrow\quad P_1=P_1+\rho\frac{V_1^{\,2}-V_2^{\,2}}{2}$$

Substituting,

$$P_2=(300\text{ kPa})+(1000\text{ kg/m}^3)\frac{(10\text{ m/s})^2-(2.5\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)=347\text{ kPa}$$

Therefore, the error in the Bernoulli equation is

$$\text{Error}=P_{2,\text{ Bernoulli}}-P_2=347-322=\mathbf{25.0\ kPa}$$

Note that the use of the Bernoulli equation results in an error of (347 – 322) / 322 = 0.078 or 7.8%.