University of Anbar
College of Engineering
Mechanical Engineering Dept.

# Fluid Mechanics-II (ME 2305)

## Handout Lectures for Year Two
## Chapter One/ Viscous Flow in Ducts

**Course Tutor**

**Assist. Prof. Dr. Waleed M. Abed**

# Chapter One
# Viscous Flow in Ducts

## 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).

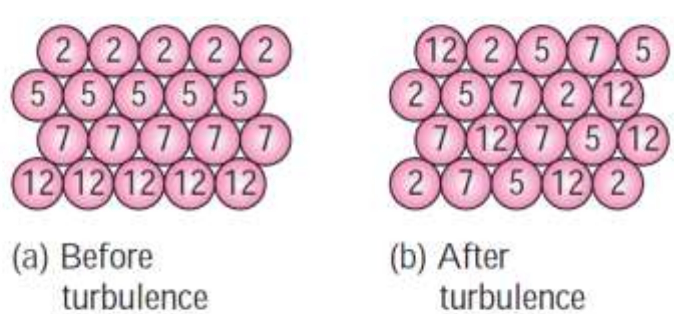



**Figure 1.1:** The intense mixing in turbulent flow brings fluid particles at different momentums into close contact and thus enhances momentum transfer.

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 $u'$,

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

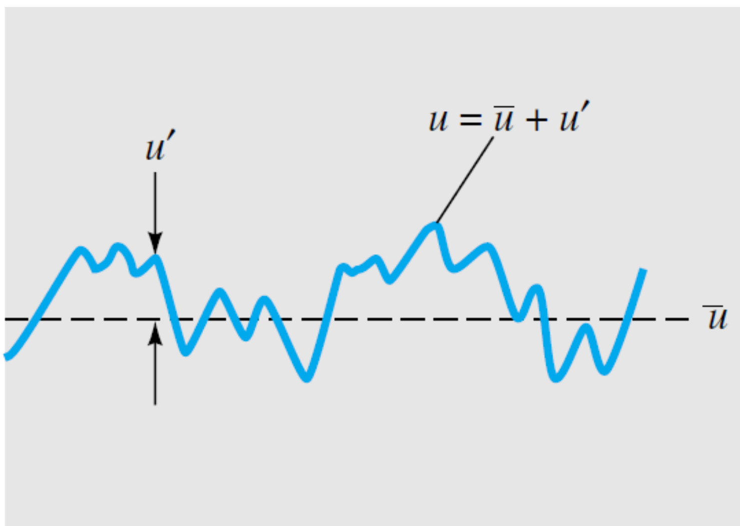



**Figure 1.2:** Fluctuations of the velocity component $u$ with time at a specified location in turbulent flow.

## 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 $\tau_{turb}$) and is related to the fluctuation components of velocity). Then the total shear stress in turbulent flow can be expressed as

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

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.

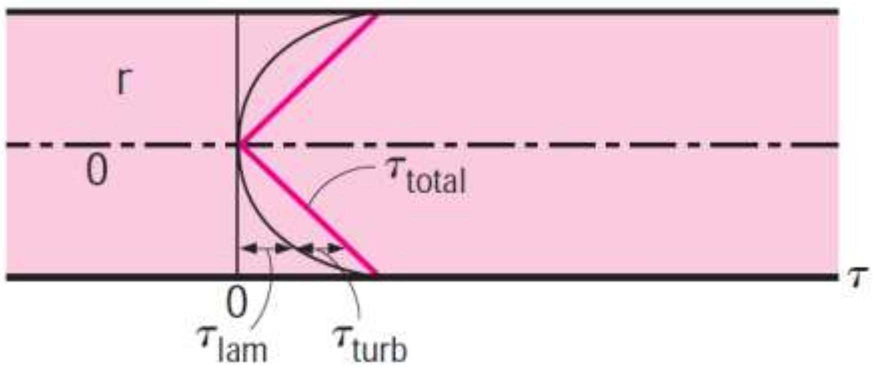



**Figure 1.3:** The velocity profile and the variation of shear stress with radial distance for turbulent flow in a pipe.

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

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

where $\mu_t$ 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

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

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

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

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

## 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.

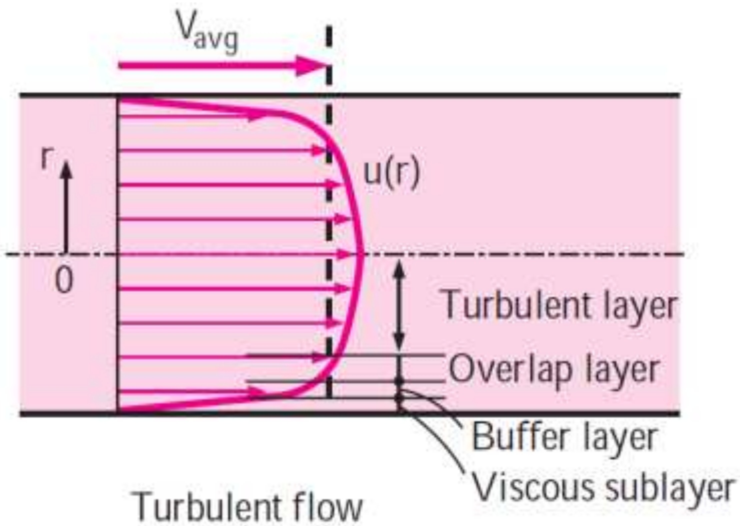



**Figure 1.4:** The velocity profile in fully developed pipe flow is parabolic in laminar flow, but much fuller in turbulent flow.

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

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

where $y$ is the distance from the wall (note that $y = R - r$ for a circular pipe). The quantity $\tau_w/\rho$ is frequently encountered in the analysis of turbulent velocity profiles. The square root of $\tau_w/\rho$ 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 $u_\sigma$ 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*** ($\varepsilon/D$), 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*.

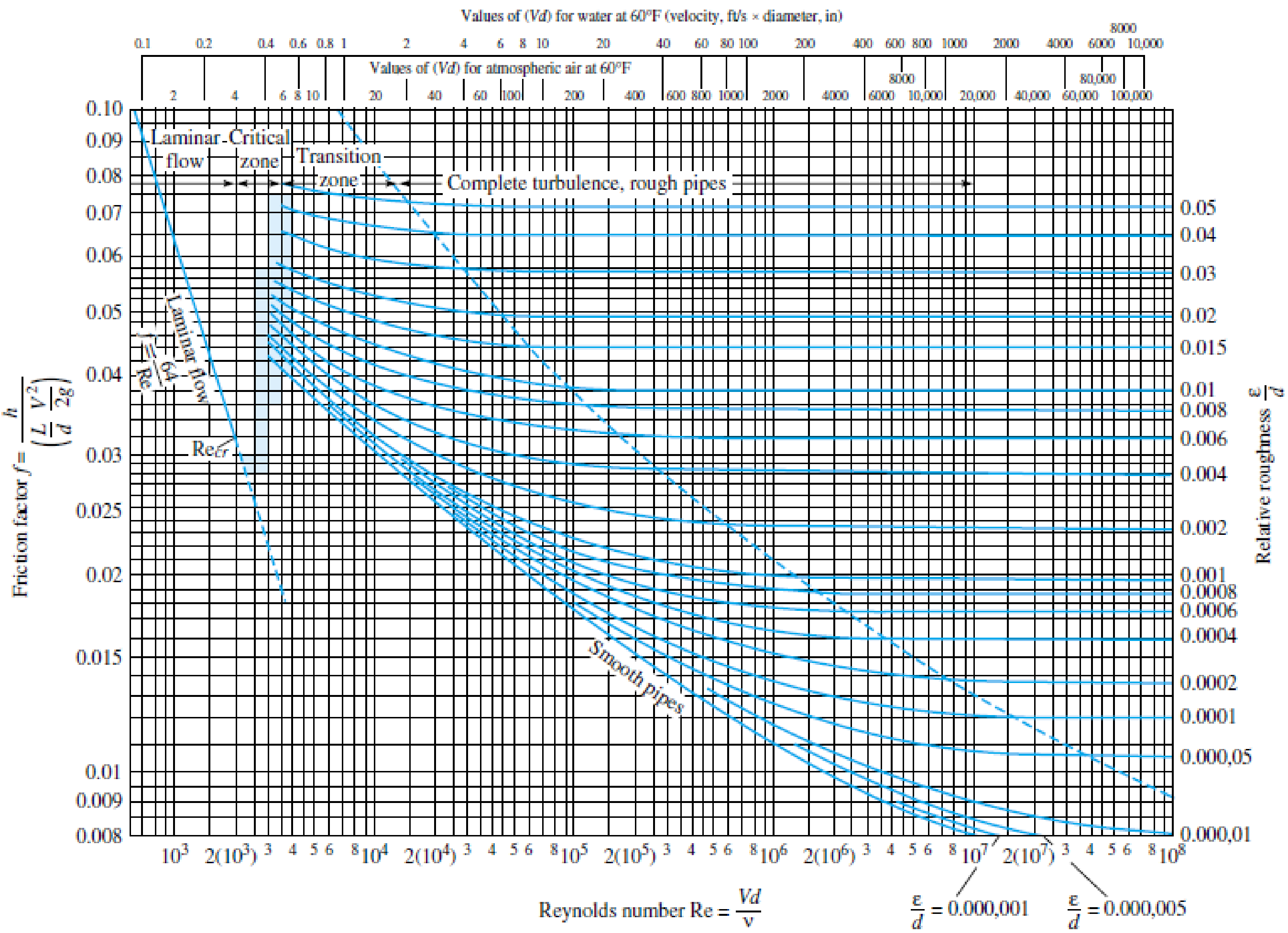



**Figure 1.5:** The Moody chart for the friction factor for fully developed flow.

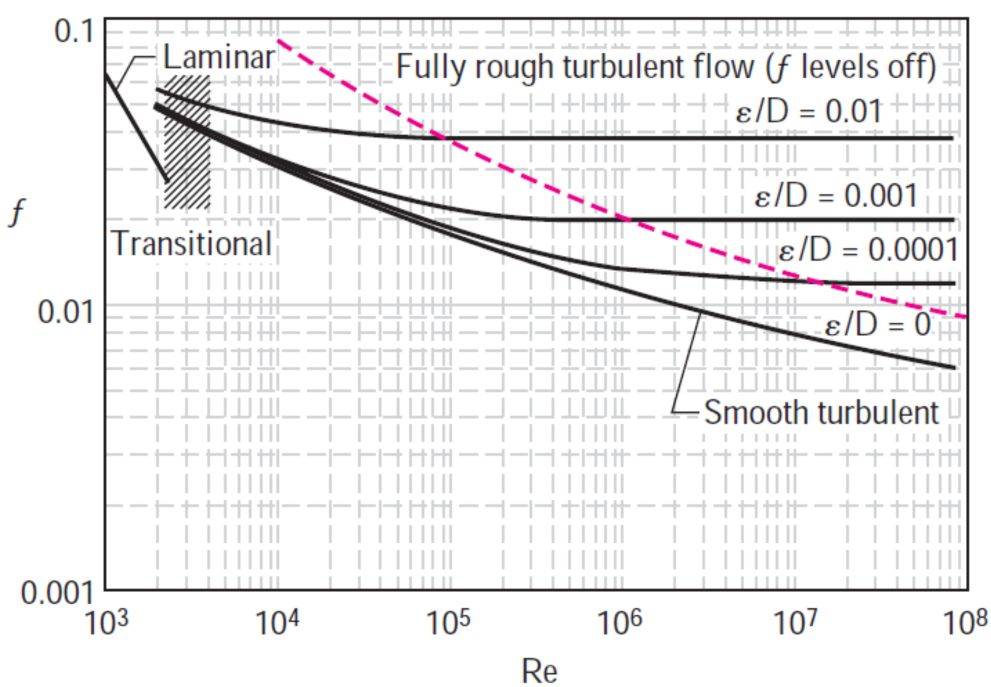



**Figure 1.6:** At very large Reynolds numbers, the friction factor curves on the Moody chart are nearly horizontal, and thus the friction factors are independent of the Reynolds number.

## 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:

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

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

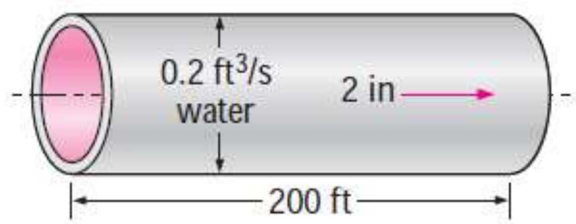
$$D = 0.66\left[\varepsilon^{1.25}\left(\frac{L\dot{V}^2}{gh_L}\right)^{4.75} + \nu\dot{V}^{9.4}\left(\frac{L}{gh_L}\right)^{5.2}\right]^{0.04} \qquad \begin{array}{l} 10^{-6} < \varepsilon/D < 10^{-2} \\ 5000 < Re < 3\times 10^{8} \end{array}$$

**Examples:**

**Example 1:**

Water ($\rho$= 62.36 lbm/ft$^3$ and $\mu$= 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:




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

$$Re = \frac{\rho V D}{\mu} = \frac{(62.36\ \text{lbm/ft}^3)(9.17\ \text{ft/s})(2/12\ \text{ft})}{7.536\times 10^{-4}\ \text{lbm/ft}\cdot\text{s}} = 126{,}400$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is calculated using Table1.1. $\varepsilon/D = \dfrac{0.000007\ \text{ft}}{2/12\ \text{ft}} = 0.000042$

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

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0174\frac{200\text{ ft}}{2/12\text{ ft}}\frac{(62.36\text{ lbm/ft}^3)(9.17\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$
$$= 1700\text{ lbf/ft}^2 = 11.8\text{ psi}$$

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

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

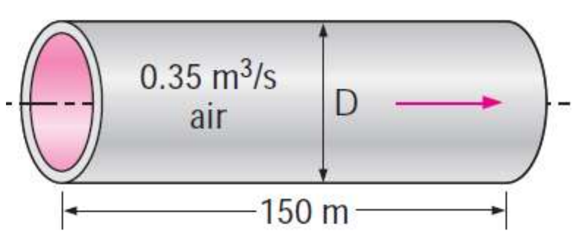
**Example 2:**

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 $\rho$ = 1.145 $kg/m^3$, $\mu$= $1.895\times10^{-5}$ kg/m·s, and $\nu$= $1.655\times10^{-5}$ $m^2/s$.

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

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

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$\times$10$^{-4}$ kg/m.s. The roughness of copper tubing is 1.5$\times$10$^{-6}$ m.

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

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

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(665.1\text{ kg/m}^3)(11.49\text{ m/s})(0.005\text{ m})}{2.361\times 10^{-4}\text{ kg/m}\cdot\text{s}} = 1.618\times 10^5$$

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

$$\varepsilon/D = \frac{1.5\times 10^{-6}\text{ m}}{0.005\text{ m}} = 3\times 10^{-4}$$

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),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \;\rightarrow\; \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{3\times 10^{-4}}{3.7} + \frac{2.51}{1.618\times 10^5\sqrt{f}}\right)$$

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

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

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

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

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

University of Anbar
College of Engineering
Mechanical Engineering Dept.

# Fluid Mechanics

## Handout Lectures for Year Two
## Chapter Three/ Fluid Kinematics

**Course Tutor**

**Assist. Prof. Dr. Waleed M. Abed**

# Chapter Three
# Fluid Kinematics

## 1.1 FUNDAMENTALS OF FLOW VISUALIZATION

While quantitative study of fluid dynamics requires advanced mathematics, much can be learned from flow visualization—the visual examination of flow field features. Flow visualization is useful not only in physical experiments (Fig. 3–1), but in numerical solutions as well [computational fluid dynamics (CFD)]. In fact, the very first thing an engineer using CFD does after obtaining a numerical solution is simulate some form of flow visualization, so that he or she can see the "whole picture" rather than merely a list of numbers and quantitative data. Why? Because the human mind is designed to rapidly process an incredible amount of visual information; as they say, a picture is worth a thousand words. There are many types of flow patterns that can be visualized, both physically (experimentally) and/or computationally.

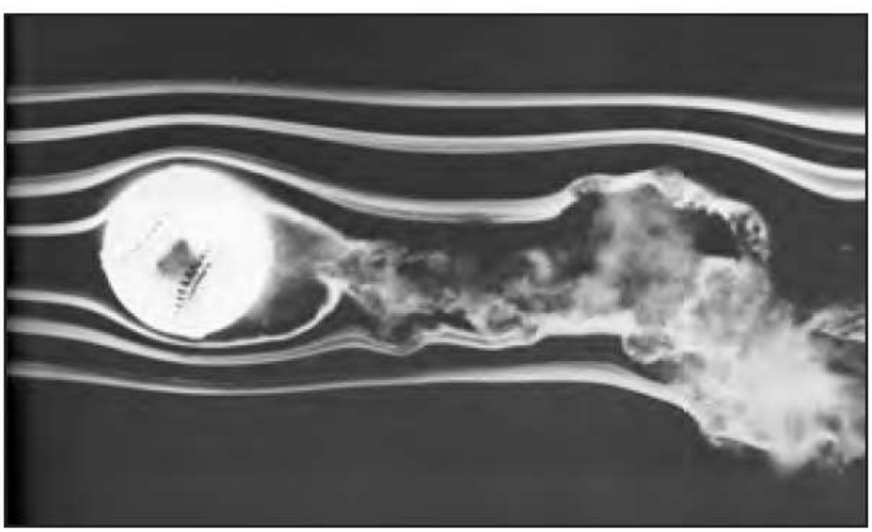


**Figure 3.1:** Spinning baseball.

### 1.2 Streamlines

A ***streamline*** is a curve that is everywhere tangent to the instantaneous local velocity vector.

Streamlines are useful as indicators of the instantaneous direction of fluid motion throughout the flow field. For example, regions of recirculating flow and separation of a fluid off of a solid wall are easily identified by the streamline pattern. Streamlines cannot be directly observed experimentally except in steady flow fields, in which they are coincident with pathlines and streaklines, to be discussed next. Mathematically, however, we can write a simple expression for a streamline based on its definition.

Consider an infinitesimal arc length $d\vec{r} = dx\vec{i} + dy\vec{j} + dz\vec{k}$ along streamline; $d\vec{r}$ must be parallel to the local velocity vector $\vec{V} = u\vec{i} + v\vec{j} + w\vec{k}$ by definition of the streamline. By simple geometric arguments using similar triangles, we know that the components of $d\vec{r}$ must be proportional to those of $\vec{V}$ (Fig. 3–2). Hence,

Equation for a Streamline:

$$\frac{dr}{V} = \frac{dr}{u} = \frac{dr}{v} = \frac{dr}{w} \qquad (3\text{–}1)$$

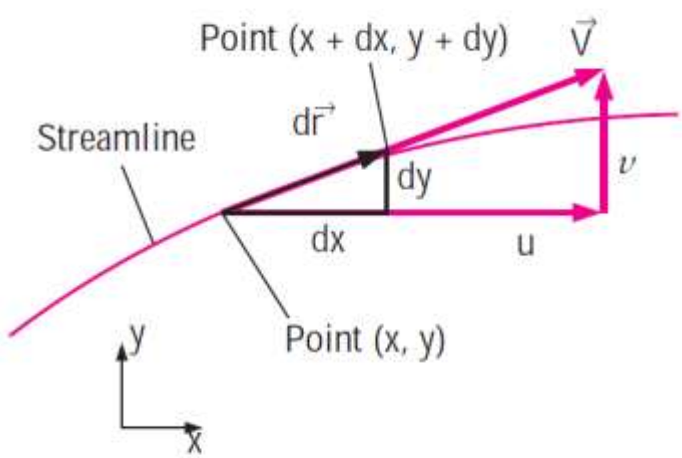



**Figure 3.2:** Two-dimensional flow in the *xy*-plane, arc length $d\vec{r}$= (d*x*, d*y*) along a streamline is everywhere tangent to the local instantaneous velocity vector $\vec{V}$ = (*u*, *v*).

where $dr$ is the magnitude of $d\vec{r}$ and $V$ is the speed, the magnitude of $\vec{V}$. Equation 3–1 is illustrated in two dimensions for simplicity in Fig. 3–2. For a known velocity field, we can integrate Eq. 3–1to obtain equations for the streamlines. In two dimensions, (x, y), (u, v), the following differential equation is obtained:

Streamline in the xy-plane: $\left(\frac{dy}{dx}\right)_{along\ a\ \text{Streamline}} = \frac{v}{u}$ (3–2)

In some simple cases, Eq. 3–2 may be solvable analytically; in the general case, it must be solved numerically. In either case, an arbitrary constant of integration appears, and the family of curves that satisfy Eq. 3–2 represents streamlines of the flow field.

**EXAMPLE 4–4 Streamlines in the *xy*-Plane—An Analytical Solution**

For the steady, incompressible, two-dimensional velocity field of Example 4–1, plot several streamlines in the right half of the flow ($x > 0$) and compare to the velocity vectors plotted in Fig. 4–4.

**SOLUTION** An analytical expression for streamlines is to be generated and plotted in the upper-right quadrant.

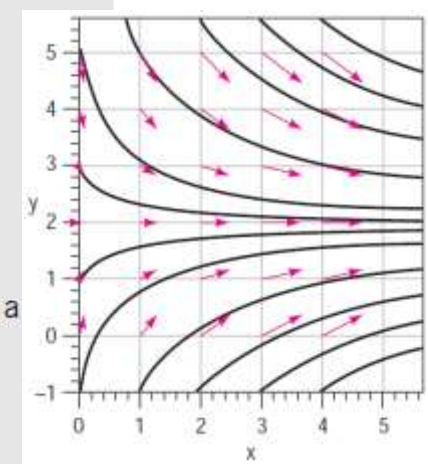
***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional, implying no *z*-component of velocity and no variation of *u* or *v* with *z*.

***Analysis*** Equation 4–16 is applicable here; thus, along a streamline,

$$\frac{dy}{dx} = \frac{1.5 - 0.8y}{0.5 + 0.8x}$$

We solve this differential equation by separation of variables:

$$\frac{dy}{1.5 - 0.8y} = \frac{dx}{0.5 + 0.8x} \quad \rightarrow \quad \int \frac{dy}{1.5 - 0.8y} = \int \frac{dx}{0.5 + 0.8x}$$




After some algebra (which we leave to the reader), we solve for *y* as a tion of *x* along a streamline,

$$y = \frac{C}{0.8(0.5 + 0.8x)} + 1.875$$

where *C* is a constant of integration that can be set to various values in order to plot the streamlines. Several streamlines of the given flow field are shown in Fig. 4–17.

### 1.3 Pathlines

A ***pathline*** is the actual path traveled by an individual fluid particle over some time period.

Pathlines are the easiest of the flow patterns to understand. A pathline is a Lagrangian concept in that we simply follow the path of an individual fluid particle as it moves around in the flow field (Fig. 3–3). Thus, a pathline is the same as the fluid particle's material position vector ($x_{particle}(t)$, $y_{particle}(t)$, $z_{particle}(t)$), traced out over some finite time interval. In a physical experiment, you can imagine a tracer fluid particle that is marked somehow—either by color or brightness—such that it is easily distinguishable from surrounding fluid particles. Now imagine a camera with the shutter open for a certain time period, $t_{start} > t > t_{end}$, in which the particle's path is recorded; the resulting curve is called a pathline. An intriguing example is shown in Fig. 3–3 for the case of waves moving along the surface of water in a tank. Neutrally buoyant white **tracer particles** are suspended in the water, and a time-exposure photograph is taken for one complete wave period. The result is pathlines that are elliptical in shape, showing that fluid particles bob up and down and forward and backward, but return to their original position upon completion of one wave period; there is no net forward motion. You may have experienced something similar while bobbing up and down on ocean waves.

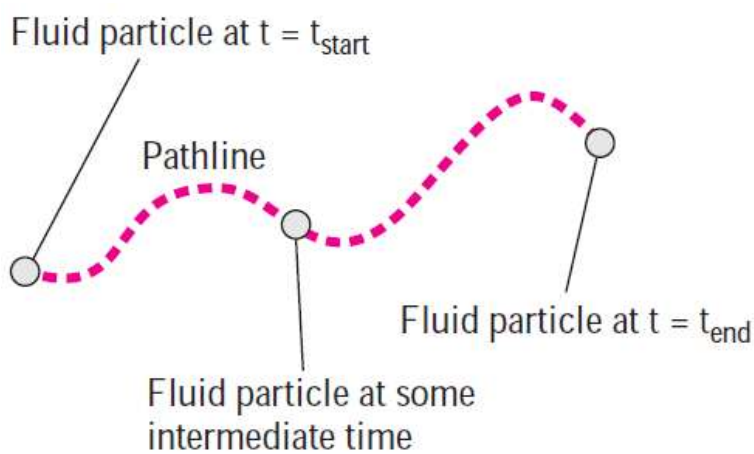



**Figure 3.3:** A pathline is formed by following the actual path of a fluid particle.

University of Anbar
College of Engineering
Mechanical Engineering Dept.

# Fluid Mechanics

## Handout Lectures for Year Two
## Chapter Three/ Mass, Bernoulli, and Energy Equations

**Course Tutor**

**Assist. Prof. Dr. Waleed M. Abed**

# Chapter Three

# Mass, Bernoulli, and Energy Equations

## 1.1 INTRODUCTION

You are already familiar with numerous ***conservation laws*** such as the laws of conservation of mass, conservation of energy, and conservation of momentum. Historically, the conservation laws are first applied to a fixed quantity of matter called a closed system or just a system, and then extended to regions in space called control volumes. The conservation relations are also called balance equations since any conserved quantity must balance during a process. We now give a brief description of the conservation of mass, momentum, and energy relations.

## 1.2 Conservation of Mass Principle

The conservation of mass principle for a control volume can be expressed as: The net mass transfer to or from a control volume during a time interval $\Delta t$ is equal to the net change (increase or decrease) in the total mass within the control volume during $\Delta t$. That is,

$$\begin{pmatrix}\text{Total mass entering}\\ \text{the CV during }\Delta t\end{pmatrix} - \begin{pmatrix}\text{Total mass leaving}\\ \text{the CV during }\Delta t\end{pmatrix} = \begin{pmatrix}\text{Net change in mass}\\ \text{within the CV during }\Delta t\end{pmatrix}$$

Or, $m_{in} - m_{out} = \Delta m_{CV}$ (kg) (3.1)

It can also be expressed in rate form as,

$\dot{m}_{in} - \dot{m}_{out} = dm_{CV}/dt$ (kg/s) (3.2) where $\dot{m}_{in}$ and m$\dot{m}_{out}$out are the total rates of mass flow into and out of the control volume, and $dm_{CV}/dt$ is the rate of change of mass within the control volume boundaries. Equations 3–1 and 3–2 are often referred to as the ***mass balance*** and are applicable to any control volume undergoing any kind of process.

Consider a control volume of arbitrary shape, as shown in Figure 3–1. The mass of a differential volume $dV$ within the control volume is d$m$= $\rho dV$. The total mass within the control volume at any instant in time $t$ is determined by integration to be

Total mass within the CV:

$$m_{CV} = \int_{CV} \rho \, dV \qquad (3.3)$$

Then the time rate of change of the amount of mass within the control volume can be expressed as

Rate of change of mass within the CV:

$$\frac{dm_{CV}}{dt} = \frac{d}{dt}\int_{CV} \rho \, dV \qquad (3.4)$$

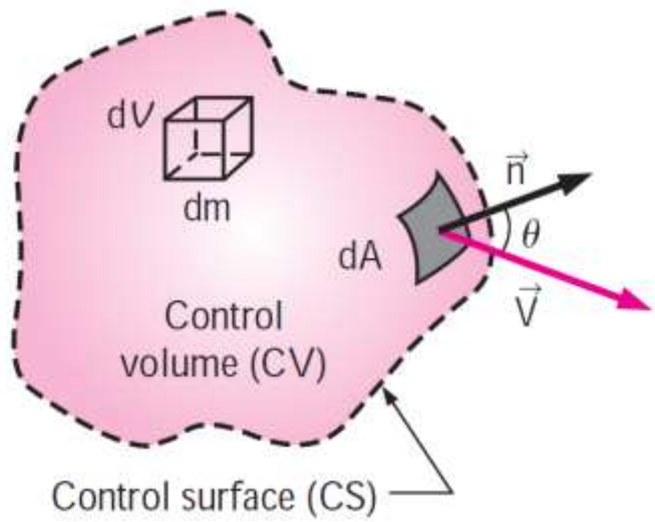



**Figure 3.1:** The differential control volume $dV$ and the differential control surface $dA$ used in the derivation of the conservation of mass relation.

Using the definition of mass flow rate as,

$$\frac{d}{dt}\int_{CV} \rho \, dV = \sum_{in} \dot{m} - \sum_{out} \dot{m} \quad \text{or} \quad \frac{dm_{CV}}{dt} = \sum_{in} \dot{m} - \sum_{out} \dot{m} \qquad (3.5)$$

There is considerable flexibility in the selection of a control volume when solving a problem. Several control volume choices may be correct, but some are more convenient to work with. A control volume should not introduce any unnecessary complications. The proper choice of a control volume can make the solution of a seemingly complicated problem rather easy. A simple rule in selecting a control volume is to make the control surface normal to flow at all locations where it crosses fluid flow, whenever possible.

### 1.3 Mass Balance for Steady-Flow Processes

During a steady-flow process, the total amount of mass contained within a control volume does not change with time ($m_{CV}$= constant). Then the conservation of mass principle requires that the total amount of mass entering a control volume equal the total amount of mass leaving it. For a garden hose nozzle in steady operation, for example, the amount of water entering the nozzle per unit time is equal to the amount of water leaving it per unit time. When dealing with steady-flow processes, we are not interested in the amount of mass that flows in or out of a device over time; instead, we are interested in the amount of mass flowing per unit time, that is, the mass flow rate $\dot{m}$. The conservation of mass principle for a general steady-flow system with multiple inlets and outlets can be expressed in rate form as (Figure 3.2)

**Steady flow:** $\sum_{in} \dot{m} = \sum_{out} \dot{m}$ (kg/s) (3.6)

It states that the total rate of mass entering a control volume is equal to the total rate of mass leaving it.

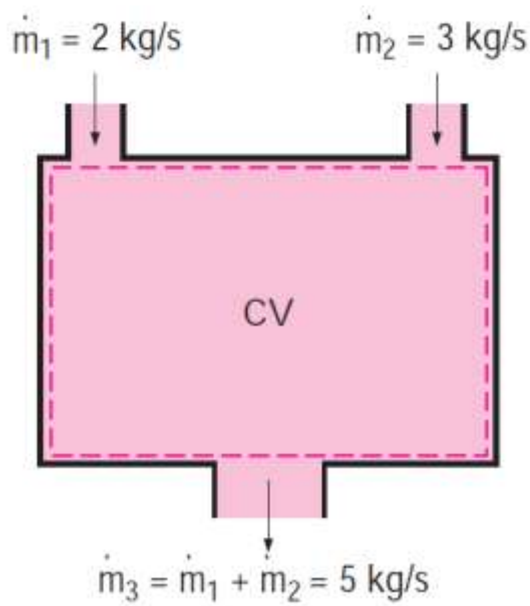



**Figure 3.2:** Conservation of mass principle for a two-inlet–one-outlet steady-flow system.

Many engineering devices such as nozzles, diffusers, turbines, compressors, and pumps involve a single stream (only one inlet and one outlet). For these cases, we denote the inlet state by the subscript 1 and the outlet state by the subscript 2, and drop the summation signs. Then Eq. 3.6 reduces, for single-stream steady-flow systems, to

**Steady flow (single stream):** $\dot{m}_1 = \dot{m}_2 \quad \Rightarrow \quad \rho_1 V_1 A_1 = \rho_2 V_2 A_2$ (3.7)

**Special Case: Incompressible Flow**

The conservation of mass relations can be simplified even further when the fluid is incompressible, which is usually the case for liquids. Canceling the density from both sides of the general steady-flow relation gives

**Steady, incompressible flow:** $\sum_{in} \dot{V} = \sum_{out} \dot{V}$ ($m^3/s$) (3.8)

For single-stream steady-flow systems it becomes

**Steady, incompressible flow (single stream):** $\dot{V}_1 = \dot{V}_2 \Rightarrow V_1 A_1 = V_2 A_2$ (3.9)

It should always be kept in mind that there is no such thing as a "*conservation of volume*" principle. Therefore, the volume flow rates into and out of a steady-flow device may be different. The volume flow rate at the outlet of an air compressor is much less than that at the inlet even though the mass flow rate of air through the compressor is constant (Figure 3.3). This is due to the higher density of air at the compressor exit. For steady flow of liquids, however, the volume flow rates, as well as the mass flow rates, remain constant since liquids are essentially incompressible (constant-density) substances. Water flow through the nozzle of a garden hose is an example of the latter case.

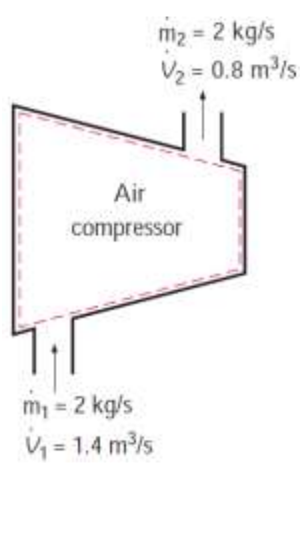



**Figure 3.3:** During a steady-flow process, volume flow rates are not necessarily conserved although mass flow rates are.

**Example 3–1:** A garden hose attached with a nozzle is used to fill a 10-gal bucket. The inner diameter of the hose is 2 cm, and it reduces to 0.8 cm at the nozzle exit. If it takes 50 s to fill the bucket with water, determine (***a***) the volume and mass flow rates of water through the hose, and (***b***) the average velocity of water at the nozzle exit.

**SOLUTION** A garden hose is used to fill a water bucket. The volume and mass flow rates of water and the exit velocity are to be determined.

***Assumptions*** **1** Water is an incompressible substance. **2** Flow through the hose is steady. **3** There is no waste of water by splashing.

***Properties*** We take the density of water to be 1000 kg/m³ = 1 kg/L.

***Analysis*** (*a*) Noting that 10 gal of water are discharged in 50 s, the volume and mass flow rates of water are

$$\dot{V} = \frac{V}{\Delta t} = \frac{10\text{ gal}}{50\text{ s}}\left(\frac{3.7854\text{ L}}{1\text{ gal}}\right) = \mathbf{0.757\ L/s}$$

$$\dot{m} = \rho\dot{V} = (1\text{ kg/L})(0.757\text{ L/s}) = \mathbf{0.757\ kg/s}$$

(*b*) The cross-sectional area of the nozzle exit is

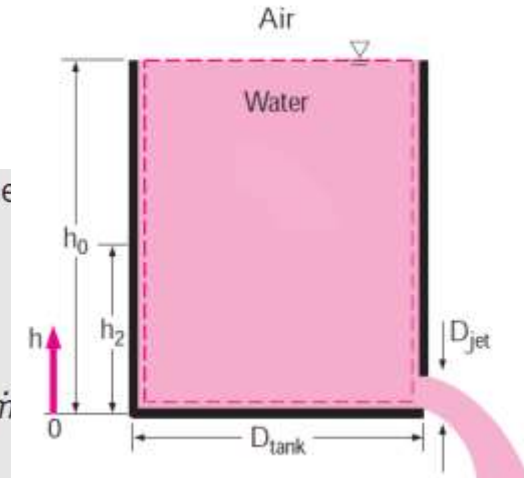
$$A_e = \pi r_e^2 = \pi(0.4\text{ cm})^2 = 0.5027\text{ cm}^2 = 0.5027 \times 10^{-4}\text{ m}^2$$

The volume flow rate through the hose and the nozzle is constant. Then the average velocity of water at the nozzle exit becomes

$$V_e = \frac{\dot{V}}{A_e} = \frac{0.757\text{ L/s}}{0.5027 \times 10^{-4}\text{ m}^2}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right) = \mathbf{15.1\ m/s}$$

**Example 3–2:** A 4-ft-high, 3-ft-diameter cylindrical water tank whose top is open to the atmosphere is initially filled with water. Now the discharge plug near the bottom of the tank is pulled out, and a water jet whose diameter is 0.5 in streams out (Fig. 3.4). The average velocity of the jet is given by $= \sqrt{2gh}$ , where $h$ is the height of water in the tank measured from the center of the hole (a variable) and $g$

is the gravitational acceleration. Determine how long it will take for the water level in the tank to drop to 2 ft from the bottom.




**Solution:**

The conservation of mass relation for a control volume process is given in the rate form as

$$\dot{m}_{in} - \dot{m}_{out} = \frac{dm_{CV}}{dt}$$

During this process no mass enters the control volume ($\dot{m}$ mass flow rate of discharged water can be expressed as

$$\dot{m}_{out} = (\rho VA)_{out} = \rho\sqrt{2gh}A_{jet} \qquad (2)$$

where $A_{jet} = \pi D_{jet}^2/4$ is the cross-sectional area of the jet, which is constant. Noting that the density of water is constant, the mass of water in the tank at any time is

$$m_{CV} = \rho V = \rho A_{tank}h \qquad (3)$$

where $A_{tank} = \pi D_{tank}^2/4$ is the base area of the cylindrical tank. Substituting Eqs. 2 and 3 into the mass balance relation (Eq. 1) gives

$$-\rho\sqrt{2gh}A_{jet} = \frac{d(\rho A_{tank}h)}{dt} \rightarrow -\rho\sqrt{2gh}(\pi D_{jet}^2/4) = \frac{\rho(\pi D_{tank}^2/4)\,dh}{dt}$$

Canceling the densities and other common terms and separating the variables give

$$dt = -\frac{D_{tank}^2}{D_{jet}^2}\frac{dh}{\sqrt{2gh}}$$

Integrating from $t = 0$ at which $h = h_0$ to $t = t$ at which $h = h_2$ gives

$$\int_0^t dt = -\frac{D_{tank}^2}{D_{jet}^2\sqrt{2g}}\int_{h_0}^{h_2}\frac{dh}{\sqrt{h}} \rightarrow t = \frac{\sqrt{h_0} - \sqrt{h_2}}{\sqrt{g/2}}\left(\frac{D_{tank}}{D_{jet}}\right)^2$$

Substituting, the time of discharge is determined to be

$$t = \frac{\sqrt{4\text{ ft}} - \sqrt{2\text{ ft}}}{\sqrt{32.2/2\text{ ft/s}^2}}\left(\frac{3 \times 12\text{ in}}{0.5\text{ in}}\right)^2 = 757\text{ s} = \mathbf{12.6\ min}$$

Therefore, half of the tank will be emptied in 12.6 min after the discharge hole is unplugged.

### 1.4 The Bernoulli equation

The ***Bernoulli equation*** is *an approximate relation between pressure, velocity, and elevation*, and is valid in *regions of steady, incompressible flow where net frictional forces are negligible* (Fig. 3-4). Despite its simplicity, it has proven to be a very powerful tool in fluid mechanics. In this section, we derive the Bernoulli equation by applying the *conservation of linear momentum principle*, and we demonstrate both its usefulness and its limitations.

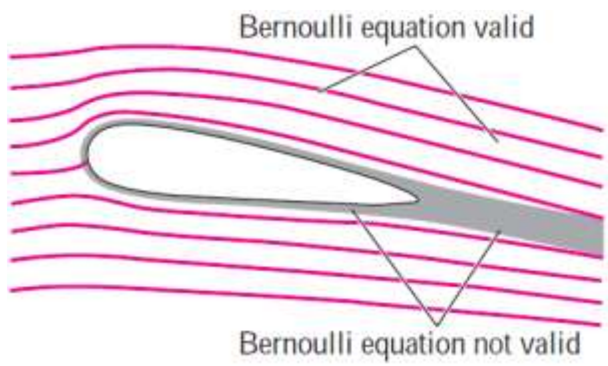



**Figure 3.4:** The Bernoulli equation is an approximate equation that is valid only in inviscid regions of flow where net viscous forces are negligibly small compared to inertial, gravitational, or pressure forces. Such regions occur

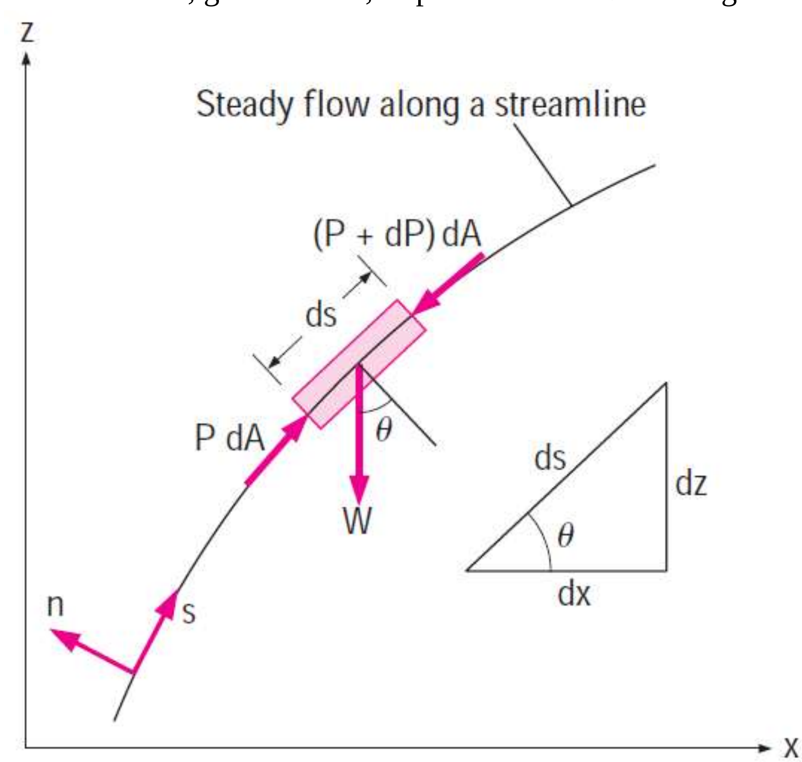



**Figure 3.5:** The forces acting on a fluid particle along a streamline.

Consider the motion of a fluid particle in a flow field in steady flow described in detail. Applying Newton's second law (which is referred to as the conservation of linear momentum relation in fluid mechanics) in the s-direction on a particle moving along a streamline gives,

$$\sum F_s = ma_s \qquad (3.10)$$

In regions of flow where net frictional forces are negligible, the significant forces acting in the ***s***-direction are the pressure (acting on both sides) and the component of the weight of the particle in the ***s***-direction (Figure 3-5). Therefore, Equation 3-10 becomes

$$PdA - (P + dP)dA - W\sin\theta = mV\frac{dV}{ds} \qquad (3.11)$$

where $\theta$ is the angle between the normal of the streamline and the vertical $z$-axis at that point, $m = \rho V = \rho dAds$ is the mass, $W = mg = \rho gdAds$ is the weight of the fluid particle, and $\sin\theta = dz/ds$. Substituting,

$$-dPdA - \rho gdA\, ds\,\frac{dz}{ds} = \rho dA\, dsV\frac{dV}{ds} \qquad (3.12)$$

Canceling $dA$ from each term and simplifying,

$$-dP - \rho g\, dz = \rho VdV \qquad (3.13)$$

Noting that $VdV = 0.5\, d(V^2)$ and dividing each term by $\rho$ gives

$$\frac{dP}{\rho} + \frac{1}{2}d(V^2) + gdz = 0 \qquad (3.14)$$

Integrating

***Steady flow:*** $\int\frac{dP}{\rho} + \frac{V^2}{2} + gz = \ constant\ (along\ a\ streamline)$ (3.15)

since the last two terms are exact differentials. In the case of incompressible flow, the first term also becomes an exact differential, and its integration gives

***Steady, incompressible flow:*** $\frac{P}{\rho} + \frac{V^2}{2} + gz = \ constant$ (3.16)

This is the famous ***Bernoulli equation***, which is commonly used in fluid mechanics for steady, incompressible flow along a streamline in inviscid regions of flow. The value of the constant can be evaluated at any point on the streamline where the pressure, density, velocity, and elevation are known. The ***Bernoulli equation*** can also be written between any two points on the same streamline as

***Steady, incompressible flow:*** $\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2$ (3.17)

The Bernoulli equation is obtained from the conservation of momentum for a fluid particle moving along a streamline. It can also be obtained from the *first law of thermodynamics* applied to a steady-flow system.

**The Bernoulli Equation According to Static, Dynamic, and Stagnation Pressures**

The Bernoulli equation states that the sum of the flow, kinetic, and potential energies of a fluid particle along a streamline is constant. Therefore, the kinetic and potential energies of the fluid can be converted to flow energy (and vice versa) during flow, causing the pressure to change. This phenomenon can be made more visible by multiplying the Bernoulli equation by the density *ρ*,

$P + \rho \frac{V^2}{2} + \rho g z = \text{constant}$ (along a streamline) (3.18)

Each term in this equation has pressure units, and thus each term represents some kind of pressure:

- ✓ $P$ is the ***static pressure*** (it does not incorporate any dynamic effects); it represents the actual thermodynamic pressure of the fluid. This is the same as the pressure used in thermodynamics and property tables.
- ✓ $\rho V^2/2$ is the ***dynamic pressure;*** it represents the pressure rise when the fluid in motion is brought to a stop isentropically.
- ✓ $\rho g z$ is the ***hydrostatic pressure,*** which is not pressure in a real sense since its value depends on the reference level selected; it accounts for the elevation effects, i.e., of fluid weight on pressure.

The sum of the static, dynamic, and hydrostatic pressures is called the **total pressure.** Therefore, the Bernoulli equation states that *the total pressure along a streamline is constant*.

The sum of the static and dynamic pressures is called the **stagnation pressure,** and it is expressed as

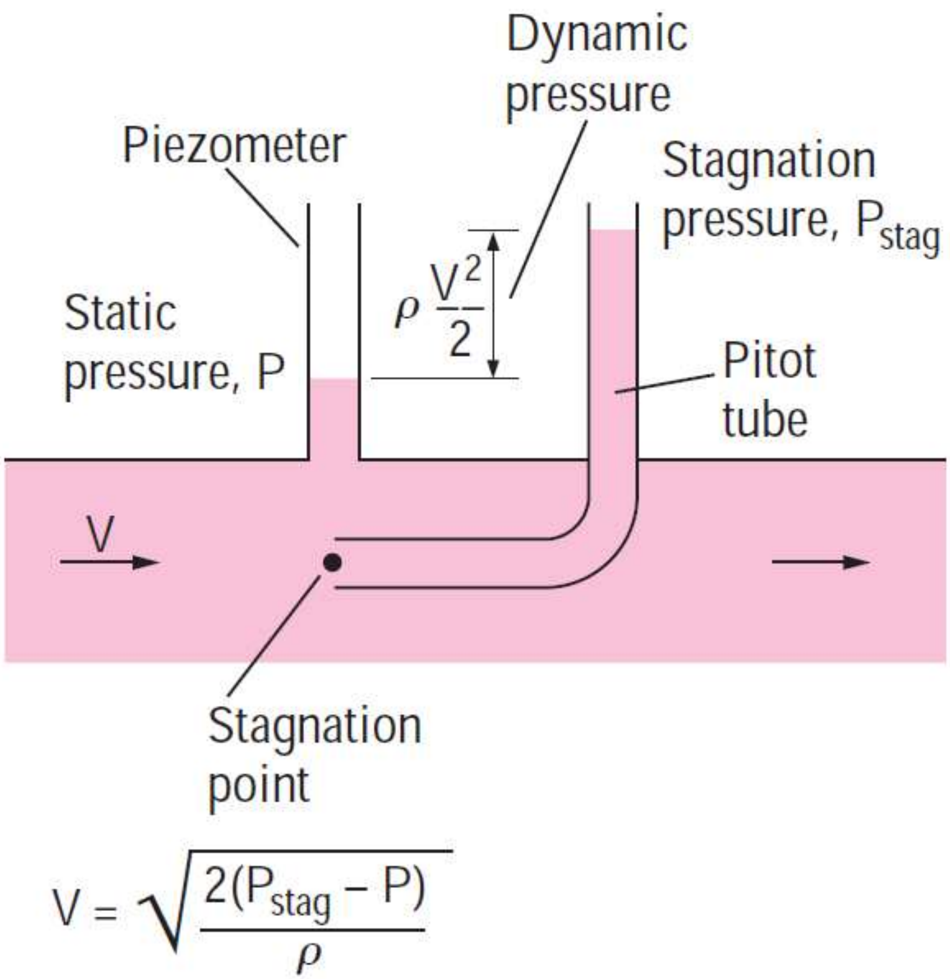



**Figure 3.6:** The static, dynamic, and stagnation pressures.

$$P_{\text{Stagnation}} = P + \rho \frac{V^2}{2} \qquad \text{(kPa)} \qquad (3.19)$$

The stagnation pressure represents the pressure at a point where the fluid is brought to a complete stop isentropically. The static, dynamic, and stagnation pressures are shown in Figure 3.6. When static and stagnation pressures are measured at a specified location, ***the fluid velocity*** at that location can be calculated from

$$V = \sqrt{\frac{2(P_{\text{Stagnation}} - P)}{\rho}} \qquad \text{(m/s)} \qquad (3.20)$$

**Example 1:**

Water is flowing from a hose attached to a water main at 400 kPa gage (Figure 3.7). A child places his thumb to cover most of the hose outlet, causing a thin jet of high-speed water to emerge. If the hose is held upward, what is the maximum height that the jet could achieve?

***Solution:***

The water height will be maximum under the stated assumptions. The velocity inside the hose is relatively low ($V_1 = 0$) and we take the hose outlet as the reference level ($z_1 = 0$). At the top of the water trajectory $V_2 = 0$, and atmospheric pressure pertains. Then the Bernoulli equation simplifies to

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

Solving for $z_2$ and substituting,

$$z_2 = \frac{P_1 - P_{atm}}{\rho g} = \frac{P_{1,\,gage}}{\rho g} = \frac{400\text{ kPa}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1000\text{ N/m}^2}{1\text{ kPa}}\right)\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)$$

$$= \mathbf{40.8\text{ m}}$$


**Figure 3.7**

**Example 2:**

A large tank open to the atmosphere is filled with water to a height of 5 m from the outlet tap (Figure 3.8 ). A tap near the bottom of the tank is now opened, and water flows out from the smooth and rounded outlet. Determine the water velocity at the outlet.

***Solution:***

We take point 1 to be at the free surface of water so that $P_1$ = $P$atm (open to the atmosphere), $V_1$ = 0 (the tank is large relative to the outlet), and $z_1$ = 5 m and $z_2$ = 0 (we take the reference level at the center of the outlet). Also, $P_2$ = $P$atm (water discharges into the atmosphere). Then the Bernoulli equation simplifies to

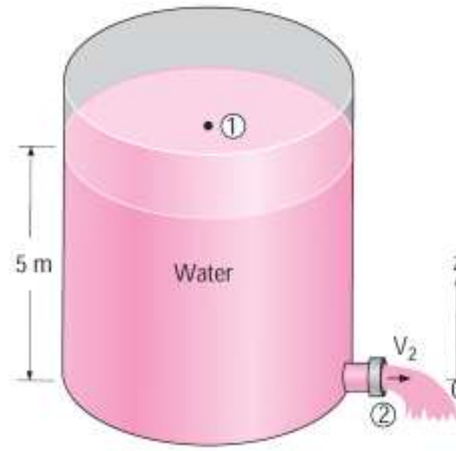



**Figure 3.8**

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

Solving for $V_2$ and substituting,

$$V_2 = \sqrt{2gz_1} = \sqrt{2(9.81 \text{ m/s}^2)(5 \text{ m})} = \mathbf{9.9 \text{ m/s}}$$

The relation $V = \sqrt{2gz}$ is called the **Toricelli equation.**

Therefore, the water leaves the tank with an initial velocity of 9.9 m/s. This is the same velocity that would manifest if a solid were dropped a distance of 5 m in the absence of air friction drag. (What would the velocity be if the tap were at the bottom of the tank instead of on the side?)

**Example 3:**

During a trip to the beach ($P$atm = 1 atm = 101.3 kPa), a car runs out of gasoline, and it becomes necessary to siphon gas out of the car of a Good Samaritan (Figure 3.9). The siphon is a small-diameter hose, and to start the siphon it is necessary to insert one siphon end in the full gas tank, fill the hose with gasoline via suction, and then place the other end in a gas can below the level of the gas tank. The difference in pressure between point 1 (at the free surface of the gasoline in the tank) and point 2 (at the outlet of the tube) causes the liquid to flow from the higher to the lower elevation. Point 2 is located 0.75 m below point 1 in this case, and point 3 is located 2 m above point 1. The siphon diameter is 4 mm, and frictional losses in the siphon are to be disregarded. Determine (***a***) the minimum time to withdraw 4 L of gasoline from the tank to the can and (***b***) the pressure at point 3. The density of gasoline is 750 kg/m$^3$.

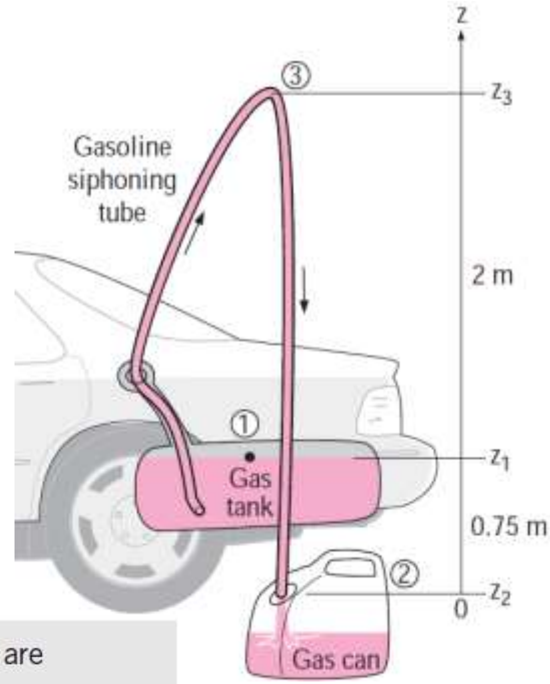



**Figure 3.9**

**S*olution:***

*(a)* We take point 1 to be at the free surface of gasoline in the tank so that $P_1$ = *P*atm (open to the atmosphere), $V_1$ = 0 (the tank is large relative to the tube diameter), and $z_2$ = 0 (point 2 is taken as the reference level). Also, $P_2$ = *P*atm (gasoline

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

Solving for $V_2$ and substituting,

$$V_2=\sqrt{2gz_1}=\sqrt{2(9.81\ \text{m/s}^2)(0.75\ \text{m})}=3.84\ \text{m/s}$$

The cross-sectional area of the tube and the flow rate of gasoline are

$$A=\pi D^2/4=\pi(5\times10^{-3}\ \text{m})^2/4=1.96\times10^{-5}\ \text{m}^2$$

$$\dot{V}=V_2A=(3.84\ \text{m/s})(1.96\times10^{-5}\ \text{m}^2)=7.53\times10^{-5}\ \text{m}^3/\text{s}=0.0753\ \text{L/s}$$

Then the time needed to siphon 4 L of gasoline becomes

$$\Delta t=\frac{V}{\dot{V}}=\frac{4\ \text{L}}{0.0753\ \text{L/s}}=\mathbf{53.1\ s}$$

(*b*) The pressure at point 3 can be determined by writing the Bernoulli equation between points 2 and 3. Noting that $V_2 = V_3$ (conservation of mass), $z_2 = 0$, and $P_2 = P_{atm}$,

$$\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2\nearrow^0=\frac{P_3}{\rho g}+\frac{V_3^2}{2g}+z_3 \quad\rightarrow\quad \frac{P_{atm}}{\rho g}=\frac{P_3}{\rho g}+z_3$$

Solving for $P_3$ and substituting,

$$P_3=P_{atm}-\rho g z_3$$

$$=101.3\ \text{kPa}-(750\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(2.75\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$

$$=\mathbf{81.1\ kPa}$$

**Example 4:**

A piezometer and a ***Pitot tube*** are tapped into a horizontal water pipe, as shown in Figure 3-10, to measure static and stagnation (static + dynamic) pressures. For the indicated water column heights, determine the velocity at the c enter of the pipe.

***Solution:***

We take points 1 and 2 along the centerline of the pipe, with point 1 directly under the piezometer and point 2 at the tip of the Pitot tube. This is a steady flow with straight and parallel streamlines, and the gage pressures at points 1 and 2 can be expressed as

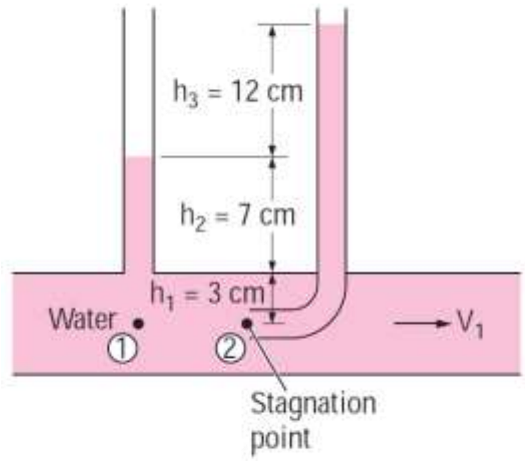



**Figure 3.10, Schematic for Example**

$$P_1 = \rho g(h_1 + h_2)$$

$$P_2 = \rho g(h_1 + h_2 + h_3)$$

Noting that point 2 is a stagnation point and thus $V_2 = 0$ and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

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

Substituting the $P_1$ and $P_2$ expressions gives

$$\frac{V_1^2}{2g} = \frac{P_2 - P_1}{\rho g} = \frac{\rho g(h_1 + h_2 + h_3) - \rho g(h_1 + h_2)}{\rho g} = h_3$$

Solving for $V_1$ and substituting,

$$V_1 = \sqrt{2gh_3} = \sqrt{2(9.81 \text{ m/s}^2)(0.12 \text{ m})} = \mathbf{1.53 \text{ m/s}}$$

### 1.5 Mechanical energy and efficiency

The *mechanical energy* can be defined as the form of energy that can be converted to mechanical work completely and directly by an ideal mechanical device such as an ideal turbine. Kinetic and potential energies are the familiar forms of mechanical energy. Thermal energy is not mechanical energy, however, since it cannot be converted to work directly and completely (the second law of thermodynamics).

A ***pump*** transfers mechanical energy to a fluid by raising its pressure, and a ***turbine*** extracts mechanical energy from a fluid by dropping its pressure. Therefore, the pressure of a flowing fluid is also associated with its mechanical energy.

The steady-flow energy equation on a unit-mass basis can be written conveniently as a mechanical energy balance as,

$$w_{\text{shaft, net in}} + \frac{P_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 = \frac{P_2}{\rho_2} + \frac{V_2^2}{2} + gz_2 + e_{\text{mech, loss}}$$

Noting that $W_{\text{shaft, net in}} = W_{\text{shaft, in}} - W_{\text{shaft, out}} = W_{\text{pump}} - W_{\text{turbine}}$, the mechanical energy balance can be written more explicitly as,

$$\frac{P_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 + w_{\text{pump}} = \frac{P_2}{\rho_2} + \frac{V_2^2}{2} + gz_2 + w_{\text{turbine}} + e_{\text{mech, loss}}$$

where $W_{\text{pump}}$ is the mechanical work input (due to the presence of a pump, fan, compressor, etc.) and $W_{\text{turbine}}$ is the mechanical work output. When the flow is incompressible, either absolute or gage pressure can be used for $P$ since $P_{\text{atm}}/\rho$ would appear on both sides and would cancel out. $e_{\text{mech, loss}}$ is the *total* mechanical power loss, which consists of pump and turbine losses as well as the frictional losses in the piping network. Multiplying above Equation by the mass flow rate $\dot{m}$ gives:

$$\dot{m}\left(\frac{P_1}{\rho_1} + \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_2}{\rho_2} + \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech, loss}}$$

By convention, irreversible pump and turbine losses are treated separately from irreversible losses due to other components of the piping system. Thus the energy equation can be expressed in its most common form in terms of heads as,

$$\frac{P_1}{\rho_1 g} + \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho_2 g} + \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L$$

where $h_{\text{pump, u}} = \frac{w_{\text{pump, u}}}{g} = \frac{\dot{W}_{\text{pump, u}}}{\dot{m}g} = \frac{\eta_{\text{pump}}\dot{W}_{\text{pump}}}{\dot{m}g}$ is the useful head delivered to the fluid by the pump. Because of irreversible losses in the pump, $h_{\text{pump, u}}$ is less than $\dot{W}_{\text{pump}}/\dot{m}g$ by the factor $\eta_{\text{pump}}$. Similarly, $h_{\text{turbine, e}} = \frac{w_{\text{turbine, e}}}{g} = \frac{\dot{W}_{\text{turbine, e}}}{\dot{m}g} = \frac{\dot{W}_{\text{turbine}}}{\eta_{\text{turbine}}\dot{m}g}$ is the extracted head removed from the fluid by the turbine. Because of irreversible losses in the turbine, $h_{\text{turbine, e}}$ is greater than $\dot{W}_{\text{turbine}}/\dot{m}g$ by the factor $\eta_{\text{turbine}}$. Finally, $h_L = \frac{e_{\text{mech loss, piping}}}{g} = \frac{\dot{E}_{\text{mech loss, piping}}}{\dot{m}g}$ is the irreversible head loss between 1 and 2 due to all components of the piping system other than the pump or turbine.

**Example 5:**

The pump of a water distribution system is powered by a 15-kW electric motor whose efficiency is 90 percent (**Figure 3.11**). The water flow rate through the pump is 50 L/s. The diameters of the inlet and outlet pipes are the same, and the elevation difference across the pump is negligible. If the pressures at the inlet and outlet of the pump are measured to be 100 kPa and 300 kPa (absolute), respectively, determine (**a**) the mechanical efficiency of the pump and (**b**) the temperature rise of water as it flows through the pump due to the mechanical inefficiency.

***Solution:***

**1** The flow is steady and incompressible.
**2** The pump is driven by an external motor so that the heat generated by the motor is dissipated to the atmosphere.
**3** The elevation difference between the inlet and outlet of the pump is negligible, $z_1 \approx z_2$.
**4** The inlet and outlet diameters are the same and thus the inlet and outlet velocities and kinetic energy correction factors are equal, $V_1 = V_2$.

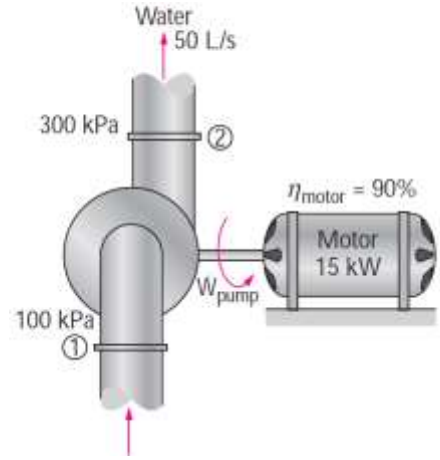



**Figure 3.11**

(**a**) The mass flow rate of water through the pump is

$$\dot{m} = \rho \dot{V} = (1 \text{ kg/L})(50 \text{ L/s}) = 50 \text{ kg/s}$$

The motor draws 15 kW of power and is 90 percent efficient. Thus the mechanical (shaft) power it delivers to the pump is

$$\dot{W}_{pump,\ shaft} = \eta_{motor}\dot{W}_{electric} = (0.90)(15 \text{ kW}) = 13.5 \text{ kW}$$

To determine the mechanical efficiency of the pump, we need to know the increase in the mechanical energy of the fluid as it flows through the pump, which is

$$\Delta\dot{E}_{mech,\ fluid} = \dot{E}_{mech,\ out} - \dot{E}_{mech,\ in} = \dot{m}\left(\frac{P_2}{\rho} + \alpha_2\frac{V_2^2}{2} + gz_2\right) - \dot{m}\left(\frac{P_1}{\rho} + \alpha_1\frac{V_1^2}{2} + gz_1\right)$$

Where α is the kinetic energy correction factor.

Simplifying it for this case and substituting the given values,

$$\Delta\dot{E}_{mech,\ fluid} = \dot{m}\left(\frac{P_2 - P_1}{\rho}\right) = (50 \text{ kg/s})\left(\frac{(300 - 100) \text{ kPa}}{1000 \text{ kg/m}^3}\right)\left(\frac{1 \text{ kJ}}{1 \text{ kPa} \cdot \text{m}^3}\right) = 10 \text{ kW}$$

Then the mechanical efficiency of the pump becomes

$$\eta_{pump} = \frac{\dot{W}_{pump,\ u}}{\dot{W}_{pump,\ shaft}} = \frac{\Delta\dot{E}_{mech,\ fluid}}{\dot{W}_{pump,\ shaft}} = \frac{10 \text{ kW}}{13.5 \text{ kW}} = 0.741 \quad \text{or} \quad 74.1\%$$

(**b**) Of the 13.5-kW mechanical power supplied by the pump, only 10 kW is imparted to the fluid as mechanical energy. The remaining 3.5 kW is converted to thermal energy due to frictional effects, and this “lost” mechanical energy manifests itself as a heating effect in the fluid,

$$\dot{E}_{\text{mech, loss}} = \dot{W}_{\text{pump, shaft}} - \Delta\dot{E}_{\text{mech, fluid}} = 13.5 - 10 = 3.5 \text{ kW}$$

The temperature rise of water due to this mechanical inefficiency is determined from the thermal energy balance, $\dot{E}_{\text{mech, loss}} = \dot{m}(u_2 - u_1) = \dot{m}c\Delta T.$

$$\Delta T = \frac{\dot{E}_{\text{mech, loss}}}{\dot{m}c} = \frac{3.5 \text{ kW}}{(50 \text{ kg/s})(4.18 \text{ kJ/ kg}\cdot{}^\circ\text{C})} = \mathbf{0.017^\circ C}$$

**Example 6:**

In a hydroelectric power plant, 100 $m^3$/s of water flows from an elevation of 120 m to a turbine, where electric power is generated (Figure 3-12). The total irreversible head loss in the piping system from point 1 to point 2 (excluding the turbine unit) is determined to be 35 m. If the overall efficiency of the turbine–generator is 80 percent, estimate the electric power output.

***Solution*** The mass flow rate of water through the turbine is

$$\dot{m} = \rho\dot{V} = (1000 \text{ kg/m}^3)(100 \text{ m}^3/\text{s}) = 10^5 \text{ kg/s}$$

We take point 2 as the reference level, and thus $z_2 = 0$. Also, both points 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the flow velocities are negligible at both points ($V_1 = V_2 = 0$). Then the energy equation for steady, incompressible flow reduces to

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

$$h_{\text{turbine, e}} = z_1 - h_L$$

Substituting, the extracted turbine head and the corresponding turbine power are

$$h_{\text{turbine, e}} = z_1 - h_L = 120 - 35 = 85 \text{ m}$$

$$\dot{W}_{\text{turbine, e}} = \dot{m}gh_{\text{turbine, e}} = (10^5 \text{ kg/s})(9.81 \text{ m/s}^2)(85 \text{ m})\left(\frac{1 \text{ kJ/kg}}{1000 \text{ m}^2/\text{s}^2}\right) = 83{,}400 \text{ kW}$$

Therefore, a perfect turbine–generator would generate 83,400 kW of electricity from this resource. The electric power generated by the actual unit is

$$\dot{W}_{electric} = \eta_{turbine-gen}\dot{W}_{turbine,\,e} = (0.80)(83.4\ \text{MW}) = \mathbf{66.7\ MW}$$

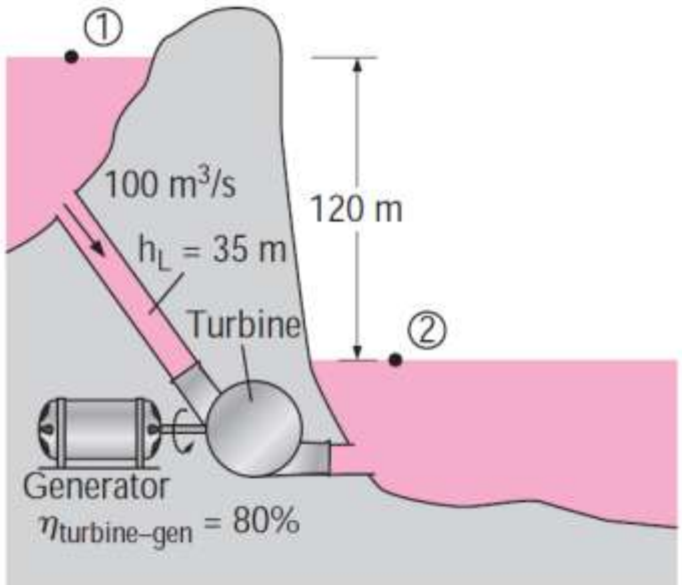



**Figure 3.12**

**Example 7:**

Water is pumped from a lower reservoir to a higher reservoir by a pump that provides 20 kW of useful mechanical power to the water (Figure 3.13). The free surface of the upper reservoir is 45 m higher than the surface of the lower reservoir. If the flow rate of water is measured to be 0.03 $m^3$/s, determine the irreversible head loss of the system and the lost mechanical power during this process.

***Solution:***

The mass flow rate of water through the system is

$$\dot{m} = \rho\dot{V} = (1000\ \text{kg/m}^3)(0.03\ \text{m}^3\text{/s}) = 30\ \text{kg/s}$$

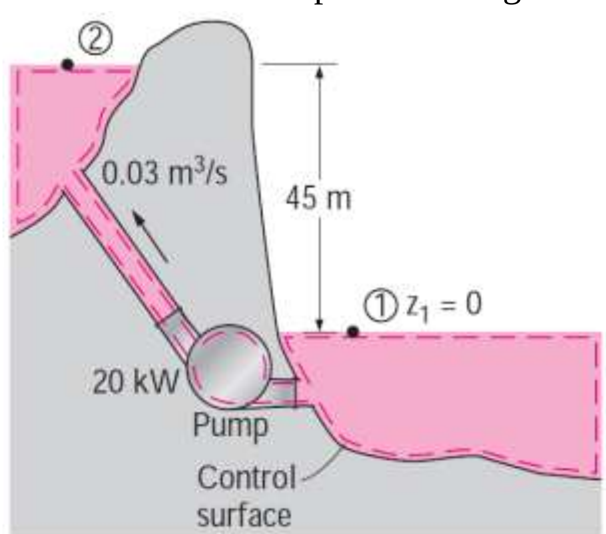



**Figure 3.13**

We choose points 1 and 2 at the free surfaces of the lower and upper reservoirs, respectively, and take the surface of the lower reservoir as the reference level ($z_1$ = 0). Both points are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the velocities at both locations are negligible ($V_1 = V_2 = 0$). Then the energy equation for steady incompressible flow for a control volume between 1 and 2 reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \alpha_1 \frac{V_1^2}{2}^{\nearrow 0} + gz_1^{\nearrow 0}\right) + \dot{W}_{pump} = \dot{m}\left(\frac{P_2}{\rho} + \alpha_2 \frac{V_2^2}{2}^{\nearrow 0} + gz_2\right) + \dot{W}_{turbine}^{\nearrow 0} + \dot{E}_{mech,\ loss}$$

$$\dot{W}_{pump} = \dot{m}gz_2 + \dot{E}_{mech,\ loss} \quad \rightarrow \quad \dot{E}_{mech,\ loss} = \dot{W}_{pump} - \dot{m}gz_2$$

Substituting, the lost mechanical power and head loss are determined to be

$$\dot{E}_{mech,\ loss} = 20\text{ kW} - (30\text{ kg/s})(9.81\text{ m/s}^2)(45\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right)$$

$$= \mathbf{6.76\ kW}$$

Noting that the entire mechanical losses are due to frictional losses in piping and thus $\dot{E}_{mech,\ loss} = \dot{E}_{mech,\ loss,\ piping}$, the irreversible head loss is determined to be

$$h_L = \frac{\dot{E}_{mech\ loss,\ piping}}{\dot{m}g} = \frac{6.76\text{ kW}}{(30\text{ kg/s})(9.81\text{ m/s}^2)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)\left(\frac{1000\text{ N}\cdot\text{m/s}}{1\text{ kW}}\right) = \mathbf{23.0\ m}$$

Consider a container of height h filled with water, as shown in Figure 3-14, with the reference level selected at the bottom surface. The gage pressure and the potential energy per unit mass are, respectively, $P_A = 0$ and $pe_A = gh$ at point A at the free surface, and $P_B = \rho gh$ and $pe_B = 0$ at point B at the bottom of the container. An ideal hydraulic turbine would produce the same work per unit mass $w_{turbine} = gh$ whether it receives water (or any other fluid with constant density) from the top or from the bottom of the container. Note that we are also assuming ideal flow (no irreversible losses) through the pipe leading from the tank to the turbine. Therefore, the total mechanical energy of water at the bottom is equivalent to that at the top.

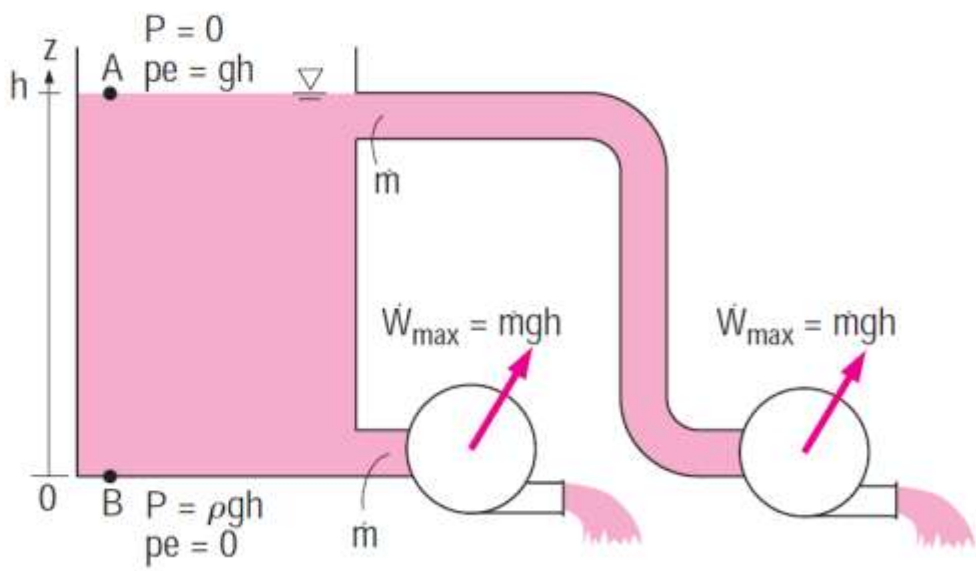



**Figure 3.14: The mechanical energy of water at the bottom of a container is equal to the mechanical energy at any depth including the free surface of the container.**

The transfer of mechanical energy is usually accomplished by a rotating shaft, and thus mechanical work is often referred to as shaft work. A pump or a fan receives shaft work (usually from an electric motor) and transfers it to the fluid as mechanical energy (less frictional losses). A turbine, on the other hand, converts the mechanical energy of a fluid to shaft work. In the absence of any irreversibilities such as friction, mechanical energy can be converted entirely from one mechanical form to another, and the ***mechanical efficiency*** of a device or process can be defined as,

$$\eta_{mech} = \frac{\text{Mechanical energy output}}{\text{Mechanical energy input}} = \frac{E_{mech,\ out}}{E_{mech,\ in}} = 1 - \frac{E_{mech,\ loss}}{E_{mech,\ in}}$$

A conversion efficiency of less than 100 percent indicates that conversion is less than perfect and some losses have occurred during conversion. A mechanical efficiency of 97 percent indicates that 3 percent of the mechanical energy input is converted to thermal energy as a result of frictional heating, and this will manifest itself as a slight rise in the temperature of the fluid.

The degree of perfection of the conversion process between the mechanical work supplied or extracted and the mechanical energy of the fluid is expressed by the ***pump efficiency*** and ***turbine efficiency***, defined as

$$\eta_{pump} = \frac{\text{Mechanical energy increase of the fluid}}{\text{Mechanical energy input}} = \frac{\Delta \dot{E}_{mech,\ fluid}}{\dot{W}_{shaft,\ in}} = \frac{\dot{W}_{pump,\ u}}{\dot{W}_{pump}}$$

where $\Delta E_{mech,fluid} = E_{mech,out} - E_{mech,\ in}$ is the rate of increase in the mechanical energy of the fluid, which is equivalent to the ***useful pumping power*** $W_{pump,\ u}$ supplied to the fluid, and

$$\eta_{turbine} = \frac{\text{Mechanical energy output}}{\text{Mechanical energy decrease of the fluid}} = \frac{W_{shaft,\ out}}{|\Delta \dot{E}_{mech,\ fluid}|} = \frac{W_{turbine}}{\dot{W}_{turbine,\ e}}$$

where $\Delta E_{mech,\ fluid} = E_{mech,\ in} - E_{mech,\ out}$ is the rate of decrease in the mechanical energy of the fluid, which is equivalent to the mechanical power extracted from the fluid by the turbine *W* turbine, *e*, and we use the absolute value sign to avoid negative values for efficiencies. A pump or turbine efficiency of 100 percent indicates perfect conversion between the shaft work and the mechanical energy of the fluid, and this value can be approached (but never attained) as the frictional effects are minimized.

**Example 8:** The water in a large lake is to be used to generate electricity by the installation of a hydraulic turbine–generator at a location where the depth of the water is 50 m (Figure **3.15**). Water is to be supplied at a rate of 5000 kg/s. If the electric power generated is measured to be 1862 kW and the generator efficiency is 95 percent, determine (***a***) the overall efficiency of the turbine– generator, (***b***) the mechanical efficiency of the turbine, and (***c***) the shaft power supplied by the turbine to the generator.

***Solution:***

(***a***) We take the bottom of the lake as the reference level for convenience. Then kinetic and potential energies of water are zero, and the change in its mechanical energy per unit mass becomes

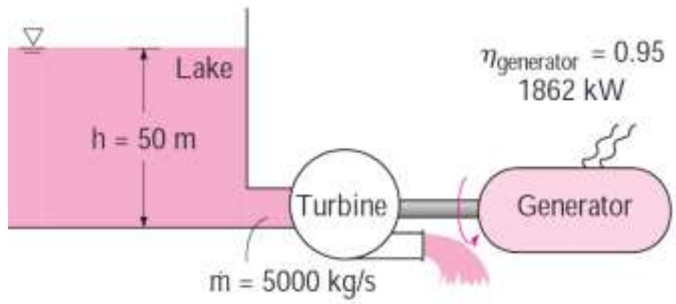



**Figure 3.15: Schematic for Example 8.**

$$e_{mech,\,in} - e_{mech,\,out} = \frac{P}{\rho} - 0 = gh = (9.81\ m/s^2)(50\ m)\left(\frac{1\ kJ/kg}{1000\ m^2/s^2}\right) = 0.491\ kJ/kg$$

Then the rate at which mechanical energy is supplied to the turbine by the fluid and the overall efficiency become

$$|\Delta \dot{E}_{mech,\,fluid}| = \dot{m}(e_{mech,\,in} - e_{mech,\,out}) = (5000\ kg/s)(0.491\ kJ/kg) = 2455\ kW$$

$$\eta_{overall} = \eta_{turbine-gen} = \frac{\dot{W}_{elect,\,out}}{|\Delta \dot{E}_{mech,\,fluid}|} = \frac{1862\ kW}{2455\ kW} = \mathbf{0.76}$$

(***b***) Knowing the overall and generator efficiencies, the mechanical efficiency of the turbine is determined from

$$\eta_{turbine-gen} = \eta_{turbine}\,\eta_{generator} \rightarrow \eta_{turbine} = \frac{\eta_{turbine-gen}}{\eta_{generator}} = \frac{0.76}{0.95} = \mathbf{0.80}$$

(***c***) The shaft power output is determined from the definition of mechanical efficiency,

$$\dot{W}_{shaft,\,out} = \eta_{turbine}|\Delta \dot{E}_{mech,\,fluid}| = (0.80)(2455\ kW) = \mathbf{1964\ kW}$$

### 1.6 The linear momentum equation

Newton's second law for a system of mass m subjected to a net force $\vec{F}$ is expressed as

$$\sum \vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d}{dt}(m\vec{V})$$

Where $m\vec{V}$ is the linear momentum of the system. Noting that both the density and velocity may change from point to point within the system, Newton's second law can be expressed more generally as

$$\sum \vec{F} = \frac{d}{dt}\int_{sys} \rho\vec{V}\, dV$$

where $\delta m = \rho\, dv$ is the mass of a differential volume element $dv$, and is its momentum. Therefore, Newton's second law can be stated as *the sum of all external forces acting on a system is equal to the time rate of change of linear momentum of the system*. This statement is valid for a coordinate system that is at rest or moves with a constant velocity, called an *inertial coordinate system* or *inertial reference frame*. Accelerating systems such as aircraft during takeoff are best analyzed using non-inertial (or accelerating) coordinate systems fixed to the aircraft. Note that the above equation is a vector relation, and thus the quantities $\vec{F}$ and $\vec{V}$ have direction as well as magnitude.

The general form of the linear momentum equation that applies to fixed, moving, or deforming control volumes is obtained to be

$$\begin{pmatrix}\text{The sum of all}\\ \text{external forces}\\ \text{acting on a CV}\end{pmatrix} = \begin{pmatrix}\text{The time rate of change}\\ \text{of the linear momentum}\\ \text{of the contents of the CV}\end{pmatrix} + \begin{pmatrix}\text{The net flow rate of}\\ \text{linear momentum out of the}\\ \text{control surface by mass flow}\end{pmatrix}$$

In General:

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V}\, dV + \int_{CS} \rho \vec{V}(\vec{V}_r \cdot \vec{n})\, dA$$

Note that the momentum equation is a *vector equation*, and thus each term should be treated as a vector. Also, the components of this equation can be resolved along orthogonal coordinates (such as *x*, *y*, and *z* in the Cartesian coordinate system) for convenience.

The above equation is exact for fixed control volumes, it is not always convenient when solving practical engineering problems because of the integrals. Instead, as we did for conservation of mass, we would like to rewrite the above equation in terms of average velocities and mass flow rates through inlets and outlets. In other words, our desire is to rewrite the equation in *algebraic* rather than *integral* form. In many practical applications, fluid crosses the boundaries of the control volume at one or more inlets and one or more outlets, and carries with it some momentum into or out of the control volume. For simplicity, we always draw our control surface such that it slices normal to the inflow or outflow velocity at each such inlet or outlet (Figure 3.16). The mass flow rate $\dot{m}$ into or out of the control volume across an inlet or outlet at which $\rho$ is nearly constant is

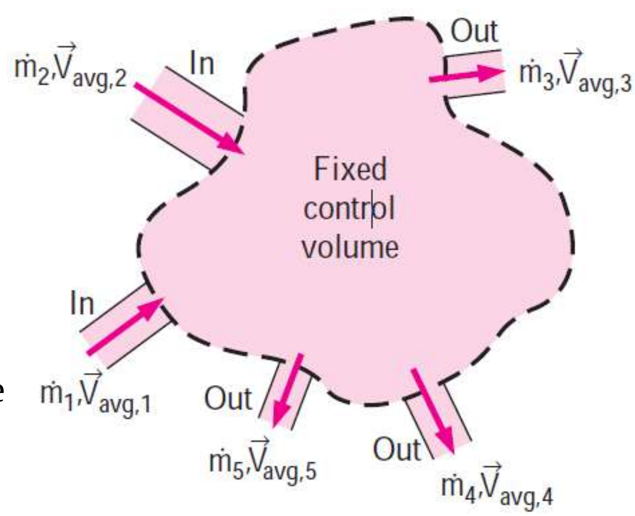



**Figure 3.16:** In a typical engineering problem, the control volume may contain many inlets and outlets; at each inlet or outlet we define the mass flow rate $\dot{m}$ and the average velocity $V_{avg}$.

Mass flow rate across an inlet or outlet:

$$\dot{m} = \int_{A_c} \rho(\vec{V} \cdot \vec{n})\, dA_c = \rho V_{avg} A_c$$

Then we could write the rate of inflow or outflow of momentum through the inlet or outlet in simple algebraic form, Momentum flow rate across a uniform inlet or outlet:

$$\int_{A_c} \rho\vec{V}(\vec{V} \cdot \vec{n})\, dA_c = \rho V_{avg} A_c \vec{V}_{avg} = \dot{m}\vec{V}_{avg}$$

The uniform flow approximation is reasonable at some inlets and outlets, e.g., the well-rounded entrance to a pipe, the flow at the entrance to a wind tunnel test section, and a slice through a water jet moving at nearly uniform speed through air (Figure 3-17).

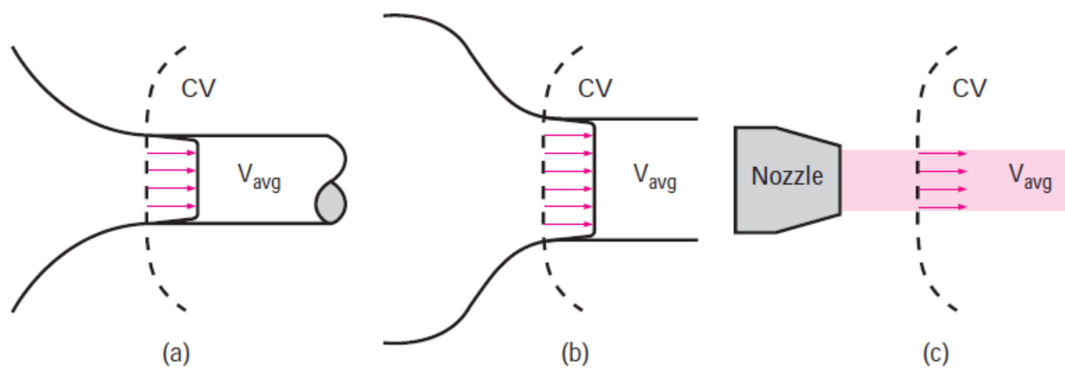



**Figure 3.17:** Examples of inlets or outlets in which the uniform flow approximation is reasonable: (a) the well-rounded entrance to a pipe, (b) the entrance to a wind tunnel test section, and (c) a slice through a free water jet in air.

### 1.7 Momentum-Flux Correction Factor, *β*

Unfortunately, the velocity across most inlets and outlets of practical engineering interest is not uniform. Nevertheless, it turns out that we can still convert the control surface integral of Equation,

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V}\, dV + \int_{CS} \rho \vec{V}(\vec{V}\cdot\vec{n})\, dA$$

into algebraic form, but a dimensionless correction factor b, called the momentum-flux correction factor, is required, as first shown by the French scientist *Joseph Boussinesq* (1842–1929). The algebraic form of the above equation for a fixed control volume is then written as,

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V}\, dV + \sum_{out} \beta \dot{m} \vec{V}_{avg} - \sum_{in} \beta \dot{m} \vec{V}_{avg}$$

where a unique value of momentum-flux correction factor is applied to each inlet and outlet in the control surface. Note that $\beta = 1$ for the case of uniform flow over an inlet or outlet, as in Figure 3-17.

Momentum-flux correction factor:

$$\beta = \frac{1}{A_c}\int_{A_c}\left(\frac{V}{V_{avg}}\right)^2 dA_c$$

It turns out that for any velocity profile you can imagine, $\beta$ is always greater than or equal to unity.

**Example 9:**

Consider laminar flow through a very long straight section of round pipe. The velocity profile through a cross-sectional area of the pipe is parabolic (Figure 3-18), with the axial velocity component given by

$$V = 2V_{avg}\left(1 - \frac{r^2}{R^2}\right)$$

where $R$ is the radius of the inner wall of the pipe and $V_{avg}$ is the average velocity. Calculate the momentum-flux correction factor through a cross section of the pipe for the case in which the pipe flow represents an outlet of the control volume, as sketched in Figure 3-18.

***Solution:***

We substitute the given velocity profile for $V$ in the above equation and integrate, noting that $dAc = 2\pi r dr$,

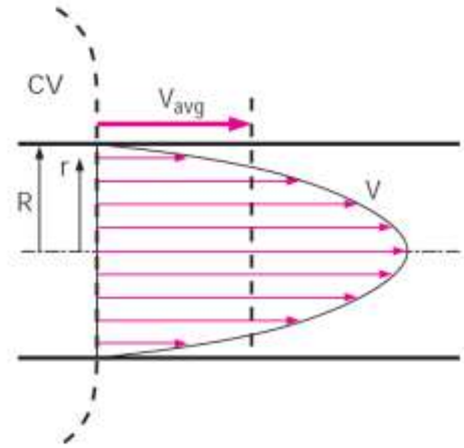



**Figure 3.18:** Velocity profile over a cross section of a pipe in which the flow is fully-developed and laminar.

$$\beta = \frac{1}{A_c}\int_{A_c}\left(\frac{V}{V_{avg}}\right)^2 dA_c = \frac{4}{\pi R^2}\int_0^R \left(1-\frac{r^2}{R^2}\right)^2 2\pi r\, dr$$

Defining a new integration variable $y = 1 - r^2/R^2$ and thus $dy = -2r\, dr/R^2$ (also, $y = 1$ at $r = 0$, and $y = 0$ at $r = R$) and performing the integration, the momentum-flux correction factor for fully developed laminar flow becomes

**Laminar flow:** $$\beta = -4\int_1^0 y^2\, dy = -4\left[\frac{y^3}{3}\right]_1^0 = \frac{4}{3}$$

<u>**Notice:**</u> For turbulent flow β may have an insignificant effect at inlets and outlets, but for laminar flow β may be important and should not be neglected. It is wise to include β in all momentum control volume problems.

## 1.8 Steady Flow

If the flow is also steady, the time derivative term in Equation:

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V}\, dV + \sum_{out} \beta \dot{m} \vec{V}_{avg} - \sum_{in} \beta \dot{m} \vec{V}_{avg}$$

vanishes and we are left with,

**Steady linear momentum equation:** $$\sum \vec{F} = \sum_{out} \beta \dot{m} \vec{V} - \sum_{in} \beta \dot{m} \vec{V}$$

where we dropped the subscript "*avg*" from average velocity. Above Equation states that the net force acting on the control volume during steady flow is equal to the difference between the rates of outgoing and incoming momentum flows. This statement is illustrated in Figure 3.19. It can also be expressed for any direction, since above equation is a vector equation.

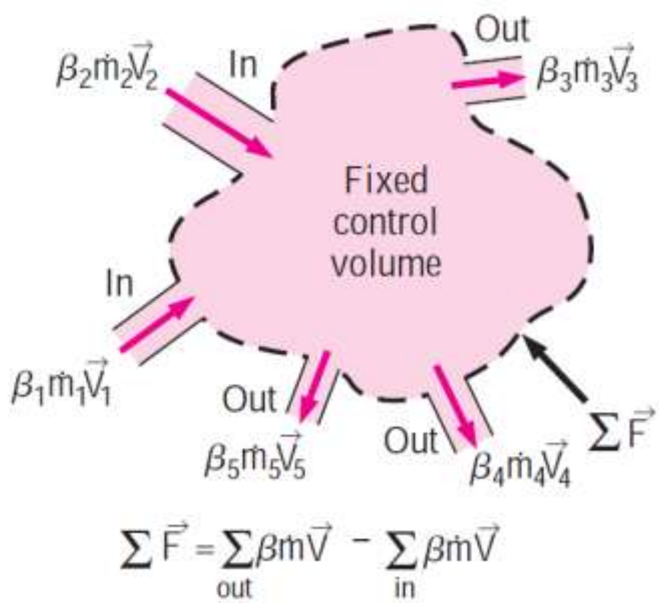



**Figure 3.19:** Velocity profile over a cross section of a pipe in which the flow is fully-developed and laminar.

***Steady Flow with One Inlet and One Outlet***: Many practical problems involve just one inlet and one outlet (Figure 3.20). The mass flow rate for such single-stream systems remains constant, and above equation reduces to,

One inlet and one outlet: $$\sum \vec{F} = \dot{m}\,(\beta_2 \vec{V}_2 - \beta_1 \vec{V}_1)$$

**Example 10:**

A reducing elbow is used to deflect water flow at a rate of 14 kg/s in a horizontal pipe upward 30° while accelerating it as shown in figure 3.20. The elbow discharges water into the atmosphere. The cross-sectional area of the elbow is 113 cm2 at the inlet and 7 cm2 at the outlet. The elevation difference between the centers of the outlet and the inlet is 30 cm. The weight of the elbow and the water in it is considered to be negligible. Determine (**a**) the gage pressure at the center of the inlet of the elbow and (**b**) the anchoring force needed to hold the elbow in place. Take the momentum-flux correction factor to be $\beta$= 1.03.

***Solution:***

(**a**) We take the elbow as the control volume and designate the inlet by ① and the outlet by ②. We also take the *x*- and *z*-coordinates as shown.

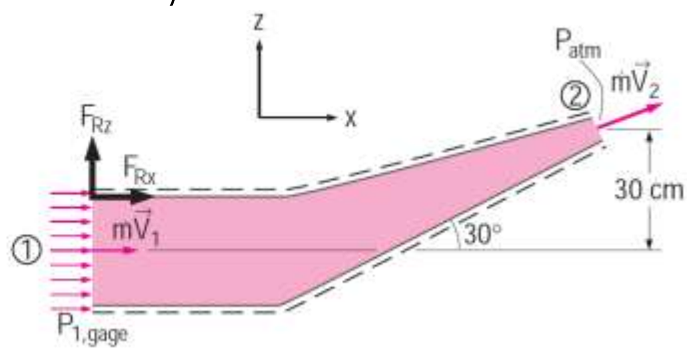



**Figure 3.20:** Schematic for Example 10.

The continuity equation for this one-inlet, one-outlet, steady-flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m} = 14$ kg/s. Noting that $\dot{m} = \rho AV$, the inlet and outlet velocities of water are

$$V_1 = \frac{\dot{m}}{\rho A_1} = \frac{14 \text{ kg/s}}{(1000 \text{ kg/m}^3)(0.0113 \text{ m}^2)} = 1.24 \text{ m/s}$$

$$V_2 = \frac{\dot{m}}{\rho A_2} = \frac{14 \text{ kg/s}}{(1000 \text{ kg/m}^3)(7 \times 10^{-4} \text{ m}^2)} = 20.0 \text{ m/s}$$

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

$$P_1 - P_2 = \rho g\left(\frac{V_2^2 - V_1^2}{2g} + z_2 - z_1\right)$$

$$P_1 - P_{atm} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2) \times \left(\frac{(20 \text{ m/s})^2 - (1.24 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} + 0.3 - 0\right)\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$P_{1,\text{ gage}} = 202.2 \text{ kN/m}^2 = \mathbf{202.2 \text{ kPa}} \quad \text{(gage)}$$

(**b**) The momentum equation for steady one-dimensional flow is

$$\sum \vec{F} = \sum_{out} \beta \dot{m} \vec{V} - \sum_{in} \beta \dot{m} \vec{V}$$

We let the x- and z-components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive direction. We also use gage pressure since the atmospheric pressure acts on the entire control surface. Then the momentum equations along the x- and z-axes become

$$F_{Rx} + P_{1,\,gage}A_1 = \beta \dot{m} V_2 \cos\theta - \beta \dot{m} V_1$$
$$F_{Rz} = \beta \dot{m} V_2 \sin\theta$$

Solving for $F_{Rx}$ and $F_{Rz}$, and substituting the given values,

$$\begin{aligned} F_{Rx} &= \beta \dot{m}(V_2 \cos\theta - V_1) - P_{1,\,gage}A_1 \\ &= 1.03(14\ \text{kg/s})[(20 \cos 30^\circ - 1.24)\ \text{m/s}]\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) \\ &\quad - (202{,}200\ \text{N/m}^2)(0.0113\ \text{m}^2) \\ &= 232 - 2285 = \mathbf{-2053\ N} \end{aligned}$$

$$F_{Rz} = \beta \dot{m} V_2 \sin\theta = (1.03)(14\ \text{kg/s})(20 \sin 30^\circ\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{144\ N}$$

**Example 11:**

A reversing elbow such that the fluid makes a 180° *U-turn* before it is discharged, as shown in Figure 3.21. The elevation difference between the centers of the inlet and the exit sections is still 0.3 m. Determine the anchoring force needed to hold the elbow in place. Take the momentum-flux correction factor to be $\beta$= 1.03.

$V_2$= 20 m/s

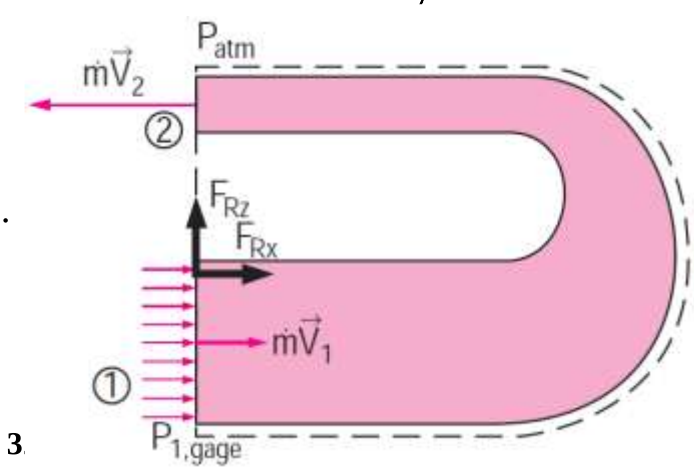



**Figure 3.21:** Schematic for Example 11.

$V_1$= 1.204 m/s
$P_{1,\,gage}$= 2022oo Pa
$A_1$= 0.0113 $m^2$

***Solution:***

The vertical component of the anchoring force at the connection of the elbow to the pipe is zero in this case ($F_{Rz}$= 0) since there is no other force or momentum flux in the vertical direction.

$$F_{Rx} + P_{1,\,gage}A_1 = \beta_2\dot{m}(-V_2) - \beta_1\dot{m}V_1 = -\beta\dot{m}(V_2 + V_1)$$

Solving for $F_{Rx}$ and substituting the known values,

$$F_{Rx} = -\beta\dot{m}(V_2 + V_1) - P_{1,\,gage}A_1$$

$$= -(1.03)(14\text{ kg/s})[(20 + 1.24)\text{ m/s}]\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) - (202{,}200\text{ N/m}^2)(0.0113\text{ m}^2)$$

$$= -306 - 2285 = \mathbf{-2591\ N}$$

Noting that the outlet velocity is negative since it is in the negative x-direction. Therefore, the horizontal force on the flange is 2591 N acting in the negative x-direction (the elbow is trying to separate from the pipe).

**Example 12:**

Water is accelerated by a nozzle to an average speed of 20 m/s, and strikes a stationary vertical plate at a rate of 10 kg/s with a normal velocity of 20 m/s (Figure 3.22). After the strike, the water stream splatters off in all directions in the plane of the plate. Determine the force needed to prevent the plate from moving horizontally due to the water stream. Take the momentum-flux correction factor to be $\beta$= 1.

***Solution***:

The momentum equation for steady one-dimensional flow is given as,

$$\sum \vec{F} = \sum_{out} \beta\dot{m}\vec{V} - \sum_{in} \beta\dot{m}\vec{V}$$

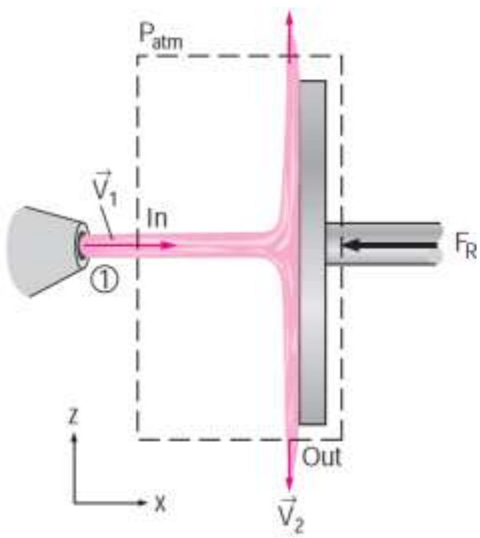



**Figure 3.22:** Schematic for Example 12.

Writing it for this problem along the *x*-direction (without forgetting the negative sign for forces and velocities in the negative *x*-direction) and noting that $V_{1,x} = V_1$ and $V_{2,x} = 0$ gives,

$$-F_R = 0 - \beta \dot{m}\vec{V}_1$$

Substituting the given values,

$$F_R = \beta \dot{m}\vec{V}_1 = (1)(10\ \text{kg/s})(20\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{200\ N}$$

**Example 13:**

A wind generator with a 30-ft-diameter blade span has a cut-in wind speed (minimum speed for power generation) of 7 mph, at which velocity the turbine generates 0.4 kW of electric power (Figure 3–23). Determine (**a**) the efficiency of the wind turbine–generator unit and (**b**) the horizontal force exerted by the wind on the supporting mast of the wind turbine. What is the effect of doubling the wind velocity to 14 mph on power generation and the force exerted? Assume the efficiency remains the same, and take the density of air to be 0.076 lbm/ft$^3$. Take the momentum-flux correction factor to be $\beta$= 1.

***Solution***:

The power potential of the wind is proportional to its kinetic energy, which is $V^2/2$ per unit mass, and thus the maximum power is $\dot{m}V^2/2$ for a given mass flow rate:

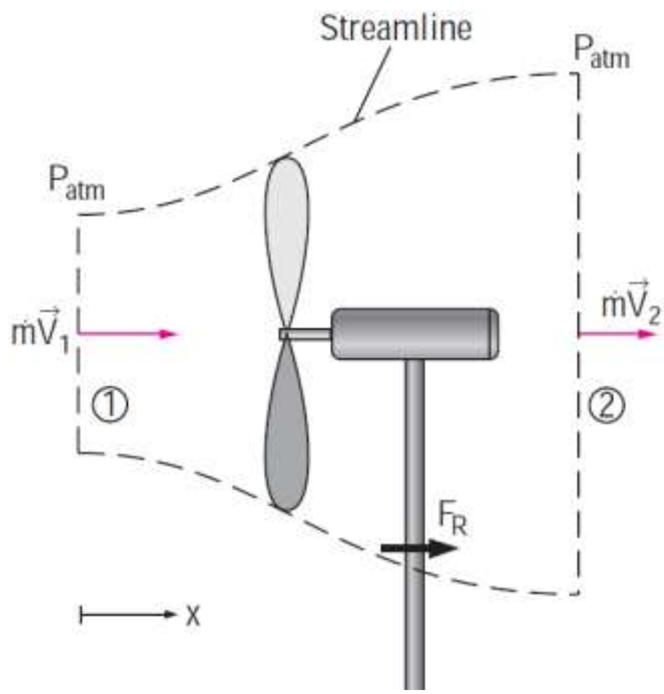



**Figure 3.23:** Schematic for Example 13.

$$V_1 = (7\text{ mph})\left(\frac{1.4667\text{ ft/s}}{1\text{ mph}}\right) = 10.27\text{ ft/s}$$

$$\dot{m} = \rho_1 V_1 A_1 = \rho_1 V_1 \frac{\pi D^2}{4} = (0.076\text{ lbm/ft}^3)(10.27\text{ ft/s})\frac{\pi(30\text{ ft})^2}{4} = 551.7\text{ lbm/s}$$

$$\dot{W}_{max} = \dot{m}ke_1 = \dot{m}\frac{V_1^2}{2}$$

$$= (551.7\text{ lbm/s})\frac{(10.27\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right)$$

$$= 1.225\text{ kW}$$

Therefore, the available power to the wind turbine is 1.225 kW at the wind velocity of 7 mph. Then the turbine–generator efficiency becomes

$$\eta_{\text{wind turbine}} = \frac{\dot{W}_{act}}{\dot{W}_{max}} = \frac{0.4\text{ kW}}{1.225\text{ kW}} = \mathbf{0.327}$$

Noting that the mass flow rate remains constant, the exit velocity is determined to be

$$\dot{m}ke_2 = \dot{m}ke_1(1 - \eta_{\text{wind turbine}}) \quad \rightarrow \quad \dot{m}\frac{V_2^2}{2} = \dot{m}\frac{V_1^2}{2}(1 - \eta_{\text{wind turbine}})$$

$$V_2 = V_1\sqrt{1 - \eta_{\text{wind turbine}}} = (10.27\text{ ft/s})\sqrt{1 - 0.327} = 8.43\text{ ft/s}$$

The momentum equation for steady one-dimensional flow is given as

$$\sum \vec{F} = \sum_{out} \beta\dot{m}\vec{V} - \sum_{in} \beta\dot{m}\vec{V} \qquad F_R = \dot{m}V_2 - \dot{m}V_1 = \dot{m}(V_2 - V_1)$$

Substituting the known values gives

$$F_R = \dot{m}(V_2 - V_1) = (551.7\text{ lbm/s})(8.43 - 10.27\text{ ft/s})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= -31.5\text{ lbf}$$

Then the force exerted by the wind on the mast becomes $F_{mast} = -F_R = 31.5$ lbf.

# Module 6 : Lecture 1
# DIMENSIONAL ANALYSIS
# (Part – I)

**Overview**

Many practical flow problems of different nature can be solved by using equations and analytical procedures, as discussed in the previous modules. However, solutions of some real flow problems depend heavily on experimental data and the refinements in the analysis are made, based on the measurements. Sometimes, the experimental work in the laboratory is not only time-consuming, but also expensive. So, the dimensional analysis is an important tool that helps in correlating analytical results with experimental data for such unknown flow problems. Also, some dimensionless parameters and scaling laws can be framed in order to predict the prototype behavior from the measurements on the model. The important terms used in this module may be defined as below;

Dimensional Analysis: The systematic procedure of identifying the variables in a physical phenomena and correlating them to form a set of dimensionless group is known as *dimensional analysis*.

Dimensional Homogeneity: If an equation truly expresses a proper relationship among variables in a physical process, then it will be *dimensionally homogeneous*. The equations are correct for any system of units and consequently each group of terms in the equation must have the same dimensional representation. This is also known as the law of *dimensional homogeneity*.

Dimensional variables: These are the quantities, which actually vary during a given case and can be plotted against each other.

Dimensional constants: These are normally held constant during a given run. But, they may vary from case to case.

Pure constants: They have no dimensions, but, while performing the mathematical manipulation, they can arise.

Let us explain these terms from the following examples:

- Displacement of a free falling body is given as, $S = S_0 + V_0 t + \frac{1}{2} g t^2$, where, $V_0$ is the initial velocity, $g$ is the acceleration due to gravity, $t$ is the time, $S$ and $S_0$ are the final and initial distances, respectively. Each term in this equation has the dimension of length $[L]$ and hence it is *dimensionally homogeneous*. Here, $S$ and $t$ are the *dimensional variables*, $g$, $S_0$ and $V_0$ are the *dimensional constants* and $\frac{1}{2}$ arises due to mathematical manipulation and is the *pure constant*.

- Bernoulli's equation for incompressible flow is written as, $\frac{p}{\rho} + \frac{1}{2} V^2 + gz = \mathrm{C}$. Here, $p$ is the pressure, $V$ is the velocity, $z$ is the distance, $\rho$ is the density and $g$ is the acceleration due to gravity. In this case, the *dimensional variables* are $p, V$ and $z$, the *dimensional constants* are $g$, $\rho$ and $C$ and $\frac{1}{2}$ is the *pure constant*. Each term in this equation including the constant has dimension of $\left[L^2 T^{-2}\right]$ and hence it is *dimensionally homogeneous*.

**Buckingham pi Theorem**

The dimensional analysis for the experimental data of unknown flow problems leads to some non-dimensional parameters. These dimensionless products are frequently referred as *pi terms*. Based on the concept of *dimensional homogeneity*, these dimensionless parameters may be grouped and expressed in functional forms. This idea was explored by the famous scientist Edgar Buckingham (1867-1940) and the theorem is named accordingly.

*Buckingham pi theorem*, states that if an equation involving $k$ variables is dimensionally homogeneous, then it can be reduced to a relationship among $(k - r)$ independent dimensionless products, where $r$ is the minimum number of reference dimensions required to describe the variable. For a physical system, involving $k$ variables, the functional relation of variables can be written mathematically as,

$$y = f\left(x_1, x_2 .........., x_k\right) \tag{6.1.1}$$

In Eq. (6.1.1), it should be ensured that the dimensions of the variables on the left side of the equation are equal to the dimensions of any term on the right side of equation. Now, it is possible to rearrange the above equation into a set of dimensionless products (*pi terms*), so that

$$\Pi_1 = \varphi\left(\Pi_2, \Pi_3 .........., \Pi_{k-r}\right) \tag{6.1.2}$$

Here, $\varphi\left(\Pi_2, \Pi_3 .........., \Pi_{k-r}\right)$ is a function of $\Pi_2$ through $\Pi_{k-r}$. The required number of *pi terms* is less than the number of original reference variables by $r$. These reference dimensions are usually the basic dimensions $M$, $L$ and $T$ (Mass, Length and Time).

**Determination of pi Terms**

Several methods can be used to form dimensionless products or *pi terms* that arise in dimensional analysis. But, there is a systematic procedure called *method of repeating variables* that allows in deciding the dimensionless and independent *pi terms*. For a given problem, following distinct steps are followed.

Step I: List out all the variables that are involved in the problem. The 'variable' is any quantity including dimensional and non-dimensional constants in a physical situation under investigation. Typically, these variables are those that are necessary to describe the "geometry" of the system (diameter, length etc.), to define fluid properties (density, viscosity etc.) and to indicate the external effects influencing the system (force, pressure etc.). All the variables must be independent in nature so as to minimize the number of variables required to describe the complete system.

Step II: Express each variable in terms of basic dimensions. Typically, for fluid mechanics problems, the basic dimensions will be either $M$, $L$ and $T$ or $F$, $L$ and $T$. Dimensionally, these two sets are related through Newton's second law $\left(F = m.a\right)$ so that $F = MLT^{-2}$ e.g. $\rho = ML^{-3}$ or $\rho = FL^{-4}T^{2}$. It should be noted that these basic dimensions should not be mixed.

Step III: Decide the required number of *pi terms*. It can be determined by using *Buckingham pi theorem* which indicates that the number of *pi terms* is equal to $\left(k - r\right)$, where $k$ is the number of variables in the problem (determined from Step I) and $r$ is the number of reference dimensions required to describe these variables (determined from Step II).

Step IV: Amongst the original list of variables, select those variables that can be combined to form *pi terms*. These are called as *repeating variables*. The required number of *repeating variables is equal to the number of reference dimensions*. Each *repeating variable* must be dimensionally independent of the others, i.e. they cannot be combined themselves to form any dimensionless product. Since there is a possibility of repeating variables to appear in more than one *pi term*, so dependent variables should not be chosen as one of the repeating variable.

Step V: Essentially, the *pi terms* are formed by multiplying one of the non-repeating variables by the product of the repeating variables each raised to an exponent that will make the combination dimensionless. It usually takes the form of $x_i \; x_1^a \, x_2^b \, x_3^c$ where the exponents $a$, $b$ and $c$ are determined so that the combination is dimensionless.

Step VI: Repeat the 'Step V' for each of the remaining non-repeating variables. The resulting set of *pi terms* will correspond to the required number obtained from Step III.

Step VII: After obtaining the required number of *pi terms*, make sure that all the *pi terms* are dimensionless. It can be checked by simply substituting the basic dimension $(M,\, L \text{ and } T)$ of the variables into the *pi terms*.

Step VIII: Typically, the final form of relationship among the *pi terms* can be written in the form of Eq. (6.1.2) where, $\Pi_1$ would contain the dependent variable in the numerator. The actual functional relationship among *pi terms* is determined from experiment.

**Illustration of *Pi Theorem***

Let us consider the following example to illustrate the procedure of determining the various steps in the *pi theorem*.

Example (Pressure drop in a pipe flow)

Consider a steady flow of an incompressible Newtonian fluid through a long, smooth walled, horizontal circular pipe. It is required to measure the pressure drop per unit length of the pipe and find the number of non-dimensional parameters involved in the problem. Also, it is desired to know the functional relation among these dimensionless parameters.

Step I: Let us express all the pertinent variables involved in the experimentation of pressure drop per unit length $(\Delta p_l)$ of the pipe, in the following form;

$$\Delta p_l = f\left(D, \rho, \mu, V\right) \tag{6.1.3}$$

where, $D$ is the pipe diameter, $\rho$ is the fluid density, $\mu$ is the viscosity of the fluid and $V$ is the mean velocity at which the fluid is flowing through the pipe.

Step II: Next step is to express all the variables in terms of basic dimensions i.e. $M$, $L$ and $T$. It then follows that

$$\Delta p_l = ML^{-2}T^{-2};\ D = L;\ \rho = ML^{-3};\ \mu = ML^{-1}T^{-1};\ V = LT^{-1} \tag{6.1.4}$$

Step III: Apply *Buckingham theorem* to decide the number of *pi terms* required. There are five variables (including the dependent variable $\Delta p_l$) and three reference dimensions. Since, $k = 5$ and $r = 3$, only *two pi terms* are required for this problem.

Step IV: The repeating variables to form pi terms, need to be selected from the list $D$, $\rho$, $\mu$ and $V$. It is to be noted that the dependent variable should not be used as one of the repeating variable. Since, there are three reference dimensions involved, so we need to select three repeating variable. These repeating variables should be dimensionally independent, i.e. dimensionless product cannot be formed from this set. In this case, $D$, $\rho$ and $V$ may be chosen as the repeating variables.

Step V: Now, first *pi term* is formed between the dependent variable and the repeating variables. It is written as,

$$\Pi_1 = \Delta p_l\, D^a V^b \rho^c \tag{6.1.5}$$

Since, this combination need to be dimensionless, it follows that

$$\left(ML^{-2}T^{-2}\right)\left(L\right)^a \left(LT^{-1}\right)^b \left(ML^{-3}\right)^c = M^0 L^0 T^0 \tag{6.1.6}$$

The exponents $a$, $b$ and $c$ must be determined by equating the exponents for each of the terms $M$, $L$ and $T$ i.e.

$$\begin{aligned} &\text{For } M: \ 1+c=0 \\ &\text{For } L: \ -2+a+b-3c=0 \\ &\text{For } T: -2-b=0 \end{aligned} \tag{6.1.7}$$

The solution of this algebraic equations gives $a=1; b=-2; c=-1$. Therefore,

$$\Pi_1 = \frac{\Delta p_l \, D}{\rho V^2} \tag{6.1.8}$$

The process is repeated for remaining non-repeating variables with other additional variable $(\mu)$ so that,

$$\Pi_2 = \mu . D^d . V^e . \rho^f \tag{6.1.9}$$

Since, this combination need to be dimensionless, it follows that

$$\left(ML^{-1}T^{-1}\right)\left(L\right)^d \left(LT^{-1}\right)^e \left(ML^{-3}\right)^f = M^0 L^0 T^0 \tag{6.1.10}$$

Equating the exponents,

$$\begin{aligned} &\text{For } M: \ 1+f=0 \\ &\text{For } L: \ -1+d+e-3f=0 \\ &\text{For } T: -1-e=0 \end{aligned} \tag{6.1.11}$$

The solution of this algebraic equation gives $d=-1; e=-1; f=-1$. Therefore,

$$\Pi_2 = \frac{\mu}{\rho V D} \tag{6.1.12}$$

<u>Step VI</u>: Now, the correct numbers of *pi term*s are formed as determined in "Step III". In order to make sure about the dimensionality of *pi terms*, they are written as,

$$\begin{aligned} \Pi_1 &= \frac{\Delta p_l \, D}{\rho V^2} = \frac{\left(ML^{-2}T^{-2}\right)\left(L\right)}{\left(ML^{-3}\right)\left(LT^{-1}\right)^2} = M^0 L^0 T^0 \\ \Pi_2 &= \frac{\mu}{\rho V \, D} = \frac{\left(ML^{-1}T^{-1}\right)\left(L\right)}{\left(ML^{-3}\right)\left(LT^{-1}\right)\left(L\right)} = M^0 L^0 T^0 \end{aligned} \tag{6.1.13}$$

<u>Step VII</u>: Finally, the result of dimensional analysis is expressed among the *pi term*s as,

$$\frac{D \, \Delta p_l}{\rho V^2} = \phi\left(\frac{\mu}{\rho V \, D}\right) = \phi\left(\frac{1}{\text{Re}}\right) \tag{6.1.14}$$

It may be noted here that $\text{Re}$ is the Reynolds number.

*Remarks*

- If the difference in the number of variables for a given problem and number of reference dimensions is equal to unity, then only *one Pi* term is required to describe the phenomena. Here, the functional relationship for the *one Pi* term is a constant quantity and it is determined from the experiment.

$$\Pi_1 = \text{Constant} \tag{6.1.15}$$

- The problems involving two *Pi* terms can be described such that

$$\Pi_1 = \phi\left(\Pi_2\right) \tag{6.1.16}$$

Here, the functional relationship among the variables can then be determined by varying $\Pi_2$ and measuring the corresponding values of $\Pi_1$.

# Module 6 : Lecture 2
# DIMENSIONAL ANALYSIS
# (Part – II)

**Non Dimensional numbers in Fluid Dynamics**

Forces encountered in flowing fluids include those due to inertia, viscosity, pressure, gravity, surface tension and compressibility. These forces can be written as follows;

$$\begin{aligned}
&\text{Inertia force: } m.a = \rho V \frac{dV}{dt} \propto \rho V^2 L^2 \\
&\text{Viscous force: } \tau A = \mu A \frac{du}{dy} \propto \mu V L \\
&\text{Pressure force: } (\Delta p) A \propto (\Delta p) L^2 \\
&\text{Gravity force: } m g \propto g \rho L^3 \\
&\text{Surface tension force: } \sigma L \\
&\text{Compressibility force: } E_v A \propto E_v L^2
\end{aligned} \tag{6.2.1}$$

The notations used in Eq. (6.2.1) are given in subsequent paragraph of this section. It may be noted that the ratio of any two forces will be dimensionless. Since, inertia forces are very important in fluid mechanics problems, the ratio of the inertia force to each of the other forces listed above leads to fundamental dimensionless groups. Some of them are defined as given below;

Reynolds number $(\mathrm{Re})$: It is defined as the ratio of inertia force to viscous force. Mathematically,

$$\mathrm{Re} = \frac{\rho V L}{\mu} = \frac{V L}{\nu} \tag{6.2.2}$$

where $V$ is the velocity of the flow, $L$ is the characteristics length, $\rho, \mu$ and $\nu$ are the density, dynamic viscosity and kinematic viscosity of the fluid respectively. If $\mathrm{Re}$ is very small, there is an indication that the viscous forces are dominant compared to inertia forces. Such types of flows are commonly referred to as “creeping/viscous flows”. Conversely, for large $\mathrm{Re}$, viscous forces are small compared to inertial effects and such flow problems are characterized as inviscid analysis. This number is also used to study the transition between the laminar and turbulent flow regimes.

Euler number $(E_u)$: In most of the aerodynamic model testing, the pressure data are usually expressed mathematically as,

$$E_u = \frac{\Delta p}{\frac{1}{2}\rho V^2} \tag{6.2.3}$$

where $\Delta p$ is the difference in local pressure and free stream pressure, $V$ is the velocity of the flow, $\rho$ is the density of the fluid. The denominator in Eq. (6.2.3) is called "dynamic pressure". $E_u$ is the ratio of pressure force to inertia force and many a times the pressure coefficient $(c_p)$ is a also common name which is defined by same manner. In the study of cavitations phenomena, similar expressions are used where, $\Delta p$ is the difference in liquid stream pressure and liquid-vapour pressure. This dimensional parameter is then called as "cavitation number".

Froude number $(F_r)$: It is interpreted as the ratio of inertia force to gravity force. Mathematically, it is written as,

$$F_r = \frac{V}{\sqrt{g.L}} \tag{6.2.4}$$

where $V$ is the velocity of the flow, $L$ is the characteristics length descriptive of the flow field and $g$ is the acceleration due to gravity. This number is very much significant for flows with free surface effects such as in case of open-channel flow. In such types of flows, the characteristics length is the depth of water. $F_r$ less than unity indicates sub-critical flow and values greater than unity indicate super-critical flow. It is also used to study the flow of water around ships with resulting wave motion.

Weber number $(W_e)$: It is defined as the ratio of the inertia force to surface tension force. Mathematically,

$$W_e = \frac{\rho V^2 L}{\sigma} \tag{6.2.5}$$

where $V$ is the velocity of the flow, $L$ is the characteristics length descriptive of the flow field, $\rho$ is the density of the fluid and $\sigma$ is the surface tension force. This number is taken as an index of droplet formation and flow of thin film liquids in which there is an interface between two fluids. The inertia force is dominant compared to surface tension force when, $W_e \gg 1$ (e.g. flow of water in a river).

Mach number $(M)$: It is the key parameter that characterizes the compressibility effects in a fluid flow and is defined as the ratio of inertia force to compressibility force. Mathematically,

$$M = \frac{V}{c} = \frac{V}{\sqrt{\frac{dp}{d\rho}}} = \frac{V}{\sqrt{\frac{E_v}{\rho}}} \qquad (6.2.6)$$

where $V$ is the velocity of the flow, $c$ is the local sonic speed, $\rho$ is the density of the fluid and $E_v$ is the bulk modulus. Sometimes, the square of the Mach number is called "Cauchy number" $(C_a)$ i.e.

$$C_a = M^2 = \frac{\rho V^2}{E_v} \qquad (6.2.7)$$

Both the numbers are predominantly used in problems in which fluid compressibility is important. When, $M_a$ is relatively small (say, less than 0.3), the inertial forces induced by fluid motion are sufficiently small to cause significant change in fluid density. So, the compressibility of the fluid can be neglected. However, this number is most commonly used parameter in compressible fluid flow problems, particularly in the field of gas dynamics and aerodynamics.

Strouhal number $(S_t)$: It is a dimensionless parameter that is likely to be important in unsteady, oscillating flow problems in which the frequency of oscillation is $\omega$ and is defined as,

$$S_t = \frac{\omega L}{V} \qquad (6.2.8)$$

where $V$ is the velocity of the flow and $L$ is the characteristics length descriptive of the flow field. This number is the measure of the ratio of the inertial forces due to unsteadiness of the flow (local acceleration) to inertia forces due to changes in velocity from point to point in the flow field (convective acceleration). This type of unsteady flow develops when a fluid flows past a solid body placed in the moving stream.

In addition, there are few other dimensionless numbers that are of importance in fluid mechanics. They are listed below;

| Parameter | Mathematical expression | Qualitative definition | Importance |
|---|---|---|---|
| Prandtl number | $P_r = \frac{\mu c_p}{k}$ | $\frac{\text{Dissipation}}{\text{Conduction}}$ | Heat convection |
| Eckert number | $E_c = \frac{V^2}{c_p T_0}$ | $\frac{\text{Kinetic energy}}{\text{Enthalpy}}$ | Dissipation |
| Specific heat ratio | $\gamma = \frac{c_p}{c_v}$ | $\frac{\text{Enthalpy}}{\text{Internal energy}}$ | Compressible flow |
| Roughness ratio | $\frac{\varepsilon}{L}$ | $\frac{\text{Wall roughness}}{\text{Body length}}$ | Turbulent rough walls |
| Grashof number | $G_r = \frac{\beta(\Delta T) g L^3 \rho^2}{\mu^2}$ | $\frac{\text{Buoyancy}}{\text{Viscosity}}$ | Natural onvection |
| Temperature ratio | $\frac{T_w}{T_0}$ | $\frac{\text{Wall temperature}}{\text{Stream temperature}}$ | Heat transfer |
| Pressure coefficient | $C_p = \frac{p - p_\infty}{(1/2)\rho V^2}$ | $\frac{\text{Static pressure}}{\text{Dynamic pressure}}$ | Hydrodynamics, Aerodynamics |
| Lift coefficient | $C_L = \frac{L}{(1/2) A \rho V^2}$ | $\frac{\text{Lift force}}{\text{Dynamic force}}$ | Hydrodynamics,Aero dynamics |
| Drag coefficient | $C_D = \frac{D}{(1/2) A \rho V^2}$ | $\frac{\text{Drag force}}{\text{Dynamic force}}$ | Hydrodynamics, Aero dynamics |

## Modeling and Similitude

A "model" is a representation of a physical system which is used to predict the behavior of the system in some desired respect. The physical system for which the predictions are to be made is called "prototype". Usually, a model is smaller than the prototype so that laboratory experiments/studies can be conducted. It is less expensive to construct and operate. However, in certain situations, models are larger than the prototype e.g. study of the motion of blood cells whose sizes are of the order of micrometers. "Similitude" is the indication of a known relationship between a model and prototype. In other words, the model tests must yield data that can be scaled to obtain the similar parameters for the prototype.

Theory of models: The dimensional analysis of a given problem can be described in terms of a set of pi terms and these non-dimensional parameters can be expressed in functional forms;

$$\Pi_1 = \phi\left(\Pi_2, \Pi_3, \ldots\ldots\ldots\Pi_n\right) \tag{6.2.9}$$

Since this equation applies to any system, governed by same variables and if the behavior of a particular prototype is described by Eq. (6.2.9), then a similar relationship can be written for a model.

$$\Pi_{1m} = \phi\left(\Pi_{2m}, \Pi_{3m}, \ldots\ldots\ldots\Pi_{nm}\right) \tag{6.2.10}$$

The form of the function remains the same as long as the same phenomenon is involved in both the prototype and the model. Therefore, if the model is designed and operated under following conditions,

$$\Pi_{2m} = \Pi_2;\ \Pi_{3m} = \Pi_3 \ldots\ldots\ldots \text{ and } \Pi_{nm} = \Pi_n \tag{6.2.11}$$

Then it follows that

$$\Pi_1 = \Pi_{1m} \tag{6.2.12}$$

Eq. (6.2.12) is the desired "prediction equation" and indicates that the measured value of $\Pi_{1m}$ obtained with the model will be equal to the corresponding $\Pi_1$ for the prototype as long as the other *pi terms* are equal. These are called "model design conditions / similarity requirements / modeling laws".

**Flow Similarity**

In order to achieve similarity between model and prototype behavior, all the corresponding pi terms must be equated to satisfy the following conditions.

Geometric similarity: A model and prototype are geometric similar if and only if all body dimensions in all three coordinates have the same linear-scale ratio. In order to have geometric similarity between the model and prototype, the model and the prototype should be of the same shape, all the linear dimensions of the model can be related to corresponding dimensions of the prototype by a constant scale factor. Usually, one or more of these *pi terms* will involve ratios of important lengths, which are purely geometrical in nature.

Kinematic similarity: The motions of two systems are kinematically similar if homogeneous particles lie at same points at same times. In a specific sense, the velocities at corresponding points are in the same direction (i.e. same streamline patterns) and are related in magnitude by a constant scale factor.

Dynamic similarity: When two flows have force distributions such that identical types of forces are parallel and are related in magnitude by a constant scale factor at all corresponding points, then the flows are dynamic similar. For a model and prototype, the dynamic similarity exists, when both of them have same length-scale ratio, time-scale ratio and force-scale (or mass-scale ratio).

In order to have complete similarity between the model and prototype, all the similarity flow conditions must be maintained. This will automatically follow if all the important variables are included in the dimensional analysis and if all the similarity requirements based on the resulting pi terms are satisfied. For example, in compressible flows, the model and prototype should have same Reynolds number, Mach number and specific heat ratio etc. If the flow is incompressible (without free surface), then same Reynolds numbers for model and prototype can satisfy the complete similarity.

**Model scales**

In a given problem, if there are two length variables $l_1$ and $l_2$, the resulting requirement based on the pi terms obtained from these variables is,

$$\frac{l_{1m}}{l_1} = \frac{l_{2m}}{l_2} = \lambda_l \qquad (6.2.13)$$

This ratio is defined as the "length scale". For true models, there will be only one length scale and all lengths are fixed in accordance with this scale. There are other 'model scales' such as velocity scale $\left(\frac{V_m}{V} = \lambda_v\right)$, density scale $\left(\frac{\rho_m}{\rho} = \lambda_\rho\right)$, viscosity scale $\left(\frac{\mu_m}{\mu} = \lambda_\mu\right)$ etc. Each of these scales needs to be defined for a given problem.

**Distorted models**

In order to achieve the complete dynamic similarity between geometrically similar flows, it is necessary to reproduce the independent dimensionless groups so that dependent parameters can also be duplicated (e.g. same Reynolds number between a model and prototype is ensured for dynamically similar flows).

In many model studies, dynamic similarity may also lead to incomplete similarity between the model and the prototype. If one or more of the similarity requirements are not met, e.g. in Eq. 6.2.9, if $\Pi_{2m} \neq \Pi_2$, then it follows that Eq. 6.2.12 will not be satisfied i.e. $\Pi_1 \neq \Pi_{1m}$. It is a case of distorted model for which one or more of the similar requirements are not satisfied. For example, in the study of free surface flows, both Reynolds number $\left(\frac{\rho V l}{\mu}\right)$ and Froude number $\left(\frac{V}{\sqrt{gl}}\right)$ are involved. Then,

Froude number similarity requires,

$$\frac{V_m}{\sqrt{g_m l_m}} = \frac{V}{\sqrt{gl}} \qquad (6.2.14)$$

If the model and prototype are operated in the same gravitational field, then the velocity scale becomes,

$$\frac{V_m}{V} = \sqrt{\frac{l_m}{l}} = \sqrt{\lambda_l} \qquad (6.2.15)$$

Reynolds number similarity requires,

$$\frac{\rho_m.V_m.l_m}{\mu_m} = \frac{\rho.V.l}{\mu} \tag{6.2.16}$$

Then, the velocity scale is,

$$\frac{V_m}{V} = \frac{\mu_m}{\mu} \cdot \frac{\rho}{\rho_m} \cdot \frac{l}{l_m} \tag{6.2.17}$$

Since, the velocity scale must be equal to the square root of the length scale, it follows that

$$\frac{\nu_m}{\nu} = \frac{(\mu_m/\rho_m)}{(\mu/\rho)} = \left(\frac{l_m}{l}\right)^{\frac{3}{2}} = (\lambda_l)^{\frac{3}{2}} \tag{6.2.18}$$

Eq. (6.2.18) requires that both model and prototype to have different kinematics viscosity scale. But practically, it is almost impossible to find a suitable fluid for the model, in small length scale. In such cases, the systems are designed on the basis of Froude number with different Reynolds number for the model and prototype where Eq. (6.2.18) need not be satisfied. Such analysis will result a “distorted model” and there are no general rules for handling distorted models, rather each problem must be considered on its own merits.

University of Anbar
College of Engineering
Mechanical Engineering Dept.

# Fluid Mechanics

## Handout Lectures for Year Two
## Chapter Five/ Flow i n pipes

**Course Tutor**

**Assist. Prof. Dr. Waleed M. Abed**

# Chapter Five
# Flow i n pipes

## 5.1. LAMINAR AND TURBULENT FLOWS

The flow regime in the first case is said to be ***laminar***, characterized by smooth streamlines and highly ordered motion, and ***turbulent*** in the second case, where it is characterized by velocity fluctuations and highly disordered motion. The ***transition*** from laminar to turbulent flow does not occur suddenly; rather, it occurs over some region in which the flow fluctuates between laminar and turbulent flows before it becomes fully turbulent. Most flows encountered in practice are turbulent. Laminar flow is encountered when highly viscous fluids such as oils flow in small pipes or narrow passages as shown in Figure 5.1.

We can verify the existence of these laminar, transitional, and turbulent flow regimes by injecting some dye streaks into the flow in a glass pipe, as the British engineer ***Osborne Reynolds*** (1842–1912) did over a century ago. We observe that the dye streak forms a ***straight and smooth line at low velocities*** when the flow is laminar (we may see some blurring because of molecular diffusion), has bursts of ***fluctuations in the transitional regime***, and ***zigzags rapidly and randomly*** when the flow becomes fully turbulent. These zigzags and the dispersion of the dye are indicative of the fluctuations in the main flow and the rapid mixing of fluid particles from adjacent layers.

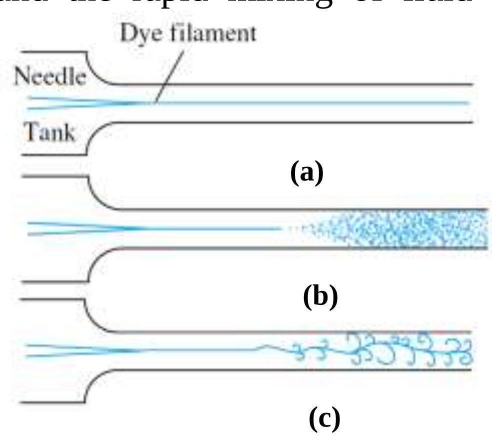



**Figure 5.1:** Spinning Reynolds' sketches of pipe-flow transition: (a) low-speed, laminar flow; (b) high-speed, turbulent flow; (c) spark photograph of condition (b).

### 5.2. Reynolds Number

After exhaustive experiments in the 1880s, Osborne Reynolds discovered that the flow regime depends mainly on the ratio of ***inertial forces* to *viscous forces*** in the fluid. This ratio is called the ***Reynolds number*** and is expressed for internal flow in a circular pipe as,

$$Re = \frac{\text{Inertial forces}}{\text{Viscous forces}} = \frac{V_{avg}D}{\nu} = \frac{\rho V_{avg}D}{\mu}$$

where $V_{avg}$= average flow velocity (m/s), $D$= characteristic length of the geometry (diameter in this case, in m), and $\upsilon = \mu/\rho$= kinematic viscosity of the fluid ($m^2/s$). Note that the Reynolds number is a ***dimensionless*** quantity. Also, kinematic viscosity has the unit $m^2/s$, and can be viewed as ***viscous diffusivity*** or ***diffusivity for momentum***.

The Reynolds number at which the flow becomes turbulent is called the ***critical Reynolds number,*** $Re_{cr}$. The value of the critical Reynolds number is different for different geometries and flow conditions. For internal flow in a circular pipe, the generally accepted value of the critical Reynolds number is $Re_{cr}$= 2300.

For flow through noncircular pipes, the Reynolds number is based on the hydraulic diameter $D_h$ defined as (Figure 5.2),

**Hydraulic diameter:** $$D_h = \frac{4A_c}{p}$$

where $A_c$ is the cross-sectional area of the pipe and $p$ is its wetted perimeter. The hydraulic diameter is defined such that it reduces to ordinary diameter $D$ for circular pipes,

**Circular pipes:** $$D_h = \frac{4A_c}{p} = \frac{4(\pi D^2/4)}{\pi D} = D$$

**Square duct:** $$D_h = \frac{4a^2}{4a} = a$$

**Rectangular duct:** $$D_h = \frac{4ab}{2(a+b)} = \frac{2ab}{a+b}$$

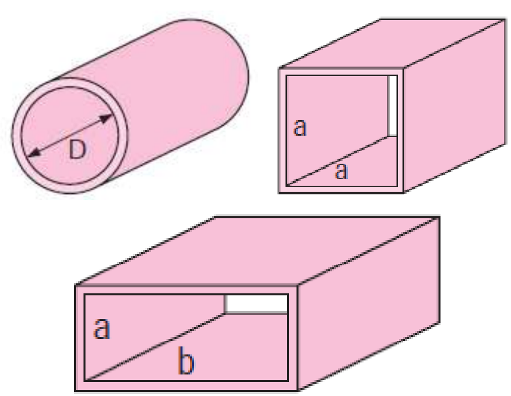





**Figure 5.2**

Under most practical conditions, the flow in a circular pipe is laminar for Re ≤ 2300, turbulent for Re ≥ 4000, and transitional in between. That is,

| | |
|---|---|
| $Re \lesssim 2300$ | laminar flow |
| $2300 \lesssim Re \lesssim 4000$ | transitional flow |
| $Re \gtrsim 4000$ | turbulent flow |

## 5.3. LAMINAR FLOW IN PIPES

We mentioned in Section 5.2. that flow in pipes is laminar for Re ≤ 2300, and that the flow is fully developed if the pipe is sufficiently long (relative to the entry length) so that the entrance effects are negligible.

In fully developed laminar flow, each fluid particle moves at a constant axial velocity along a streamline and the velocity profile $u(r)$ remains unchanged in the flow direction. There is no motion in the radial direction, and thus the velocity component in the direction normal to flow is everywhere *zero*. There is no acceleration since the flow is steady and fully developed.

Now consider a ring-shaped differential volume element of radius $r$, thickness $dr$, and length $dx$ oriented coaxially with the pipe, as shown in Figure 5.3. The volume element involves only pressure and viscous effects and thus the pressure and shear forces must balance each other. The pressure force acting on a submerged plane surface is the product of the pressure at the centroid of the surface and the surface area. A force balance on the volume element in the flow direction gives

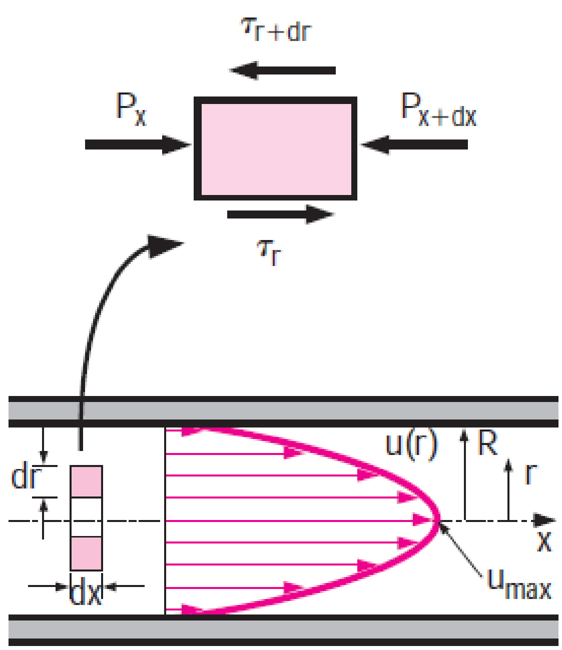



**Figure 5.3:** Free-body diagram of a ring-shaped differential fluid element of radius $r$, thickness $dr$, and length $dx$ oriented coaxially with a horizontal pipe in fully developed laminar flow.

$$(2\pi r\, dr\, P)_x - (2\pi r\, dr\, P)_{x+dx} + (2\pi r\, dx\, \tau)_r - (2\pi r\, dx\, \tau)_{r+dr} = 0$$

which indicates that in fully developed flow in a horizontal pipe, the viscous and pressure forces balance each other. Dividing by $2\pi drdx$ and rearranging,

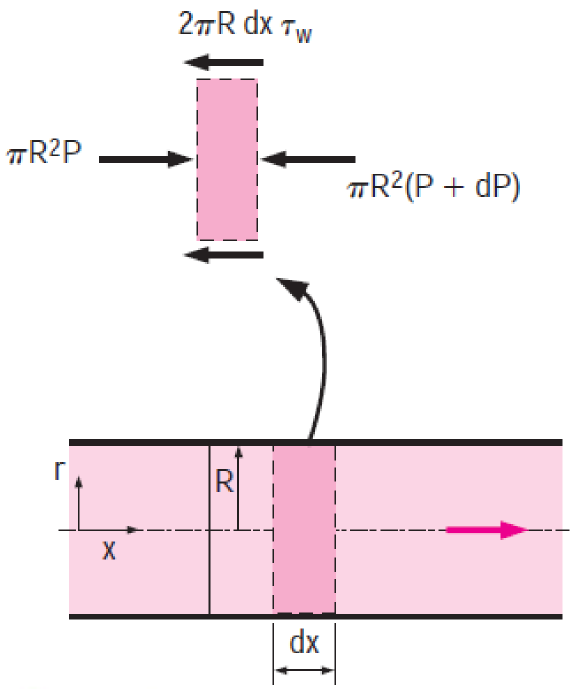
$$r\frac{P_{x+dx} - P_x}{dx} + \frac{(r\tau)_{r+dr} - (r\tau)_r}{dr} = 0$$

Taking the limit as $dr, dx \to 0$ gives

$$r\frac{dP}{dx} + \frac{d(r\tau)}{dr} = 0$$

Substituting $\tau = -\mu(du/dr)$ and taking $\mu$= constant gives the desired equation,

$$\frac{\mu}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{dP}{dx}$$




Force balance:

$$\pi R^2 P - \pi R^2(P + dP) - 2\pi R\, dx\, \tau_w = 0$$

Simplifying:

$$\frac{dP}{dx} = -\frac{2\tau_w}{R}$$

The quantity $du/dr$ is negative in pipe flow, and the negative sign is included to obtain positive values for t. (Or, $du/dr = - du/dy$ since $y= R - r$.) The left side of above Equation is a function of $r$, and the right side is a function of $x$. The equality must hold for any value of $r$ and $x$, and an equality of the form $f(r) = g(x)$ can be satisfied only if both $f(r)$ and $g(x)$ are equal to the same constant. Thus we conclude that $dP/dx$ = constant. This can be verified by writing a force balance on a volume element of radius $R$ and thickness $dx$ (a slice of the pipe), which gives

$$\frac{dP}{dx} = -\frac{2\tau_w}{R}$$

Here $\tau_w$ is constant since the viscosity and the velocity profile are constants in the fully developed region. Therefore, $dP/dx$= constant.

by rearranging and integrating it twice to give

$$u(r) = \frac{1}{4\mu}\left(\frac{dP}{dx}\right) + C_1 \ln r + C_2$$

The velocity profile u(r) is obtained by applying the boundary conditions $\partial u/\partial r = 0$ at r = 0 (because of symmetry about the centerline) and u = 0 at r = R (the no-slip condition at the pipe surface). We get

$$u(r) = -\frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1 - \frac{r^2}{R^2}\right)$$

Therefore, the velocity profile in fully developed laminar flow in a pipe is parabolic with a maximum at the centerline and minimum (*zero*) at the pipe wall. Also, the axial velocity u is positive for any r, and thus the axial pressure gradient dP/dx must be negative (i.e., pressure must decrease in the flow direction because of viscous effects).

$$V_{avg} = \frac{2}{R^2}\int_0^R u(r)r\,dr = \frac{-2}{R^2}\int_0^R \frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1 - \frac{r^2}{R^2}\right)r\,dr = -\frac{R^2}{8\mu}\left(\frac{dP}{dx}\right)$$

Combining the last two equations, the velocity profile is rewritten as

$$u(r) = 2V_{avg}\left(1 - \frac{r^2}{R^2}\right)$$

This is a convenient form for the velocity profile since $V_{avg}$ can be determined easily from the flow rate information. The maximum velocity occurs at the centerline and is determined from the velocity profile equation (equation above) by substituting $r = 0$,

$$u_{max} = 2V_{avg}$$

**Therefore, the average velocity in fully developed laminar pipe flow is one half of the maximum velocity.**

## 5.4. Pressure Drop and Head Loss

A quantity of interest in the analysis of pipe flow is the pressure drop (P since it is directly related to the power requirements of the fan or pump to maintain flow. We note that dP/dx= constant, and integrating from x= $x_1$ where the pressure is $P_1$ to x= $x_1$ + L where the pressure is $P_2$ gives

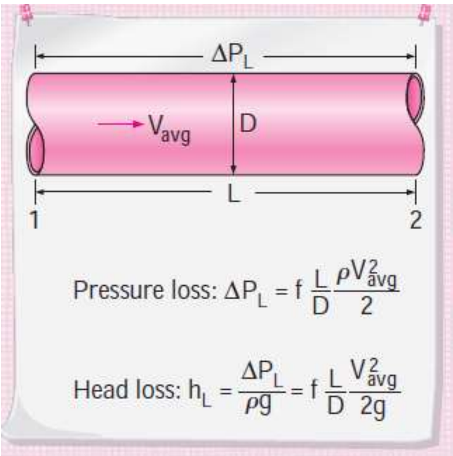
$$\frac{dP}{dx} = \frac{P_2 - P_1}{L}$$

Substituting above equation into the $V_{avg}$ expression, the pressure drop can be expressed as,

**Laminar flow:** $$\Delta P = P_1 - P_2 = \frac{8\mu L V_{avg}}{R^2} = \frac{32\mu L V_{avg}}{D^2}$$

In fluid flow, $\Delta P$ is used to designate pressure drop, and thus it is $P_1$ & $P_2$. A pressure drop due to viscous effects represents an irreversible pressure loss, and it is called **pressure loss** $\Delta P_L$ to emphasize that it is a *loss* (just like the head loss $h_L$, which is proportional to it). Therefore, the drop of pressure from $P_1$ to $P_2$ in this case is due entirely to viscous effects, and above equation represents the pressure loss $\Delta P_L$ when a fluid of viscosity m flows through a pipe of constant diameter *D* and length *L* at average velocity $V_{avg}$.

In practice, it is found convenient to express the pressure loss for all types of fully developed internal flows (laminar or turbulent flows, circular or noncircular pipes, smooth or rough surfaces, horizontal or inclined pipes).




**Pressure loss:** $$\Delta P_L = f\frac{L}{D}\frac{\rho V_{avg}^2}{2}$$

where $\rho V^2{}_{avg}/2$ is the *dynamic pressure*

*f* is the **Darcy friction factor,** $$f = \frac{8\tau_w}{\rho V_{avg}^2}$$

It is also called the **Darcy–Weisbach friction factor**, It should not be confused with the friction coefficient $C_f$ [also called the Fanning friction factor] which is defined as $C_f = 2\tau_w/(rV^2_{avg}) = f/4$.

Solving for *f* gives the friction factor for fully developed laminar flow in a circular pipe,

**Circular pipe, laminar:** $$f = \frac{64\mu}{\rho D V_{avg}} = \frac{64}{Re}$$

This equation shows that in laminar flow, the friction factor is a function of the Reynolds number only and is independent of the roughness of the pipe surface.

**Head loss:** $$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2_{avg}}{2g}$$

Once the pressure loss (or head loss) is known, the required pumping power to overcome the pressure loss is determined from

$$\dot{W}_{pump,\,L} = \dot{V}\,\Delta P_L = \dot{V}\rho g h_L = \dot{m} g h_L$$

where V is the volume flow rate and ṁ is the mass flow rate.

**Example:**

Water properties ($\rho$= 62.42 lbm/ft$^3$ and $\mu$=1.038×10$^{-3}$ lbm/ft .s) is flowing through a 0.12 in (= 0.010 ft) diameter 30 ft long horizontal pipe steadily at an average velocity of 3.0 ft/s (see Figure 5.4). Determine (**a**) the head loss, (**b**) the pressure drop, and (**c**) the pumping power requirement to overcome this pressure drop.

**Solution:**

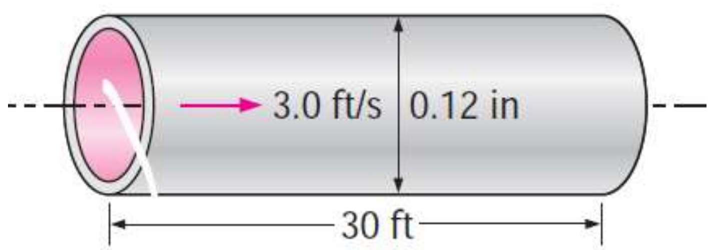



**Figure 5.4:** Schematic for above Example.

(**a**) First we need to determine the flow regime. The Reynolds number is

$$Re = \frac{\rho V_{avg} D}{\mu} = \frac{(62.42\ lbm/ft^3)(3\ ft/s)(0.01\ ft)}{1.038 \times 10^{-3}\ lbm/ft \cdot s} = 1803$$

which is less than 2300. Therefore, the flow is laminar. Then the friction factor and the head loss become

$$f = \frac{64}{Re} = \frac{64}{1803} = 0.0355$$

$$h_L = f\frac{L}{D}\frac{V_{avg}^2}{2g} = 0.0355\frac{30\ ft}{0.01\ ft}\frac{(3\ ft/s)^2}{2(32.2\ ft/s^2)} = \mathbf{14.9\ ft}$$

(**b**) Noting that the pipe is horizontal and its diameter is constant, the pressure drop in the pipe is due entirely to the frictional losses and is equivalent to the pressure loss,

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V_{avg}^2}{2} = 0.0355\frac{30\ ft}{0.01\ ft}\frac{(62.42\ lbm/ft^3)(3\ ft/s)^2}{2}\left(\frac{1\ lbf}{32.2\ lbm \cdot ft/s^2}\right)$$
$$= \mathbf{929\ lbf/ft^2 = 6.45\ psi}$$

(**c**) The volume flow rate and the pumping power requirements are

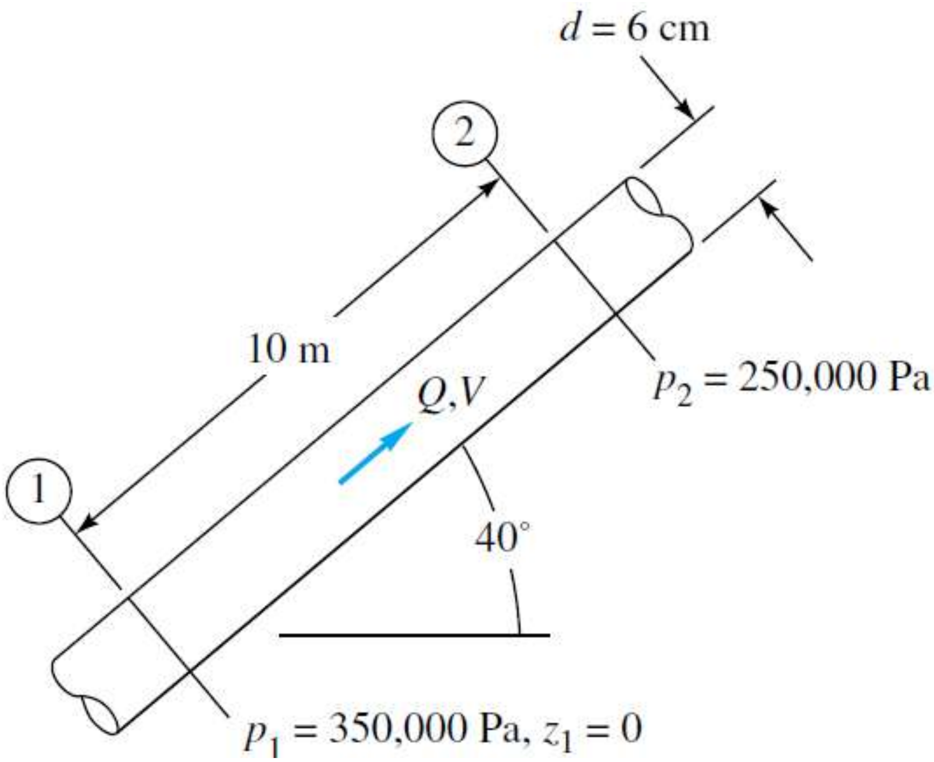
$$\dot{V} = V_{avg}A_c = V_{avg}(\pi D^2/4) = (3\ ft/s)[\pi(0.01\ ft)^2/4] = 0.000236\ ft^3/s$$

$$\dot{W}_{pump} = \dot{V}\,\Delta P = (0.000236\ ft^3/s)(929\ lbf/ft^2)\left(\frac{1\ W}{0.737\ lbf \cdot ft/s}\right) = \mathbf{0.30\ W}$$

**Example:**

An oil with $\rho$= 900 kg/m$^3$ and $\upsilon$= 0.0002 m$^2$/s flows upward through an inclined pipe as shown in Figure below. The pressure and elevation are known at sections 1 and 2, 10 m apart. Assuming steady laminar flow, (a) verify that the flow is up, (b) compute $h_f$ between 1 and 2, and compute (c) volume flow rate, (d) Velocity, and (e) Reynolds number. Is the flow really laminar?

**Solution:**




For later use, calculate

$$\mu = \rho\nu = (900\ \text{kg/m}^3)(0.0002\ \text{m}^2/\text{s}) = 0.18\ \text{kg/(m}\cdot\text{s)}$$

$$z_2 = \Delta L \sin 40° = (10\ \text{m})(0.643) = 6.43\ \text{m}$$

The flow goes in the direction of falling HGL; therefore compute the hydraulic grade-line height at each section

$$\text{HGL}_1 = z_1 + \frac{p_1}{\rho g} = 0 + \frac{350{,}000}{900(9.807)} = 39.65\ \text{m}$$

$$\text{HGL}_2 = z_2 + \frac{p_2}{\rho g} = 6.43 + \frac{250{,}000}{900(9.807)} = 34.75\ \text{m}$$

The HGL is lower at section 2; hence the flow is from 1 to 2 as assumed. *Ans. (a)*

The head loss is the change in HGL:

$$h_f = \text{HGL}_1 - \text{HGL}_2 = 39.65\ \text{m} - 34.75\ \text{m} = 4.9\ \text{m} \qquad \textit{Ans. (b)}$$

Half the length of the pipe is quite a large head loss.

We can compute $Q$ from the various laminar-flow formulas, notably Eq. (6.47)

We can compute $Q$ from the various laminar-flow formulas, notably Eq. (6.47)

$$Q = \frac{\pi \rho g d^4 h_f}{128 \mu L} = \frac{\pi(900)(9.807)(0.06)^4(4.9)}{128(0.18)(10)} = 0.0076 \text{ m}^3/\text{s} \qquad \textit{Ans. (c)}$$

Divide $Q$ by the pipe area to get the average velocity

$$V = \frac{Q}{\pi R^2} = \frac{0.0076}{\pi(0.03)^2} = 2.7 \text{ m/s} \qquad \textit{Ans. (d)}$$

With $V$ known, the Reynolds number is

$$\text{Re}_d = \frac{Vd}{\nu} = \frac{2.7(0.06)}{0.0002} = 810 \qquad \textit{Ans. (e)}$$

This is well below the transition value Re= 2300, and so we are fairly certain the flow is laminar.