Fluid Mechanics

Chapter One: Introduction

Dr. Amer Khalil Ababneh

Fluid Mechanics

Mechanics is the field of science focused on the motion of material bodies. **Mechanics** involves force, energy, motion, deformation, and material properties.

When mechanics applies to material bodies in the **solid phase**, the discipline is called *solid mechanics*.

When the material body is in the **gas or liquid** phase, the discipline is called ***fluid mechanics***

More about Definition of Fluids

Unlike solid, a *fluid* is a substance whose molecules move freely past each other.

More specifically, a fluid is a substance that will continuously deform—that is, flow under the action of a shear stress.

Alternatively, a solid will deform under the action of a shear stress but will not flow like a fluid.

Both liquids and gases are classified as fluid

1.1 Liquids and Gases

The difference between Liquids and gases is because of forces between the molecules.

liquid will take the **shape of a container** whereas a gas will expand to fill a closed container.

The **behavior** of the liquid is produced by strong attractive force between the molecules. This strong attractive force also explains why the density of a liquid is much higher than the density of gas.

A *gas* is a phase of material in which molecules are widely spaced, molecules move about freely, and forces between molecules are minuscule, except during collisions. Alternatively, a *liquid* is a phase of material in which molecules are closely spaced, molecules move about freely, and there are strong attractive forces between molecules. See Table 1.1.

1.2 The Continuum Assumption

While a body of fluid is comprised of molecules, most characteristics of fluids are due to average molecular behavior. That is, a fluid often behaves as if it were comprised of continuous matter that is infinitely divisible into smaller and smaller parts. This idea is called the *continuum assumption*.

When the continuum assumption is valid, engineers can apply limit concepts from differential calculus. Recall that a limit concept, for example, involves letting a length, an area, or a volume approach zero.

Because of the continuum assumption, fluid parameters such as density and velocity can be considered continuous functions of position with a value at each point in space.

More on Continuum Assumption

To gain insight into the validity of the continuum assumption, consider a hypothetical experiment to find density.

Fig. 1.1*a* shows a container of gas in which a volume has been identified. The idea is to find the mass of the molecules ΔM inside the volume and then to calculate density by

$$\rho = \frac{\Delta M}{\Delta V}$$

The calculated density is plotted in Fig. 1.1*b*. When the measuring volume is very small (approaching zero), the number of molecules in the volume will vary with time because of the random nature of molecular motion.

Thus, the density will vary as shown. As volume increases, the variations in calculated density will decrease until the calculated density is independent of the measuring volume. This condition corresponds to the vertical line at . If the volume is too large, as shown by , then the value of density may change due to spatial variations.

Continuum Assumption

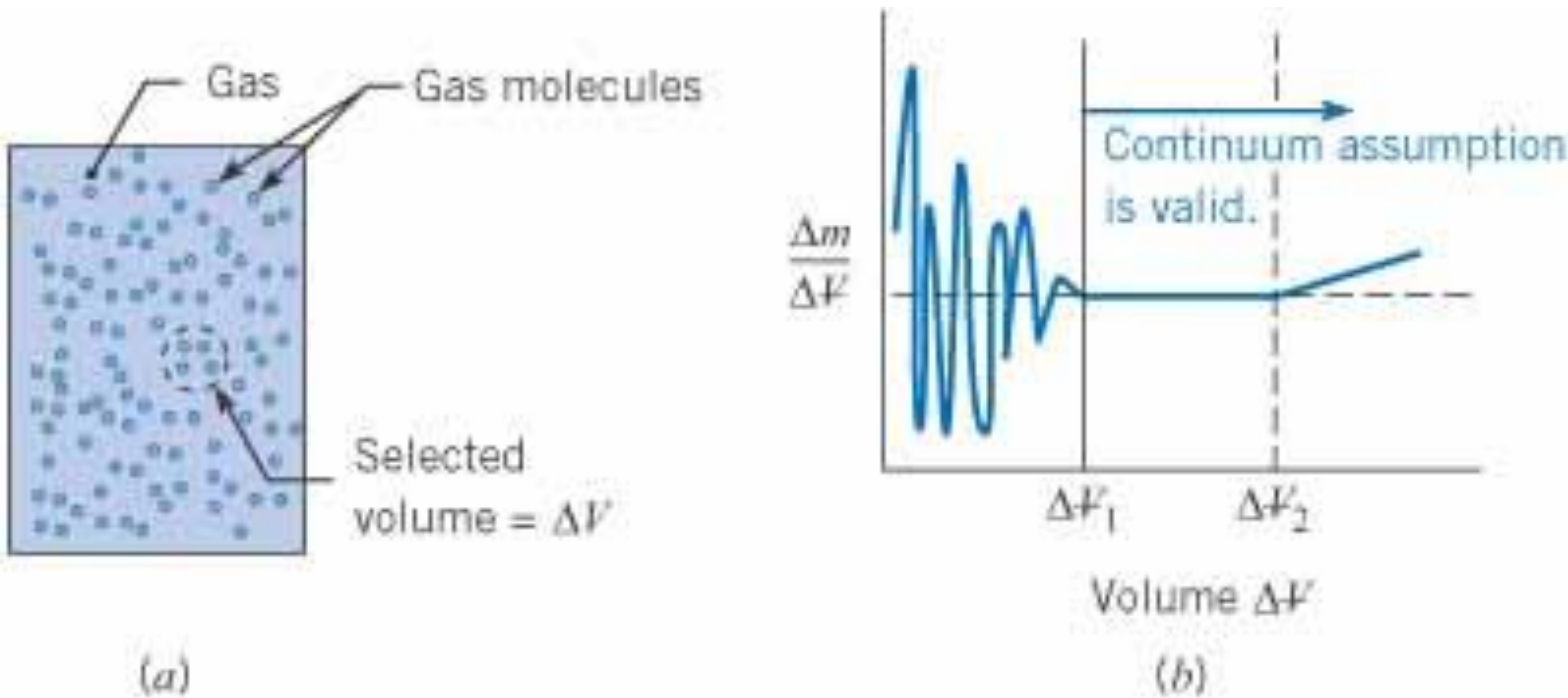


Figure 1.1

Continuum Assumption

In most engineering applications, continuum assumption is valid. To demonstrate this, compute the volume required to have 10^6 molecules. Using Avogadro number ($\sim 6\times10^{23}$ molecules/mole), the limiting volume for water is 10^{-13} mm^3 (or $<10^{-4}$ mm cube). For ideal gas at standard conditions, the limiting volume is 10^{-10} mm^3 (or $<10^{-3}$ mm cube).

1.3 Dimensions and Units

A *dimension* is a category that represents a physical quantity such as mass, length, time, momentum, force, acceleration, and energy. To simplify matters, engineers express dimensions using a limited set that are called ***primary dimensions***. Table 1.2 lists one common set of primary dimensions.

Table 1.2 PRIMARY DIMENSIONS

Dimension Symbol	Unit (SI)
Length L	meter (m)
Mass M	kilogram (kg)
Time T	second (s)
Temperature θ	kelvin (K)
Electric current i	ampere (A)
Amount of light C	candela (cd)
Amount of matter	mole (mol)

Secondary dimensions such as momentum and energy can be related to primary dimensions by using equations.

For example, the secondary dimension "force" is expressed in primary dimensions by using Newton's second law of motion, $F = ma$. The primary dimensions of acceleration are L/T^2, so

$$[F] = [ma] = M\frac{L}{T^2} = \frac{ML}{T^2}$$

Units

While a dimension expresses a specific type of physical quantity, a unit assigns a number so that the dimension can be measured. For example, measurement of volume (a dimension) can be expressed using units of liters.

Similarly, measurement of energy (a dimension) can be expressed using units of joules. Most dimensions have multiple units that are used for measurement. For example, the dimension of “force” can be expressed using units of newtons, pounds-force, or dynes.

Suggested Problems

1.1 For each variable below, list three common units.

a. Volume flow rate (Q), mass flow rate (m), and pressure (p).

b. Force, energy, power.

c. Viscosity.

1.5 Find the primary dimensions of each of the following terms.

a. $(\rho V2)/2$ (kinetic pressure), where ρ is fluid density and V is velocity.

b. T (torque).

c. P (power).

d. $(\rho V2L)/\delta$ (Weber number), where ρ is fluid density, V is velocity, L is length, and σ is surface tension.

Fluid Mechanics

Chapter Two: Fluid Properties

Dr. Amer Khalil Ababneh

Fluid Properties

A fluid has certain characteristics by which its physical condition may be described. These **characteristics are called *properties*** of the fluid.

2.1 Properties Involving Mass and Weight

Mass Density ρ

Mass density is defined as the ratio of mass to volume at a point, given by

$$\rho = \lim_{\Delta V \to 0} \frac{\Delta m}{\Delta V}$$

Mass density has units of kilograms per cubic meter (kg/m3).

The mass <u>density of water</u> at 4°C is 1000 kg/m^3 and it decreases slightly with increasing temperature, as shown in Table A.5. The mass <u>density of air</u> at 20°C and standard atmospheric pressure is 1.2 kg/m^3

Specific Weight, γ

The gravitational force per unit volume of fluid, or simply the weight per unit volume, is defined as *specific weight*. It is given the Greek symbol γ (gamma).

$$\gamma = \frac{W}{\forall} = \frac{mg}{\forall} = \frac{\rho \forall g}{\forall} = \rho g$$

Water at 20°C has a specific weight of 9790 N/m^3. In contrast, the specific weight of air at 20°C and standard atmospheric pressure is 11.8 N/m^3.

Specific weights of common liquids are given in Table A.4.

Variation in Liquid Density

In practice, engineers need to decide whether or not to model a fluid as constant density or variable density. Usually, a liquid such as water requires a large change in pressure in order to change the density. Thus, for most applications, liquids can be considered **incompressible** and can be assumed to have constant density. An exception to this occurs when different solutions, such as saline and fresh water, are mixed. <u>A mixture of salt in water changes the density of the water without changing its volume</u>. Therefore in some flows, such as in estuaries, density variations may occur within the flow field even though the fluid is essentially incompressible. A fluid wherein density varies spatially is described as ***nonhomogeneous***. This text emphasizes the flow of ***homogeneous*** fluids, so the term ***incompressible***, used throughout the text, implies constant density.

Specific Gravity, S

The ratio of the specific weight of a given fluid to the specific weight of water at the standard reference temperature 4°C is defined as *specific gravity*, S:

$$S = \frac{\gamma_{fluid}}{\gamma_{water}} = \frac{\rho_{fluid}}{\rho_{water}}$$

The specific weight of water at atmospheric pressure is 9790 N/m^3. The specific gravity of mercury at 20°C is

$$S = \frac{113\, kN/m^3}{9.79\, kN/m^3} = 13.6$$

Because specific gravity is a ratio of specific weights, it is dimensionless and therefore independent of the system of units used.

2.2 Ideal Gas Law

The *ideal gas law* relates important thermodynamic properties, and is often used to calculate density. One form of the law is

$$p\forall = nR_uT$$

where *p* is the absolute pressure, $\forall$ is the volume, *n* is the number of moles, R_u is the universal gas constant (the same for all gases), and *T* is absolute temperature. Absolute pressure, introduced in Chapter 3, is referred to absolute zero. The universal gas constant is 8.314 kJ/kmol-K in the SI system.

A second form of the *ideal gas law* is

$$p = \rho RT$$

EXAMPLE 2.1 DESITY OF AIR

Air at standard sea-level pressure (p = 101 kN/m^2) has a temperature of 4°C. What is the density (kg/m^3). of the air? R = 287 J/kg K.

Solution: Apply ideal gas law

$$\rho = \frac{p}{RT}$$

$$\rho = \frac{101 \times 10^3 \text{ N/m}^2}{287 \text{ J/kg k} \times (273 + 4) \text{ K}} = \boxed{1.27 \text{ kg/m}^3}$$

2.3 Properties Involving Thermal Energy

Specific Heat, *c*

The property that describes the capacity of a substance to store thermal energy is called ***specific heat***. By definition, it is the amount of thermal energy that must be transferred to a unit mass of substance to raise its temperature by one degree. The specific heat of a gas depends on the process accompanying the change in temperature. If the ***specific volume v*** of the gas ($v = 1/\rho$) remains constant while the temperature changes, then the specific heat is identified as c_v. However, if the pressure is held constant during the change in state, then the specific heat is identified as c_p. The ratio c_p/c_v is given the symbol *k*. Values for c_p and *k* for various gases are given in Table A.2.

Internal Energy

The energy that a substance possesses because of the state of the molecular activity in the substance is termed *internal energy.* Internal energy is usually expressed as a **specific quantity; i.e.,** per unit mass. In the SI system, the *specific internal energy, u*, is given in joules per kilogram; in Traditional Units it is given in Btu/lbm. The internal energy is generally a function of temperature and pressure. However, for an ideal gas, it is a function of temperature alone.

Enthalpy

The combination $u + p/\rho$ is encountered frequently in equations for thermodynamics and compressible flow; it has been given the name *specific enthalpy.* For an ideal gas, u and p/ρ are functions of temperature alone. Consequently their sum, specific enthalpy, is also a function solely of temperature.

2.4 Viscosity

The property of viscosity is important to engineering practice because it leads to significant energy loss when moving fluids contact a solid boundary, or when different zones of fluid are flowing at different velocities.

Viscosity, μ

The symbol used to represent viscosity is μ (mu). *Viscosity* (also called *dynamic viscosity*, or *absolute viscosity*) is a measure of a fluid's resistance to deformation under shear stress. For example, crude oil has a higher resistance to shear than does water. Crude oil will pour *more slowly* than water from an identical beaker held at the same angle. This relative slowness of the oil implies a low "speed" or rate of strain.

Viscosity, μ (continue)

To understand the physics of viscosity, it is useful to refer back to solid mechanics and the concepts of shear stress and shear strain. Shear stress, τ (tau) is the ratio of force/area on a surface when the force is aligned parallel to the area. Shear strain is a change in an interior angle of a cubical element, (φ, that was originally a right angle. The shear stress on a material element in solid mechanics is proportional to the strain, and the constant of proportionality is the shear modulus:

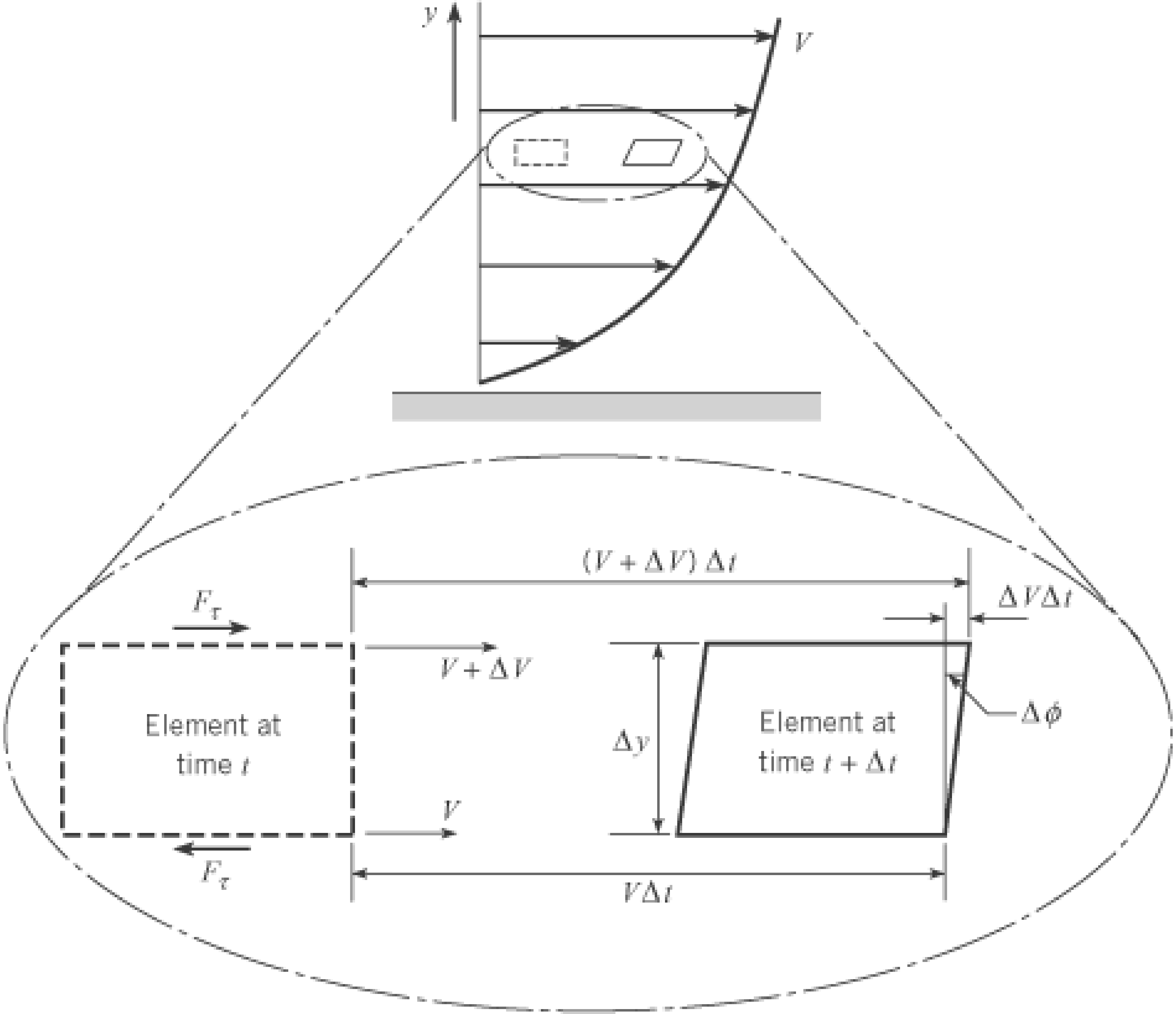
In solids,

$$\{\text{shear stress}\} = \{\text{shear modulus}\} \times \{\text{strain}\}$$

In fluid flow, however, the shear stress on a fluid element is proportional to the rate (speed) of strain, and the constant of proportionality is the viscosity:

$$\{\text{shear stress}\} = \{\text{viscosity}\} \times \{\text{rate of strain}\}$$

To derive an expression for the shear stress in fluids, consider Figure 2.1



y
V
(V + ΔV) Δt
ΔVΔt
F_τ
V + ΔV
Element at time t
Δy
Element at time t + Δt
Δφ
V
F_τ
VΔt

Figure 2.1

Viscosity, μ (continue)

Consider an initially rectangular element in a parallel flow field. As the element moves downstream, a shear force on the top of the element (and a corresponding shear stress in the opposite direction on the bottom of the element) causes the top surface to move faster (with velocity $V + \Delta V$) than the bottom (at velocity V). The forward and rearward edges become inclined at an angle $\Delta\varphi$ with respect to the vertical. The rate at which $\Delta\varphi$ changes with time, given by φ, is the *rate of strain*, and can be related to the velocity difference between the two surfaces.

In time (Δt) the upper surface moves ($V + \Delta V$) Δt while the bottom surface moves $V\Delta t$, so the net difference is $\Delta V \Delta t$. The strain $\Delta\varphi$ is

$$\Delta\phi \approx \frac{\Delta V \Delta t}{\Delta y}$$

where Δy is the distance between the two surfaces. The rate of strain is

$$\frac{\Delta\phi}{\Delta t} \approx \frac{\Delta V}{\Delta y}$$

In the limit as Δt, $\Delta y \rightarrow 0$, the rate of strain is related to the velocity gradient by $\varphi = d\varphi/dt = dV/dy$, so the shear stress (shear force per unit area) is

$$\tau = \mu \frac{dV}{dy}$$

The term *dV/dy* represents the velocity gradient (or change of velocity with distance from the wall), where *V* is the fluid velocity and *y* is the distance measured from the wall, see Figure 2.2. The velocity distribution shown is characteristic of flow next to a stationary solid boundary, such as fluid flowing through a pipe. Several observations: First, the velocity gradient at the boundary is finite. The curve of velocity variation cannot be tangent to the boundary because this would imply an infinite velocity gradient and, in turn, an infinite shear stress, which is impossible.

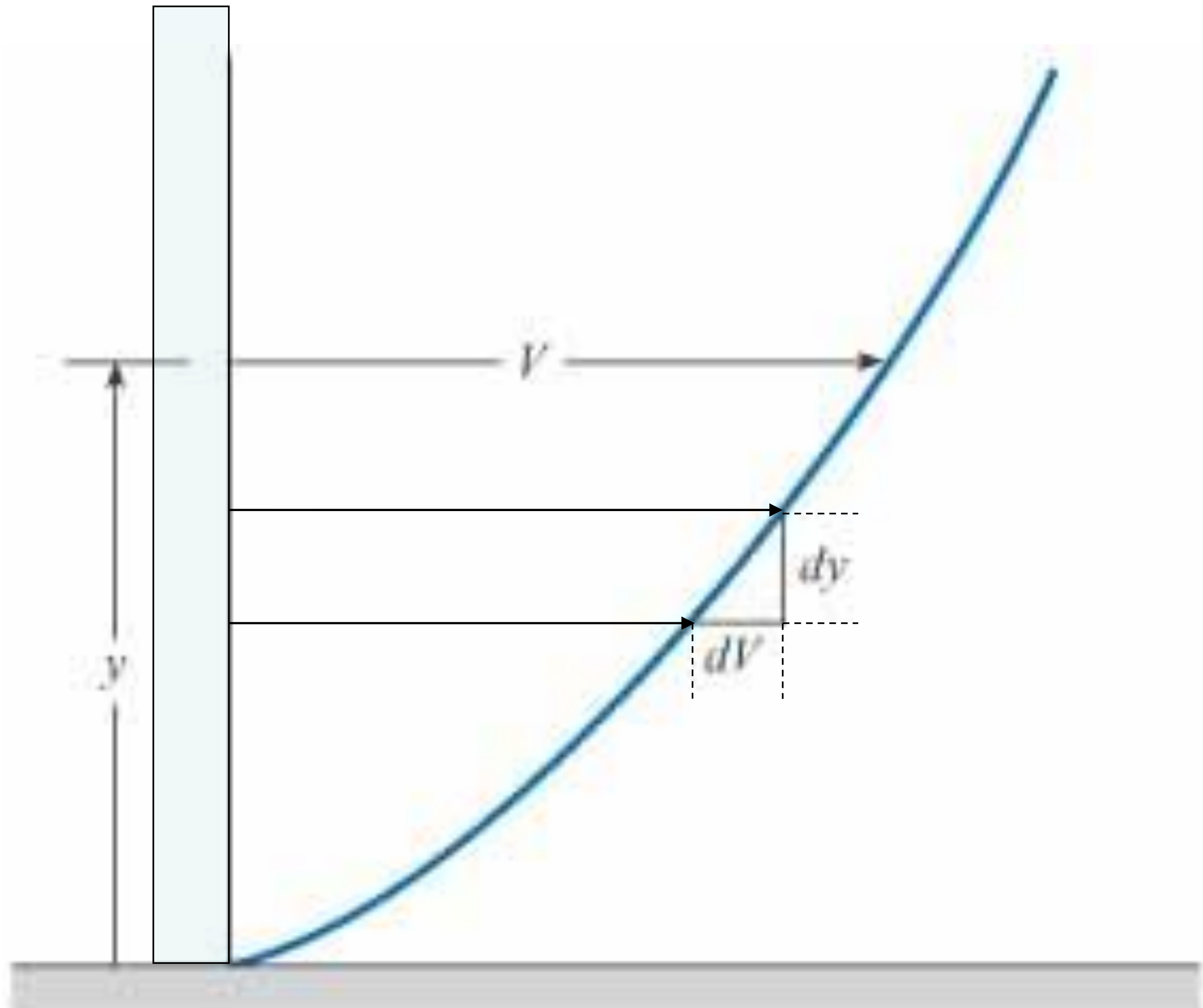


Figure 2.2

Second, a velocity gradient (*dV/dy*) that decreases with distance from the boundary has a maximum shear stress at the boundary, and the shear stress decreases with distance from the boundary. Also note that the velocity of the fluid is zero at the stationary boundary. That is, at the boundary surface the fluid has the velocity of the boundary—no slip occurs between the fluid and the boundary. This is referred to as the ***no-slip condition***. The no-slip condition is characteristic of all flows used in this text.

The units for the viscosity can be derived,

$$\mu = \frac{\tau}{dV/dy} \Rightarrow \frac{N/m^2}{(m/s)/m} = N \cdot s/m^2$$

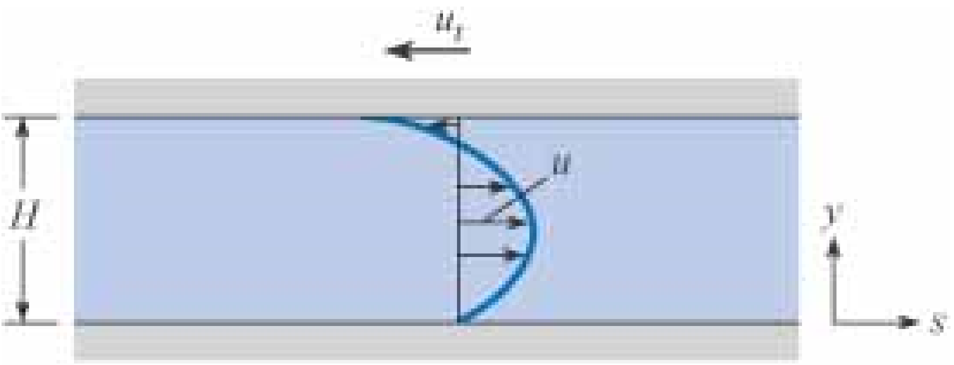
A common unit of viscosity is the *poise,* which is 1 dyne-s/cm^2 or 0.1 N · s/m^2. The viscosity of water at 20°C is one centipoise (10^{-2} poise) or 10^{-3} N · s/m^2. The unit of viscosity in the traditional system is lbf · s/ft^2.

Think?

Problem 2.38 A laminar flow occurs between two horizontal parallel plates under a pressure gradient *dp*/*ds* (*p* decreases in the positive *s* direction). The upper plate moves left (negative) at velocity *ut*. The expression for local velocity *u* is given as

$$u = -\frac{1}{2\mu}\frac{dp}{ds}(Hy - y^2) + u_t\frac{y}{H}$$

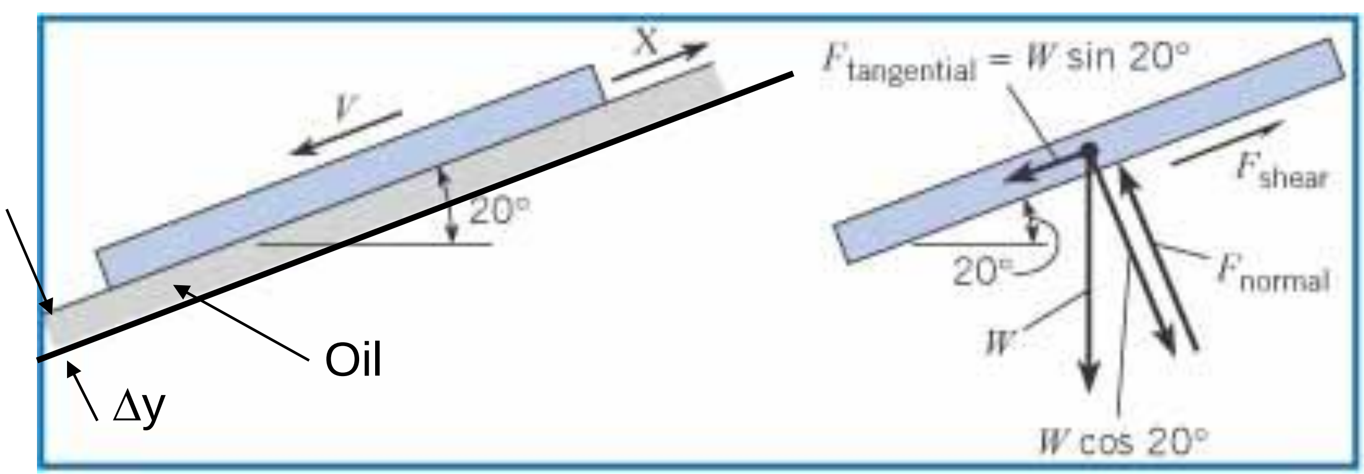
1) Where is the shear stress is maximum?
2) Where is the shear stress is zero?
3) Derive the expression for shear stress in terms of y.



$$\tau = \mu\frac{dV}{dy}$$

EXAMPLE 2.3 MODELING A BOARD SLIDING ON A LIQUID LAYER

A board 1 m by 1 m that weighs 25 N slides down an inclined ramp (slope = 20°) with a velocity of 2.0 cm/s. The board is separated from the ramp by a thin film of oil with a viscosity of 0.05 N · s/m². Neglecting edge effects, calculate the space between the board and the ramp. Velocity profile is linear.



Solution

1. Freebody analysis

$$F_{\text{tangential}} = F_{\text{shear}}$$

$$W \sin 20° = \tau A$$

$$W \sin 20° = \mu \frac{dV}{dy} A$$

2. Substitution of *dV/dy* as Δ*V*/Δ*y*

$$W \sin 20° = \mu \frac{\Delta V}{\Delta y} A$$

3. Solution for Δ*y*

$$\tau = \mu \frac{dV}{dy}$$

$$\Delta y = \frac{\mu \Delta V A}{W \sin 20°}$$

$$\Delta y = \frac{0.05 \text{ N} \cdot \text{s/m}^2 \times 0.020 \text{ m/s} \times 1 \text{ m}^2}{25 \text{ N} \times \sin 20°}$$

$$\Delta y = 0.000117 \text{ m}$$

$$\Delta y = \boxed{0.117 \text{ mm}}$$

Kinematic Viscosity, ν

Many equations of fluid mechanics include the ratio μ/ρ. Because it occurs so frequently, this ratio has been given the special name *kinematic viscosity*. The symbol used to identify kinematic viscosity is ν (nu). Units of kinematic viscosity ν are m^2/s, as shown.

$$\nu = \frac{\mu}{\rho} \Rightarrow \frac{\mathrm{N \cdot s/m^2}}{\mathrm{kg/m^3}} = \mathrm{m^2/s}$$

The units for kinematic viscosity in the traditional system are ft^2/s.

Temperature Dependency of Viscosity

The effect of temperature on viscosity is **different** for **liquids and gases**. The viscosity of liquids decreases as the temperature increases, whereas the viscosity of gases increases with increasing temperature; this trend is also true for kinematic viscosity (see Fig. 2.3 and Figs. A.2 and A.3).

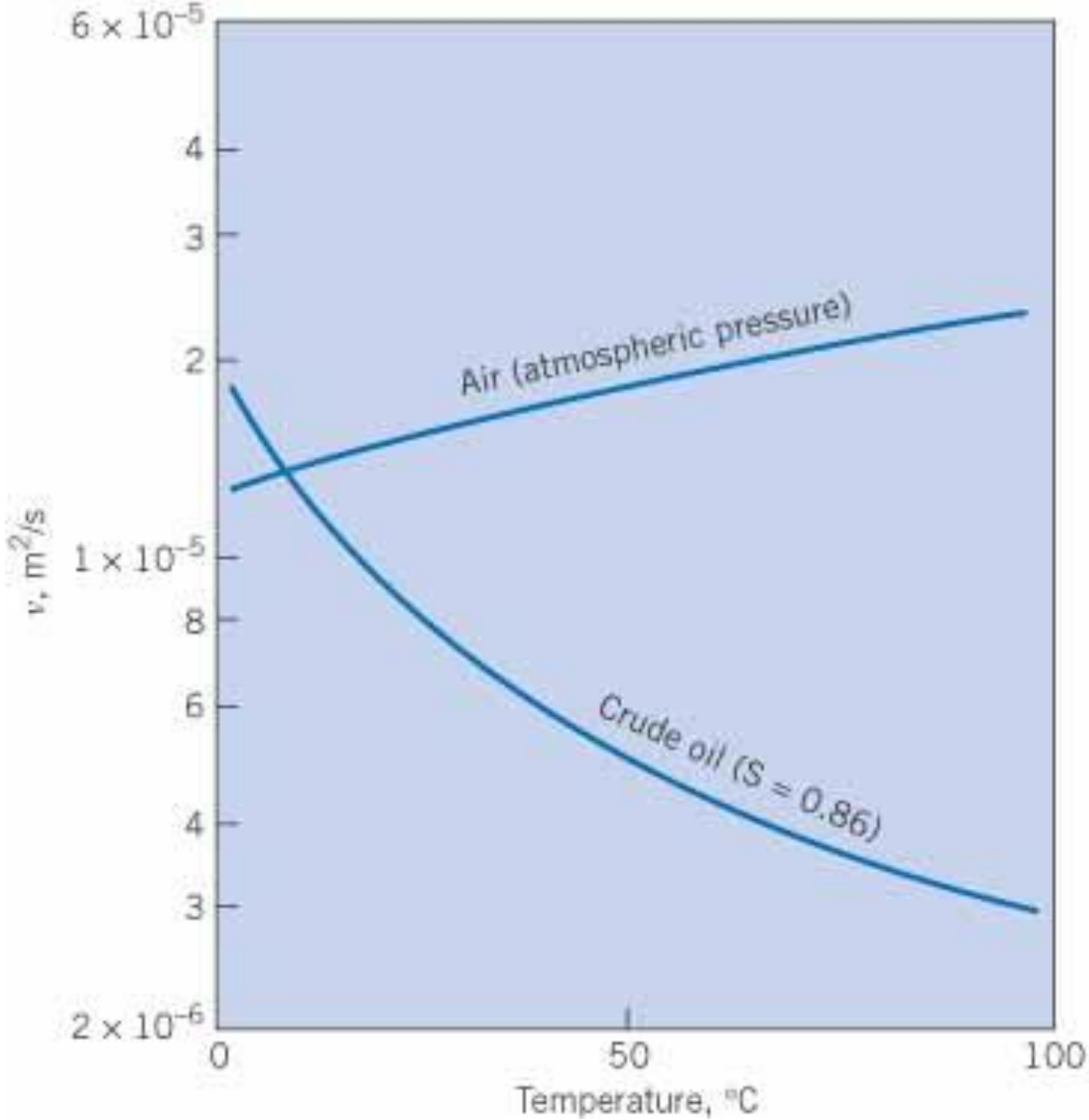


Figure 2.3 *Kinematic viscosity for air and crude oil.*

An equation for the variation (interpolation) of **liquid viscosity** with temperature is

$$\mu = Ce^{b/T}$$

where C and b are empirical constants that require viscosity data at two temperatures for evaluation. This equation should be used primarily for data interpolation. The variation of viscosity (dynamic and kinematic) for other fluids is given in Figs. A.2 and A.3.

An estimate for the variation of **gas viscosity** with temperature is *Sutherland's equation,*

$$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S}{T + S}$$

where μ_0 is the viscosity at temperature T_0, and S is Sutherland's constant. All temperatures are absolute. Sutherland's constant for air is 111 K; values for other gases are given in Table A.2. Using Sutherland's equation for air yields viscosities with an accuracy of ±2% for temperatures between 170 K and 1900 K. In general, the **effect of pressure** on the viscosity of common gases is minimal for pressures less than 10 atmospheres.

Newtonian Versus Non-Newtonian Fluids

Fluids for which the shear **stress is directly proportional to the rate of strain** are called N*ewtonian fluids*. Because shear stress is directly proportional to the shear strain, *dV/dy*, a plot relating these variables (see Fig. 2.6) results in a straight line passing through the origin. The **slope of this line** is the value of the dynamic (absolute) viscosity.

For some fluids the shear stress may not be directly proportional to the rate of strain; these are called *non-Newtonian fluids*. One class of non-Newtonian fluids, **shear-thinning fluids**, has the property that the ratio of shear stress to shear strain decreases as the shear strain increases (see Fig. 2.6). Some common shear-thinning fluids are toothpaste, catsup, paints, and printer's ink. Fluids for which the viscosity increases with shear rate are **shear-thickening fluids**. Some examples of these fluids are mixtures of glass particles in water and gypsum-water mixtures. Another type of non-Newtonian fluid, called a **Bingham** plastic, acts like a solid for small values of shear stress and then behaves as a fluid at higher shear stress. The shear stress versus shear strain rate for a Bingham plastic is also shown in Fig. 2.6. This book will focus on the theory and applications involving Newtonian fluids.

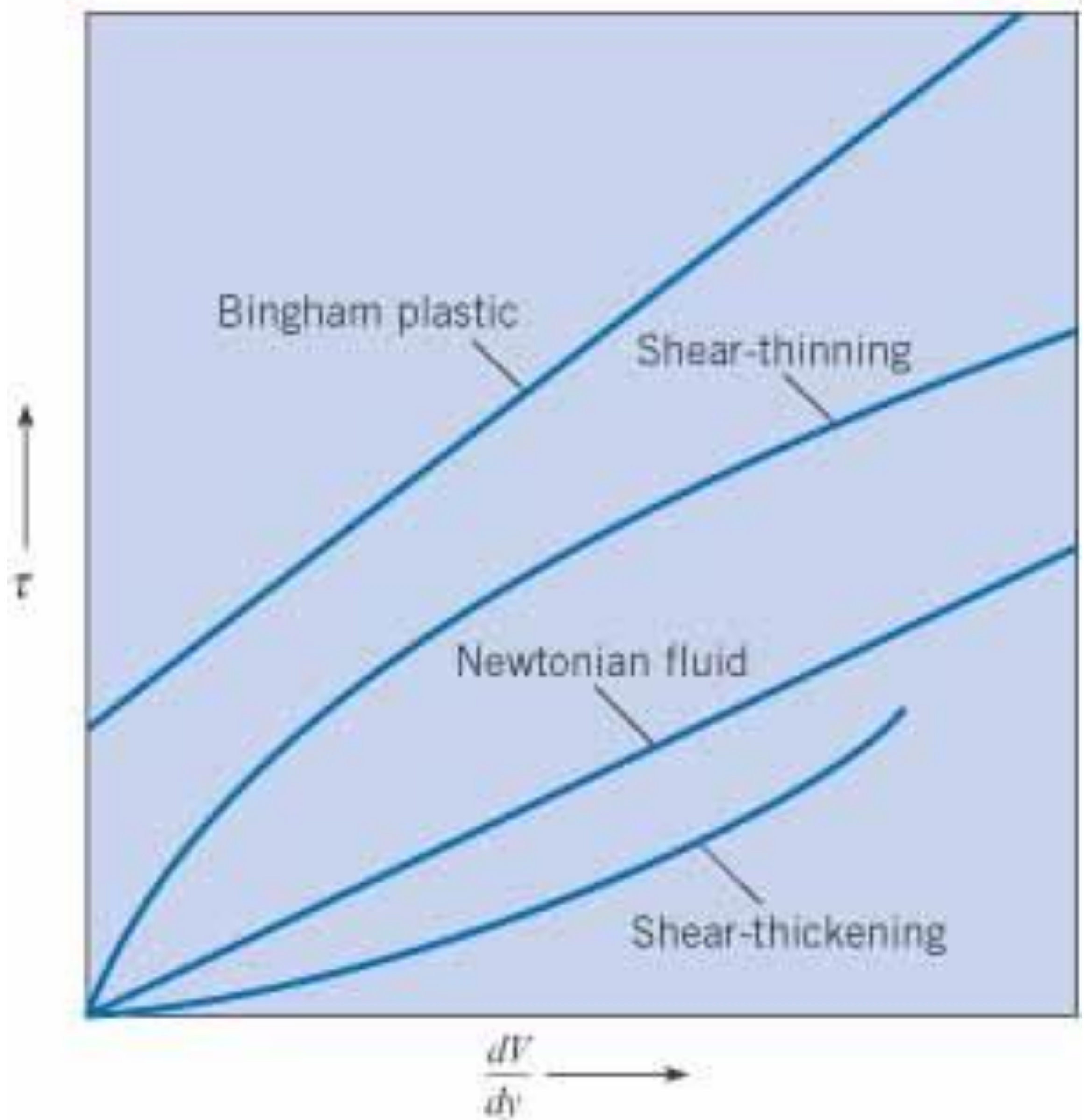


Figure 2.6 *Shear stress relations for different types of fluids.*

2.5 Bulk Modulus of Elasticity

The *bulk modulus of elasticity, E_v*, is a property that relates **changes in pressure** to **changes in volume** (e.g., expansion or contraction)

$$E_v = -\frac{dp}{d\mathcal{V} / \mathcal{V}} = -\frac{\text{change in pressure}}{\text{fractional change in volume}}$$

where *dp* is the differential pressure change, $d\mathcal{V}$ is the differential volume change, and $\mathcal{V}$ is the volume of fluid. Because $d\mathcal{V} / \mathcal{V}$ is negative for a positive *dp*, a negative sign is used in the definition to yield a positive E_v. The elasticity is often called the **compressibility** of the fluid. Also, it can be shown that E_v equals to

$$E_v = \frac{dp}{d\rho / \rho} = \frac{\text{change in pressure}}{\text{fractional change in density}}$$

The bulk modulus of elasticity of water is approximately 2.2 GN/m^2, which corresponds to a 0.05% change in volume for a change of 1 MN/m^2 in pressure. Obviously, the term *incompressible* is justifiably applied to water because it has such a small change in volume for a very large change in pressure. The elasticity of an ideal gas is proportional to the pressure, according to the ideal gas law. For an isothermal (constant-temperature) process,

$$p = \rho R T$$

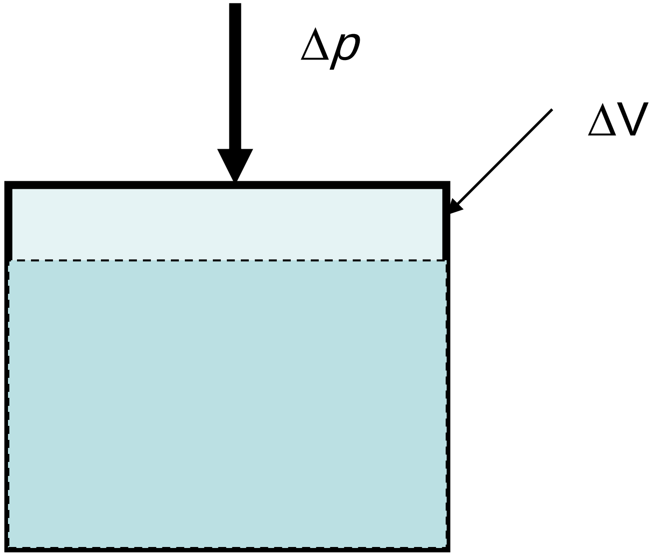
$$\frac{dp}{d\rho} = RT \quad \text{which implies} \quad E_v = \rho \frac{dp}{d\rho} = \rho R T = p$$

Think?

Problem 2.46 Calculate the pressure increase that must be applied to water to reduce its volume by 2%. $E_v = 2.2$ GPa

Answer:

$\Delta p = 44$ MPa



2.6 Surface Tension

Surface tension, σ (sigma), is a material property whereby a liquid at a material interface, usually liquid-gas, exerts a force per unit length along the surface. According to the theory of molecular attraction, molecules of liquid considerably below the surface act on each other by forces that are equal in all directions. However, molecules near the surface have a greater attraction for each other than they do for molecules below the surface because of the presence of a different substance above the surface. This produces a layer of surface molecules on the liquid that acts like a stretched membrane. Because of this membrane effect, each portion of the liquid surface exerts “tension” on adjacent portions of the surface or on objects that are in contact with the liquid surface. This tension acts in the plane of the surface, and is given by:

$$F_\sigma = \sigma L$$

where *L* is the length over which the surface tension acts.

Surface tension for a water–air surface is 0.073 N/m at room temperature; σ decreases with increasing temperature. The effect of surface tension is illustrated for the case of *capillary action* (rise above a static water level at atmospheric pressure) in a small tube (Fig. 2.7). The relatively greater **attraction** of the water molecules for the glass rather than the air causes the water surface to curve upward in the region of the glass wall. It may be assumed that the *contact angle* θ (theta) is equal to 0° for water against glass. The surface tension force produces a net upward force on the water that causes the water in the tube to rise above the water surface in the reservoir.

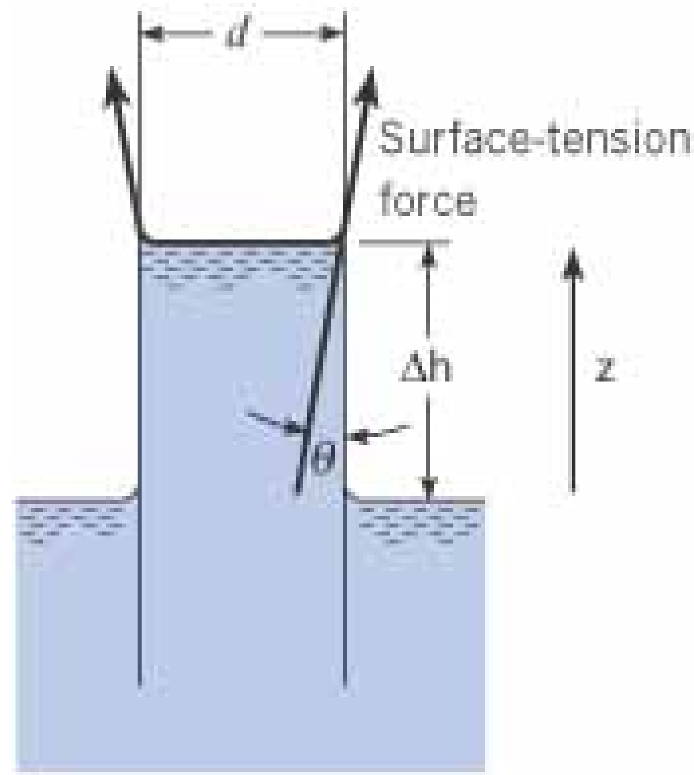


Figure 2.7 *Capillary action in a small tube.*

EXAMPLE 2.4 CAPILLARY RISE IN A TUBE

To what height above the reservoir level will water (at 20°C) rise in a glass tube, such as that shown in Fig. 2.7, if the inside diameter of the tube is 1.6 mm?

Properties: Water (20 °C), Table A.5, σ = 0.073 N/*m*; γ = 9790 N/m3.

Solution

1. Force balance: Weight of water (down) is balanced by surface tension force (up).

$$F_{\sigma,z} - W = 0$$

$$\sigma \pi d \cos\theta - \gamma(\Delta h)(\pi d^2 / 4) = 0$$

Because θ~0, hence cos θ =1. Therefore:

$$\Delta h = \frac{4\sigma}{\gamma d} = \frac{4 \times 0.073 \text{ N / m}}{9790 \text{ N / m}^3 \times 1.6 \times 10^{-3} \text{ m}} = \boxed{18.6 \text{ mm}}$$

2.7 Vapor Pressure

The pressure at which a liquid will vaporize, or boil, at a given temperature, is called its *vapor pressure*. This means that boiling occurs whenever the local pressure equals the vapor pressure. Vapor pressure increases with temperature. Note that there are two ways to boil a liquid. One way is to raise the temperature, assuming that the pressure is fixed. For water at 101.3 kPa, this can be accomplished by increasing the temperature of water to 100°C, thus reaching the temperature where the vapor pressure is equal to the same value. However, boiling can also occur in water at temperatures much below 100°C if the pressure in the water is reduced to the vapor pressure of water corresponding to that lower temperature. For example, the vapor pressure of water at 10°C is 1.230 kPa. Therefore, if the pressure in water at 10°C is reduced to 1.230 kPa, the water boils. Such boiling often occurs in localized low-pressure zones of flowing liquids, such as on the suction side of a pump. When localized low-pressure boiling does occur in flowing liquids, vapor bubbles start growing in local regions of very low pressure and then collapse in regions of higher pressure downstream. This phenomenon, which is called *cavitation,* can cause extensive damage to fluids systems. Table A.5 gives values of vapor pressure for water.

Suggest problems

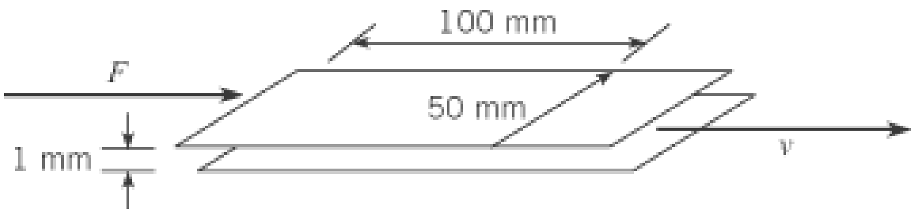
2.6) Determine the density and specific weight of methane gas at at a pressure of 300 kN/m2 absolute and 60°C.

Answer: $\rho_{methane} = 1.74 \text{ kg/m}^3,$

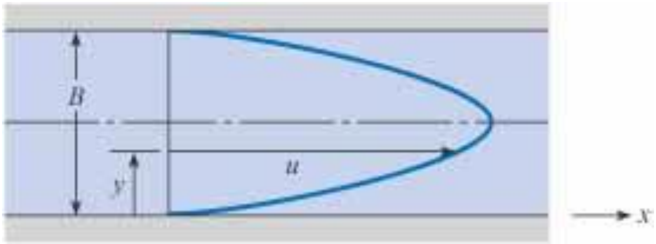
$\gamma_{methane} = 17.1 \text{ N/m}^3$

2.11 What are the specific weight and density of air at an absolute pressure of 600 kPa and a temperature of 50°C?

2.33 The sliding plate viscometer shown below is used to measure the viscosity of a fluid. The top plate is moving to the right with a constant velocity of 10 m/s in response to a force of 3 N. The bottom plate is stationary. What is the viscosity of the fluid? Assume a linear velocity distribution.



2.34 The velocity distribution for water (20°C) near a wall is given by $u = a(y/b)1/6$, where $a = 10$ m/s, $b = 2$ mm, and y is the distance from the wall in mm. Determine the shear stress in the water at $y = 1$ mm.



2.45 A pressure of 2 × 106 N/m2 is applied to a mass of water that initially filled a 2000 cm3 volume. Estimate its volume after the pressure is applied.

Fluid Mechanics

Chapter Three: Fluid Statics

Dr. Amer Khalil Ababneh

This chapter deals with mechanics of fluids by introducing concepts related to **pressure** and by describing how to calculate **forces** associated with distributions of pressure. This chapter is restricted to fluids that are in **hydrostatic equilibrium**.

As shown in Fig. 3.1, the **hydrostatic condition** involves equilibrium of a fluid particle. A *fluid particle,* is defined as a body of fluid having finite mass and internal structure but negligible dimensions. Thus, a fluid particle is very small, but large enough so that the continuum assumption applies. The *hydrostatic condition* means that each fluid particle is in force equilibrium with the net force due to pressure balancing the weight of the fluid particle.

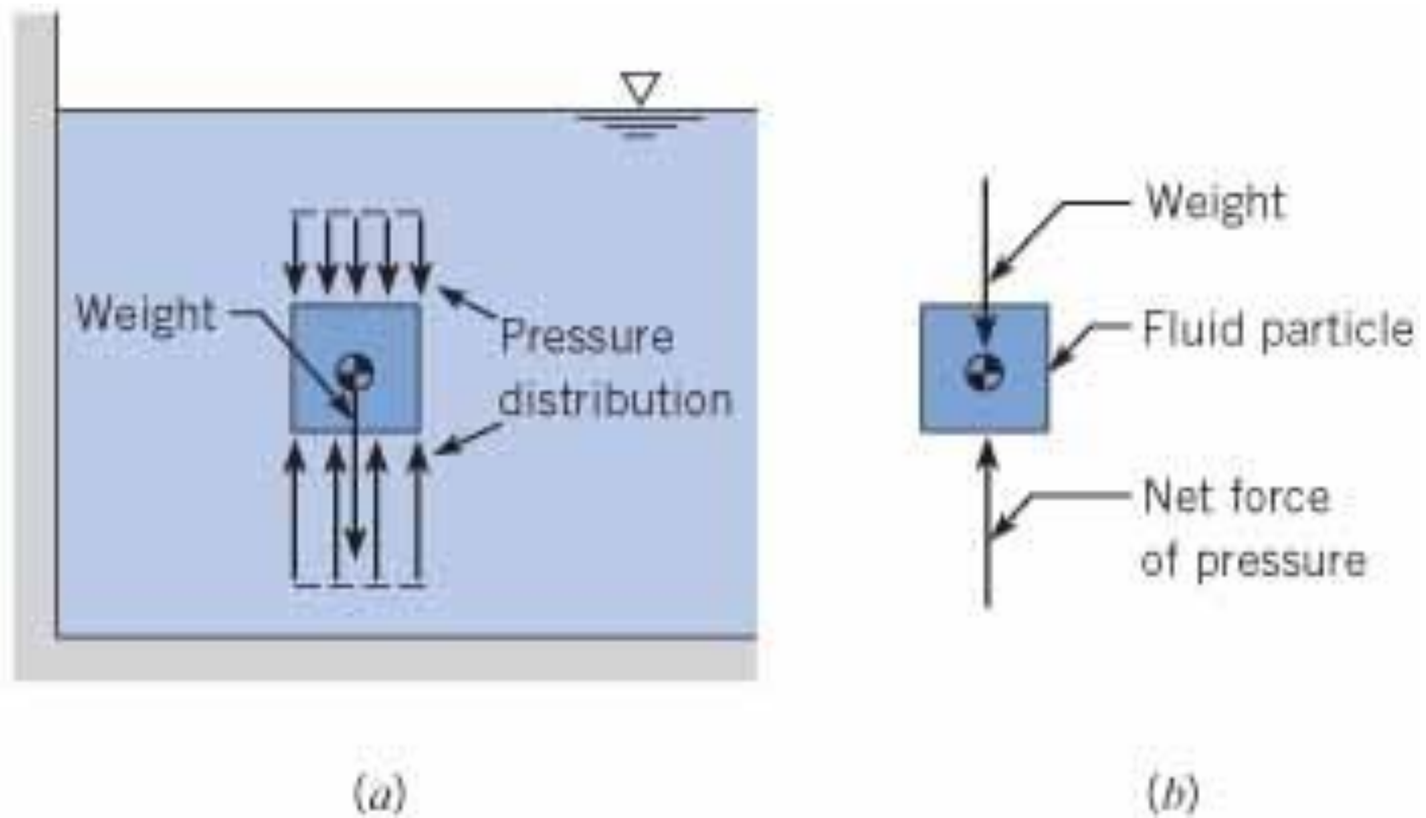


Figure 3.1 *The hydrostatic condition.*

(a) A fluid particle in a body of fluid.
(b) Forces acting on the fluid paricle.

3.1 Pressure

Definition of Pressure

Pressure is defined as the ratio of normal force to area at a point.

$$p = \lim_{\Delta A \longrightarrow 0} \frac{|\Delta \overrightarrow{F}_{normal}|}{\Delta A} = \frac{dF_{normal}}{dA}$$

Pressure is a scalar quantity; that is, it has magnitude only. Pressure is not a force; rather it is a scalar that produces a resultant force by its action on an area. The resultant force is normal to the area and acts in a direction toward the surface (compressive).

The SI units for pressure give a ratio of force to area. Newtons per square meter of area, or pascals (Pa.

Absolute Pressure, Gage Pressure, and Vacuum Pressure

Engineers use several different scales for pressure. Absolute pressure is referenced to regions such as outer space, where the pressure is essentially zero because the region is devoid of gas. The pressure in a perfect vacuum is called absolute zero, and pressure measured relative to this zero pressure is termed *absolute pressure.*

When pressure is measured relative to prevailing local atmospheric pressure, the pressure value is called *gage pressure*. For example, when a tire pressure gage gives a value of 300 kPa (44 psi), this means that the absolute pressure in the tire is 300 kPa greater than local atmospheric pressure. To convert gage pressure to absolute pressure, add the local atmospheric pressure. For example, a gage pressure of 50 kPa recorded in a location where the atmospheric pressure is 100 kPa is expressed as either

$$p = 50 \text{ kPa gage} \quad \text{or} \quad p = 150 \text{ kPa abs}$$

Gage and absolute pressures are often identified after the unit as shown above in the equation.

When pressure is less than atmospheric, the pressure can be described using vacuum pressure. *Vacuum pressure* is defined as the difference between atmospheric pressure and actual pressure. Vacuum pressure is a positive number and equals the absolute value of gage pressure (which will be negative). For example, if a gage connected to a tank indicates a vacuum pressure of 31.0 kPa, this can also be stated as 70.0 kPa absolute, or - 31.0 kPa gage.

Figure 3.4 provides a visual description of the three pressure scales. Notice that p_A = of 301 kPa abs is equivalent to 200 kPa gage. Gage, absolute, and vacuum pressure can be related using equations labeled as the "pressure equations."

$$p_{abs} = p_{atm} + p_{gage}$$

$$p_{abs} = p_{atm} - p_{vacuum}$$

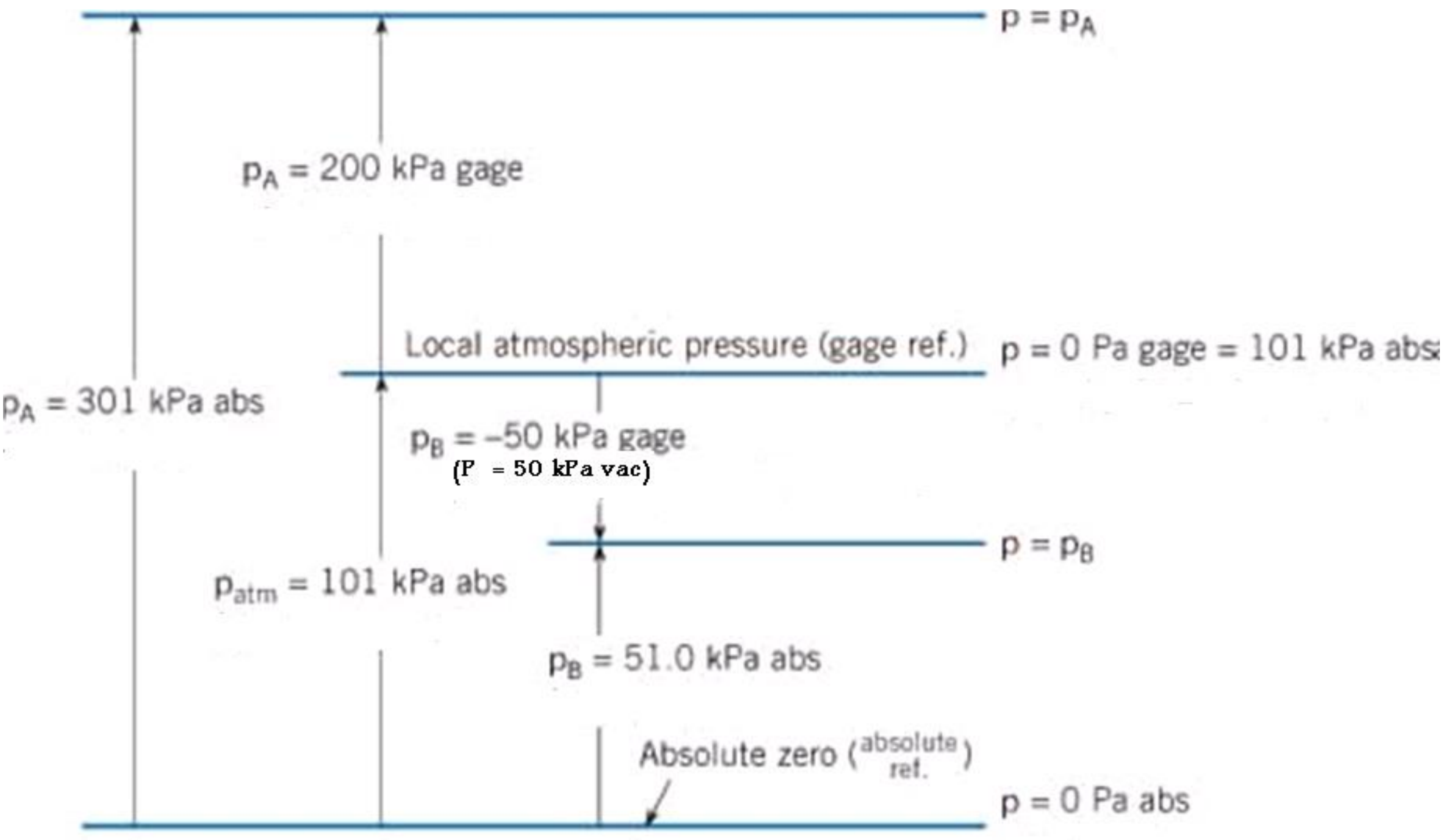
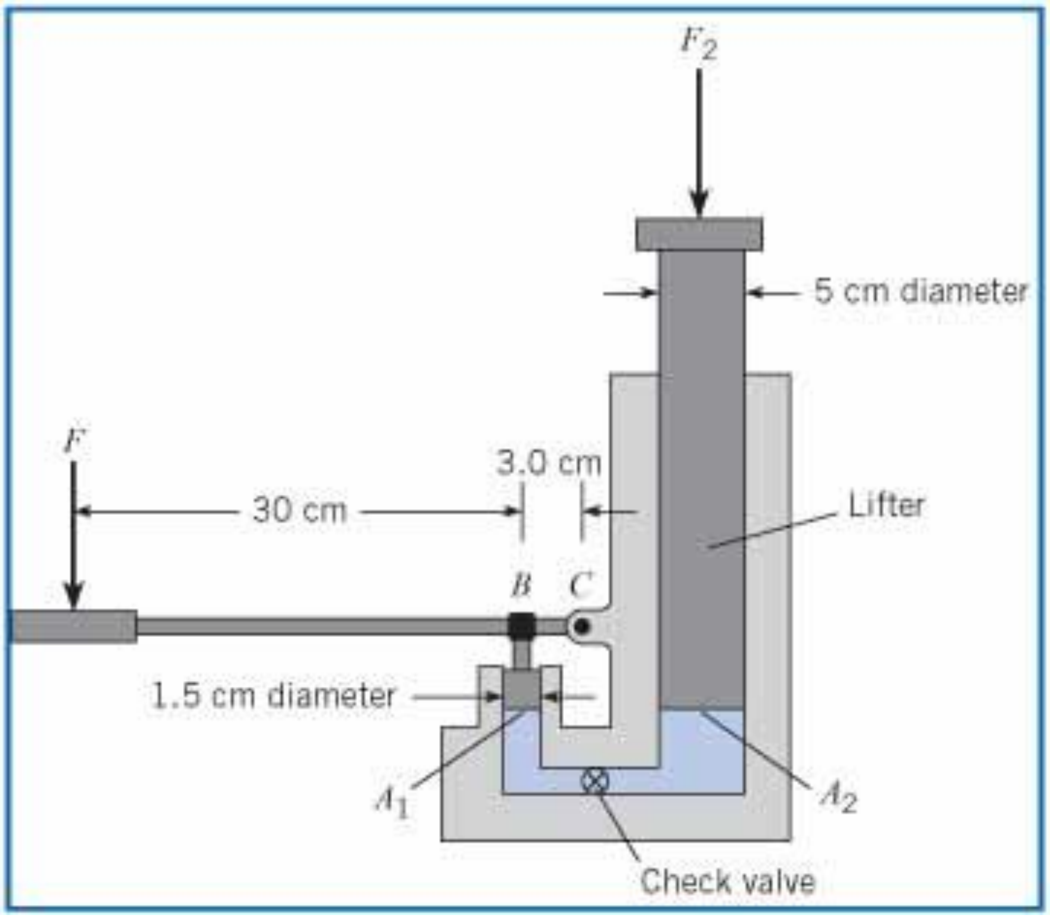


Figure 3.4 *Example of pressure relations.*

- **Hydraulic Machines**
- A *hydraulic machine* uses components such as pistons, pumps, and hoses to transmit forces and energy using fluids. Hydraulic machines are applied, for example, to braking systems, forklift trucks, power steering systems, and airplane control systems 3. Hydraulic machines provide an example of Pascal's law. This law states that pressure applied to an enclosed and continuous body of fluid is transmitted undiminished to every portion of that fluid and to the walls of the containing vessel.
- Hydraulic machines provide mechanical advantage. For example, a person using a hydraulic jack can lift a much larger load, as shown in Example 3.1.

EXAMPLE 3.1 LOAD LIFTED BY A HYDRAULIC JACK

A hydraulic jack has the dimensions shown. If one exerts a force *F* of 100 N on the handle of the jack, what load, *F*2, can the jack support? Neglect lifter weight.



Solution

1. Moment equilibrium

$$\sum M_C = 0$$

$$(0.33 \text{ m}) \times (100 \text{ N}) - (0.03 \text{ m}) F_1 = 0$$

$$F_1 = \frac{0.33 \text{ m} \times 100 \text{ N}}{0.03 \text{ m}} = 1100 \text{ N}$$

2. Force equilibrium (small piston)

$$\sum F_{\text{small piston}} = p_1 A_1 - F_1 = 0$$

$$p_1 A_1 = F_1 = 1100 \text{ N}$$

Thus,

$$p_1 = \frac{F_1}{A_1} = \frac{1100 \text{ N}}{\pi d^2 / 4} = 6.22 \times 10^6 \text{ N/m}^2$$

3. Force equilibrium (lifter)

Note that $p_1 = p_2$ because they are at the same elevation (this fact will be established in the next section). Apply force equilibrium:

$$\sum F_{\text{lifter}} = F_2 - p_1 A_2 = 0$$

$$F_2 = p_1 A_2 = \left(6.22 \times 10^6 \frac{\text{N}}{\text{m}^2}\right) \left(\frac{\pi}{4} \times (0.05 \text{ m})^2\right) = \boxed{12.2 \text{ kN}}$$

3.2 Pressure Variation with Elevation

The Hydrostatic Differential Equation

- The hydrostatic differential equation is derived by applying force equilibrium to a static body of fluid. To begin the derivation, visualize a cylindrical body of fluid, and then sketch a free-body diagram (FBD) as shown in Fig. 3.5. Notice that the cylindrical body is oriented so that its longitudinal axis is parallel to an arbitrary ℓ direction. The body is $\Delta\ell$ long, ΔA in cross-sectional area, and inclined at an angle α with the horizontal. Apply force equilibrium in the ℓ direction:

$$\sum F_\ell = 0$$

$$F_{\text{pressure}} - F_{\text{Weight}} = 0$$

$$p\,\Delta A - (p + \Delta p)\Delta A - \gamma \Delta A \Delta \ell \sin \alpha = 0$$

$$\frac{\Delta p}{\Delta \ell} = -\gamma \sin \alpha$$

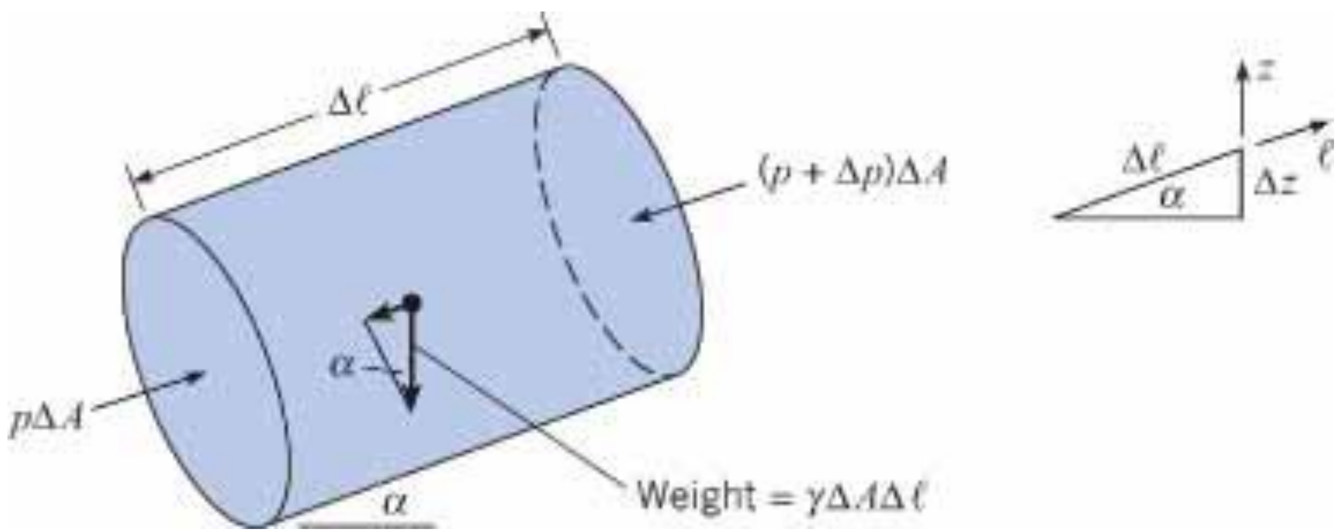


Figure 3.5 *Variation in pressure with elevation.*

It is evident from the figure that:

$$\sin\alpha = \frac{\Delta z}{\Delta \ell}$$

Combining the above equation and letting Δz approaches zero,

$$\frac{dp}{dz} = -\gamma \qquad \text{(hydrostatic differential equation)}$$

The equation is valid for hydrostatic conditions and it means that changes in pressure correspond to changes in elevation. If one travels upward in the fluid (positive *z* direction), the pressure decreases; if one goes downward (negative *z*), the pressure increases; if one moves along a horizontal plane, the pressure remains constant.

Uses of the Hydrostatic Equation

Case 1. Constant density

In this case γ is constant and then by integrating the hydrostatic equation gives

$$p + \gamma z = p_z = \text{constant}$$

where the term *z* is elevation, which is the height (vertical distance) above a fixed reference point called a datum, and p_z is *piezometric pressure*. Dividing the equation by γ gives

$$\frac{p_z}{\gamma} = \left(\frac{p}{\gamma} + z\right) = h = \text{constant}$$

where *h* is the *piezometric head.*

Since *h* is constant in the previous equation, then

$$\frac{p_1}{\gamma} + z_1 = \frac{p_2}{\gamma} + z_2$$

where the subscripts 1 and 2 identify any two points in a static fluid of constant density. Multiplying the equation by γ gives

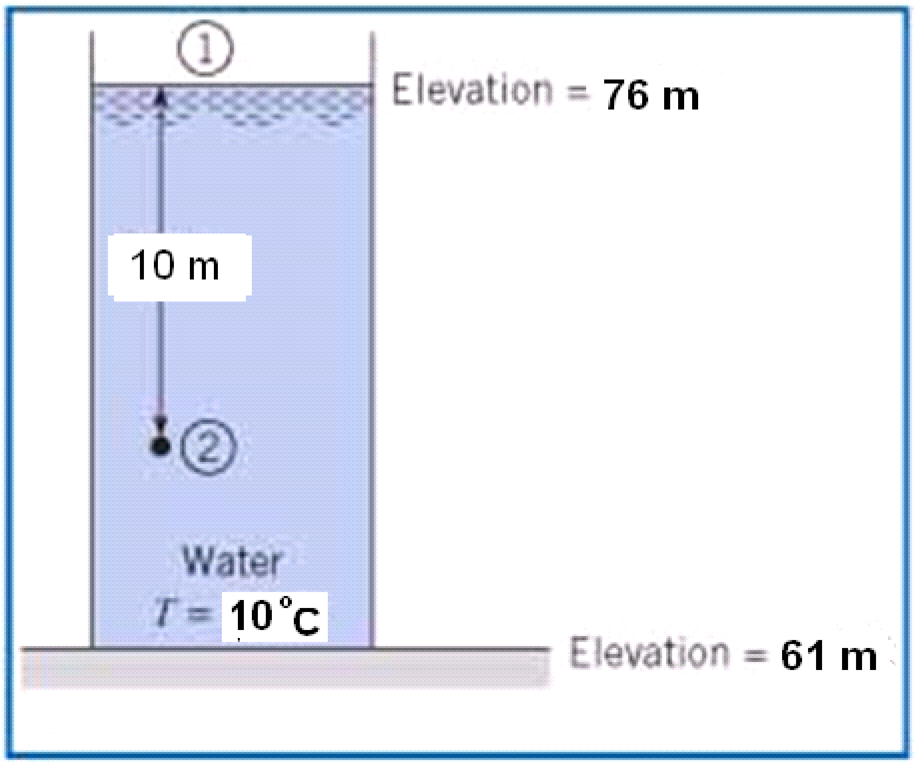
$$p_1 + \gamma z_1 = p_2 + \gamma z_2$$

The hydrostatic equation is given by either of the above equations are equivalent, because any one of the equations can be used to derive the other. The hydrostatic equation is valid for any constant density fluid in hydrostatic equilibrium.

To calculate piezometric head or piezometric pressure, an engineer identifies a specific location in a body of fluid and then uses the value of pressure and elevation at that location

EXAMPLE 3.2 WATER PRESSURE IN A TANK

What is the water pressure at a depth of 10 m in the tank shown?



Solution

The hydrostatic equation: $\frac{p_1}{\gamma} + z_1 = \frac{p_2}{\gamma} + z_2$

$p_1 = p_{atm} = 0$ kPa gage

$z_1 = 76$ m

$z_2 = 73.9$ m

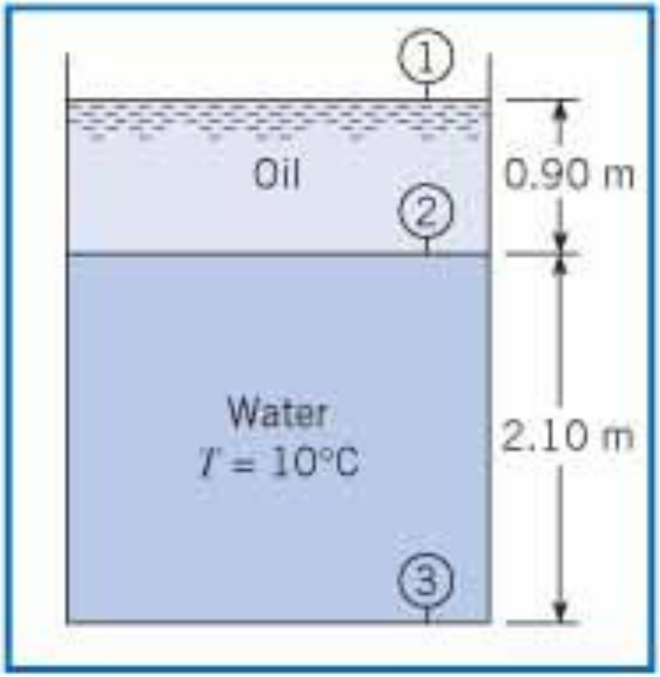
Substituting, $0 + 76 = p_2/\gamma + 73.9$; $\gamma = 9.81$ kN/m3

Hence, $p_2 = 98.1$ kPa gage

Remember! Gage pressure at the free surface of a liquid exposed to the atmosphere is zero.

- **EXAMPLE 3.3 PRESSURE IN TANK WITH TWO FLUIDS**

Oil with a specific gravity of 0.80 forms a layer 0.90 m deep in an open tank that is otherwise filled with water. The total depth of water and oil is 3 m. What is the gage pressure at the bottom of the tank?



Solution

1. Hydrostatic equation (oil)

$$\frac{p_1}{\gamma_{oil}} + z_1 = \frac{p_2}{\gamma_{oil}} + z_2$$

$$\frac{0\ \text{Pa}}{\gamma_{oil}} + 3\ \text{m} = \frac{p_2}{0.8 \times 9810\ \text{N/m}^3} + 2.1\ \text{m}$$

$$p_2 = 7.063\ \text{kPa}$$

2. Oil-water interface

$$p_{2_{oil}} = p_{2_{water}} = 7.063\ \text{kPa}$$

3. Hydrostatic equation (water)

$$\frac{p_2}{\gamma_{water}} + z_2 = \frac{p_3}{\gamma_{water}} + z_3$$

$$\frac{7.063 \times 10^3\ \text{Pa}}{9810\ \text{N/m}^3} + 2.1\ \text{m} = \frac{p_3}{9810\ \text{N/m}^3} + 0\ \text{m}$$

$$\boxed{p_3 = 27.7\ \text{kPa gage}}$$

Uses of the Hydrostatic Equation

Case 2. Variable density

Example, pressure variation in atmospheric air. In this case, one uses ideal gas for density, $\rho = \frac{p}{RT}$ hence $\gamma = \frac{pg}{RT}$

And

$$\frac{dp}{dz} = -\frac{pg}{RT}$$

To solve this equation one must have the variation of temperature as a function of elevation (z).

3.3 Pressure Measurements

This section describes **five scientific** instruments for measuring pressure: the barometer, Bourdon-tube gage, piezometer, manometer, and transducer.

1) Barometer

An instrument that is used to measure atmospheric pressure is called a *barometer*. The most common types are the mercury barometer and the aneroid barometer. A mercury barometer is made by inverting a mercury-filled tube in a container of mercury as shown in Fig. 3.8. The pressure at the top of the mercury barometer will be the vapor pressure of mercury, which is very small: $p_v = 2.4 \times 10\text{-}6$ atm at 20°C. Thus, atmospheric pressure will push the mercury up the tube to a height *h*. The mercury barometer is analyzed by applying the hydrostatic equation:

$$p_{atm} = \gamma_{Hg} h + p_v \approx \gamma_{Hg} h$$

Thus, by measuring *h*, local atmospheric pressure can be determined using the above equation)

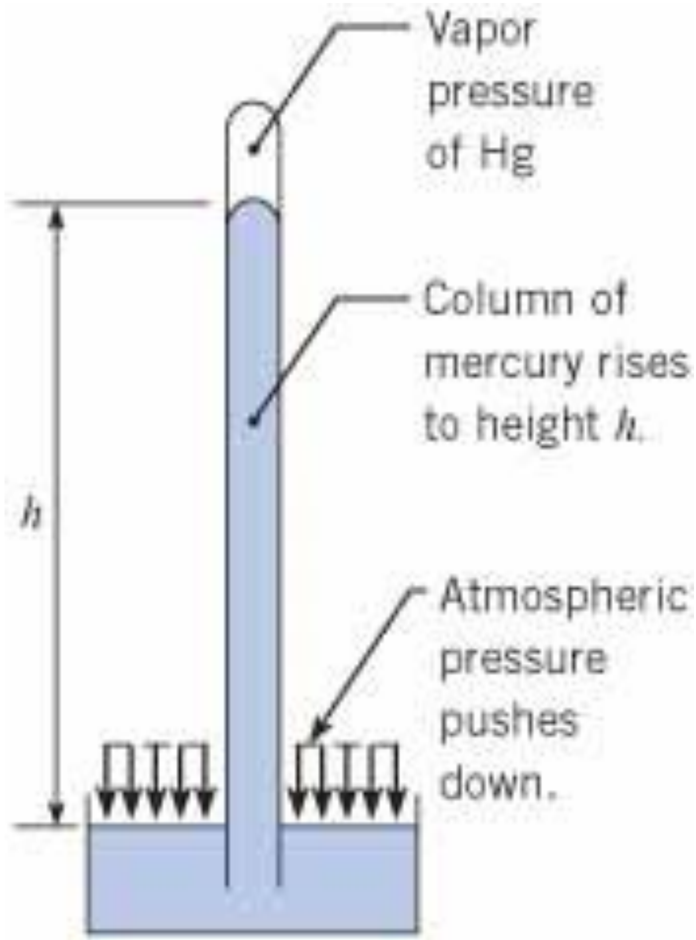


Figure 3.8 *A mercury barometer.*

2) Bourdon-Tube Gage

A *Bourdon-tube* gage, Fig. 3.9, measures pressure by sensing the deflection of a coiled tube. The tube has an elliptical cross section and is bent into a circular arc, as shown in Fig. 3.9*b*. When atmospheric pressure (zero gage pressure) prevails, the tube is undeflected, and for this condition the gage pointer is calibrated to read zero pressure. When pressure is applied to the gage, the curved tube tends to straighten (much like blowing into a party favor to straighten it out), thereby actuating the pointer to read a positive gage pressure. The Bourdon-tube gage is common because it is low cost, reliable, easy to install, and available in many different pressure ranges. There are disadvantages: dynamic pressures are difficult to read accurately; accuracy of the gage can be lower than other instruments; and the gage can be damaged by excessive pressure pulsations.

Bourdon-Tube Gage

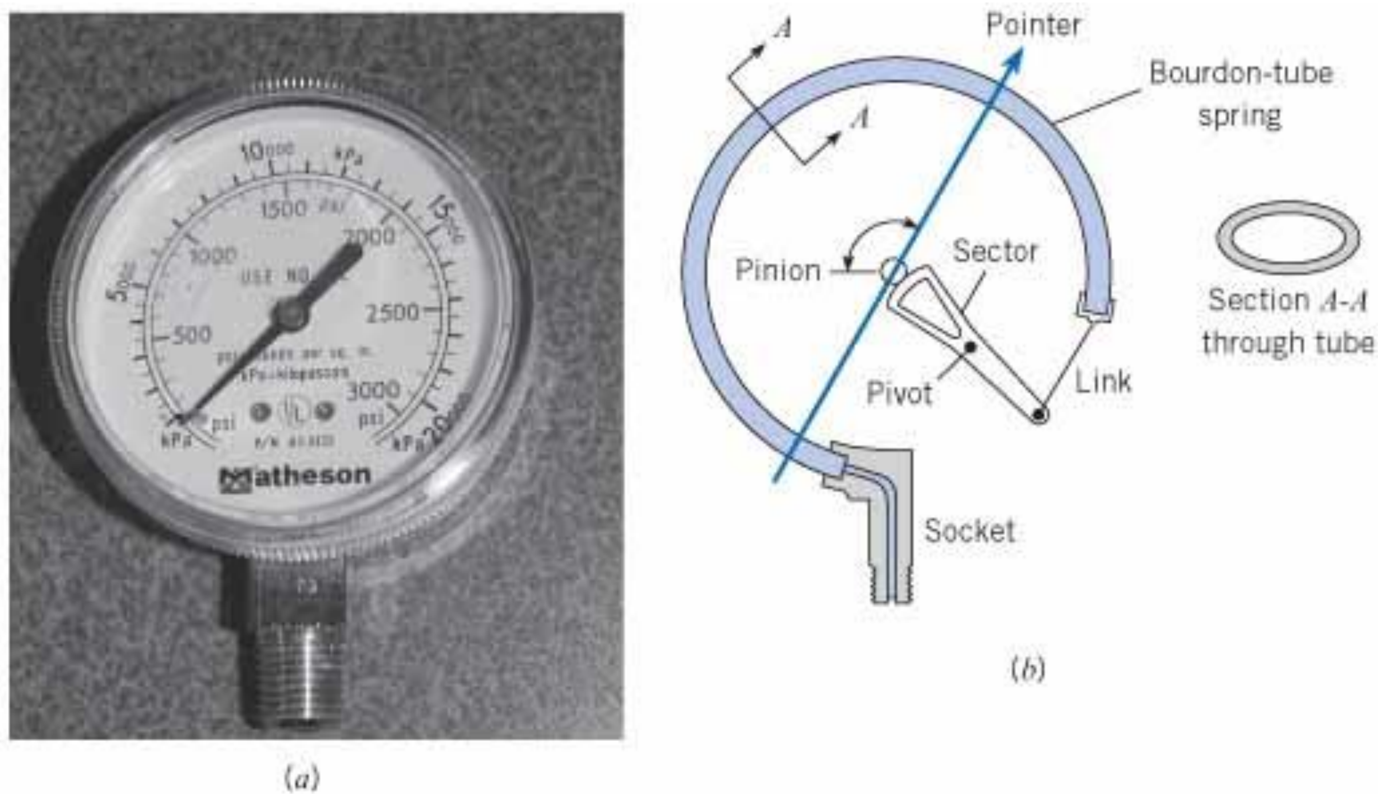


Figure 3.9 *Bourdon-tube gage.*
(a) View of typical gage.
(b) Internal mechanism (schematic).

3) Piezometer

A *piezometer* is a vertical tube, usually transparent, in which a liquid rises in response to a positive gage pressure. For example, Fig. 3.10 shows a piezometer attached to a pipe. Pressure in the pipe pushes the water column to a height *h,* and the gage pressure at the center of the pipe is $p = \gamma h$, which follows directly from the hydrostatic equation (3.7c). The piezometer has several advantages: simplicity, direct measurement (no need for calibration), and accuracy. However, a piezometer cannot easily be used for measuring pressure in a gas, and a piezometer is limited to low pressures because the column height becomes too large at high pressures.

Think? This chapter about static fluid, then how does moving fluid affect pressure measurement?

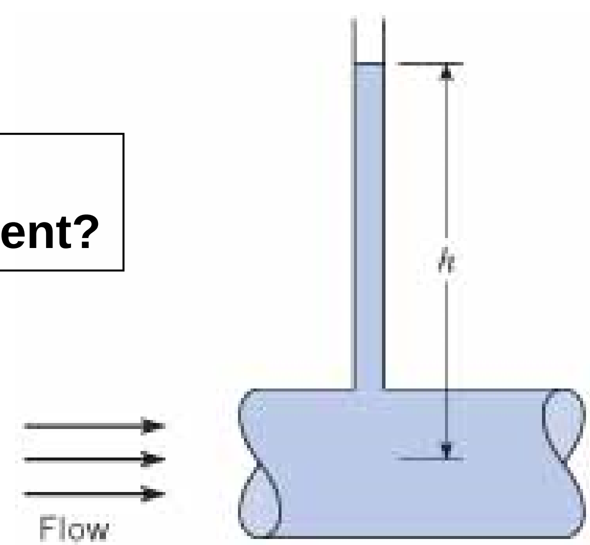


Figure 3.10 *Piexometer attached to a pipe.*

4) Manometer

A *manometer*, often shaped like the letter "U," is a device for measuring pressure by raising or lowering a column of liquid. For example, Fig. 3.11 shows a U-tube manometer that is being used to measure pressure in a flowing fluid. In the case shown, positive gage pressure in the pipe pushes the manometer liquid up a height Δh. To use a manometer, engineers relate the height of the liquid in the manometer to pressure.

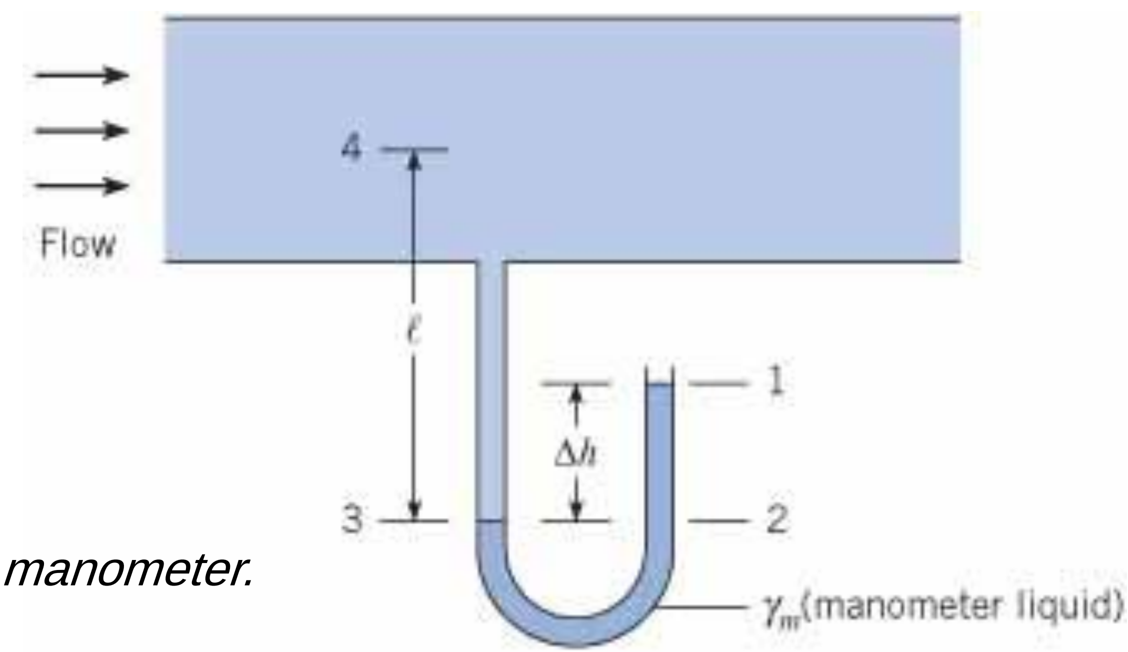


Figure 3.11 *U-tube manometer.*

EXAMPLE 3.6 PRESSURE MEASUREMENT (U-TUBE MANOMETER)

Water at 10°C is the fluid in the pipe of Fig. 3.11, and mercury is the manometer fluid. If the deflection Δh is 60 cm and ℓ is 180 cm, what is the gage pressure at the center of the pipe?

1. Water (10°C), Table A.5, γ = 9810 N/m³.
2. Mercury, Table A.4: γ = 133, 000 N/m³.

Solution

1. Calculate the pressure at point 2 using the hydrostatic equation

$$
\begin{aligned}
p_2 &= p_1 + \text{pressure increase between 1 and 2} = 0 + \gamma_m \Delta h_{12} \\
&= \gamma_m (0.6 \text{ m}) = (133{,}000 \text{ N/m}^3)(0.6 \text{ m}) \\
&= 79.8 \text{ kPa}
\end{aligned}
$$

2. Find the pressure at point 3.

$$p_3|_{\text{water}} = p_2|_{\text{water}} = 79.8 \text{ kPa}$$

When a fluid-fluid interface is flat, pressure is constant across the interface. Thus, at the oil-water interface

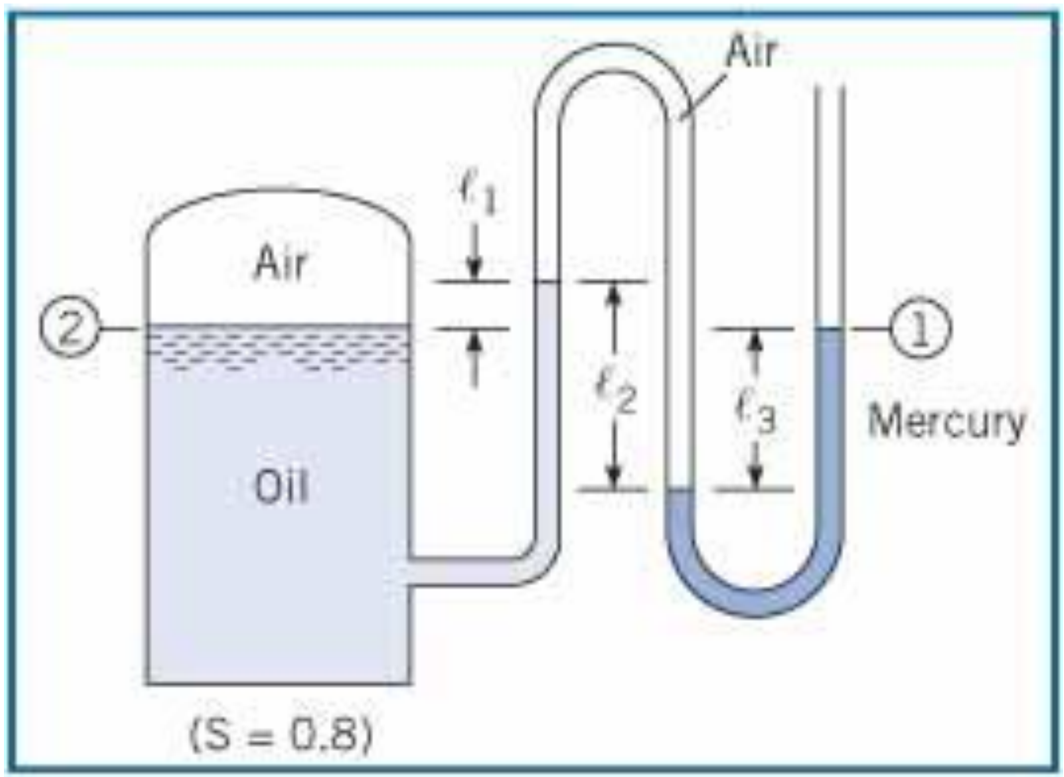
$$p_{3_{\text{mercury}}} = p_{3_{\text{water}}} = 79.8 \text{ kPa}$$

3. Find the pressure at point 4 using the hydrostatic equation.

$$p_4 = p_3 - \text{pressure decrease between 3 and 4} = p_3 - \gamma_w \ell$$
$$= 79{,}800 \text{ Pa} - (9810 \text{ N/m}^3)(1.8 \text{ m})$$
$$= 62.1 \text{ kPa gage}$$

EXAMPLE 3.7 MA%OMETER A%ALYSIS

Sketch: What is the pressure of the air in the tank if ℓ1 = 40 cm, ℓ2 = 100 cm, and ℓ3 = 80 cm?



Solution

Manometer equation

$$p_1 + \sum_{\text{down}} \gamma_i h_i - \sum_{\text{up}} \gamma_i h_i = p_2$$

$$p_1 + \gamma_{\text{mercury}} \ell_3 - \gamma_{\text{air}} \ell_2 + \gamma_{\text{oil}} \ell_1 = p_2$$

$$0 + (133{,}000 \text{ N/m}^3)(0.8 \text{ m}) - 0 + (7850 \text{ N/m}^3)(0.4 \text{ m}) = p_2$$

$$\boxed{p_2 = p_{\text{air}} = 110 \text{ kPa gage}}$$

Because the manometer configuration shown in Fig. 3.12 is common, it is useful to derive an equation specific to this application. To begin, apply the manometer equation (3.18) between points 1 and 2:

$$p_1 + \sum_{\text{down}} \gamma_i h_i - \sum_{\text{up}} \gamma_i h_i = p_2$$

$$p_1 + \gamma_A(\Delta y + \Delta h) - \gamma_B \Delta h - \gamma_A(\Delta y + z_2 - z_1) = p_2$$

Simplifying gives

$$(p_1 + \gamma_A z_1) - (p_2 + \gamma_A z_2) = \Delta h(\gamma_B - \gamma_A)$$

Dividing through by γ_A gives

$$\left(\frac{p_1}{\gamma_A} + z_1\right) - \left(\frac{p_2}{\gamma_A} + z_2\right) = \Delta h\left(\frac{\gamma_B}{\gamma_A} - 1\right)$$

Recognize that the terms on the left side of the equation are piezometric head and rewrite to give the final result:

$$h_1 - h_2 = \Delta h\left(\frac{\gamma_B}{\gamma_A} - 1\right)$$

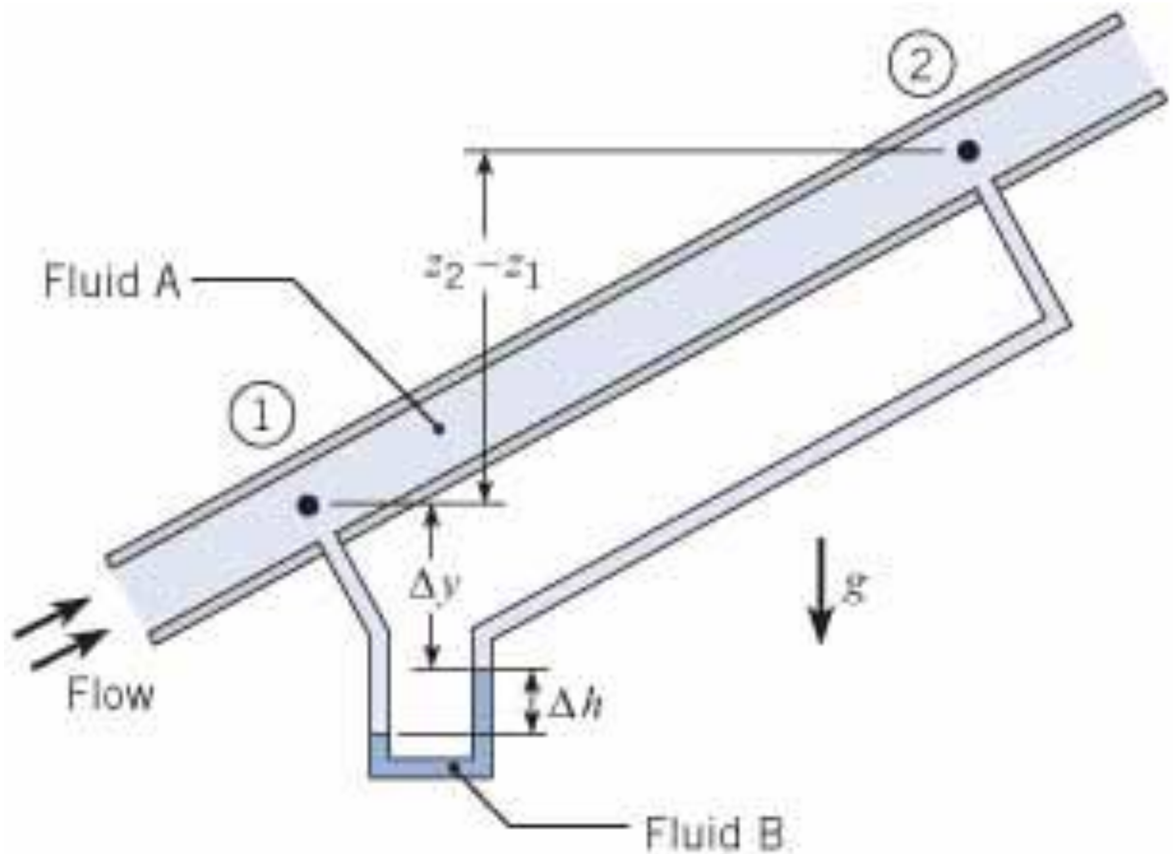


Figure 3.12 *Apparatus for determining change in piezometric head corresponding to flow in a pipe.*

EXAMPLE 3.8 CHANGE IN PIEZOMETRIC HEAD FOR PIPE FLOW

A differential mercury manometer is connected to two pressure taps in an inclined pipe as shown in Fig. 3.12. Water at 10°C is flowing through the pipe. The deflection of mercury in the manometer is 2.5 cm. Find the change in piezometric pressure and piezometric head between points 1 and 2.

1. Water (10°C), Table A.5, $\gamma = 9.81\ kN/m^3$.
2. Mercury, Table A.4: $\gamma = 133\ kN/m^3$.

Solution

Difference in piezeometric head

$$h_1 - h_2 = \Delta h \left(\frac{\gamma_{Hg}}{\gamma_{water}} - 1\right) = \left(\frac{2.5}{100}\ \text{m}\right)\left(\frac{133\ \text{kN/m}^3}{9.81\ \text{kN/m}^3} - 1\right)$$

$$= \boxed{0.314\ \text{m}}$$

Piezometric pressure

$$p_z = h\gamma_{water}$$

$$= (0.314\ \text{m})(9.81\ \text{kN/m}^3) = \boxed{\mathbf{3.08\ kPa}}$$

5) Pressure Transducers

A *pressure transducer* is a device that converts pressure to an electrical signal. Modern factories and systems that involve flow processes are controlled automatically, and much of their operation involves sensing of pressure at critical points of the system. Therefore, pressure-sensing devices, such as pressure transducers, are designed to produce electronic signals that can be transmitted to oscillographs or digital devices for recordkeeping or to control other devices for process operation. Basically, most transducers are tapped into the system with one side of a small diaphragm exposed to the active pressure of the system. When the pressure changes, the diaphragm flexes, and a sensing element connected to the other side of the diaphragm produces a signal that is usually linear with the change in pressure in the system. There are many types of sensing elements; one common type is the resistance-wire strain gage attached to a flexible diaphragm as shown in Fig. 3.13. As the diaphragm flexes, the wires of the strain gage change length, thereby changing the resistance of the wire. This change in resistance is converted into a voltage change that can then be used in various ways.

Another type of pressure transducer used for measuring rapidly changing high pressures, such as the pressure in the cylinder head of an internal combustion engine, is the piezoelectric transducer 2. These transducers operate with a quartz crystal that generates a charge when subjected to a pressure. Sensitive electronic circuitry is required to convert the charge to a measurable voltage signal.

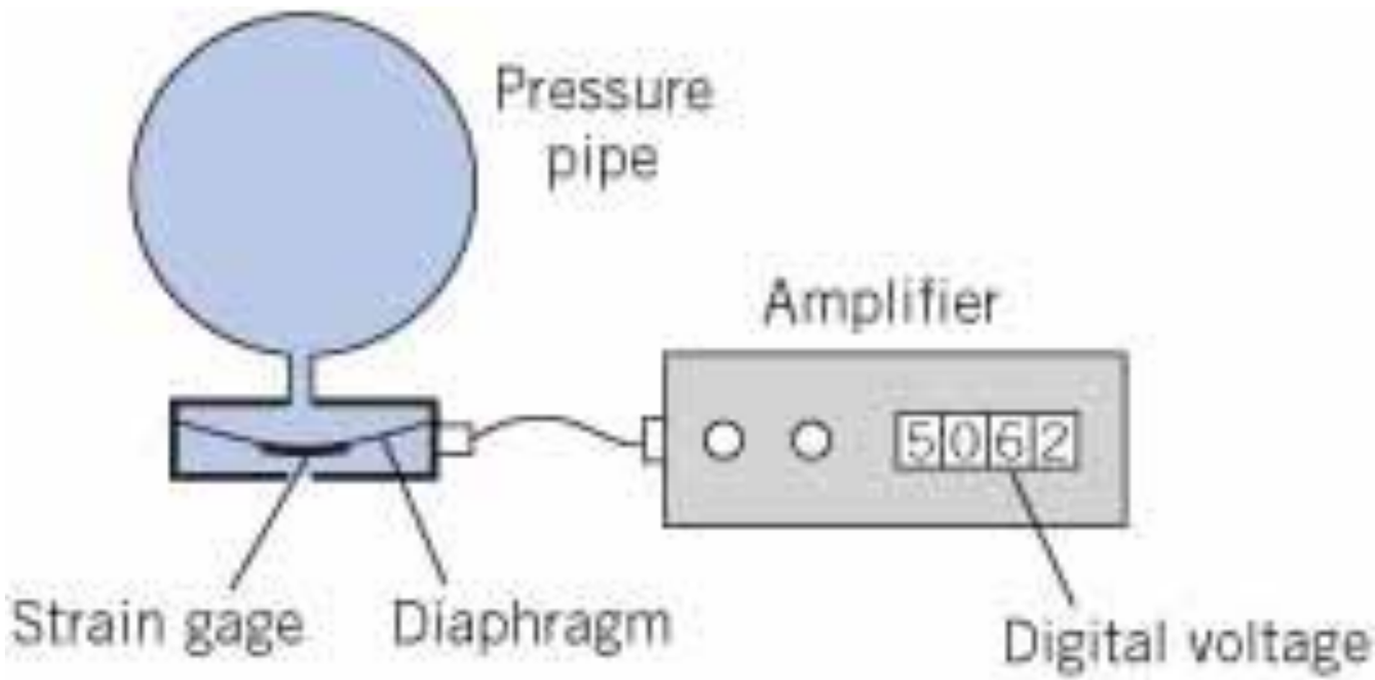


Figure 3.13 *Schematic diagram of strain-gage pressure transducer.*

3.4 Forces on Plane Surfaces (Panels)

This section explains how to represent hydrostatic pressure distributions on one face of a panel with a resultant force that passes through a point called the center of pressure.

Uniform Pressure Distribution

A plane surface or *panel* is a flat surface of arbitrary shape. A description of the pressure at all points along a surface is called a *pressure distribution*. When pressure is the same at every point, as shown in Fig. 3.14*a*, the pressure distribution is called a uniform pressure distribution. The pressure distribution in Fig. 3.14*a* can be represented by a resultant force as shown in Fig. 3.14*b*. For a uniform pressure distribution, the magnitude of the resultant force is *F* where

$$F = \int_A p\,dA = \bar{p}A$$

and $\bar{p}$ is the average pressure. The resultant force *F* passes through a point called the *center of pressure (CP)*. Notice that the CP is represented using a circle with a “plus” inside. For a uniform pressure distribution, the CP is located at the centroid of area of the panel.

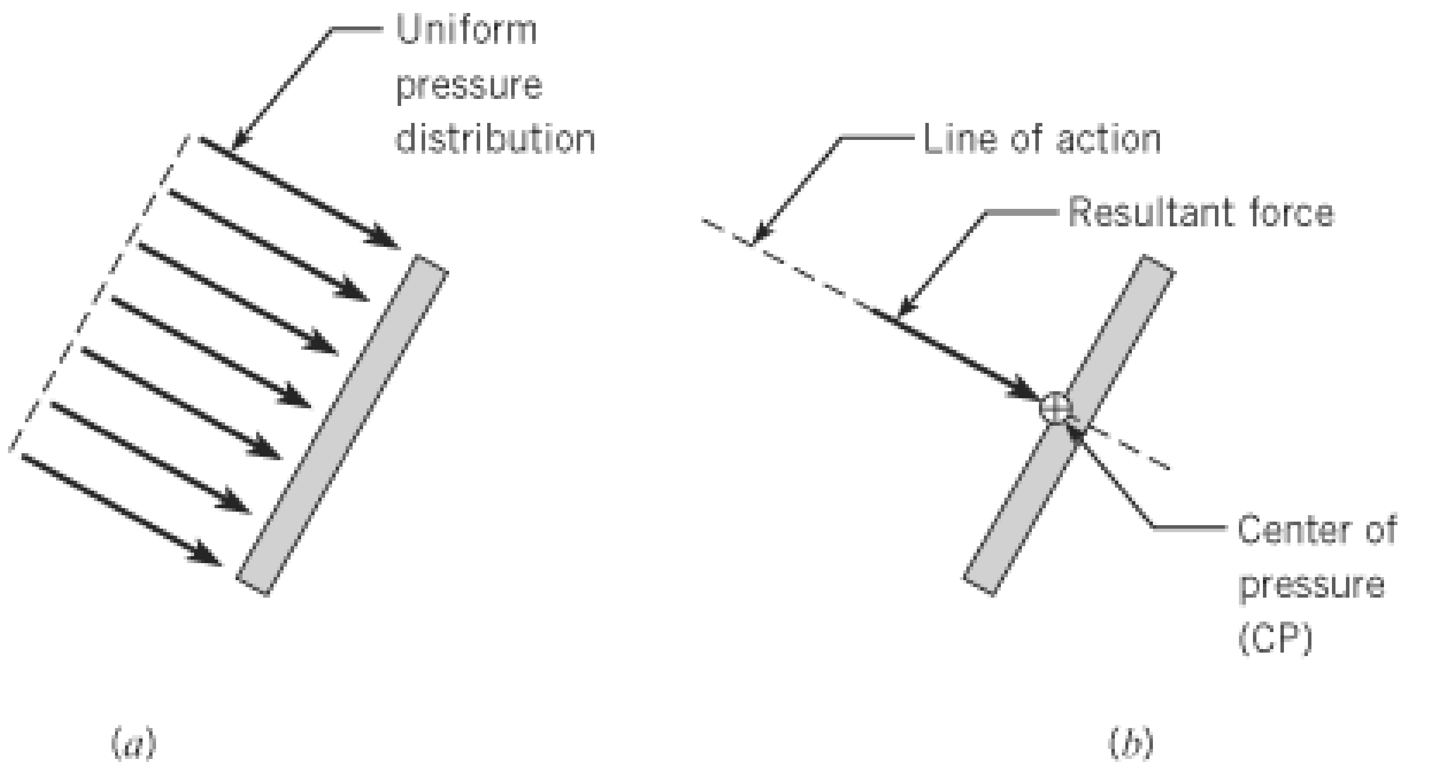


Figure 3.14
(a) Uniform pressure distribution, and (b) equivalent force.

Hydrostatic Pressure Distribution

When a pressure distribution is produced by a fluid in hydrostatic equilibrium, then the pressure distribution is called a *hydrostatic pressure distribution*. Notice that a hydrostatic pressure distribution is linear and that the arrows representing pressure act normal to the surface. In Fig. 3.15*b*, the pressure distribution is represented by a resultant force that acts at the CP. Notice that the CP is located below the centroid of area.

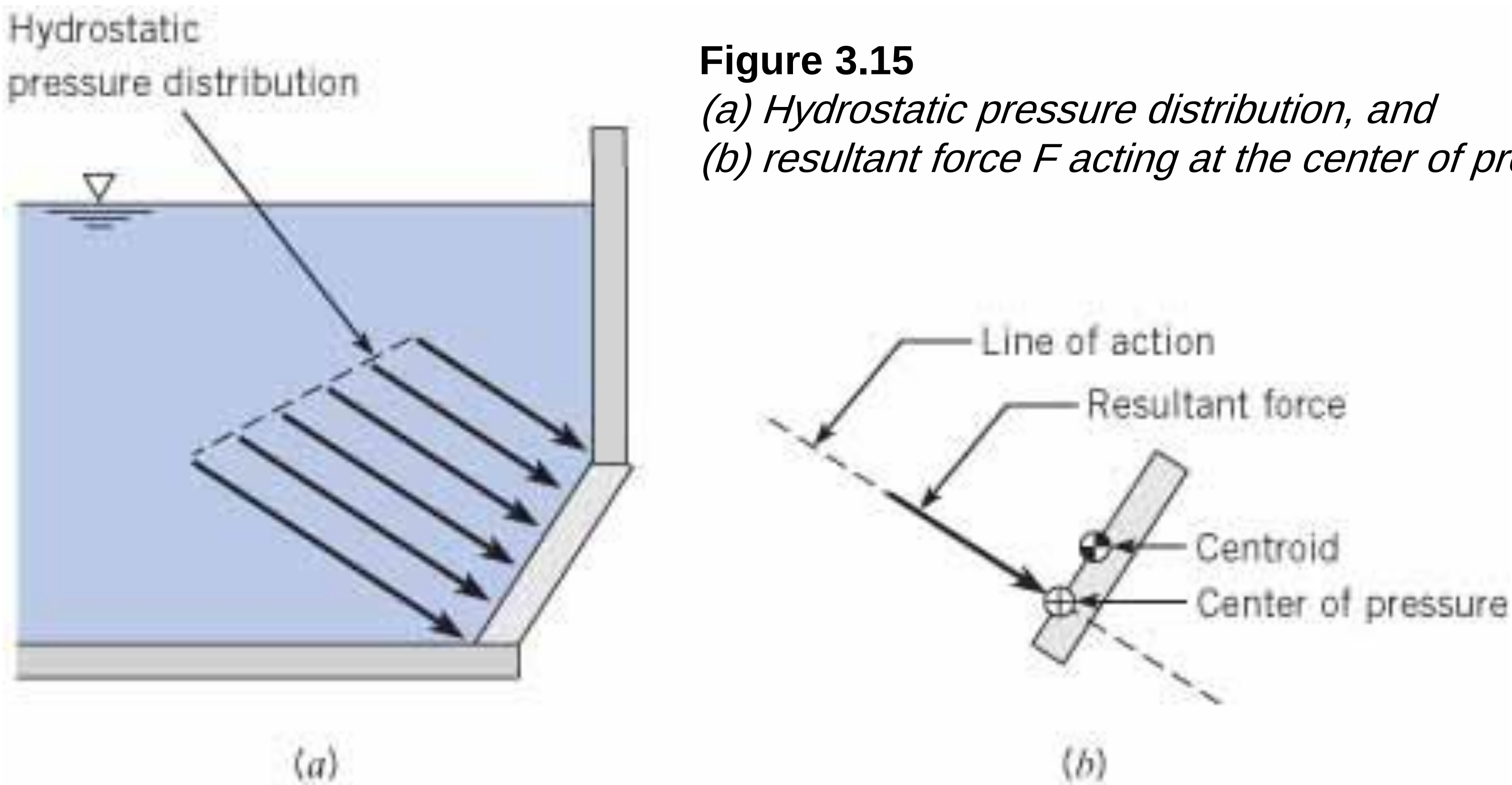


Figure 3.15
(a) Hydrostatic pressure distribution, and
(b) resultant force F acting at the center of pressure.

Magnitude of Resultant Hydrostatic Force

To derive an equation for the resultant force on a panel under hydrostatic loading, sum-up forces using an integral. The situation is shown in Fig. 3.16. Line *AB* is the edge view of a panel submerged in a liquid. The plane of this panel intersects the horizontal liquid surface at axis 0-0 with an angle α. The distance from the axis 0-0 to the horizontal axis through the centroid of the area is given by $\bar{y}$. The distance from 0-0 to the differential area *dA* is *y*. The pressure on the differential area is:

$$p = \gamma y \sin \alpha$$

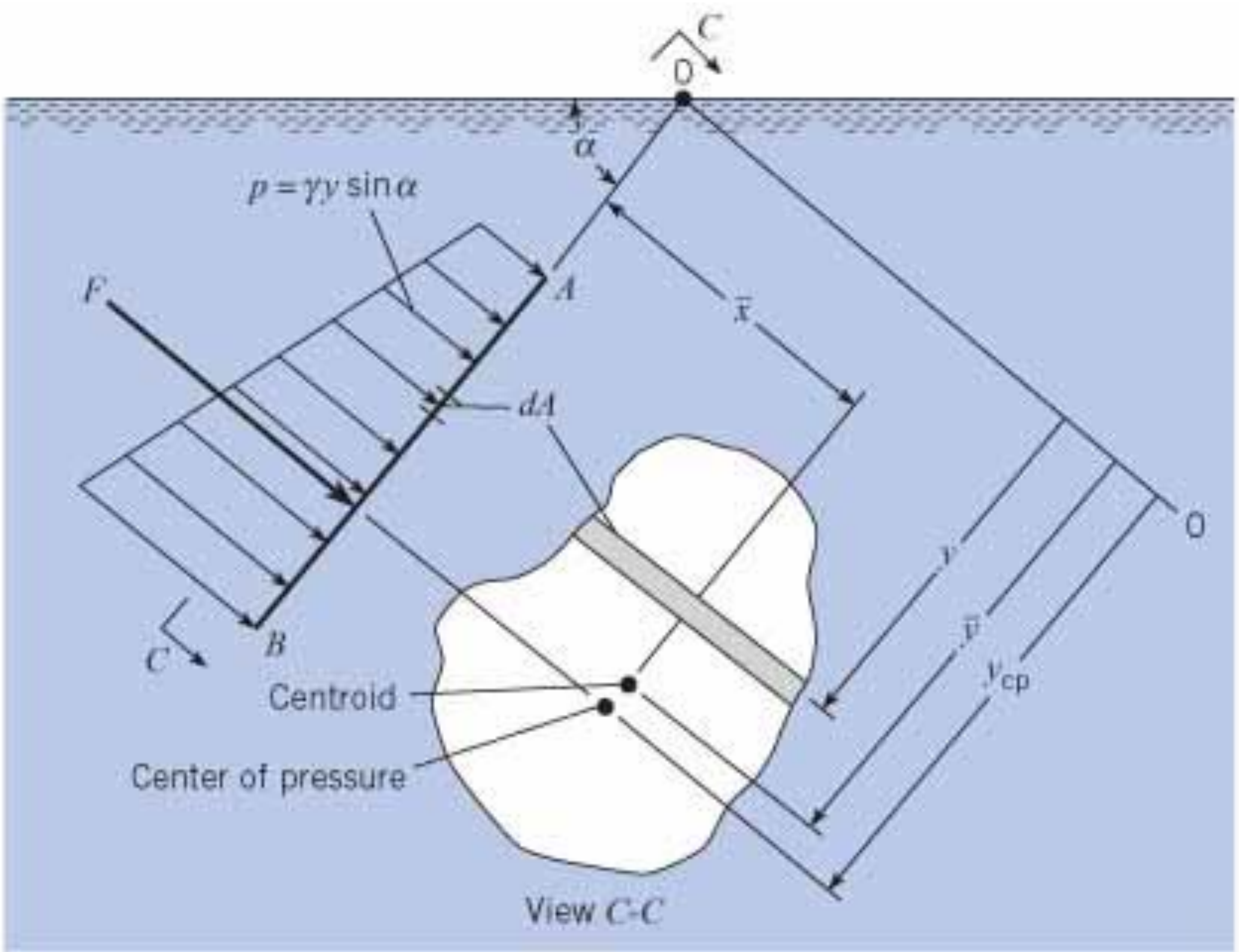


Figure 3.16 *Distribution of hydrostatic pressure on a plane surface.*

The differential force is

$$dF = p\ dA = \gamma y\ \sin\alpha\ dA$$

The total force on the area is

$$F = \int_A p dA = \int_A \gamma y\ \sin\alpha\ dA$$

In the above equation, γ and sin α are constants. Thus

$$F = \gamma\ \sin\alpha\ \int_A y\ dA$$

Now the integral in right hand side is the first moment of the area. Consequently, this is replaced by its equivalent, $\bar{y}$***A***. Therefore

$$F = \gamma\ \bar{y}\ A\ \sin\alpha$$

or

$$F = (\gamma\ \bar{y}\ \sin\alpha) A$$

The product of the variables within the parentheses is the pressure at the centroid of the area. Thus

$$F = \bar{p}\ A$$

Line of Action of the Resultant Force

A general equation for the vertical location of the CP is derived next. The initial situation is shown in Fig. 3.16. The torque due to the resultant force *F* will balance the torque due to the pressure distribution.

$$y_{cp}F = \int y \, dF$$

The differential force *dF* is given by *dF* = *p dA*; therefore,

$$y_{cp}F = \int_A yp \, dA$$

Also, $p = \gamma y \sin \alpha$ so

$$y_{cp}F = \int_A \gamma y^2 \sin \alpha \, dA$$

Since γ and $\sin\alpha$ are constants,

$$y_{cp}F = \gamma \sin \alpha \int_A y^2 \, dA$$

The integral on the right-hand side of Eq. (3.25) is the second moment of the area (often called the area moment of inertia). This shall be identified as *I*0. However, for engineering applications it is convenient to express the second moment with respect to the horizontal centroidal axis of the area. Hence by the parallel-axis theorem,

$$I_0 = \bar{I} + \bar{y}^2 A$$

which leads to

$$y_{cp} F = \gamma \sin\alpha (\bar{I} + \bar{y}^2 A)$$

However, since $F = \gamma$ sinαA. Therefore,

$$y_{cp} (\gamma \bar{y} \sin\alpha A) = \gamma \sin\alpha (\bar{I} + \bar{y}^2 A)$$

$$y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y} A}$$

Notes on previous equations

The area moment of inertia $\bar{I}$ s taken about a horizontal axis that passes through the centroid of area. Formulas for $\bar{I}$ are presented in Fig. A.1. The slant distance $\bar{y}$ measures the length from the surface of the liquid to the centroid of the panel along an axis that is aligned with the "slant of the panel" as shown in Fig. 3.16.

It is seen that the Center of Pressure (CP) will be situated below the centroid. The distance between the CP and the centroid depends on the depth of submersion, which is characterized by $\bar{y}$ and on the panel geometry, which is characterized by I/A.

Due to assumptions in the derivations, there are several limitations on the previous equations. First, they only apply to a single fluid of constant density. Second, the pressure at the liquid surface needs to be $p = 0$ gage to correctly locate the CP. Third, the last equation gives only the vertical location of the CP, not the lateral location.

EXAMPLE 3.9 HYDROSTATIC FORCE DUE TO CONCRETE

Determine the force acting on one side of a concrete form 2.44 m high and 1.22 m wide that is used for pouring a basement wall. The specific weight of concrete is 23.6 kN/m^3.

Solution

The force,

$$F = \bar{p}A$$

$$\bar{p} = \text{pressure at depth of the centroid}$$

$$\bar{p} = (\gamma_{\text{concrete}})(z_{\text{centroid}}) = (23.6 \text{ kN / m}^3)(2.44 / 2 \text{ m})$$
$$= 28.79 \text{ kPa}$$

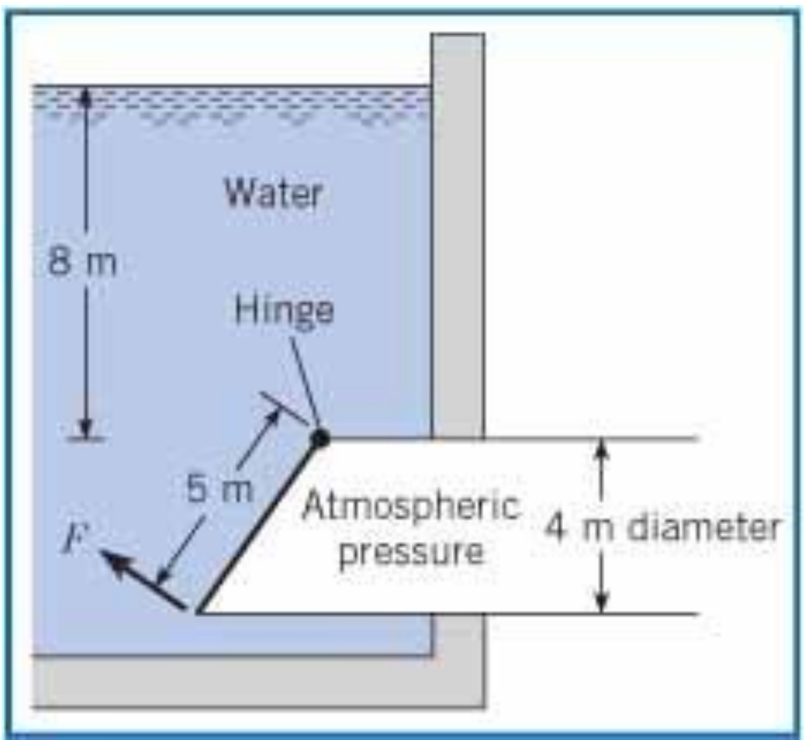
Hence, the resultant force

$$F = \bar{p}A = (28.79 \text{ kPa})(2.977 \text{ m}^2) = \boxed{85.7 \text{ kN}}$$

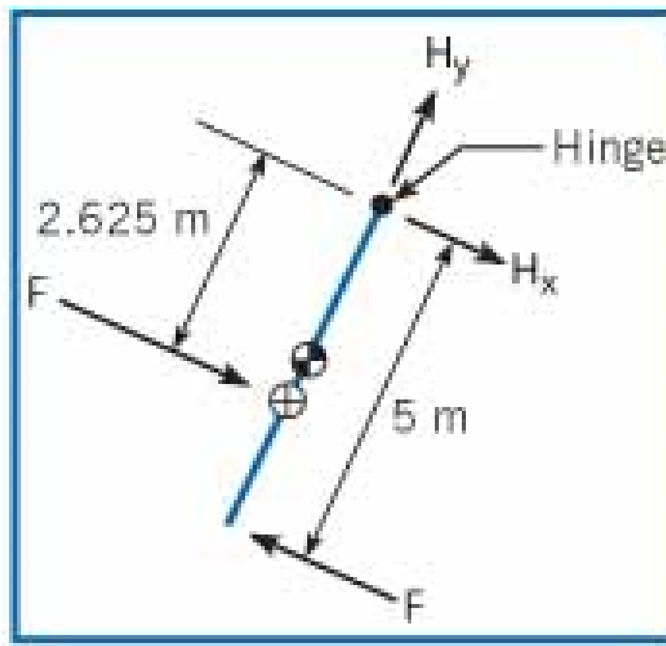
EXAMPLE 3.10 FORCE TO OPEN AN ELLIPTICAL GATE

An elliptical gate covers the end of a pipe 4 m in diameter. If the gate is hinged at the top, what normal force *F* is required to open the gate when water is 8 m deep above the top of the pipe and the pipe is open to the atmosphere on the other side? Neglect the weight of the gate.

Properties: Water (10°C), Table A.5: γ = 9810 N/m³.



Free Body Diagram



Solution

1) Hydrostatic (resultant) force: $\bar{p}$ = pressure at depth of the centroid

$$\bar{p} = (\gamma_{\text{water}})(z_{\text{centroid}}) = (9810\ \text{N/m}^3)(10\ \text{m}) = 98.1\ \text{kPa}$$

A = area of elliptical panel (using Fig. A.1 to find formula)

$$A = \pi ab$$

$$= \pi(2.5\ \text{m})(2\ \text{m}) = 15.71\ \text{m}^2$$

Calculate resultant force

$$F_p = \bar{p}A = (98.1\ \text{kPa})(15.71\ \text{m}^2) = \boxed{1.54\ \text{MN}}$$

2) Center of pressure

$\bar{y}$ = 12.5 m, where $\bar{y}$ is the slant distance from the water surface to the centroid. Area moment of inertia $\bar{I}$ of an elliptical panel using a formula from Fig. A.1

$$\bar{I} = \frac{\pi a^3 b}{4} = \frac{\pi(2.5\ \text{m})^3(2\ \text{m})}{4} = 24.54\ \text{m}^4$$

Finding center of pressure

$$y_{cp} - \bar{y} = \frac{\bar{I}}{\bar{y}A} = \frac{25.54\ \text{m}^4}{(12.5\ \text{m})(15.71\ \text{m}^2)} = 0.125\ \text{m}$$

3.5 Forces on Curved Surfaces

This section describes how to calculate forces on surfaces that have curvature. Consider the curved surface *AB* in Fig. 3.17*a*. The goal is to represent the pressure distribution with a resultant force that passes through the center of pressure. One approach is to integrate the pressure force along the curved surface and find the equivalent force. However, it is easier to sum forces for the free body shown in the upper part of Fig. 3.17*b*. The lower sketch in Fig. 3.17*b* shows how the force acting on the curved surface relates to the force *F* acting on the free body. Using the FBD and summing forces in the horizontal direction shows that

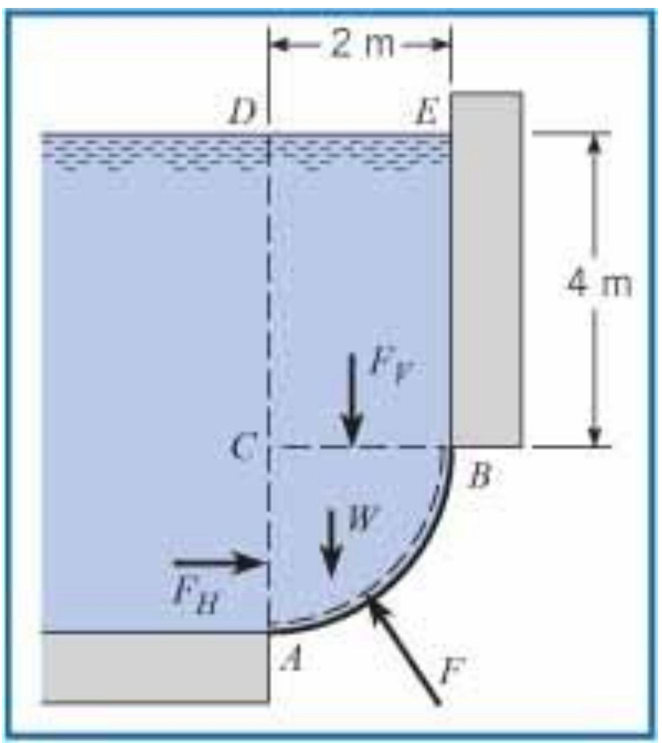
$$F_x = F_{AC}$$

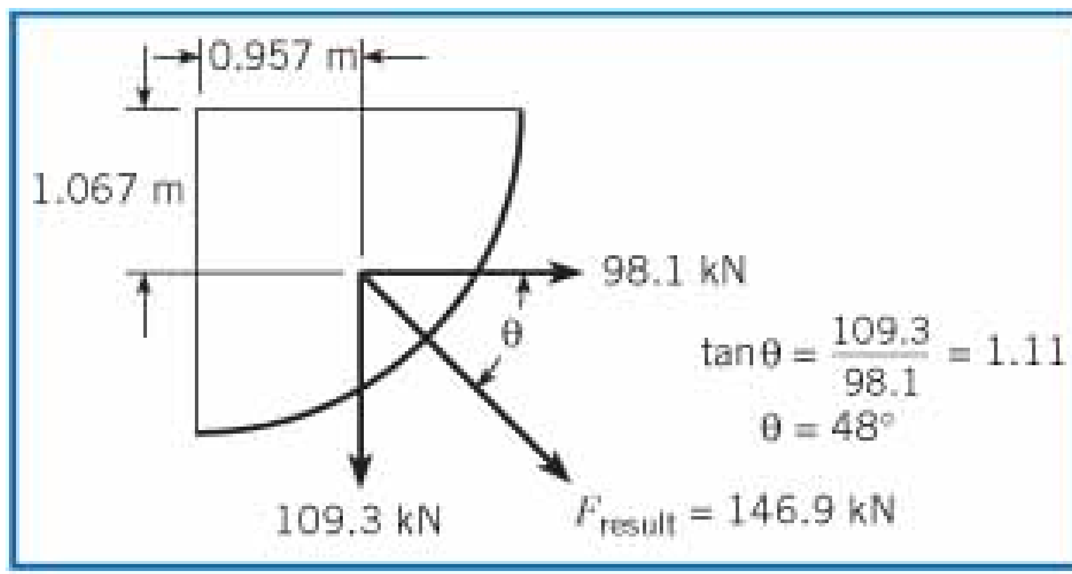
The line of action for the force *FAC* is through the center of pressure for side *AC*, as discussed in the previous

- section, and designated as *y*cp.

EXAMPLE 3.11 HYDROSTATIC FORCE ON A CURVED SURFACE

Surface *AB* is a circular arc with a radius of 2 m and a width of 1 m into the paper. The distance *EB* is 4 m. The fluid above surface *AB* is water, and atmospheric pressure prevails on the free surface of the water and on the bottom side of surface *AB*. Find the magnitude and line of action of the hydrostatic force acting on surface *AB*.





Solution

1. Equilibrium in the horizontal direction

$$F_x = F_H = \bar{p}A = (5\ \text{m})(9810\ \text{N/m}^3)(2 \times 1\ \text{m}^2)$$

$$= 98.1\ \text{kN}$$

2. Equilibrium in the horizontal direction

-Vertical force on side *CB*

$$F_v = \bar{p}_0 A = 9.81\ \text{kN/m}^3 \times 4\ \text{m} \times 2\ \text{m} \times 1\ \text{m} = 78.5\ \text{kN}$$

Weight of the water in volume *ABC:*

$$W = \gamma V_{ABC} = (\gamma)(\frac{1}{4}\pi r^2)(w)$$

$$= (9.81\ \text{kN/m}^3) \times (0.25 \times \pi \times 4\ \text{m}^2)(1\ \text{m}) = 30.8\ \text{kN}$$

Thus, total force in vertical direction F_y is:

$$F_y = W + F_V = 109.3\ \text{kN}$$

3.6 Buoyancy

A *buoyant force* is defined as the upward force that is produced on a body that is totally or partially submerged in a fluid when the fluid is in a gravity field. Buoyant forces are significant for most problems that involve liquids. Buoyant forces are sometimes significant in problems involving gases, for example, a weather balloon.

The Buoyant Force Equation

The initial situation for the derivation is shown in Fig. 3.20. Consider a body *ABCD* submerged in a liquid of specific weight γ. The sketch on the left shows the pressure distribution acting on the body. The pressures acting on the lower portion of the body create an upward force equal to the weight of liquid needed to fill the volume above surface *ADC*. The upward force is

$$F_{\text{up}} = \gamma(\forall_b + \forall_a)$$

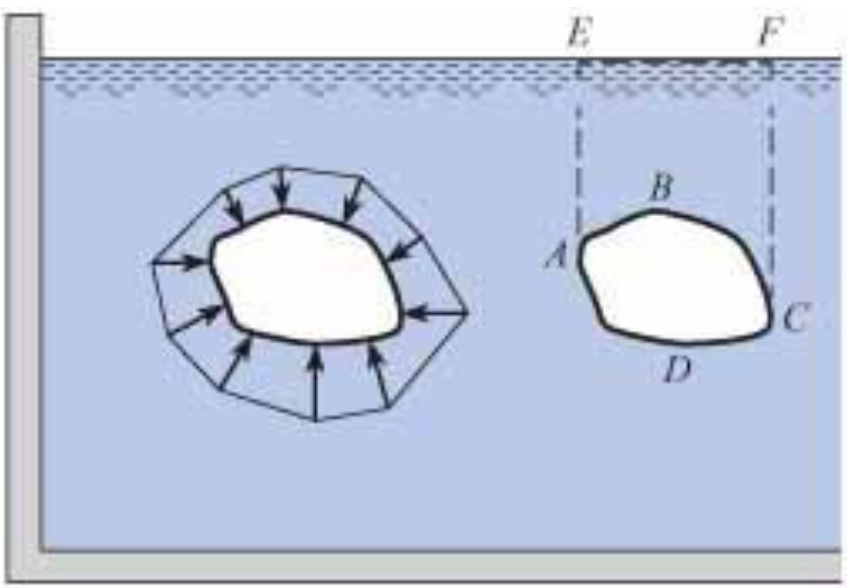
where $\forall_b$ is the volume of the body (i.e., volume *ABCD*) and $\forall_a$ is the volume of liquid above the body (i.e., volume *ABCFE*). The pressures acting on the top surface of the body create a downward force equal to the weight of the liquid above the body:

$$F_{\text{down}} = \gamma\forall_a$$

Subtracting the downward force from the upward force gives the net or buoyant force *FB* acting on the body:

$$F_B = F_{up} - F_{down} = \gamma \, V_b$$

Hence, the net force or buoyant force (*FB*) equals the weight of liquid that would be needed to occupy the volume of the body.



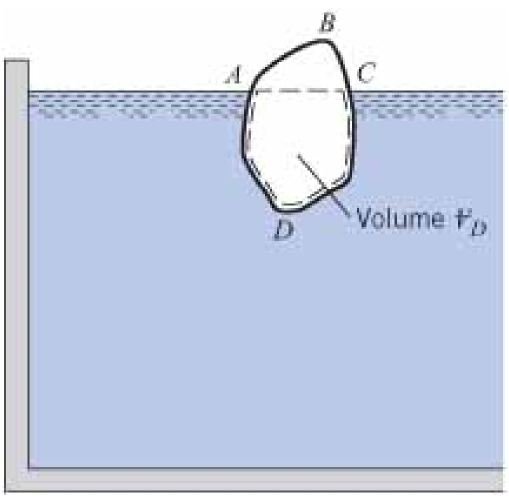


Figure 3.20 *Two views of a body immersed in a liquid.*

The Hydrometer

A *hydrometer* (Fig. 3.22) is an instrument for measuring the specific gravity of liquids. It is typically made of a glass bulb that is weighted on one end so the hydrometer floats in an upright position. A stem of constant diameter is marked with a scale, and the specific weight of the liquid is determined by the depth at which the hydrometer floats. The operating principle of the hydrometer is buoyancy. In a heavy liquid (i.e., high γ), the hydrometer will float shallower because a lesser volume of the liquid must be displaced to balance the weight of the hydrometer. In a light liquid, the hydrometer will float deeper.

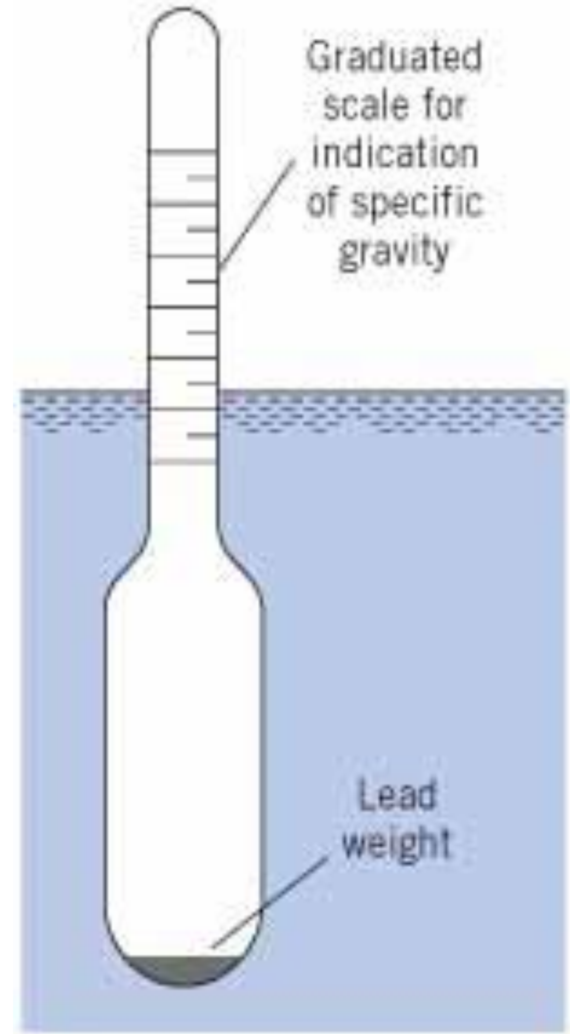


Figure 3.22 *Hydrometer*

3.7 Stability of Immersed and Floating Bodies

This section describes how to determine whether an object will tip over or remain in an upright position when placed in a liquid. This topic is important for the design of objects such as ships and buoys.

Immersed Bodies

When a body is completely immersed in a liquid, its stability depends on the relative positions of the center of gravity of the body and the centroid of the displaced volume of fluid, which is called the *center of buoyancy*. If the center of buoyancy is above the center of gravity, such as in Fig. 3.23*a*, any tipping of the body produces a righting couple, and consequently, the body is stable. However, if the center of gravity is above the center of buoyancy, any tipping produces an increasing overturning moment, thus causing the body to turn through 180°. This is the condition shown in Fig. 3.23*c.* Finally, if the center of buoyancy and center of gravity are coincident, the body is neutrally stable—that is, it lacks a tendency for righting or for overturning, as shown in Fig. 3.23*b*.

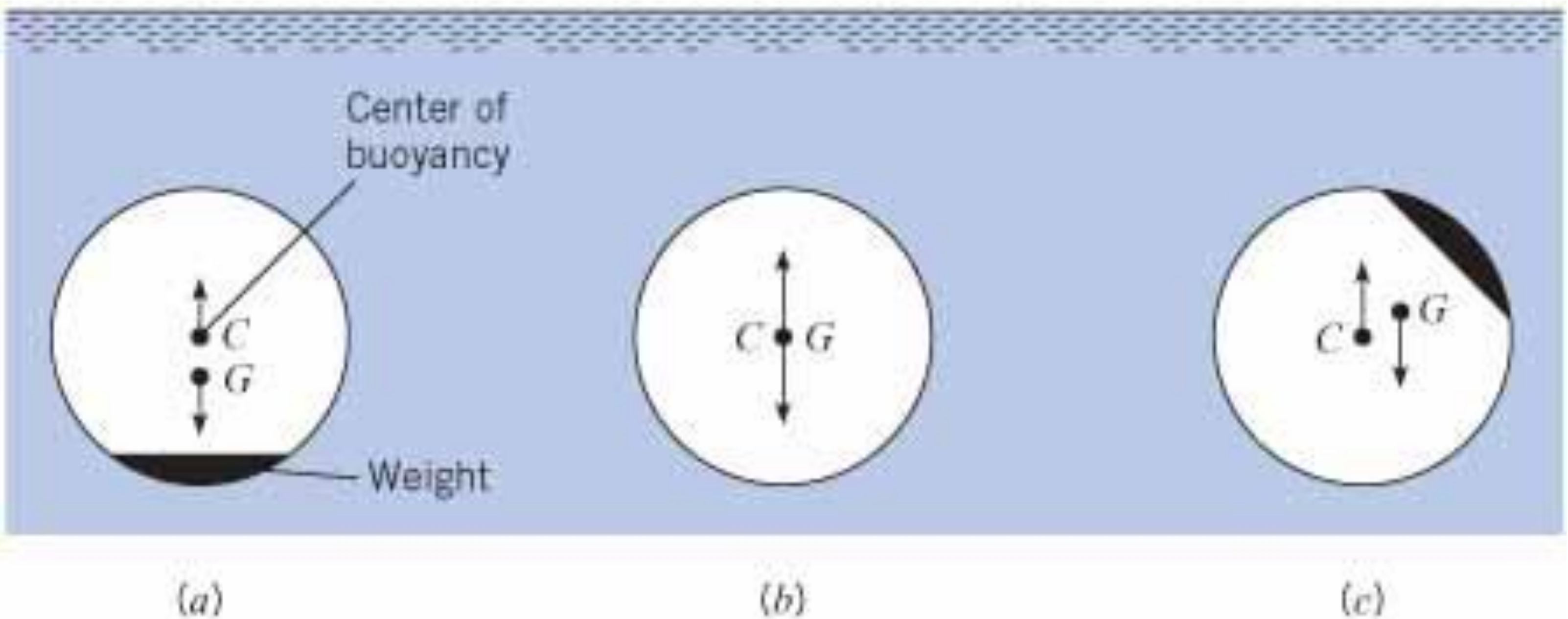


Figure 3.23 *Conditions of stability for immersed bodies.*
(a) Stable.
(b) eutral.
(c) Unstable.

Floating Bodies

The question of stability is more involved for floating bodies than for immersed bodies because the center of buoyancy may take different positions with respect to the center of gravity, depending on the shape of the body and the position in which it is floating. For example, consider the cross section of a ship shown in Fig. 3.24*a*. Here the center of gravity *G* is above the center of buoyancy *C*. Therefore, at first glance it would appear that the ship is unstable and could flip over. However, notice the position of *C* and *G* after the ship has taken a small angle of heel. As shown in Fig. 3.24*b*, the center of gravity is in the same position, but the center of buoyancy has moved outward of the center of gravity, thus producing a righting moment. A ship having such characteristics is stable.

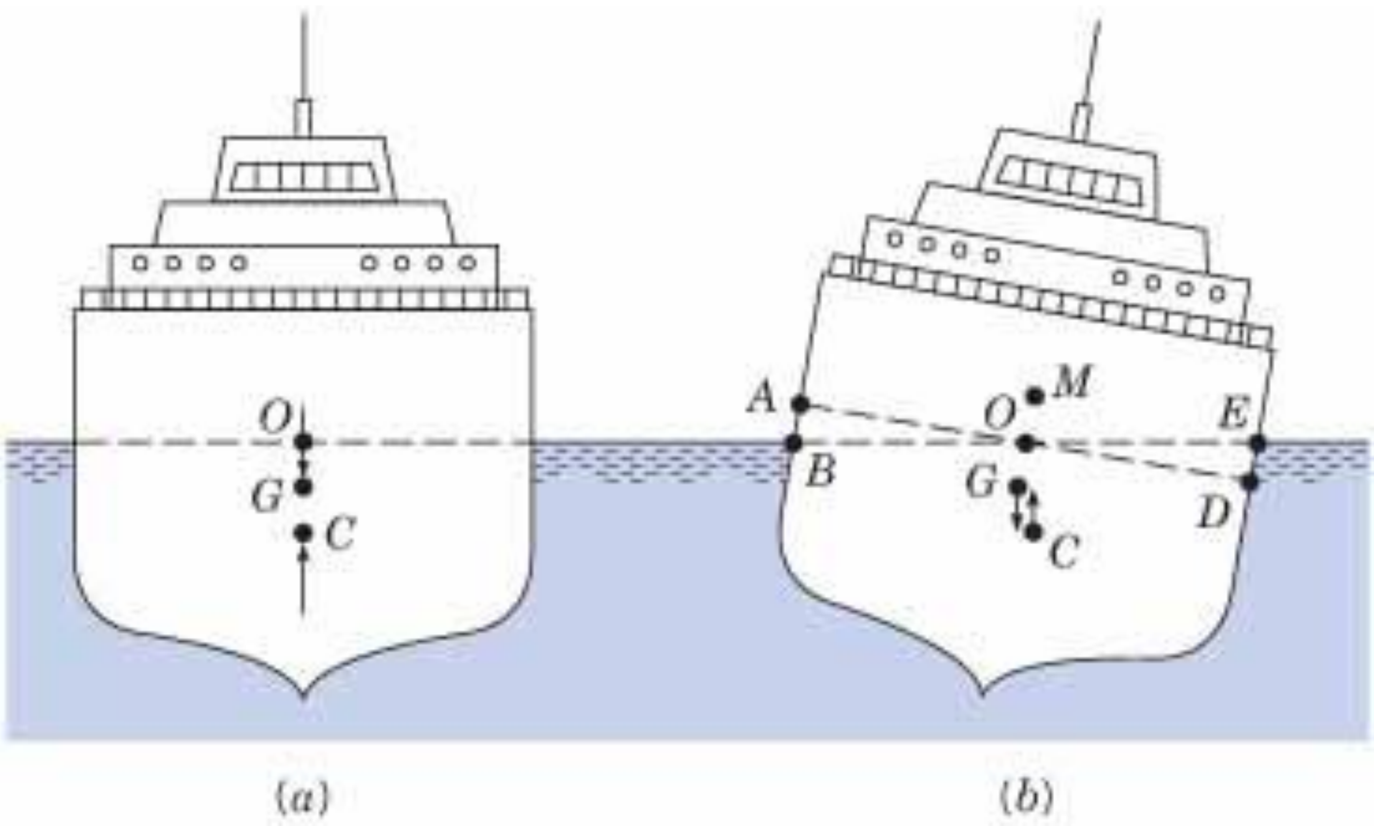


Figure 3.24 *Ship stability relations.*

EXAMPLE 3.12 BUOYA&T FORCE O& A METAL PART

A metal part (object 2) is hanging by a thin cord from a floating wood block (object 1). The wood

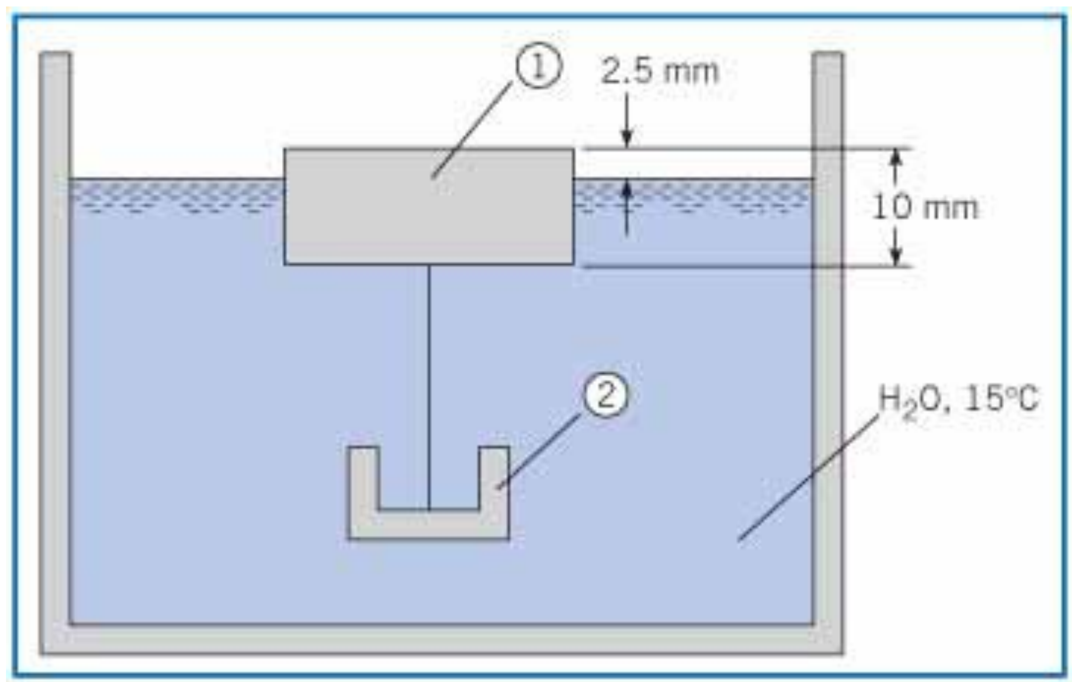
block has a specific gravity $S_1 = 0.3$ and dimensions of 50 × 50 × 10 mm. The metal part has a volume of 6600 mm3. Find the mass m_2 of the metal part and the tension T in the cord.

Properties:

1. Water (15°C),

Table A.5: $\gamma = 9800$ N/m3.

2. Wood: $S_1 = 0.3$.



Solution

1. FBDs

2. Force equilibrium (vertical direction) applied to block

$$T = F_{B1} - W_1$$

- Buoyant force F_{B1} for the submerged

$$F_{B1} = \gamma V_{D1}$$
$$= (9800\ \text{N/m}^3)(50 \times 50 \times 7.5\ \text{mm}^3)(10^{-9}\text{m}^3/\text{mm}^3)$$
$$= 0.184\ \text{N}$$

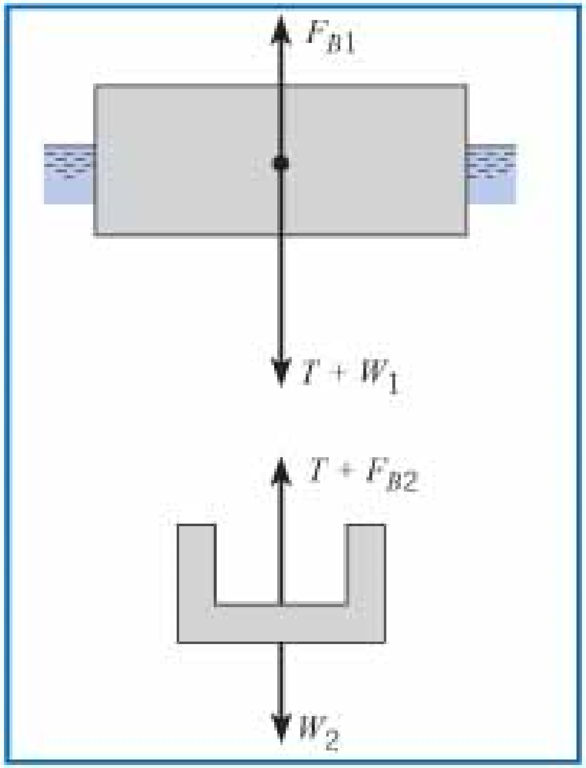
- Weight of the block

$$W_1 = \gamma S_1 V_1$$
$$= (9800\ \text{N/m}^3)(0.3)(50 \times 50 \times 10\ \text{mm}^3)(10^{-9}\text{m}^3/\text{mm}^3)$$
$$= 0.0735\ \text{N}$$

- Tension in the cord

$$T = (0.184 - 0.0735) = \boxed{0.110\ \text{N}}$$

FBD



3. Force equilibrium (vertical direction) applied to metal part

· Buoyant force

$$F_{B2} = \gamma \forall_2 = (9800 \text{ N/m}^3)(6600 \text{ mm}^3)(10^{-9}) = 0.0647 \text{ N}$$

· Equilibrium equation

$$W_2 = T + F_{B2} = (0.110 \text{ N}) + (0.0647 \text{ N})$$

4. Mass of metal part

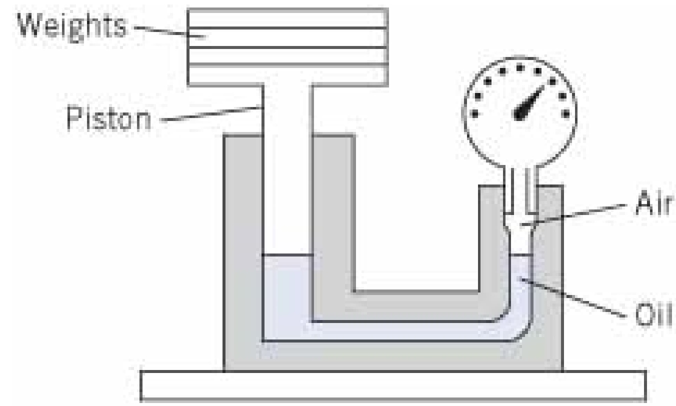
$$m_2 = W_2 / g = \boxed{17.8 \text{ g}}$$

Suggested Problems

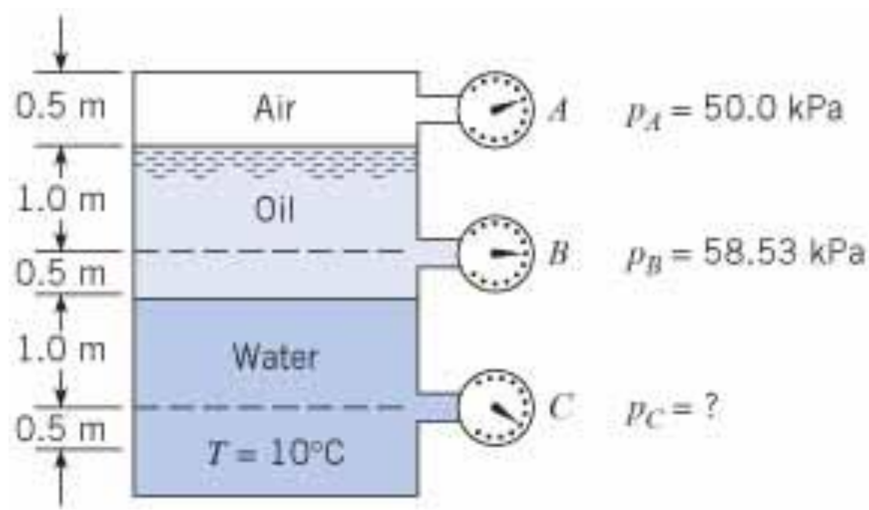
3.4 The Crosby gage tester shown in the figure is used to calibrate or to test pressure gages. When the weights and the piston together weigh 140 N, the gage being tested indicates 200 kPa. If the piston diameter is 30 mm, what percentage of error exists in the gage?

Answer:

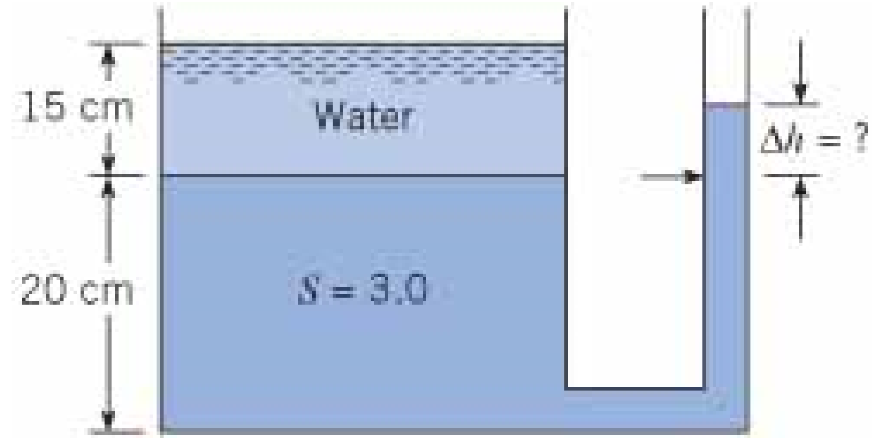
% error = 1.01%



3.11 For the closed tank with Bourdon-tube gages tapped into it, what is the specific gravity of the oil and the pressure reading on gage *C*?



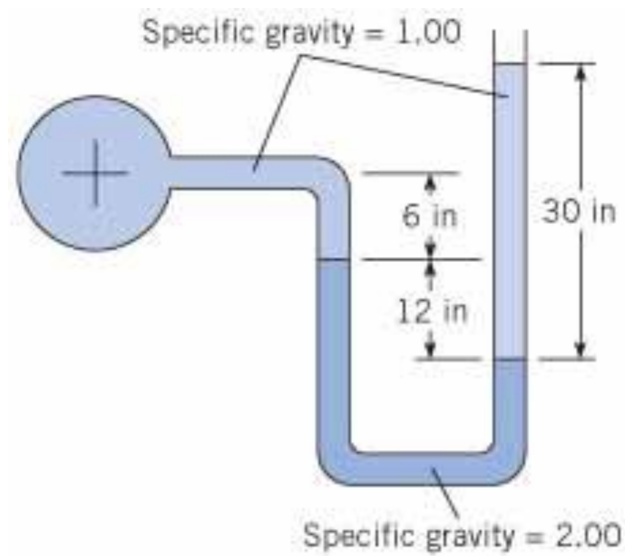
3.18 A tank is fitted with a manometer on the side, as shown. The liquid in the bottom of the tank and in the manometer has a specific gravity (S) of 3.0. The depth of this bottom liquid is 20 cm. A 15 cm layer of water lies on top of the bottom liquid. Find the position of the liquid surface in the manometer.



Answer:

Fh = 5.00 cm

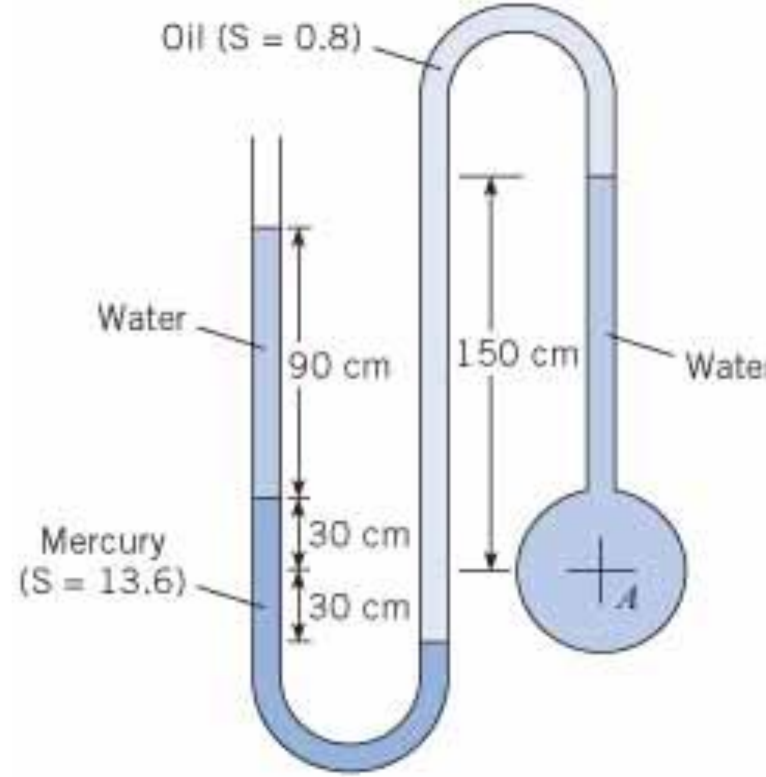
3.30 Is the gage pressure at the center of the pipe (a) negative, (b) zero, or (c) positive? Neglect surface tension effects and state your rationale. Note, solve in SI units.



Answer:

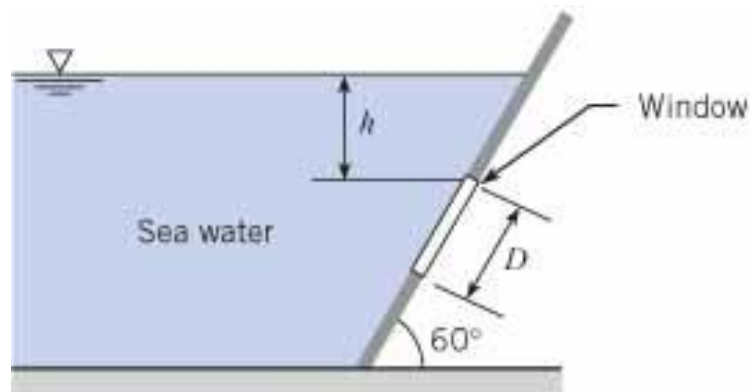
p(center of pipe) = 0.0 lbf/ft2= 0.0 kPa

3.39 Find the pressure at the center of pipe *A*. $T = 10°C$.



3.58 As shown, a round viewing window of diameter $D = 0.8$ m is situated in a large tank of seawater ($S = 1.03$). The top of the window is 1.2 m below the water surface, and the window is angled at 60° with respect to the horizontal. Find the hydrostatic force acting on the window and locate the corresponding CP.

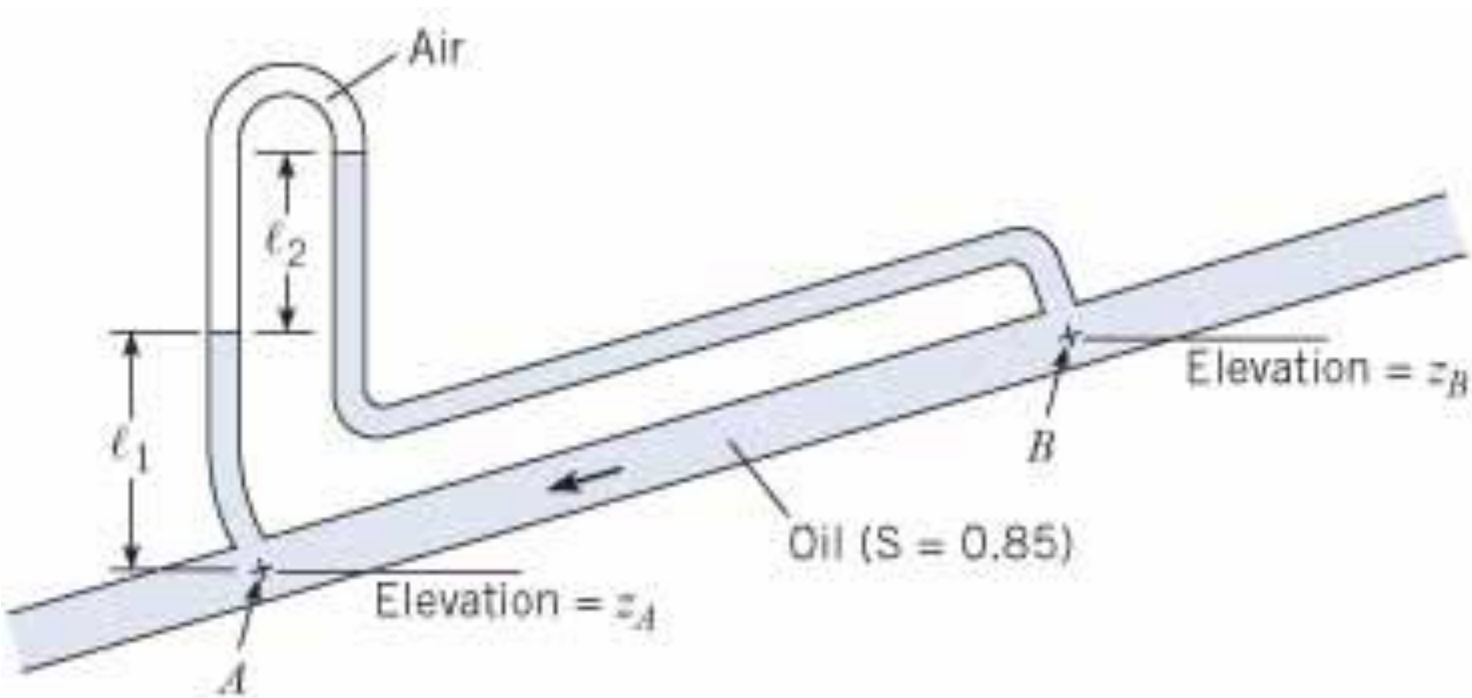
Answer:



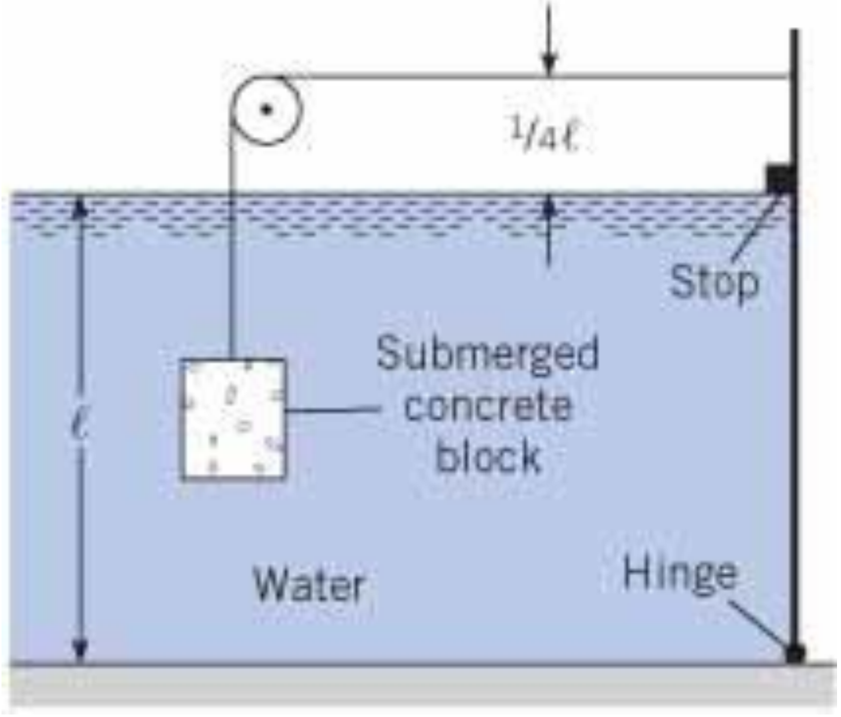
3.40 Determine (a) the difference in pressure and (b) the difference in piezometric head between points *A* and *B.* The elevations z_A and z_B are 10 m and 11 m, respectively, $\ell 1$ = 1 m, and the manometer deflection ℓ_2 is 50 cm.

Answer:

$p_A - p_B = 4.17$ kPa, $h_A - h_B = -0.50$ m



3.93 Determine the minimum volume of concrete (g = 23.6 kN/m3) needed to keep the gate (1 m wide) in a closed position, with ℓ = 2 m. Note the hinge at the bottom of the gate.



Fluid Mechanics

Chapter 4
Flowing Fluids and Pressure Variation

Dr. Amer Khalil Ababneh

This photograph shows the eye of a hurricane. The motion is the result of pressure variations.

Overall Look

In this chapter the pressure variation in flowing fluids will be addressed. The concepts of pathlines and streamlines are introduced to visualize and understand fluid motion. The definition of fluid velocity and acceleration leads to an application of Newton's second law relating forces on a fluid element to the product of mass and acceleration. These relationships lead to the Bernoulli equation, which relates local pressure and elevation to fluid velocity and is fundamental to many fluid mechanic applications. This chapter also introduces the idea of fluid rotation and the concept of irrotationality.

4.1 Descriptions of Fluid Motion

Engineers have developed ways to describe fluid flow patterns and to identify important **characteristics** of the flow field.

Streamlines and Flow Patterns

To visualize the flow field it is desirable to construct lines that show the flow direction. Such a construction is called a flow pattern, and the lines are called streamlines. The *streamline* is defined as a line drawn through the flow field in such a manner that the local velocity vector is tangent to the streamline at every point along the line at that instant. Thus the tangent of the streamline at a given time gives the direction of the velocity vector. A streamline, however, **does not indicate the magnitude** of the velocity. The flow pattern provided by the streamlines is an instantaneous visualization of the flow field.

An example of streamlines and a flow pattern is shown in Fig. 4.1*a* for water flowing through a slot in the side of a tank. The velocity vectors have been sketched at three different locations: *a*, *b*, and *c*. The streamlines, according to their definition, are tangent to the velocity vectors at these points. Also, the velocities are parallel to the wall in the wall region, so the streamlines adjacent to the wall follow the contour of the wall.

Whenever flow occurs around a body, part of it will go to one side and part to the other as shown in Fig. 4.1*b* for flow over an airfoil section. The streamline that follows the flow division (that divides on the upstream side and joins again on the downstream side) is called the *dividing streamline.* At the location where the dividing streamline intersects the body, the velocity will be zero with respect to the body. This is the *stagnation point.*

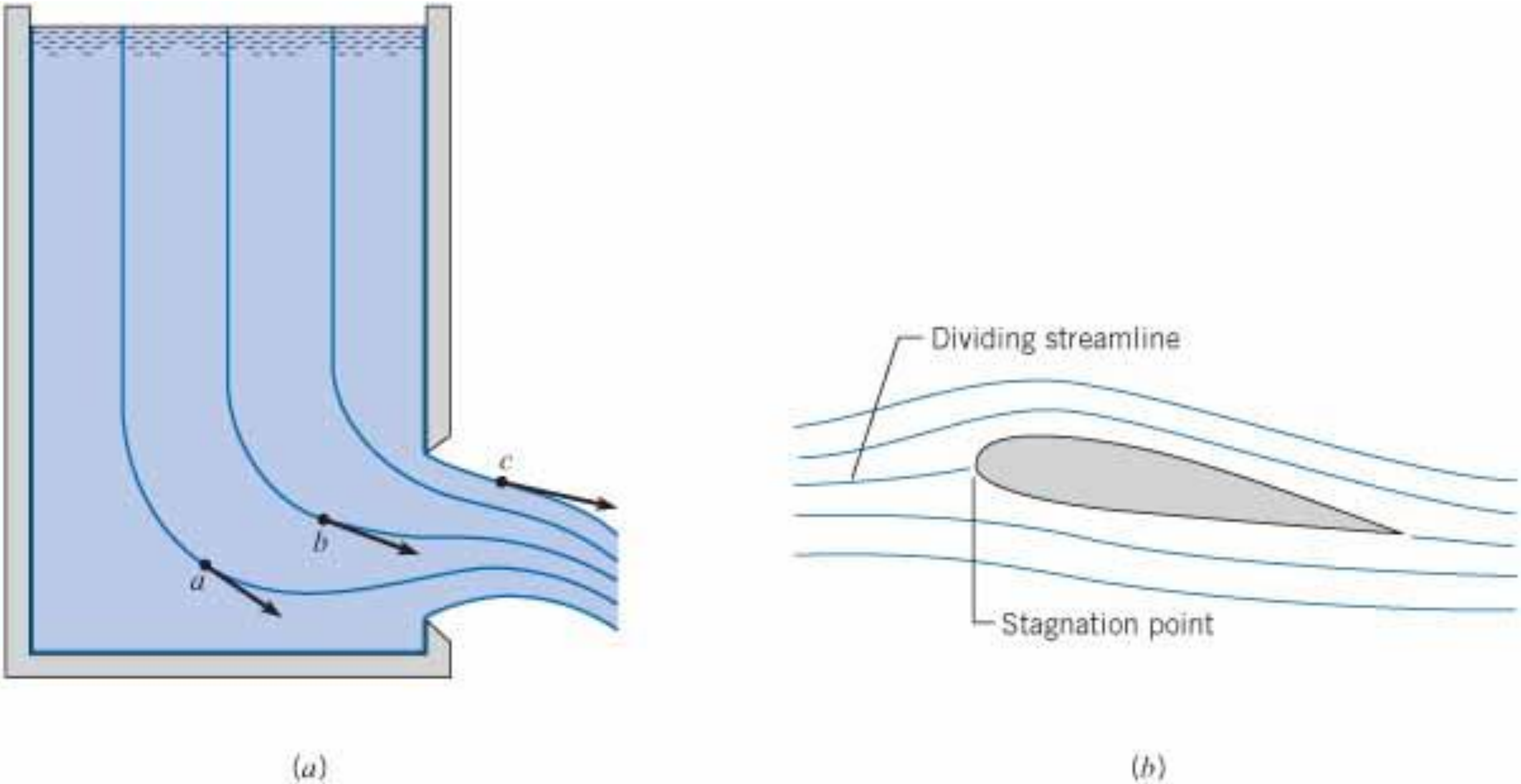


Figure 4.1 *Flow through an opening in a tank and over an airfoil section.*

Another example of streamlines is shown in Fig. 4.2. These are the streamlines predicted for the flow over an Volvo ECC prototype. Flow patterns of this nature allow the engineer to assess various aerodynamic features of the flow and possibly change the shape to achieve better performance, such as reduced drag.

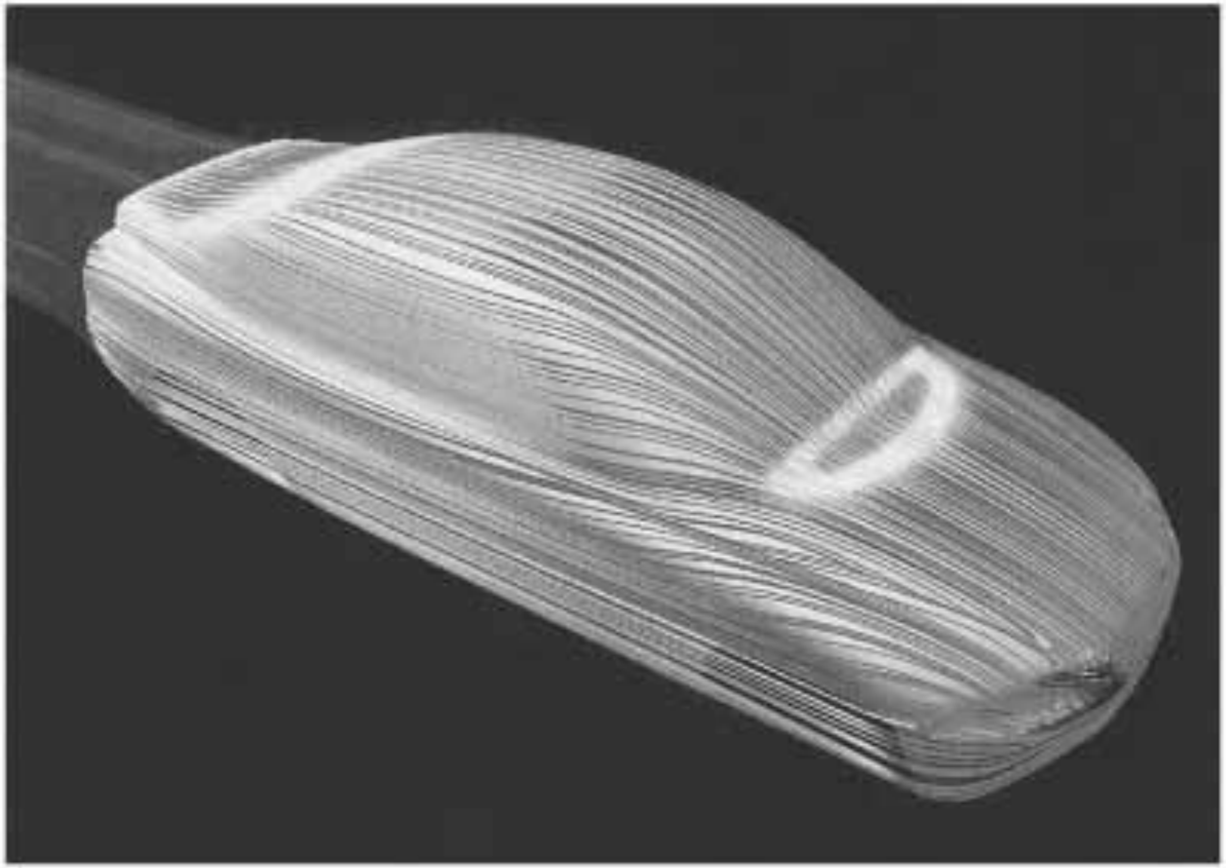


Figure 4.2 *Predicted streamline pattern over the Volvo ECC prototype.* (Courtesy J. Michael Summa, Analytic Methods, Inc.)

The velocity of the fluid may be expressed in the form

$$\boldsymbol{V} = V(s,t)$$

where *s* is the distance traveled by a fluid particle along a path, and *t* is the time, as shown in Fig. 4.3. Flows can be either uniform or nonuniform. In a *uniform flow*, the velocity does not change (magnitude and direction) along a fluid path; that is,

$$\frac{\partial \boldsymbol{V}}{\partial s} = 0$$

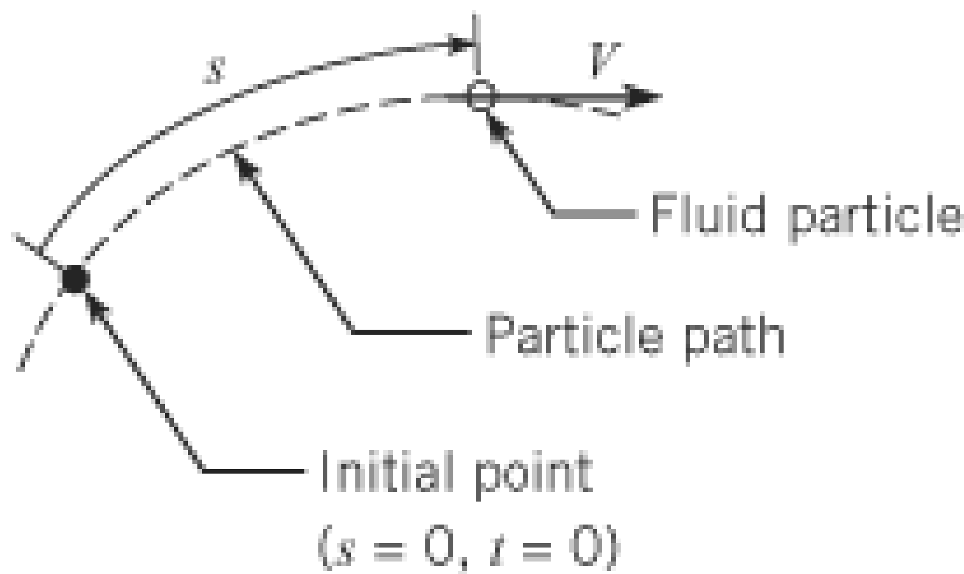


Figure 4.3 *Fluid particle moving along a pathline.*

It follows that in uniform flow the fluid paths are straight and parallel as shown in Fig. 4.4 for flow in a pipe.

$$\frac{\partial V}{\partial s} = 0 \qquad \text{(repeated)}$$

Figure 4.4 *Uniform flow in a pipe.*

In *nonuniform flow*, the velocity changes along a fluid path, so

$$\frac{\partial V}{\partial s} \neq 0$$

For the converging duct in Fig. 4.5*a*, the magnitude of the velocity increases as the duct converges, so the flow is nonuniform. For the vortex flow shown in Fig. 4.5*b*, the magnitude of the velocity does not change along the fluid path, but the direction does, so the flow is nonuniform.

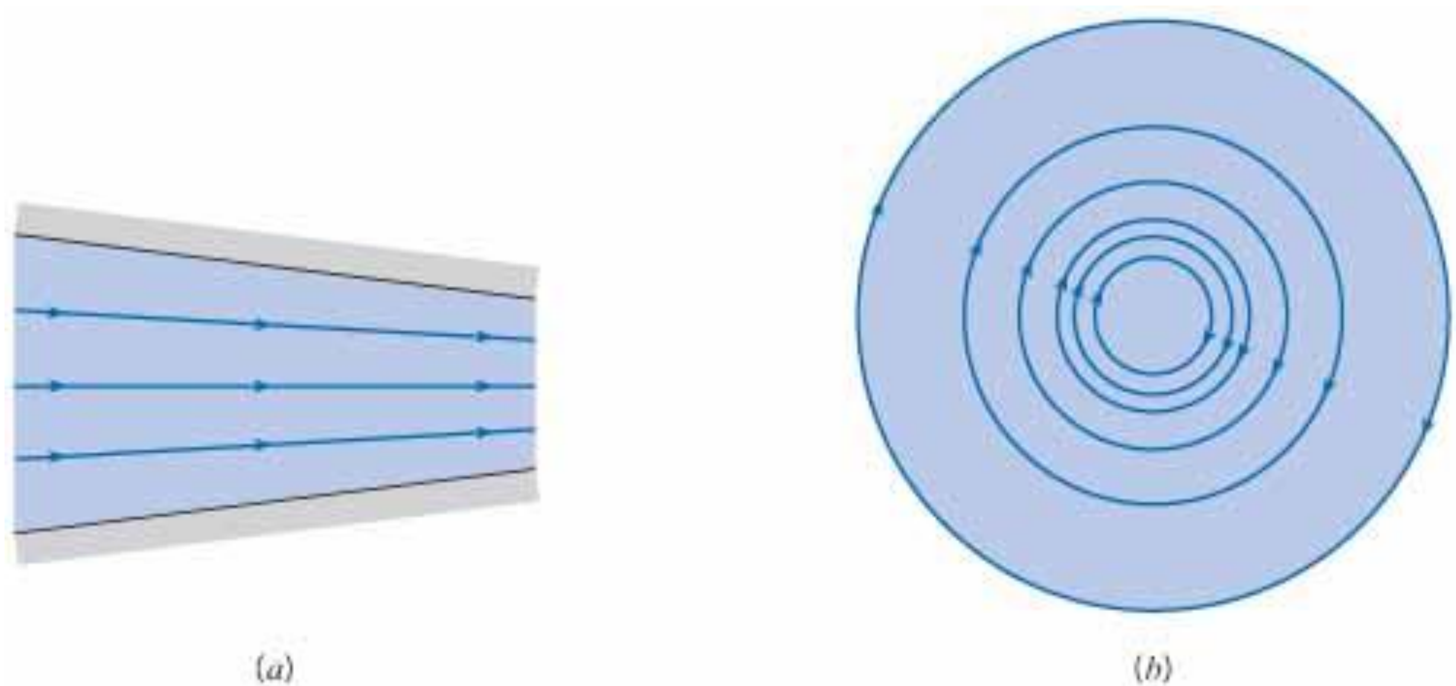


Figure 4.5 *Flow patterns for nonuniform flow. (a) Converging flow. (b) Vortex flow.*

Flows can be either steady or unsteady. In a *steady flow* the velocity at a given point on a fluid path does not change with time:

$$\frac{\partial V}{\partial t} = 0$$

The flow in a pipe, shown previously in Fig. 4.4, would be an example of steady flow if there was no change in velocity with time. An *unsteady flow* exists if

$$\frac{\partial V}{\partial t} \neq 0$$

If the flow in the pipe changed with time due to a valve opening or closing, the flow would be unsteady.

Pathlines and Streaklines

The *pathline* simply is the path of a fluid particle as it moves through the flow field. In other words, if a light were attached to a fluid particle, the trace that it makes would be the pathline.

The *streakline* is the line generated by a tracer fluid, such as a dye, continuously injected into the flow field at a single point.

In general, streamlines, pathlines are streaklines are different in an unsteady flow. But in a steady flow they are identical.

Laminar and Turbulent Flow

Laminar flow is a well-ordered state of flow in which adjacent fluid layers move smoothly with respect to each other. A typical laminar flow would be the flow of honey or thick syrup from a pitcher. Laminar flow in a pipe has a smooth, parabolic velocity distribution as shown in Fig. 4.7*a*.

Turbulent flow is an unsteady flow characterized by intense cross-stream mixing. For example, the flow in the wake of a ship is turbulent. The eddies observed in the wake cause intense mixing. The transport of smoke from a smoke stack on a windy day also exemplifies a turbulent flow. The mixing is apparent as the plume widens and disperses. An instantaneous velocity profile for turbulent flow in a pipe is shown in Fig. 4.7*b*.

In general, laminar pipe flows are associated with low velocities and turbulent flows with high velocities. Laminar flows can occur in small tubes, highly viscous flows, or flows with low velocities, but turbulent flows are, by far, the most common.

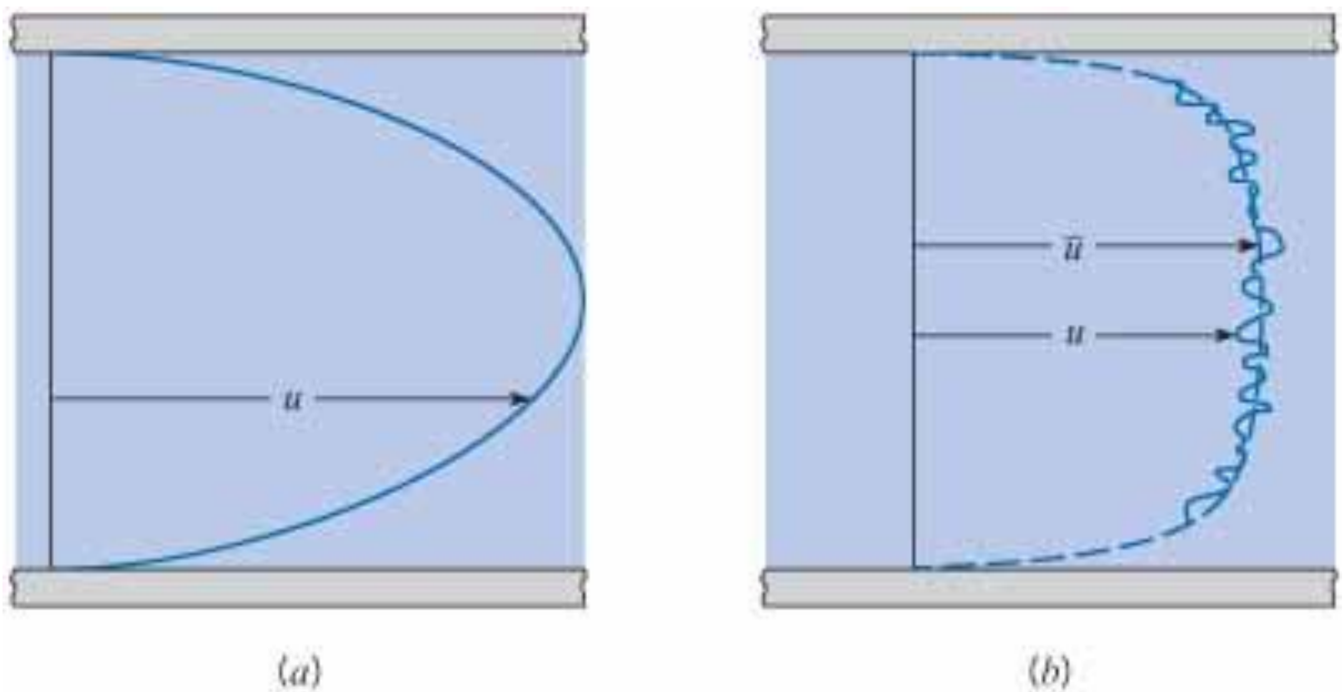
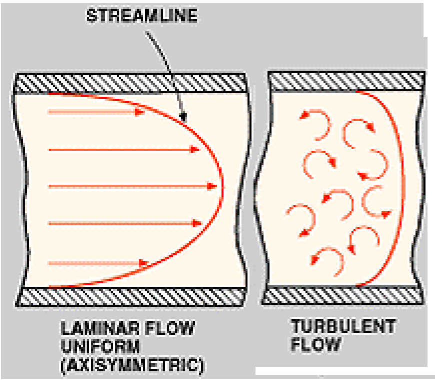
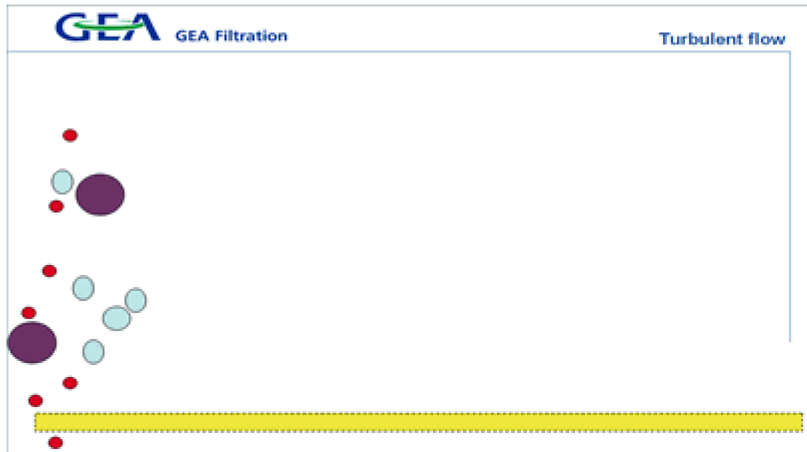
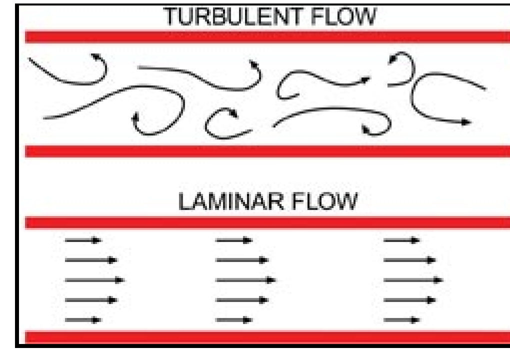


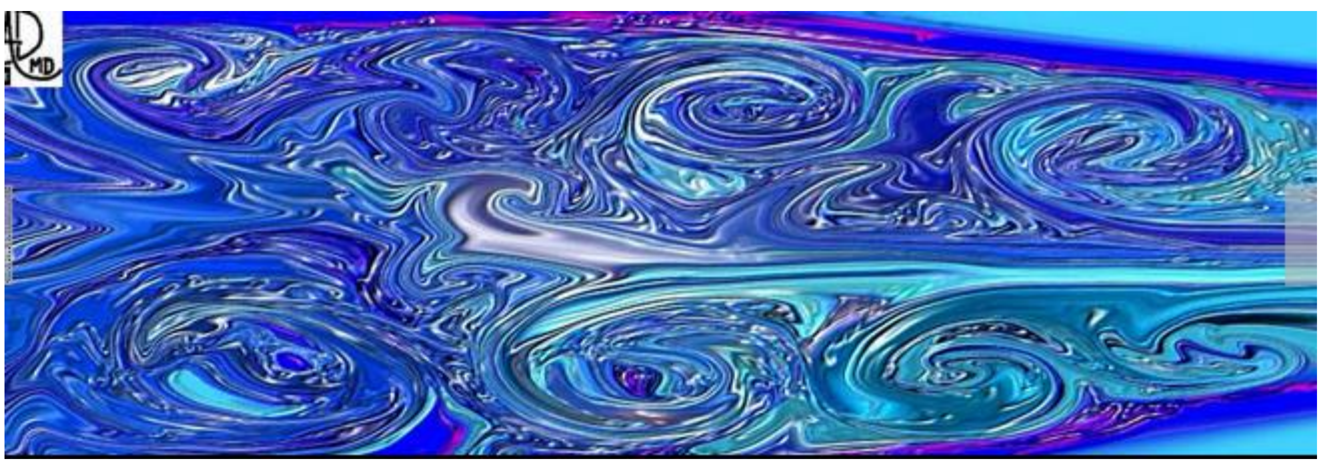
Figure 4.7 *Laminar and turbulent flow in a straight pipe. (a) Laminar flow. (b) Turbulent flow.*

Examples and Demonstration of Turbulent Flows









← Flow direction

One-Dimensional and Multi-Dimensional Flows

The dimensionality of a flow field is characterized by the number of spatial dimensions needed to describe the velocity field.

The definition is best illustrated by example. Fig. 4.8*a* shows the velocity distribution for an axisymmetric flow in a circular duct. The flow is uniform, or fully developed, so the velocity does not change in the flow direction (z). The velocity depends on only one dimension, namely the radius r, so the flow is one-dimensional. Fig. 4.8*b* shows the velocity distribution for uniform flow in a square duct. In this case the velocity depends on two dimensions, namely x and y, so the flow is two-dimensional. Figure 4.8*c* also shows the velocity distribution for the flow in a square duct but the duct cross-sectional area is expanding in the flow direction so the velocity will be dependent on z as well as x and y. This flow is three-dimensional.

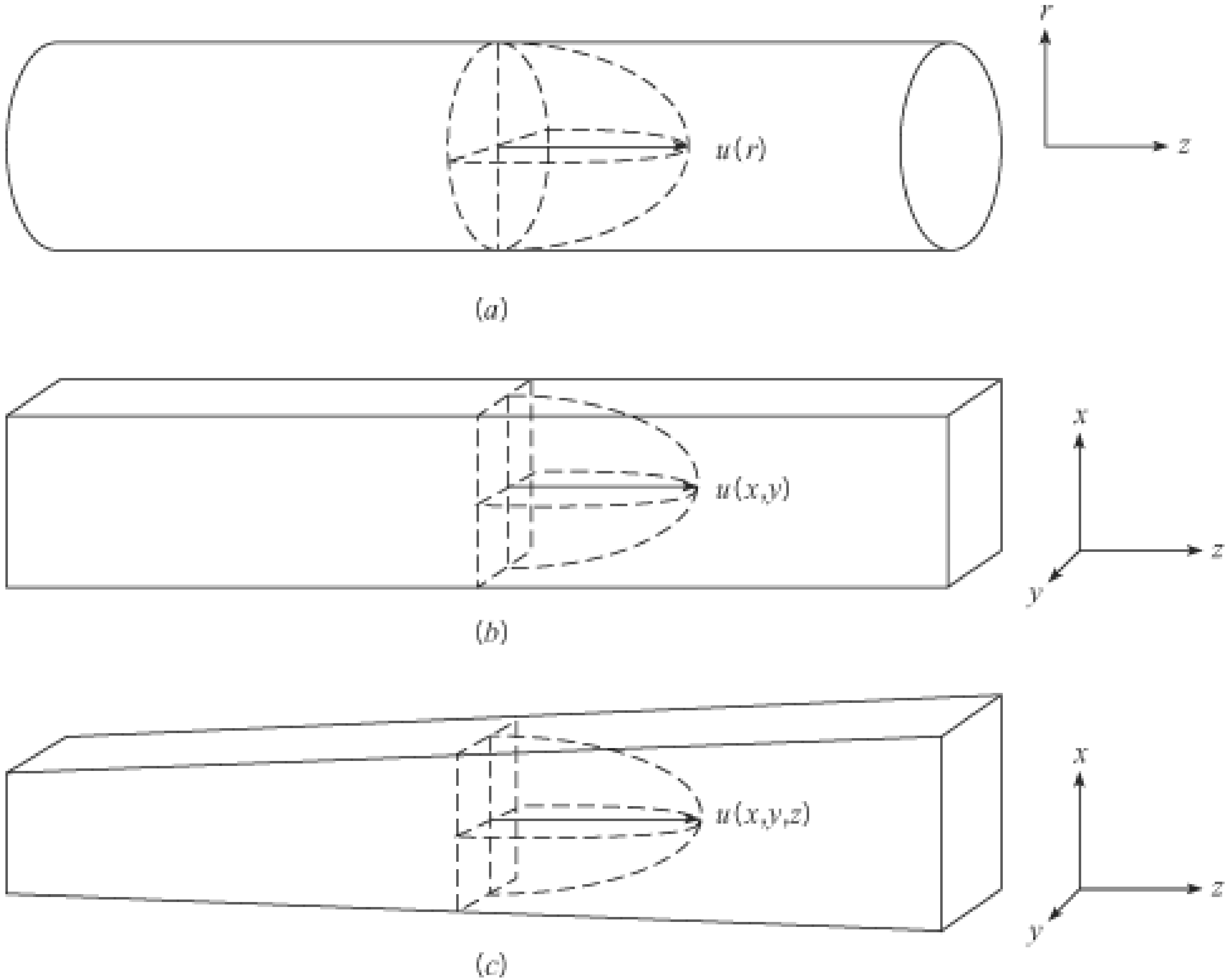


Figure 4.8 *Flow dimensionality: (a) one-dimensional flow, (b) two-dimensional flow, and (c) Three-dimensional flow.*

4.2 Acceleration

The *acceleration* of a fluid particle as it moves along a pathline, as shown in Fig. 4.9, is the rate of change of the particle's velocity with time. The local velocity of the fluid particle depends on the distance traveled, *s*, and time, *t*. The local radius of curvature of the pathline is *r*. The components of the acceleration vector are shown in Fig. 4.9*b*. The normal component of acceleration a_n will be present anytime a fluid particle is moving on a curved path (i.e., centripetal acceleration). The tangential component of acceleration a_t will be present if the particle is changing speed

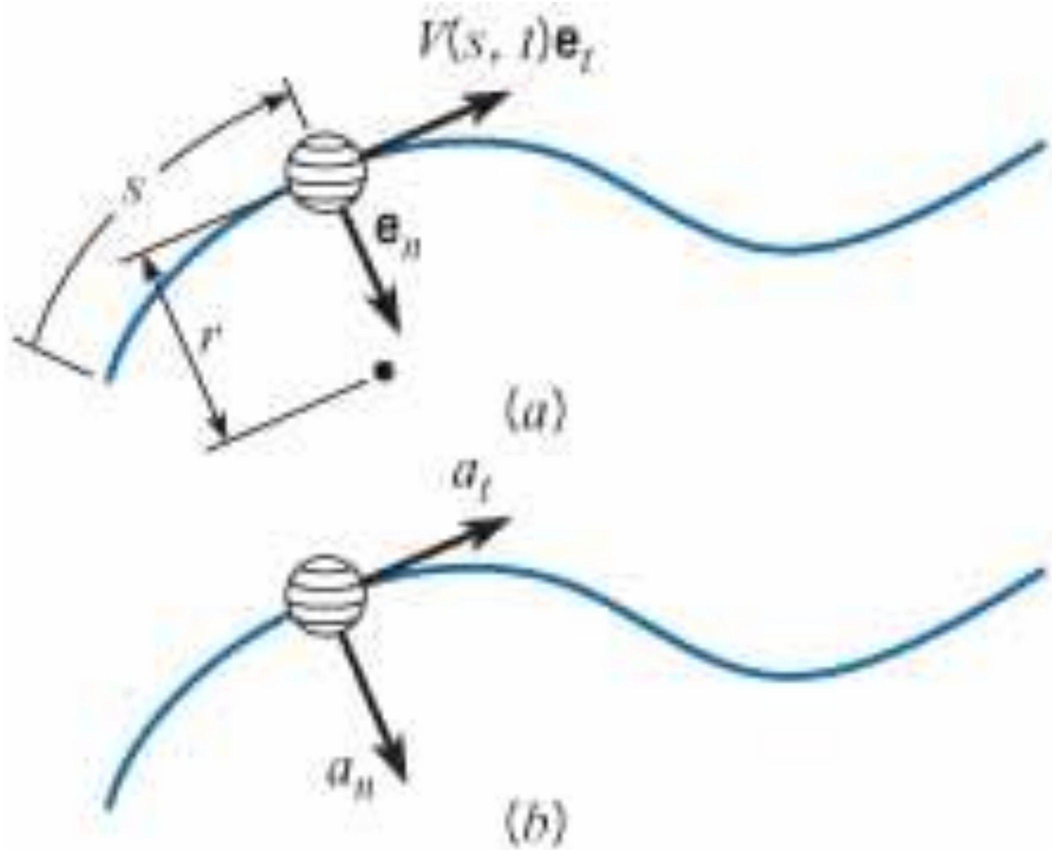


Figure 4.9 *Particle moving on a pathline. (a) Velocity. (b) Acceleration.*

Using normal and tangential components, the velocity of a fluid particle on a pathline (Fig. 4.9*a*) may be written as

$$\boldsymbol{V} = V(s, t)\, \boldsymbol{e}_t$$

where $V(s, t)$ is the speed of the particle, which can vary with distance along the pathline, *s*, and time, *t*. The direction of the velocity vector is given by a unit vector $\mathbf{e}_t$.

Using the definition of acceleration,

$$\mathbf{a} = \frac{dV}{dt} = \left(\frac{dV}{dt}\right)\mathbf{e}_t + V\left(\frac{d\mathbf{e}_t}{dt}\right) \qquad (4.1)$$

To evaluate the derivative of speed in the above equation the chain rule for a function of two variables can be used.

$$\frac{dV(s,\ t)}{dt} = \left(\frac{\partial V}{\partial s}\right)\left(\frac{ds}{dt}\right) + \frac{\partial V}{\partial t}$$

In a time *dt*, the fluid particle moves a distance *ds*, so the derivative *ds/dt* corresponds to the speed *V* of the particle; becomes

$$\frac{dV}{dt} = V\left(\frac{\partial V}{\partial s}\right) + \frac{\partial V}{\partial t}$$

In Eq. (4.1), the derivative of the unit vector $d\mathbf{e}_t/dt$ is nonzero because the direction of the unit vector changes with time as the particle moves along the pathline. The derivative is

$$\frac{d\mathbf{e}_t}{dt} = \frac{V}{r}\,\mathbf{e}_n$$

where $\mathbf{e}_n$ is the unit vector perpendicular to the pathline and pointing inward toward the center of curvature 1.

Substituting the above into Eq. (4.1) gives the acceleration of the fluid particle:

$$\mathbf{a} = \left(V\frac{\partial V}{\partial s} + \frac{\partial V}{\partial t}\right)\mathbf{e}_s + \left(\frac{V^2}{r}\right)\mathbf{e}_n \qquad (4.5)$$

The interpretation of this equation is as follows. The acceleration on the left side is the value recorded at a point in the flow field if one were moving with the fluid particle past that point. The terms on the right side represent another way to evaluate the fluid particle acceleration at the same point by measuring the velocity, the velocity gradient, and the velocity change with time at that point and reducing the acceleration according the terms to the equation.

Convective, Local, and Centripetal Acceleration

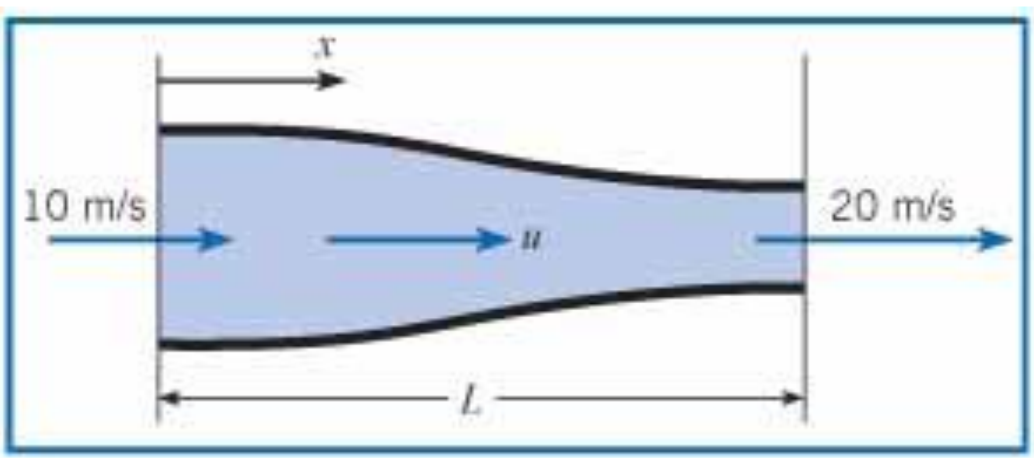
Inspection of Eq. (4.5) reveals that the acceleration component along a pathline depends on two terms. The variation of velocity with time at a point on the pathline, namely $\partial V / \partial t$, is called the ***local acceleration***. In steady flow the local acceleration is zero. The other term, $V \partial V / \partial s$, depends on the variation of velocity along the pathline and is called the ***convective acceleration***. In a uniform flow, the convective acceleration is zero. The acceleration with magnitude V^2 / r, which is normal to the pathline and directed toward the center of rotation, is the ***centripetal acceleration***.

EXAMPLE 4.1 EVALUATING ACCELERATION IN A NOZZLE

A nozzle is designed such that the velocity in the nozzle varies as

$$u(x) = \frac{u_0}{1.0 - 0.5x / L}$$

where the velocity u_0 is the entrance velocity and L is the nozzle length. The entrance velocity is 10 m/s, and the length is 0.5 m. The velocity is uniform across each section. Find the acceleration at the station halfway through the nozzle ($x/L = 0.5$).



Solution

The distance along the pathline is *x*, so *s* in Eq. 4.5 becomes *x* and *V* becomes *u*. The pathline is straight, so $r \rightarrow \infty$.

1. Evaluation of terms:

Convective acceleration

$$\frac{\partial u}{\partial x} = -\frac{u_0}{(1-0.5x/L)^2} \times \left(-\frac{0.5}{L}\right)$$

$$= \frac{1}{L}\frac{0.5u_0}{(1-0.5x/L)^2}$$

$$u\frac{\partial u}{\partial x} = 0.5\frac{u_0^2}{L}\frac{1}{(1-0.5x/L)^3}$$

Evaluation at $x/L = 0.5$:

$$u\frac{\partial u}{\partial x} = 0.5 \times \frac{10^2}{0.5} \times \frac{1}{0.75^3}$$
$$= 237 \text{ m / s}^2$$

Local acceleration:

$$\frac{\partial u}{\partial t} = 0$$

Centripetal acceleration:

$$\frac{u^2}{r} = 0$$

2. Acceleration:

$$a_x = 237 \text{ m / s}^2 + 0$$
$$= \boxed{237 \text{ m / s}^2}$$

$$a_n \text{ (normal to pathline)} = \boxed{0}$$

4.3 Euler's Equation

In Chapter 3 the hydrostatic equations were derived by equating the sum of the forces on a fluid element equal to zero. The same ideas are applied in this section to a moving fluid by equating the sum of the forces acting on a fluid element to the element's acceleration, according to Newton's second law. The resulting equation is Euler's equation, which can be used to predict pressure variation in moving fluids.

Development of Euler's Equation

Consider the cylindrical element in Fig. 4.11*a* oriented in an arbitrary direction ℓ with cross-sectional area ΔA in a flowing fluid. The element is oriented at an angle α with respect to the horizontal plane (the *x-y* plane) as shown in Fig. 4.11*b*. The element has been isolated from the flow field and can be treated as a "free body" where the presence of the surrounding fluid is replaced by pressure forces acting on the element. Neglect viscous forces.

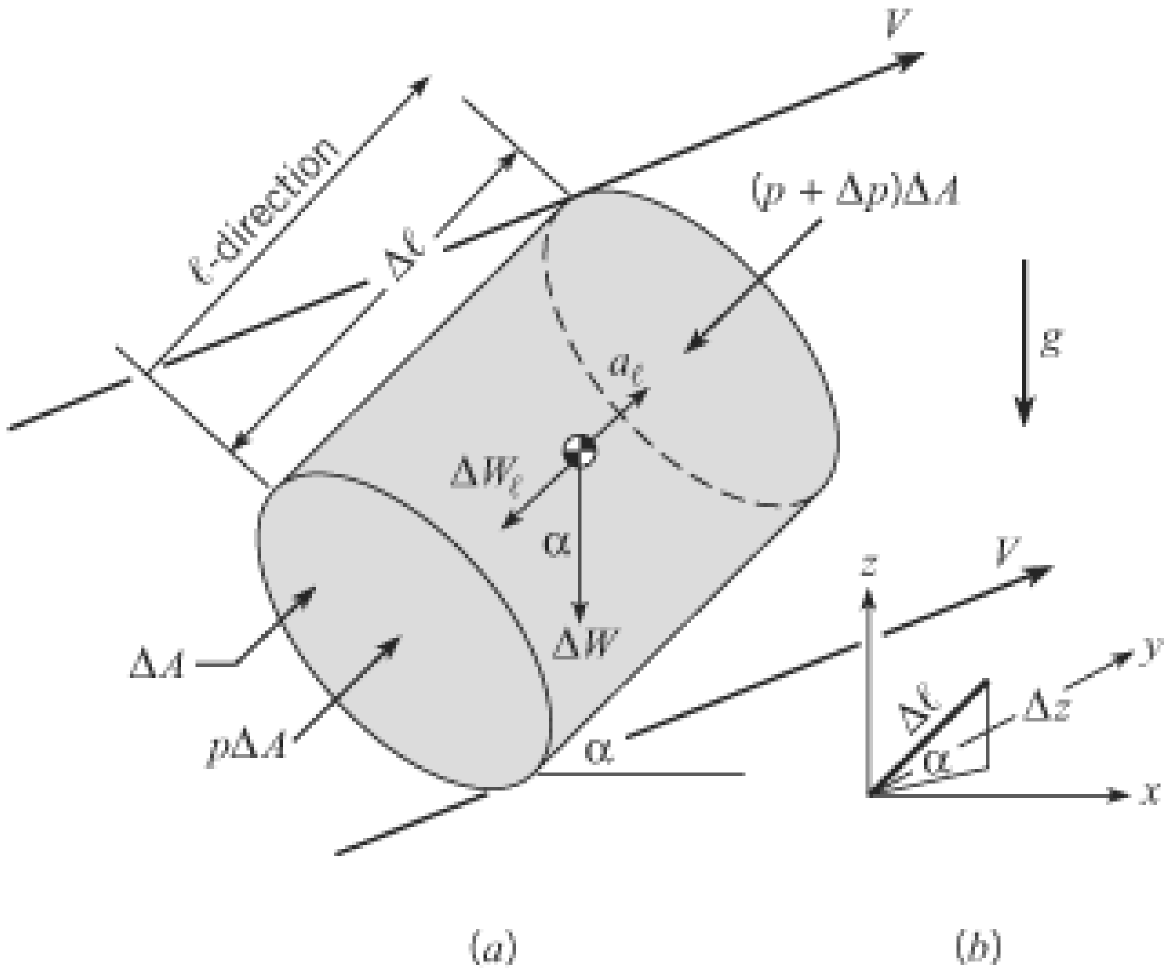


Figure 4.11 ***Free-body diagram for fluid element accelerating the ℓ-direction. (a) Fluid element. (b) Orientation of element in coordinate system.***

Here the element is being accelerated in the ℓ-direction. Note that the coordinate axis *z* is vertically upward and that the pressure varies along the length of the element. Applying Newton's second law in the ℓ-direction results in

$$\sum F_\ell = m\, a_\ell$$

$$F_{pressure} + F_{gravity} = m\, a_\ell$$

The mass of the fluid element is

$$m = \rho\, \Delta A\, \Delta \ell$$

The net force due to pressure in the ℓ-direction is

$$F_{\text{pressure}} = p\Delta A - (p + \Delta p)\Delta A = -\Delta p \Delta A$$

Any pressure forces acting on the side of the cylindrical element will not contribute to a force in the ℓ-direction.

The force due to gravity is the component of weight in the ℓ-direction

$$F_{gravity} = -\Delta W_\ell = \Delta W \sin \alpha$$

notes that sin α = $\Delta z/\Delta \ell$, and substitute back leads to

$$-\Delta p \,\boldsymbol{\Delta A} + \Delta W\, \Delta z/\Delta \ell = \rho\, \boldsymbol{\Delta A\, \Delta \ell\, a_\ell}$$

Substitute for the weight $\Delta W = \gamma \Delta \ell \Delta A$ and rearrange;

$$-\frac{\Delta p}{\Delta \ell} - \gamma \frac{\Delta z}{\Delta \ell} = \rho a_\ell$$

Taking the limit as 'ℓ approaches zero

$$-\frac{\partial p}{\partial \ell} - \gamma \frac{\partial z}{\partial \ell} = \rho a_\ell$$

This equation applies to both incompressible and compressible fluids.

For incompressible fluids γ is constant, hence

$$-\frac{\partial}{\partial l}\left(p+\gamma z\right)=\rho a_l$$

This is the Euler's equation for motion of fluid. It shows that the acceleration is equal to the change in piezometric pressure with distance, and the minus sign means that the acceleration is in the direction of decreasing piezometric pressure. The assumption went into this equations are:

1- In viscid (neglect viscous forces)

2- Incompressible

In a static body of fluid, Euler's equation reduces to the hydrostatic differential equation, Euler's equation can be applied to find the pressure distribution across streamlines in rectilinear flow. Consider the flow with parallel streamlines adjacent a wall, Fig. 4.12. In the direction normal to the wall, the *n* direction, the acceleration is zero. Applying Euler's eqn in the *n* direction gives $\partial/\partial n(p + \gamma z) = 0$, so the piezometric pressure is constant in the normal direction.

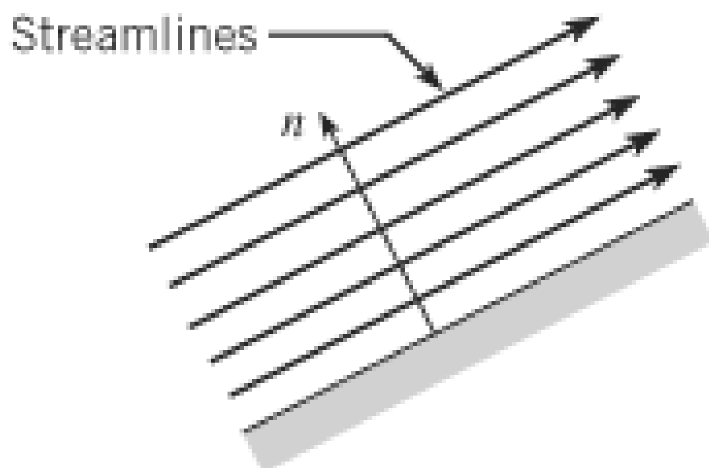


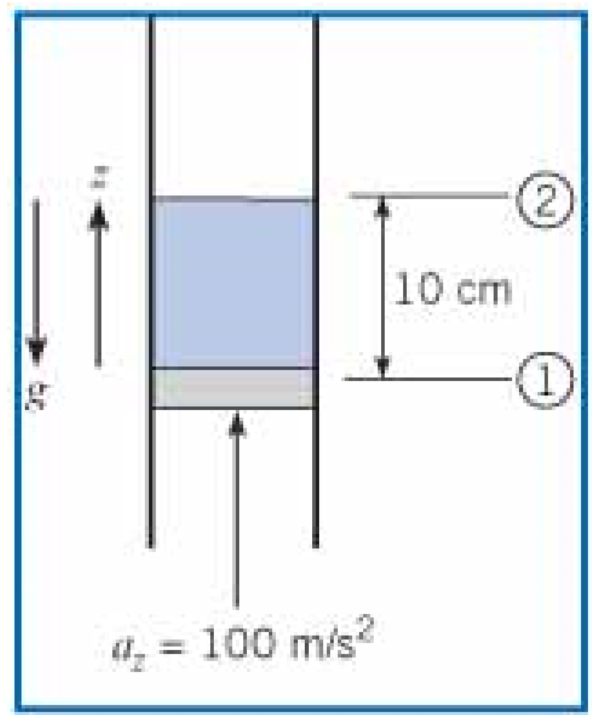
Figure 4.12 *"Normal direction to parallel stream surface*

EXAMPLE 4.2 APPLICATIO OF EULER'S EQUATIO TO ACCELERATIO OF A FLUID

A column of water in a vertical tube is being accelerated by a piston in the vertical direction at 100 m/s². The depth of the water column is 10 cm. Find the gage pressure on the piston. The water density is 10^3 kg/m³.

Solution

1. Because the acceleration is constant there is no dependence on time so the partial derivative in Euler's equation can be replaced by an ordinary derivative. Euler's equation in *z*-direction:



$$-\frac{\partial}{\partial \ell}(p+\gamma z)=\rho a_{\ell} \longrightarrow \frac{d}{dz}(p+\gamma z)=-\rho a_z$$

2. Integration between sections 1 and 2:

$$\int_1^2 d(p+\gamma z)=\int_1^2(-\rho a_z)dz$$

$$(p_2+\gamma z_2)-(p_1+\gamma z_1)=-\rho a_z(z_2-z_1)$$

3. Substitution of limits:

$$p_1=(\gamma+\rho a_z)\Delta z=\rho(g+a_z)\Delta z$$

4. Evaluation of pressure:

$$p_1=10^3 \text{ kg/m}^3\times(9.81+100)\text{ m/s}^2\times 0.1\text{ m}$$

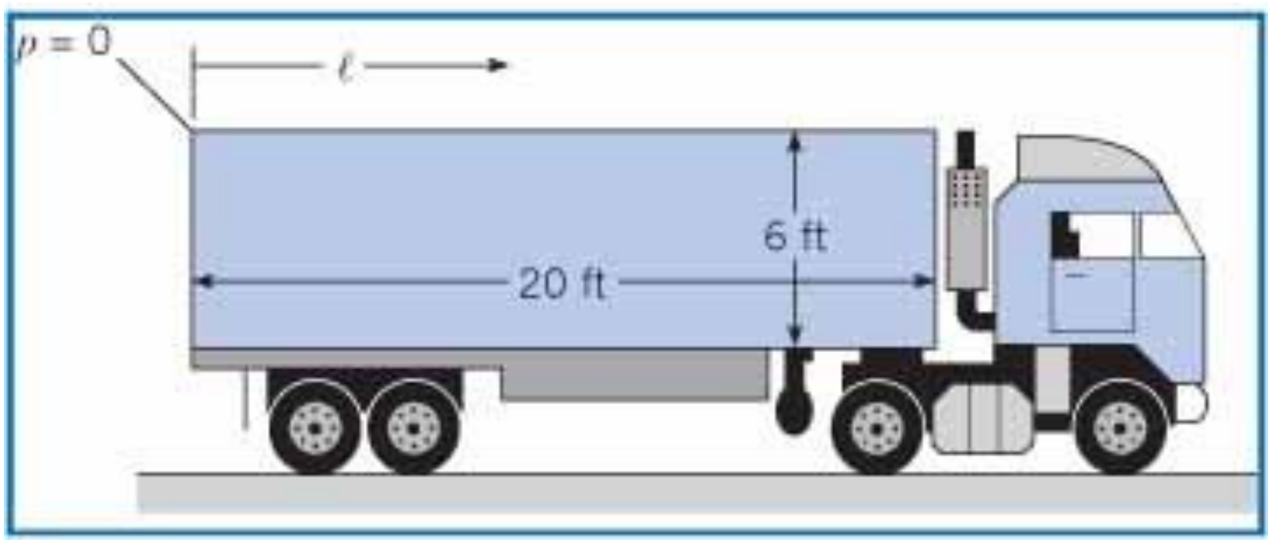
$$p_1=\boxed{10.9\times 10^3\text{ Pa}=10.9\text{ kPa, gage}}$$

EXAMPLE 4.3 PRESSURE IN A DECELERATIG TANK OF LIQUID

The tank on a trailer truck is filled completely with gasoline, which has a specific weight of 42 lbf/ft^3 (6.60 kN/m^3). The truck is decelerating at a rate of 10 ft/s^2 (3.05 m/s^2).

(a) If the tank on the trailer is 20 ft (6.1 m) long and if the pressure at the top rear end of the tank is atmospheric, what is the pressure at the top front?

(b) (b) If the tank is 6 ft (1.83 m) high, what is the maximum pressure in the tank?



Solution

1. Euler's equation along the top of the tank

$$\frac{dp}{d\ell} = -\rho a_\ell$$

Integration from back 1 to front 2

$$p_2 - p_1 = -\rho a_\ell \Delta\ell = -\frac{\gamma}{g} a_\ell \Delta\ell$$

2. Evaluation of p_2 with $p_1 = 0$

$$p_2 = -\left(\frac{6.60 \text{ kN}/\text{m}^3}{9.81 \text{ m}/\text{s}^2}\right) \times (-3.05 \text{ m}/\text{s}^2) \times 6.1 \text{ m}$$

$$= \boxed{12.5 \text{ kPa, gage}}$$

3. Euler's equation in vertical direction

$$\frac{d}{dz}(p + \gamma z) = -\rho a_z$$

4. For vertical direction, $a_z = 0$. Integration from top of tank 2 to bottom 3:

$$p_2 + \gamma z_2 = p_3 + \gamma z_3$$

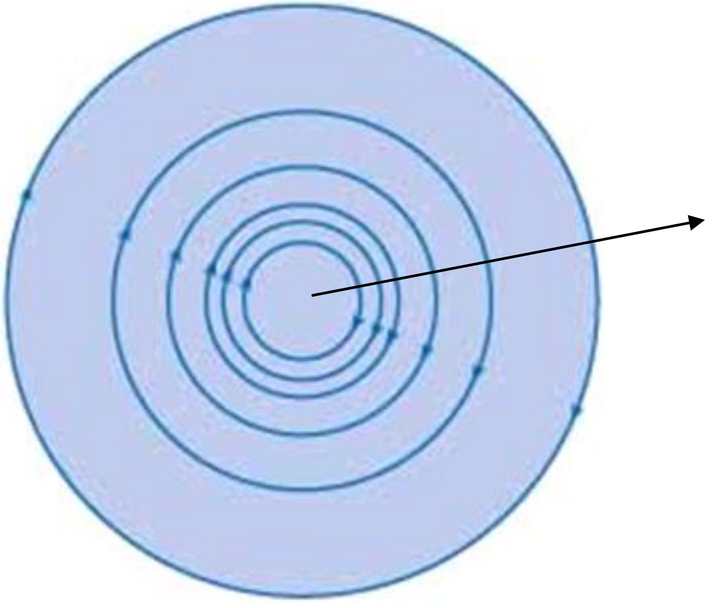
$$p_3 = p_2 + \gamma(z_2 - z_3)$$

$$p_3 = 12.5 \text{ kN/m}^2 + 6.6 \text{ kN/m}^3 \times 1.83 \text{ m}$$

$$p_3 = \boxed{24.6 \text{ kPa, gage}}$$

4.4 Pressure Distribution in Rotating Flows

Situations in which a fluid rotates as a solid body are found in many engineering applications. One common application is the centrifugal separator. The centripetal accelerations resulting from rotating a fluid separate the heavier elements from the lighter elements as the heavier elements move toward the outside and the lighter elements are displaced toward the center. A milk separator operates in this fashion, as does a cyclone separator for removing particulates from an air stream.



Apply Euler's equation in the direction normal to

the streamlines and outward from the center of rotation. In this case the fluid elements rotate as a rigid body, so the direction ℓ in Euler's equation is replaced by *r* giving

$$-\frac{d}{dr}(p+\gamma z)=\rho a_r$$

where the partial derivative has been replaced by an ordinary derivative since the flow is steady and a function only of the radius *r*. The acceleration in the radial direction (away from the center of curvature) is opposite to the centripetal acceleration,

$$a_r=-\frac{V^2}{r}$$

Hence the Euler's equation becomes,

$$-\frac{d}{dr}(p+\gamma z)=-\rho\frac{V^2}{r}$$

For a liquid rotating as a rigid body $V = \omega r$, and substituting back into the Euler's equation

$$\frac{d}{dr}(p + \gamma z) = \rho r \omega^2 = \frac{d(\rho r^2 \omega^2 / 2)}{dr}$$

Separating variables and integrating yields

$$p + \gamma z = \frac{\rho r^2 \omega^2}{2} + \text{const}$$

or,

$$\frac{p}{\gamma} + z - \frac{\omega^2 r^2}{2g} = C$$

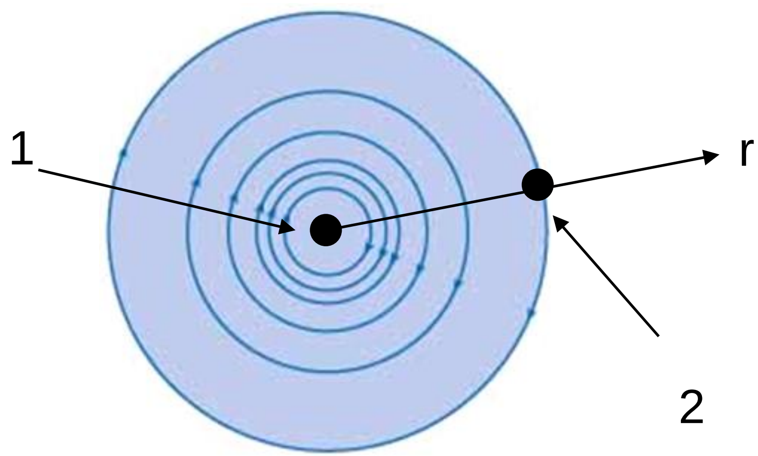
The equation can also be written as,

$$p + \gamma z - \rho \frac{\omega^2 r^2}{2} = C$$

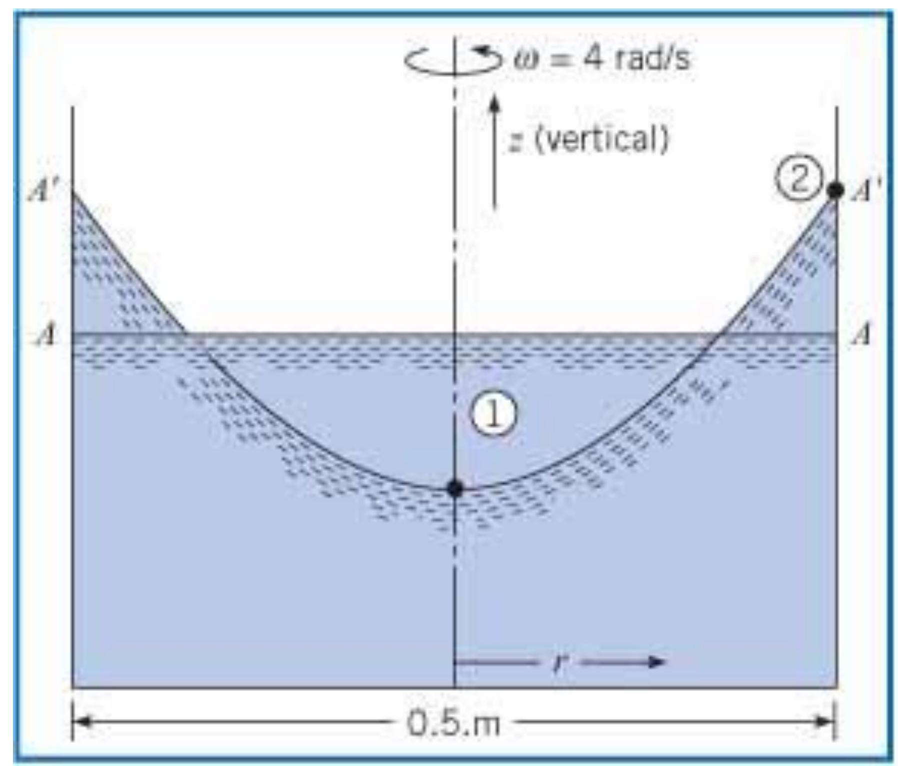
This equation describes how pressure changes in rotating flows.

EXAMPLE 4.4 SURFACE PROFILE OF ROTATING LIQUID

A cylindrical tank of liquid shown in the figure is rotating as a solid body at a rate of 4 rad/s. The tank diameter is 0.5 m. The line *AA* depicts the liquid surface before rotation, and the line *A′A′* shows the surface profile after rotation has been established. Find the elevation difference between the liquid at the center and the wall during rotation.



Pathlines when viewed from top



Solution

1. Equation (4.13a) applied between points 1 and 2.

$$\frac{p_1}{\gamma} + z_1 - \frac{\omega^2 r_1^2}{2g} = \frac{p_2}{\gamma} + z_2 - \frac{\omega^2 r_2^2}{2g}$$

The pressure at both points is atmospheric, so $p_1 = p_2$ and the pressure terms cancel out. At point 1, $r_1 = 0$, and at point 2, $r = r_2$. The equation reduces to

$$z_2 - \frac{\omega^2 {r_2}^2}{2g} = z_1$$

$$z_2 - z_1 = \frac{\omega^2 {r_2}^2}{2g}$$

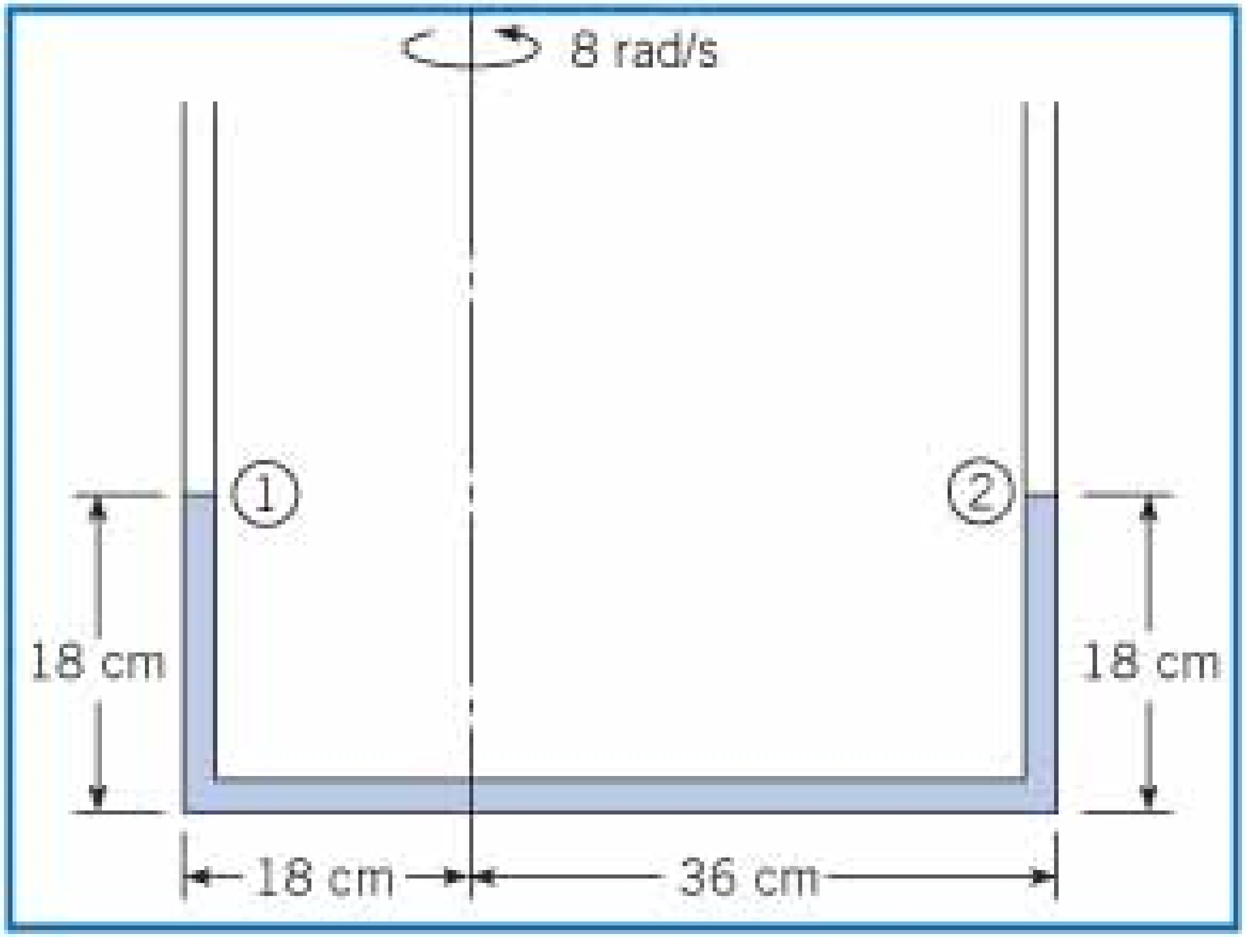
2. Evaluation of elevation difference:

$$z_2 - z_1 = \frac{(4 \text{ rad/s})^2 \times (0.25 \text{ m})^2}{2 \times 9.81 \text{ m/s}^2}$$

$$= \boxed{0.051 \text{ m or } 5.1 \text{ cm}}$$

EXAMPLE 4.5 ROTATING MANOMETER TUBE

When the U-tube is not rotated, the water stands in the tube as shown. If the tube is rotated about the eccentric axis at a rate of 8 rad/s, what are the new levels of water in the tube?



Solution

1. Application of pressure variation for rotating flows between top of leg on left (1) and on right (2):

$$z_1 - \frac{r_1^2\omega^2}{2g} = z_2 - \frac{r_2^2\omega^2}{2g}$$

$$z_2 - z_1 = \frac{\omega^2}{2g}(r_2^2 - r_1^2)$$

$$= \frac{(8\ \text{rad}/\text{s})^2}{2 \times 9.81\ \text{m}/\text{s}^2}(0.36^2\ \text{m}^2 - 0.18^2\ \text{m}^2) = 0.317\ \text{m}$$

2. The sum of the heights in each leg is 36 cm.

$$z_2 + z_1 = 0.36\ \text{m}$$

Solution for the leg heights:

$$z_2 = 0.338\ \text{m}$$

$$z_1 = 0.022\ \text{m}$$

4.5 The Bernoulli Equation Along a Streamline

From the dynamics of particles in solid-body mechanics, one knows that integrating Newton's second law for particle motion along a path provides a relationship between the change in kinetic energy and the work done on the particle. Integrating Euler's equation along a pathline in the steady flow of an incompressible fluid yields an equivalent relationship called the **Bernoulli equation**.

Derivation

The Bernoulli equation is developed by applying Euler's equation along a pathline with the direction , ℓ replaced by *s*, the distance along the pathline, and the acceleration a_ℓ replaced by a_t, the direction tangent to the pathline. Euler's equation becomes

$$-\frac{\partial}{\partial s}(p+\gamma z)=\rho a_t$$

The tangential component of acceleration is given by:

$$a_t=V\frac{\partial V}{\partial s}+\frac{\partial V}{\partial t}$$

For a steady flow, the local acceleration is zero and the pathline becomes a streamline. Also, the properties along a streamline depend only on the distance *s*, so the partial derivatives become ordinary derivatives.

$$-\frac{d}{ds}(p+\gamma z)=\rho V\frac{dV}{ds}=\rho\frac{d}{ds}\left(\frac{V^2}{2}\right)$$

Moving all the terms to one side yields

$$\frac{d}{ds}\left(p + \gamma z + \rho \frac{V^2}{2}\right) = 0$$

or

$$p + \gamma z + \rho \frac{V^2}{2} = C$$

where C is a constant. This is known as the *Bernoulli equation,* which states that the sum of the piezometric pressure ($p + \gamma z$) and kinetic pressure ($\rho V^2/2$)* is constant along a streamline for the *steady* flow of an *incompressible, inviscid* fluid. Dividing the by the specific weight yields the equivalent form of the Bernoulli equation along a streamline, namely

$$\frac{p}{\gamma} + z + \frac{V^2}{2g} = h + \frac{V^2}{2g} = C$$

where h is the piezometric head and ($V^2/2\,g$) is the velocity head.

In words,

$$\begin{pmatrix}\text{Pressure}\\ \text{head}\end{pmatrix}+\begin{pmatrix}\text{Elevation}\\ \text{head}\end{pmatrix}+\begin{pmatrix}\text{Velocity}\\ \text{head}\end{pmatrix}=\begin{pmatrix}\text{Constant along}\\ \text{streamline}\end{pmatrix}$$

- The concept underlying the Bernoulli equation can be illustrated by considering the flow through the inclined venturi (contraction-expansion) section as shown in Fig. 4.13. This configuration is often used as a flow metering device. The reduced area of the throat section leads to an increased velocity and attendant pressure change. The streamline is the centerline of the venturi. Piezometers are tapped into the wall at three locations, and the height of the liquid in the tube above the centerline is p/γ. The elevation of the centerline (streamline) above a datum is z. The location of the datum line is arbitrary.

The constant in the Bernoulli equation is the same at all three locations, so

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} = \frac{p_3}{\gamma} + z_3 + \frac{V_3^2}{2g}$$

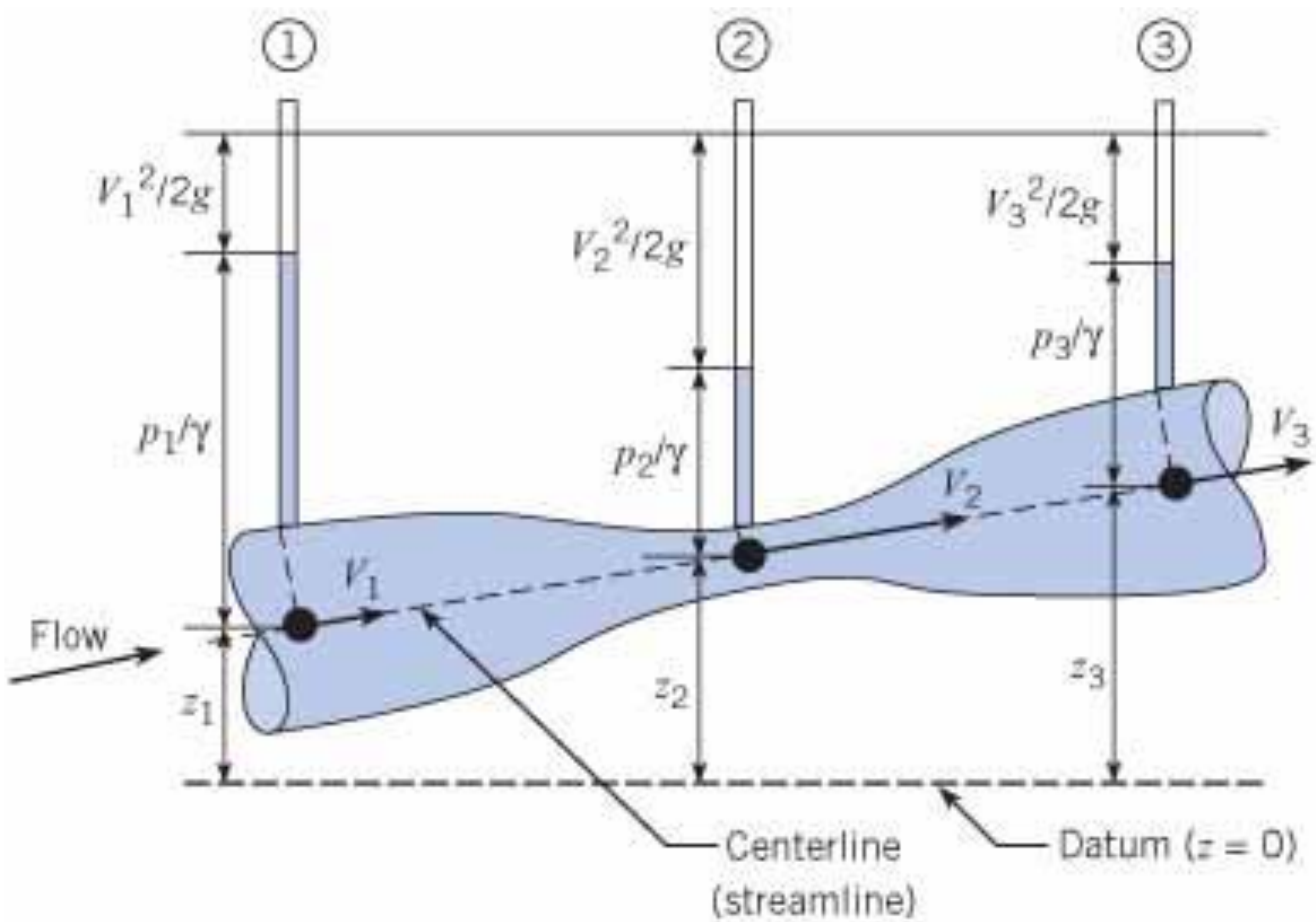


Figure 4.13 *Piezometric and velocity head variation for flow through a venturi section.*

The assumptions for Bernoulli equation:

1) Flow is steady
2) Flow along streamlines
3) Fluid is inviscid
4) Fluid is incompressible

NOTE:

The fact that the Bernoulli equation has been derived for an inviscid fluid does not limit its application here. Even though the real fluid is viscous, the effects of viscosity are small for short distances. Also, the effects of viscosity on pressure change are negligible compared to the pressure change due to velocity variation.

Application of the Bernoulli Equation

The Bernoulli equation is often used to calculate the velocity in venturi configurations given the pressure difference between the upstream section and the throat section, as shown in Example 4.6

EXAMPLE 4.6 VELOCITY IN A VENTURI SECTION

Piezometric tubes are tapped into a venturi section as shown in the figure. The liquid is incompressible. The upstream piezometric head is 1 m, and the piezometric head at the throat is 0.5 m. The velocity in the throat section is twice large as in the approach section. Find the velocity in the throat section.

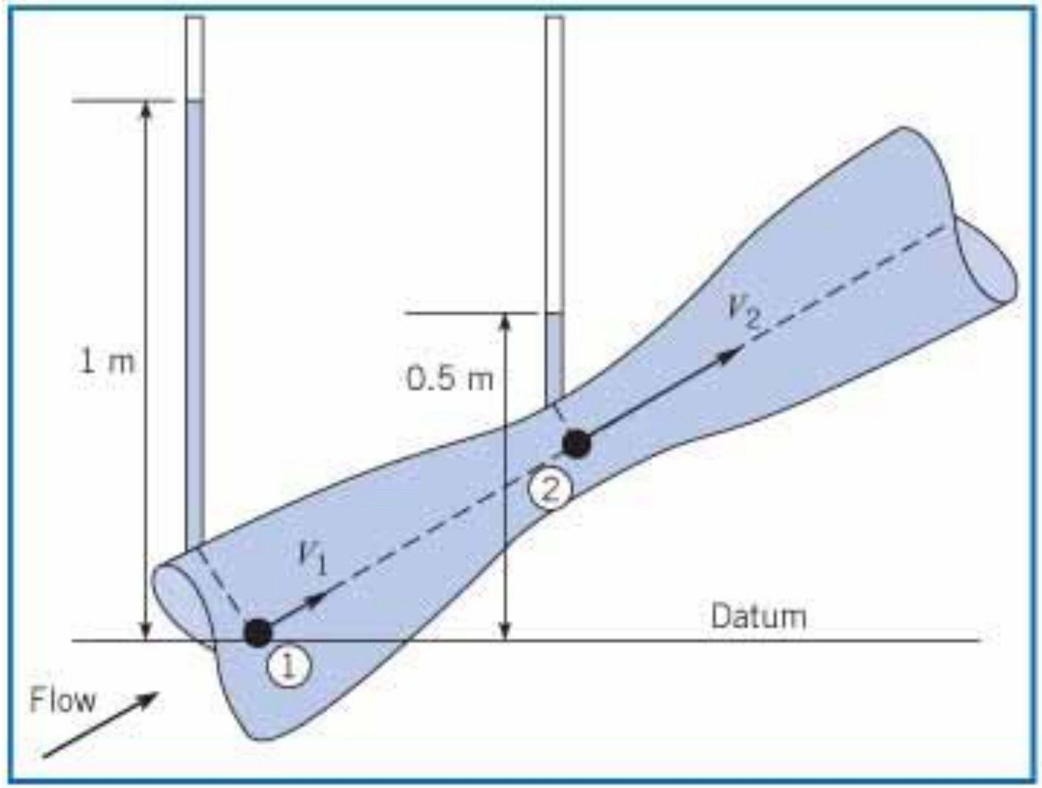
Solution

1. The Bernoulli equation with $V_2 = 2V_1$ gives

$$h_1 - h_2 = \frac{V_2^2 - V_1^2}{2g} = \frac{3\,V_1^2}{2g}$$

$$V_1^2 = \frac{2g}{3}(h_1 - h_2)$$

$$V_2 = 2\sqrt{\frac{2g}{3}(h_1 - h_2)}$$

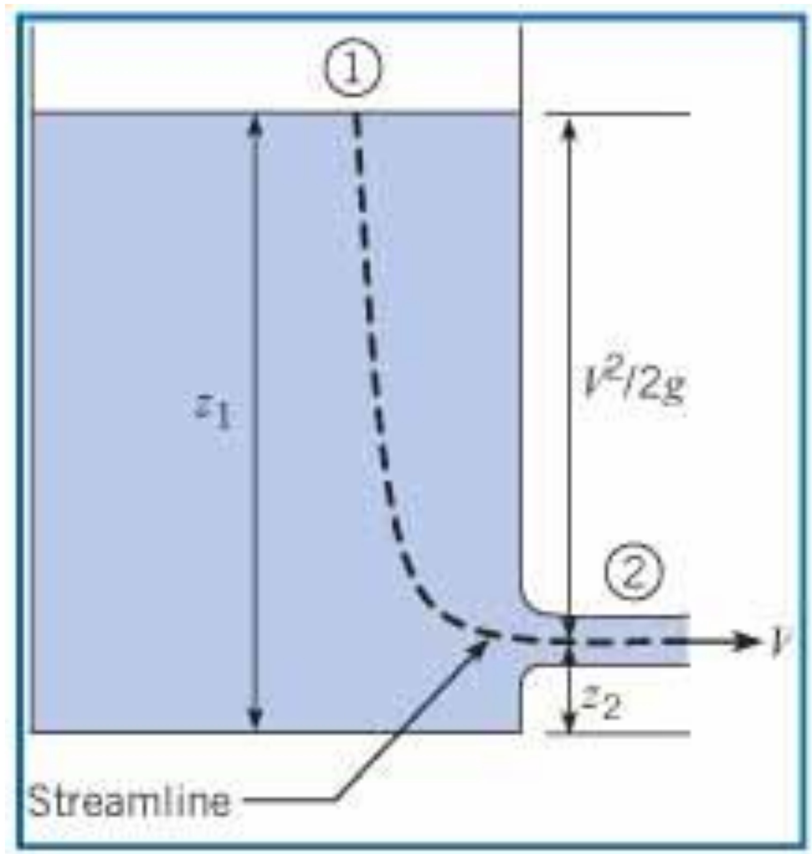


$$V_2 = 2\sqrt{\frac{2 \times 9.81\ \mathrm{m/s^2}\ (1 - 0.5)\ \mathrm{m}}{3}}$$

$$= \boxed{3.62\ \mathrm{m/s}}$$

EXAMPLE 4.7 OUTLET VELOCITY FROM DRAININGTANK

A open tank filled with water and drains through a port at the bottom of the tank. The elevation of the water in the tank is 10 m above the drain. The drain port is at atmospheric pressure. Find the velocity of the liquid in the drain port.



Solution

1. The Bernoulli equation between points 1 and 2 on streamline:

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^{\,2}}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^{\,2}}{2g}$$

2. The pressure at the outlet and the tank surface are the same (atmospheric), so $p_1 = p_2$. The velocity at the tank surface is much less than in the drain port so . Solution for V_2:

$$z_1 - z_2 = \frac{V_2^{\,2}}{2g}$$

$$V_2 = \sqrt{2g(z_1 - z_2)}$$

3. Velocity calculation:

$$V_2 = \sqrt{2 \times 9.81\ \mathrm{m/s^2} \times 10\ \mathrm{m}}$$

$$= \boxed{14\ \mathrm{m/s}}$$

Application of the Bernoulli Equation to Velocity Measurement Devices

The Bernoulli equation can be used to reduce data for flow velocity measurements from a stagnation tube and a Pitot-static tube.

Stagnation Tube

A *Stagnation tube* (sometimes call a total head tube) is an open-ended tube directed upstream in a flow and connected to a pressure sensor. Because the velocity is zero at the tube opening, the pressure measured corresponds to stagnation conditions.

Consider the stagnation tube shown in Fig. 4.14. In this case the pressure sensor is a piezometer. The rise of the liquid in the vertical leg is a measure of the pressure. When the Bernoulli equation is written between points 0 and 1 on the streamline, one notes that $z_0 = z_1$.

Therefore, the Bernoulli equation reduces to

$$p_1 + \frac{\rho V_1^2}{2} = p_0 + \frac{\rho V_0^2}{2}$$

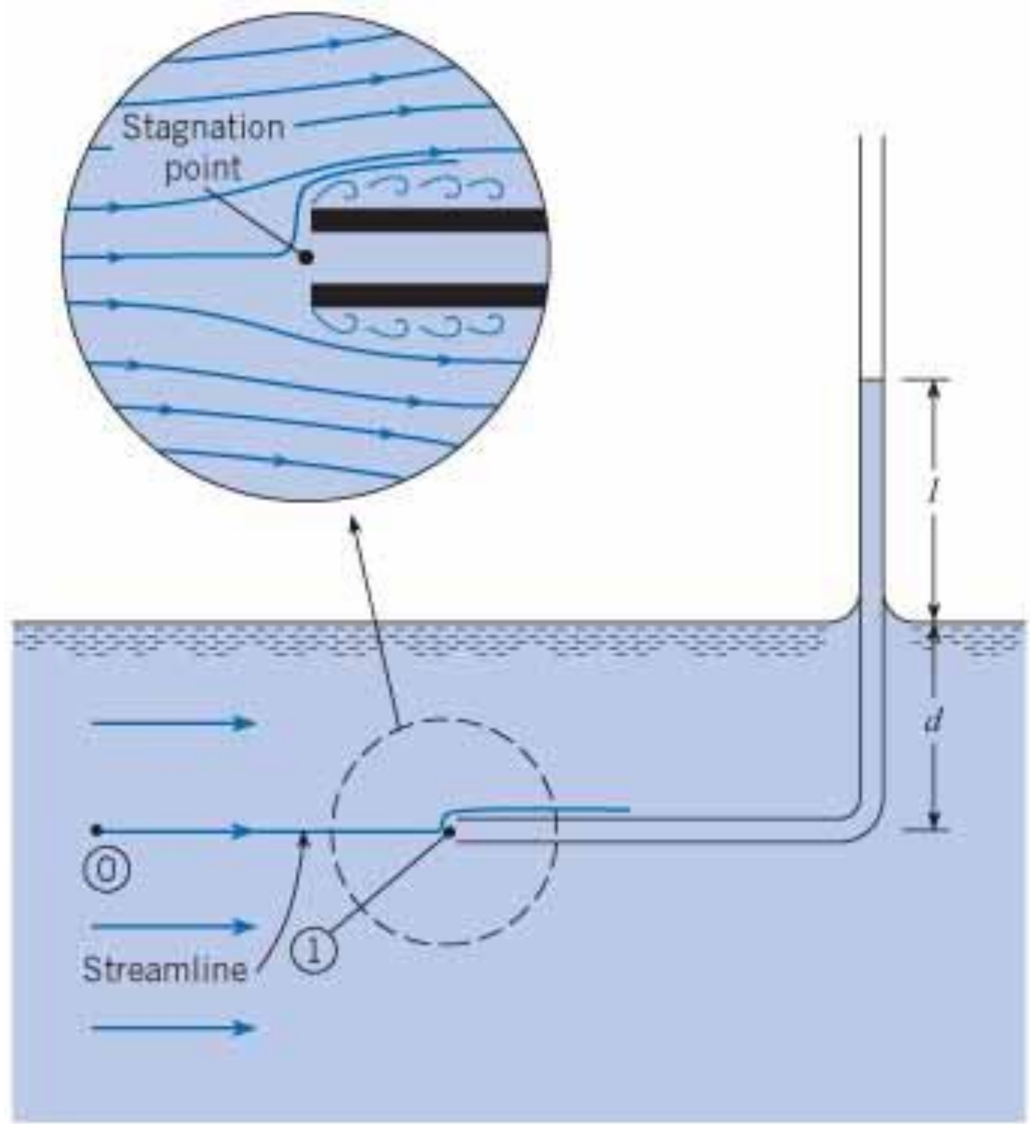


Figure 4.14 *Stagnation tube*

The velocity at point 1 is zero (stagnation point). Hence, Eq. (4.19) simplifies to

$$V_0^2 = \frac{2}{\rho}(p_1 - p_0)$$

By the equations of hydrostatics (there is no acceleration normal to the streamlines where the streamlines are straight and parallel), $p_0 = \gamma d$ and $p_1 = \gamma(l + d)$. Therefore,

$$V_0^2 = \frac{2}{\rho}(\gamma(l + d) - \gamma d)$$

which reduces to

$$V_0 = \sqrt{2gl}$$

This equation will be referred to as the *stagnation tube equation*. Thus it is seen that a very simple device such as this curved tube can be used to measure the velocity of flow.

Pitot-Static Tube

The *Pitot-static tube*, named after the eighteenth-century French hydraulic engineer who invented it, is based on the same principle as the stagnation tube, but it is much more versatile than the stagnation tube. The Pitot-static tube, shown in Fig. 4.15, has a pressure tap at the upstream end of the tube for sensing the stagnation pressure. There are also ports located several tube diameters downstream of the front end of the tube for sensing the static pressure in the fluid where the velocity is essentially the same as the approach velocity. When the Bernoulli equation is applied between points 1 and 2 along the streamline shown in Fig. 4.15, the result is

The result is

$$p_1 + \gamma z_1 + \frac{\rho V_1^2}{2} = p_2 + \gamma z_2 + \frac{\rho V_2^2}{2}$$

But $V_1 = 0$, so solving that equation for V_2 gives the *Pitot-static tube equation*

$$V_2 = \left[\frac{2}{\rho}(p_{z,1} - p_{z,2})\right]^{1/2}$$

Here $V_2 = V$, where V is the velocity of the stream and $p_{z,1}$ and $p_{z,2}$ are the piezometric pressures at points 1 and 2, respectively.

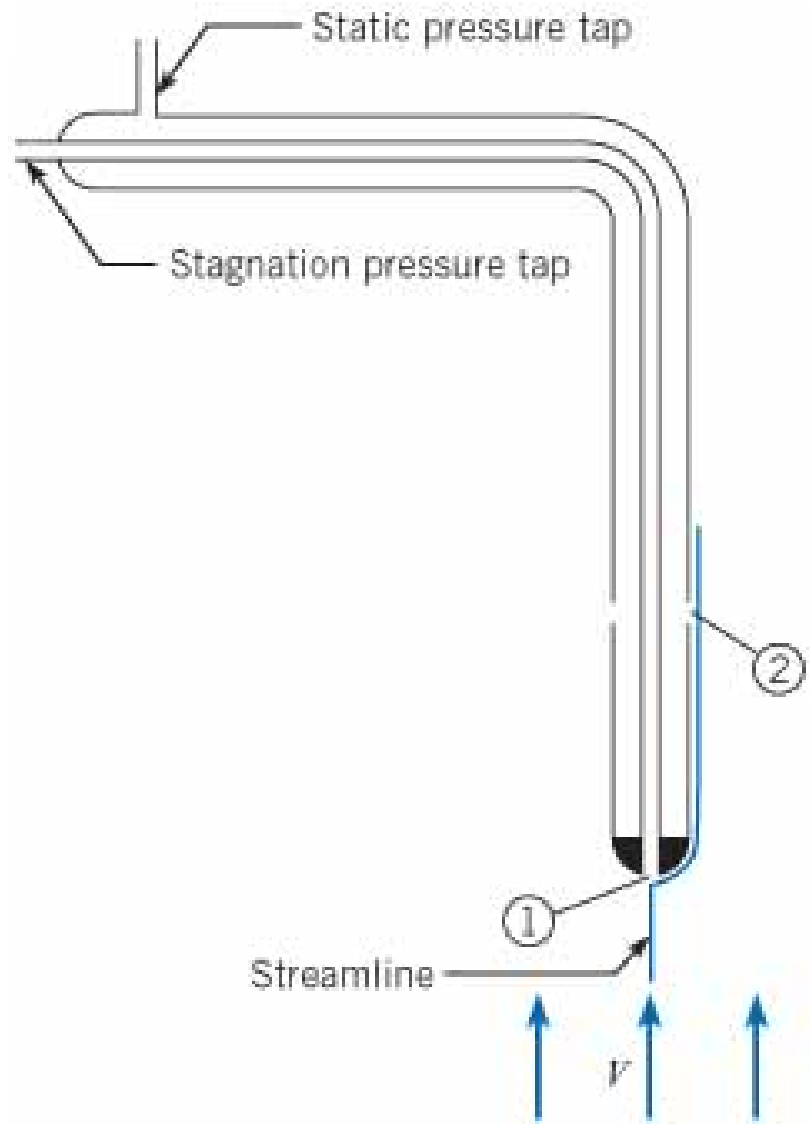


Figure 4.15 *Pitot-static tube.*

By connecting a pressure gage or manometer between the pressure taps shown in Fig. 4.15 one can easily measure the flow velocity with the Pitot-static tube. A major advantage of the Pitot-static tube is that it can be used to measure velocity in a pressurized pipe.

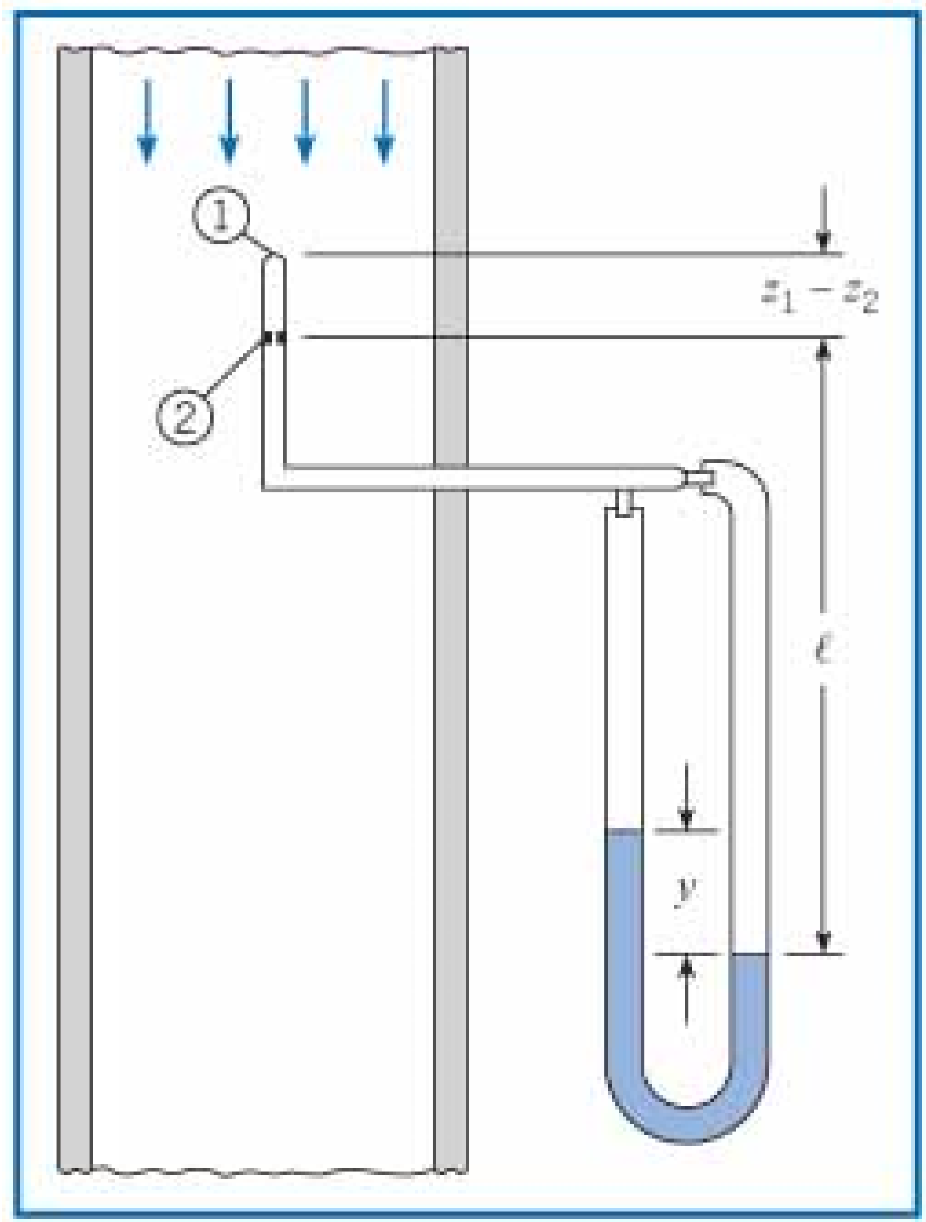
If a differential pressure gage is connected across the taps, the gage measures the difference in piezometric pressure directly. Therefore Eq. (4.22) simplifies to

$$V = \sqrt{2\Delta p / \rho}$$

where Δp is the pressure difference measured by the gage.

EXAMPLE 4.8 APPLICATION OF PITOT EQUATION WITH MANOMETER

A mercury manometer is connected to the Pitot-static tube in a pipe transporting kerosene as shown. If the deflection on the manometer is 7 in., what is the kerosene velocity in the pipe? Assume that the specific gravity of the kerosene is 0.81.



Solution

1. Manometer equation between points 1 and 2 on Pitot-static tube:

$$p_1 + (z_1 - z_2)\gamma_{\text{kero}} + \ell\gamma_{\text{kero}} - y\gamma_{\text{Hg}} - (\ell - y)\gamma_{\text{kero}} = p_2$$

Or

$$p_1 + \gamma_{\text{kero}} z_1 - (p_2 + \gamma_{\text{kero}} z_2) = y(\gamma_{\text{Hg}} - \gamma_{\text{kero}})$$

$$p_{z,1} - p_{z,2} = y(\gamma_{\text{Hg}} - \gamma_{\text{kero}})$$

2. Substitution into the Pitot-static tube equation:

$$V = \left[\frac{2}{\rho_{\text{kero}}} y(\gamma_{\text{Hg}} - \gamma_{\text{kero}})\right]^{1/2}$$

$$= \left[2gy\left(\frac{\gamma_{\text{Hg}}}{\gamma_{\text{kero}}} - 1\right)\right]^{1/2}$$

3. Velocity evaluation:

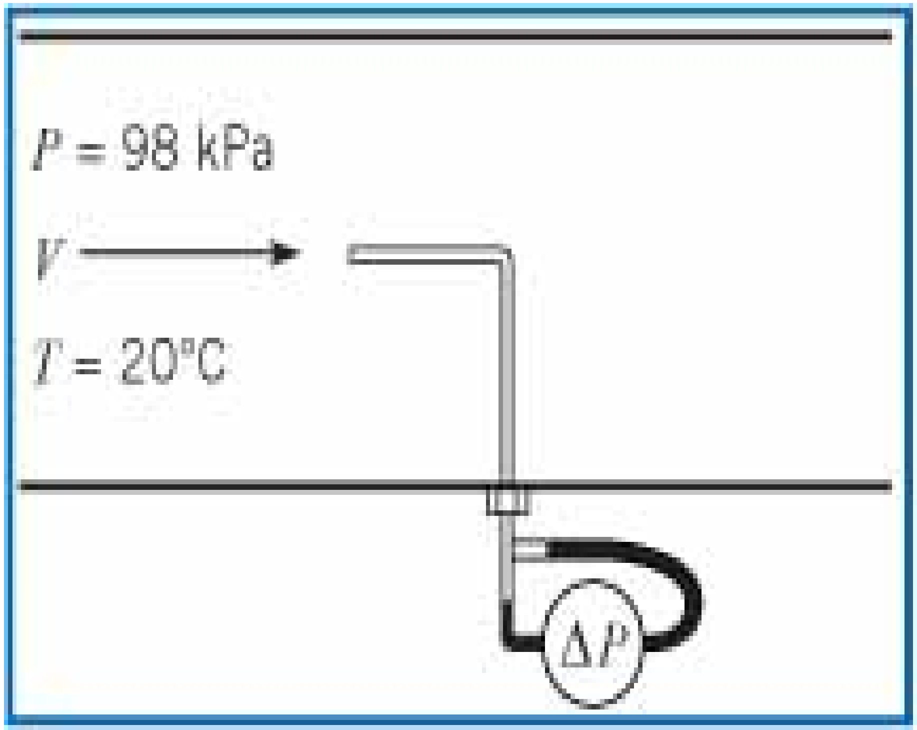
$$V = \left[2 \times 32.2 \text{ ft/s}^2 \times \frac{7}{12} \text{ ft} \left(\frac{13.55}{0.81} - 1\right)\right]^{1/2}$$

$$= \left[2 \times 32.2 \times \frac{7}{12}(16.7 - 1)\text{ft}^2/s^2\right]^{1/2}$$

$$= \boxed{24.3 \text{ ft/s}}$$

EXAMPLE 4.9 PITOT TUBE APPLICATION WITH PRESSURE GAGE

A differential pressure gage is connected across the taps of a Pitot-static tube. When this Pitot-static tube is used in a wind tunnel test, the gage indicates a Δp of 730 Pa. What is the air velocity in the tunnel? The pressure and temperature in the tunnel are 98 kPa absolute and 20°C, respectively.



Solution

1. Density calculation::

$$\rho = \frac{p}{RT}$$

$$= \frac{98 \times 10^3 \text{ N/m}^2}{(287 \text{ J/kg K}) \times (20 + 273 \text{ K})}$$

$$= 1.17 \text{ kg/m}^3$$

2. Pitot-static tube equation with differential pressure gage:

$$V = \sqrt{2\Delta p / \rho}$$

$$V = \sqrt{(2 \times 730 \text{ N/m}^2)/(1.17 \text{ kg/m}^3)} = \boxed{35.3 \text{ m/s}}$$

Application of the Bernoulli Equation to Flow of Gases

In the flow of gases, the contribution of pressure change due to elevation difference is generally very small compared with the change in kinetic pressure. Thus it is reasonable when applying the Bernoulli equation to gas flow (such as air) to use the simpler formulation

$$p + \frac{1}{2}\rho V^2 = C$$

Applicability of the Bernoulli Equation to Rotating Flows

The Bernoulli equation relates pressure, elevation, and kinetic pressure along streamlines in steady, incompressible flows where viscous effects are negligible. The question arises as to whether it can be used across streamlines; that is, could it be applied between two points on adjacent streamlines? The answer is provided by the form of the equation for pressure variation in a rotating flow, where the equation can be written as

$$p + \gamma z - \frac{1}{2}\rho V^2 = C$$

where ωr has been replaced by the velocity, V. Obviously the sign on the kinetic pressure term is different than the Bernoulli equation, so the Bernoulli equation does not apply across streamlines in a rotating flow.

In the next section the concept of flow rotation is introduced. There is a situation in which flows have concentric, circular streamlines and yet the fluid elements do not rotate. In this "irrotational" flow, the Bernoulli equation is applicable across streamlines as well a along streamlines.

4.6 Rotation and Vorticity

Concept of Rotation

The idea of fluid rotation is clear when a fluid rotates as a solid body. However, in other flow configurations it may not be so obvious. Consider fluid flow between two horizontal plates, Fig. 4.16, the bottom plate is stationary and the top is moving to the right with a velocity *V.* The velocity is linear; therefore, an element of fluid will deform as shown. Here it is seen that the element face that was initially vertical rotates clockwise, whereas the horizontal face does not. Is this a case of rotational motion?

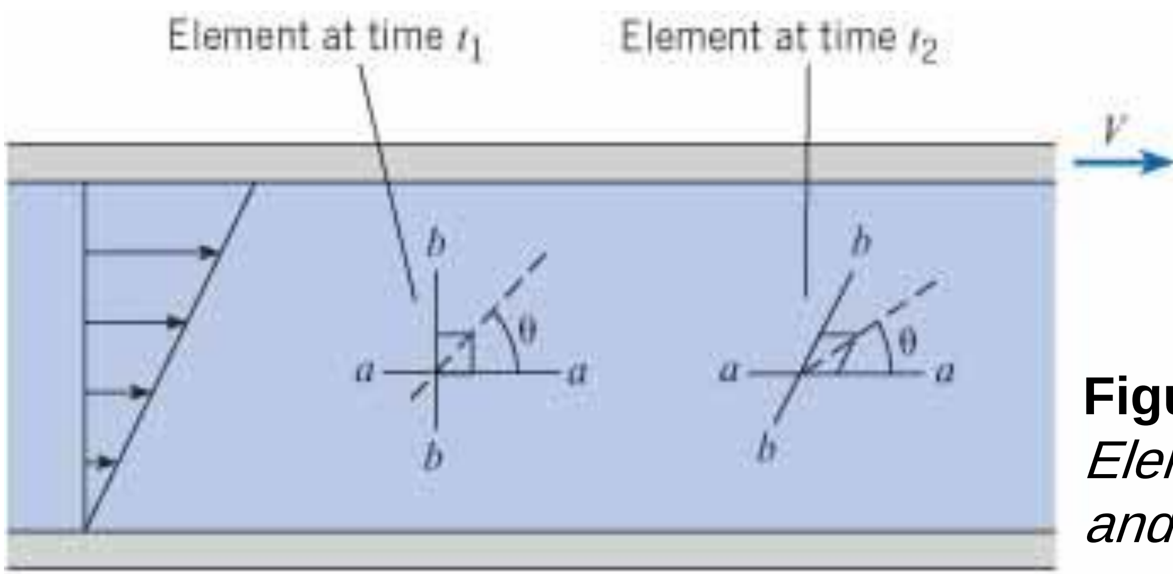


Figure 4.16 *Rotation of a fluid Element in flow between a moving and stationary parallel plate.*

Rotation is defined as the average rotation of two initially mutually perpendicular faces of a fluid element. The test is to look at the rotation of the line that bisects both faces (*a-a* and *b-b* in Fig. 4.16). The angle between this line and the horizontal axis is the rotation, θ.

The general relationship between θ and the angles defining the sides is shown in Fig. 4.17, where θ_A is the angle of one side with the *x*-axis and the angle θ_B is the angle of the other side with the *y*-axis. The angle between the sides is

$$\beta = \frac{\pi}{2} + \theta_B - \theta_A$$

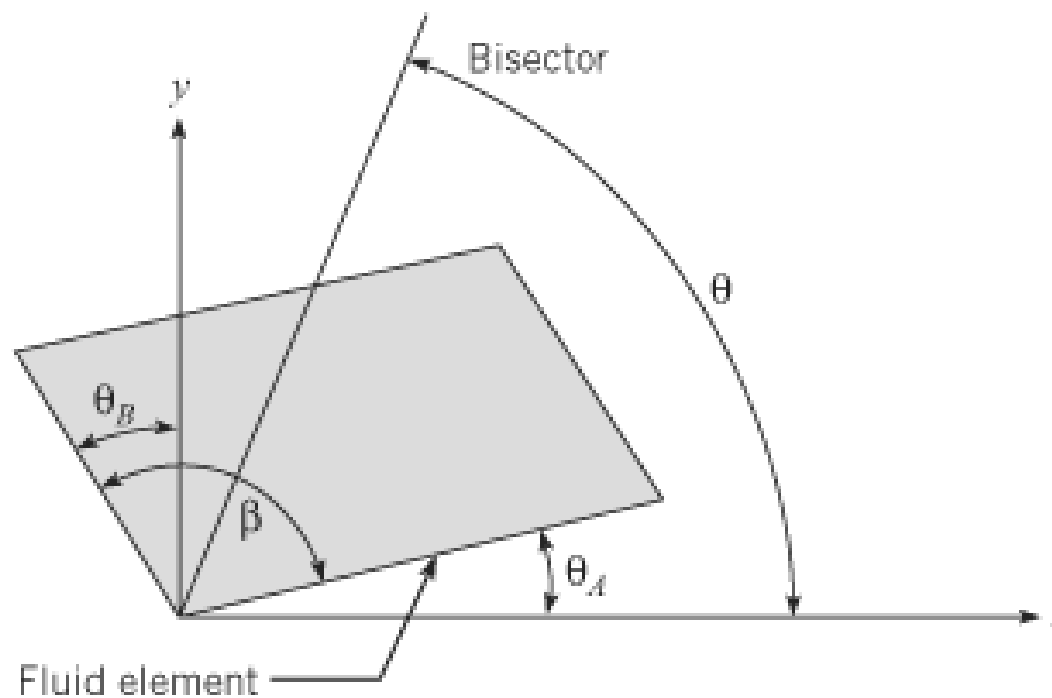


Figure 4.17 ***Orientation of*** *rotated fluid element.*

so the orientation of the element (bisector) with respect to the *x*-axis is

$$\theta = \frac{1}{2}\beta + \theta_A = \frac{\pi}{4} + \frac{1}{2}(\theta_A + \theta_B)$$

The rotational rate of the element is,

$$\dot{\theta} = \frac{1}{2}(\dot{\theta}_A + \dot{\theta}_B)$$

If $\dot{\theta} = 0$, the flow is *irrotational.*

An expression will now be derived that will give the rate of rotation of the bisector in terms of the velocity gradients in the flow. Consider the element shown in Fig. 4.18. The sides of the element are initially perpendicular with lengths Δx and Δy. Then the element moves with time and deforms as shown with point 0 going to 0′, point 1 to 1′, and point 2 to 2′. After time Δt the horizontal side has rotated counterclockwise by $\Delta\theta_A$ and the vertical side clockwise (negative direction) by $-\Delta\theta_B$.

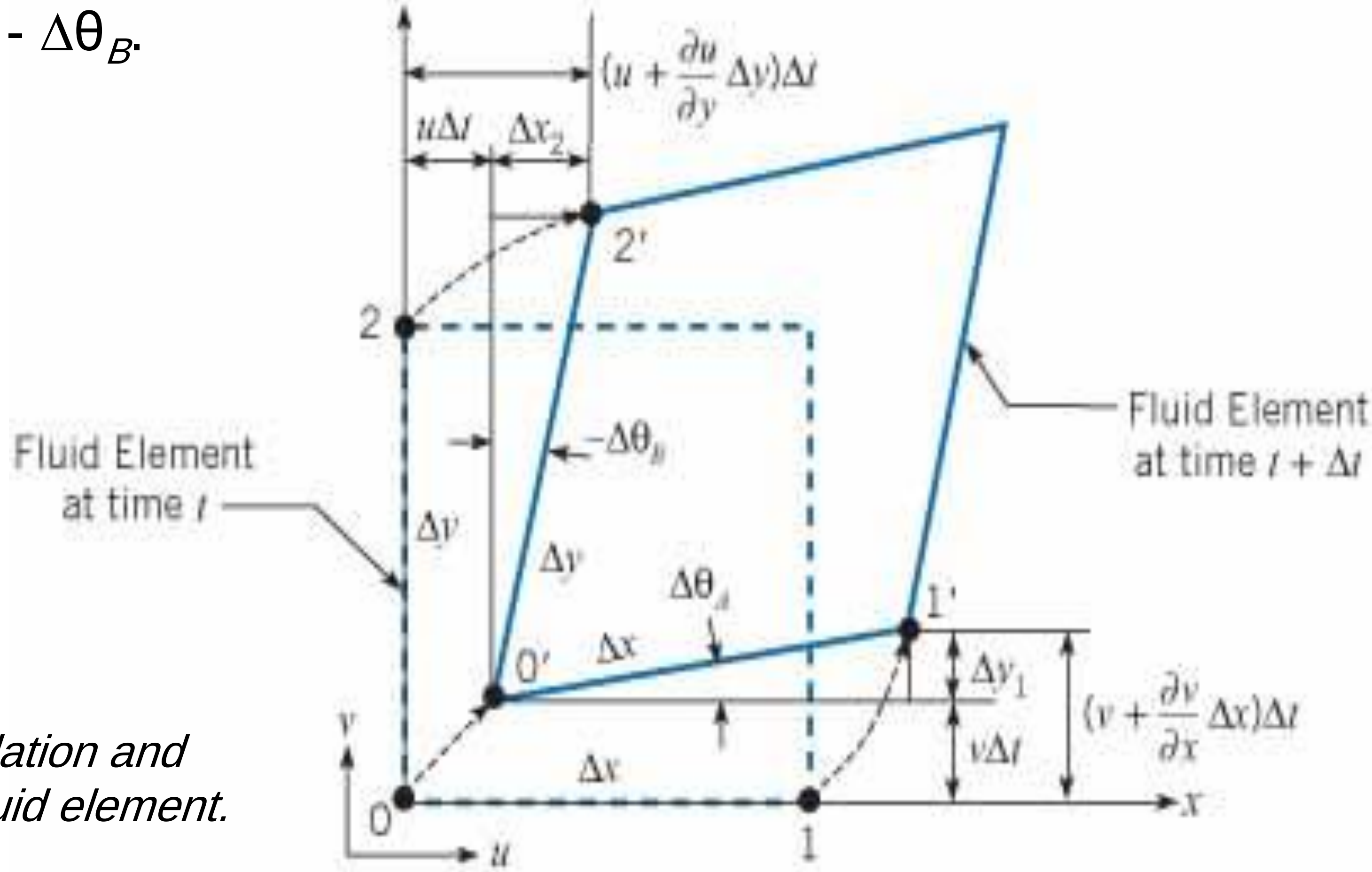


Figure 4.18 *Translation and deformation of a fluid element.*

The y velocity component of point 1 is v + (∂v/∂x)6x, and the x component of point 2 is u + (∂u/∂y)6y. The net displacements of points 1 and 2 are:

$$\Delta y_1 \sim \left[\left(v + \frac{\partial v}{\partial x}\Delta x\right)\Delta t - v\Delta t\right] = \frac{\partial v}{\partial x}\Delta x \Delta t$$

$$\Delta x_2 \sim \left[\left(u + \frac{\partial u}{\partial y}\Delta y\right)\Delta t - u\Delta t\right] = \frac{\partial u}{\partial y}\Delta y \Delta t$$

Referring to Fig. 4.18, the angles $\Delta\theta_A$ and $\Delta\theta_B$ are given by:

$$\Delta\theta_A = \operatorname{asin}\left(\frac{\Delta y_1}{\Delta x}\right) \sim \frac{\Delta y_1}{\Delta x} \sim \frac{\partial v}{\partial x}\Delta t \quad \text{for small angles}$$

$$-\Delta\theta_B = \operatorname{asin}\left(\frac{\Delta x_2}{\Delta y}\right) \sim \frac{\Delta x_2}{\Delta y} \sim \frac{\partial u}{\partial y}\Delta t \quad \text{for small angles}$$

Dividing the angles by 6*t* and taking the limit as 6$t \rightarrow 0$,

$$\dot{\theta}_A = \lim_{\Delta t \to 0} \frac{\Delta \theta_A}{\Delta t} = \frac{\partial v}{\partial x}$$

$$\dot{\theta}_B = \lim_{\Delta t \to 0} \frac{\Delta \theta_B}{\Delta t} = \frac{\partial u}{\partial y}$$

Substituting these results into Eq. (4.24) gives the rotational rate of the element about the *z*-axis (normal to the page),

$$\dot{\theta} = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

This component of rotational velocity is defined as Ω_z, so

$$\Omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

Likewise, the rotation rates about the other axes are

$$\Omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) \quad \text{and} \quad \Omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

Note rotation is a vector, thus

$$\boldsymbol{\Omega} = \Omega_x \mathbf{i} + \Omega_y \mathbf{j} + \Omega_z \mathbf{k}$$

An irrotational flow (Ω = 0) requires that all its components equal to zero, therefore,

$$\frac{\partial v}{\partial x} = \frac{\partial u}{\partial y} \quad \text{and} \quad \frac{\partial w}{\partial y} = \frac{\partial v}{\partial z} \quad \text{and} \quad \frac{\partial u}{\partial z} = \frac{\partial w}{\partial x}$$

The most extensive application of these equations is in ideal flow theory. An ideal flow is the flow of an irrotational, incompressible fluid. Flow fields in which viscous effects are small can often be regarded as irrotational. In fact, if a flow of an incompressible, inviscid fluid is initially irrotational, it will remain irrotational.

Vorticity

Another property used frequently in fluid mechanics is *vorticity,* which is a vector equal to twice the rate-of rotation vector. The magnitude of the vorticity indicates the rotationality of a flow and is very important in flows where viscous effects dominate, such as boundary layer, separated, and wake flows. The vorticity equation is

$$\begin{aligned}\omega &= 2\Omega \\ &= \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\mathbf{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\mathbf{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\mathbf{k} \\ &= \nabla \times V\end{aligned}$$

where $\nabla \times \mathbf{V}$, from vector calculus means the curl of the vector ***V***.

An irrotational flow signifies that the vorticity vector is everywhere zero.

EXAMPLE 4.10 EVALUATION OF ROTATION OF VELOCITY FIELD

The vector $\boldsymbol{V} = 10x\mathbf{i} - 10y\mathbf{j}$ represents a two-dimensional velocity field. Is the flow irrotational?

Solution

Velocity components and derivatives

$$u = 10x \qquad \frac{\partial u}{\partial y} = 0$$

$$v = -10y \qquad \frac{\partial v}{\partial x} = 0$$

Thus flow is irrotational.

EXAMPLE 4.11 ROTATION OF A FLUID ELEMENT

A fluid exists between stationary and moving parallel flat plates, and the velocity is linear as shown. The distance between the plates is 1 cm, and the upper plate moves at 2 cm/s. Find the amount of rotation that the fluid element located at 0.5 cm will undergo after it has traveled a distance of 1 cm.

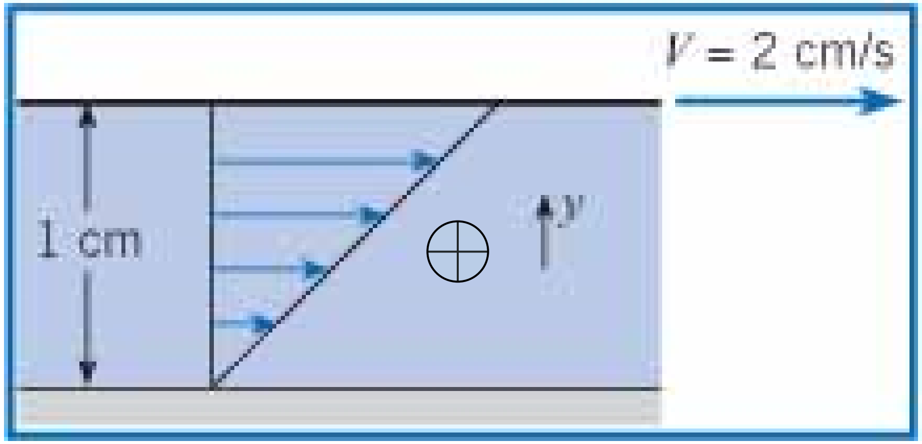
Solution

1. Velocity distribution: $u = 0.02 \text{ m/s} \times \frac{y}{0.01 \text{ m}} = 2y \ (1/\text{s})$

Rotational rate

$$\Omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = -1 \text{ rad/s}$$

2. Time to travel 1 cm:



$$u = 2\ (1/\text{s}) \times 0.005 \text{ m} = 0.01 \text{ m/s}$$

$$\Delta t = \frac{\Delta x}{u} = \frac{0.01 \text{ m}}{0.01 \text{ m/s}} = 1 \ s$$

3. Amount of rotation

$$\Delta\theta = \Omega_z \times \Delta t = -1 \times 1 = -1 \text{ rad}$$

Rotation in Flows with Concentric Streamlines

It is interesting to realize that a flow field rotating with circular streamlines can be irrotational; that is, the fluid elements do not rotate. Consider the two-dimensional flow field shown in Fig. 4.19. The circumferential velocity on the circular streamline is *V*, and its radius is *r*. The *z*-axis is perpendicular to the page. As before, the rotation of the element is quantified as before, which is

$$\dot{\theta} = \frac{1}{2}(\dot{\theta}_A + \dot{\theta}_B)$$

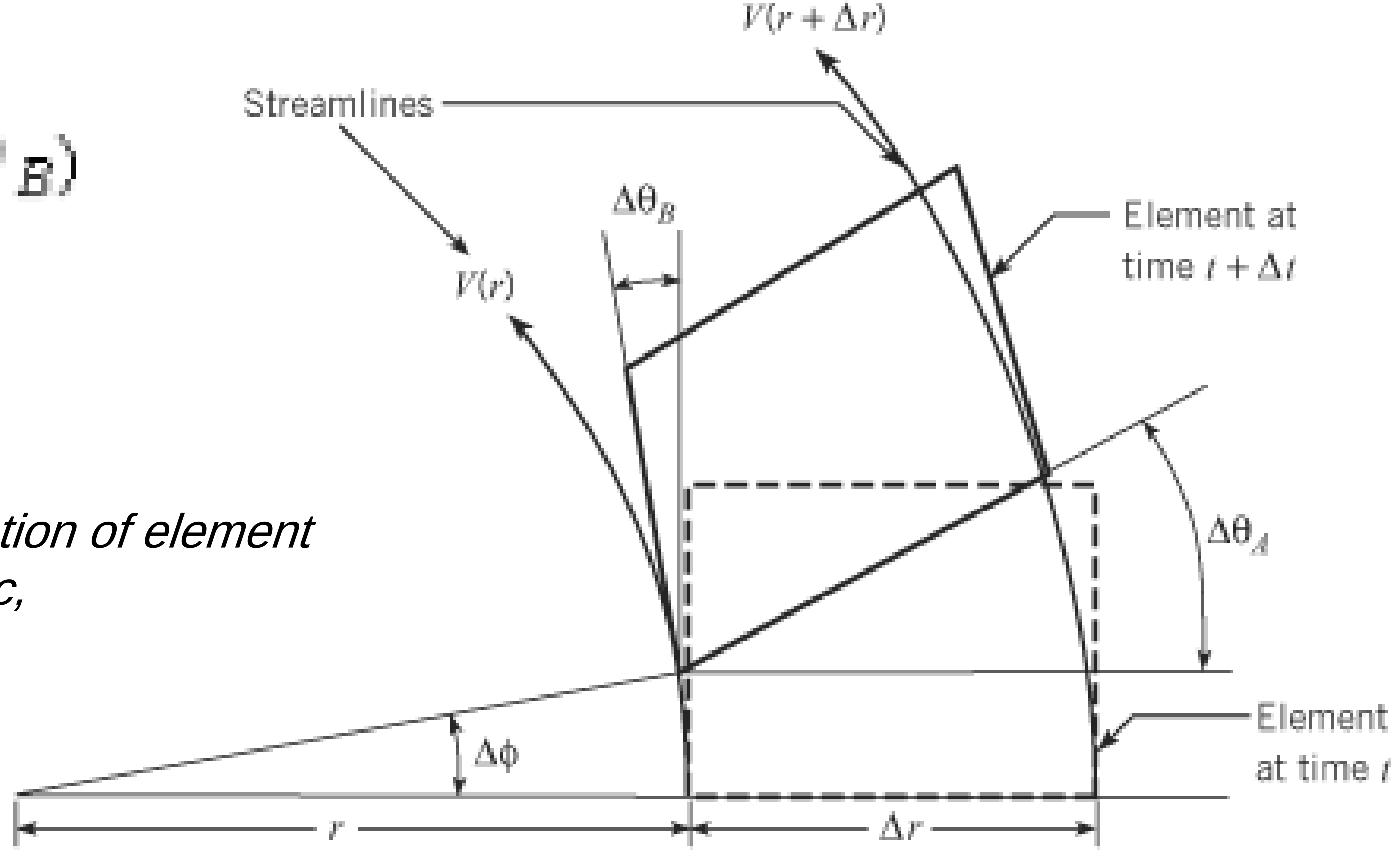


Figure 4.19 *Deformation of element in flow with concentric, circular streamlines.*

From geometry, the angle $\Delta\theta_B$ is equal to the angle $\Delta\varphi$. The rotational rate of angle φ is V/r, so

$$\dot{\theta}_B = \frac{V}{r}$$

Using the same analysis for $\dot{\theta}_A$, with r replacing x, yields

$$\dot{\theta}_A = \frac{\partial V}{\partial r}$$

Since V is a function of r only, the partial derivative can be replaced by the ordinary derivative. Therefore, the rotational rate about the z-axis is

$$\Omega_z = \frac{1}{2}\left(\frac{dV}{dr} + \frac{V}{r}\right)$$

As a check on this equation, apply it to a flow rotating as a solid body. The velocity distribution is $V = \omega r$, so the rate of rotation is

$$\Omega_z = \frac{1}{2}\left[\frac{d}{dr}(\omega r) + \omega\right] = \omega$$

as expected. This type of circular motion is called a "forced" vortex.

If the flow is irrotational, then

$$\frac{dV}{dr} = -\frac{V}{r}$$

or

$$\frac{dV}{V} = -\frac{dr}{r}$$

Integrating this equation leads to velocity distribution in this case,

$$V = \frac{C}{r}$$

where *C* is a constant. In this case, the circumferential velocity varies inversely with *r*, so the velocity decreases with increasing radius. This flow field is known as a “free” vortex. The fluid elements go around in circles, but do not rotate.

In a general flow there is both deformation and rotation. An ideal fluid is one that has no viscosity and is incompressible. If the flow of an ideal fluid is initially irrotational, it will remain irrotational. This is the foundation for many classical studies of flow fields in fluid mechanics.

4.7 TheBernoulli Equation in Irrotational Flow

Previously, the Bernoulli equation was developed for pressure variation between any two points along a streamline in steady flow with no viscous effects. In an **irrotational flow**, the Bernoulli equation **is not limited to flow along streamlines** but can be applied between any two points in the flow field. This feature of the Bernoulli equation is used extensively in classical hydrodynamics, the aerodynamics of lifting surfaces (wings), and atmospheric winds.

The Euler equation applied in the *n* direction (normal to the streamline) is

$$-\frac{d}{dn}(p + \gamma z) = \rho a_n \qquad (4.35)$$

where the partial derivative of *n* is replaced by the ordinary derivative because the flow is assumed steady (no time dependence). Two adjacent streamlines and the direction *n* is the same as *r* as shown in Fig. 4.21.

The local fluid speed is *V,* and the local radius of curvature of the streamline is *r.* The acceleration normal to the streamline is the centripetal acceleration, so

$$a_n = -\frac{V^2}{r}$$

where the negative sign occurs because the direction *n* is outward from the center of curvature and the centripetal acceleration is toward the center of curvature.

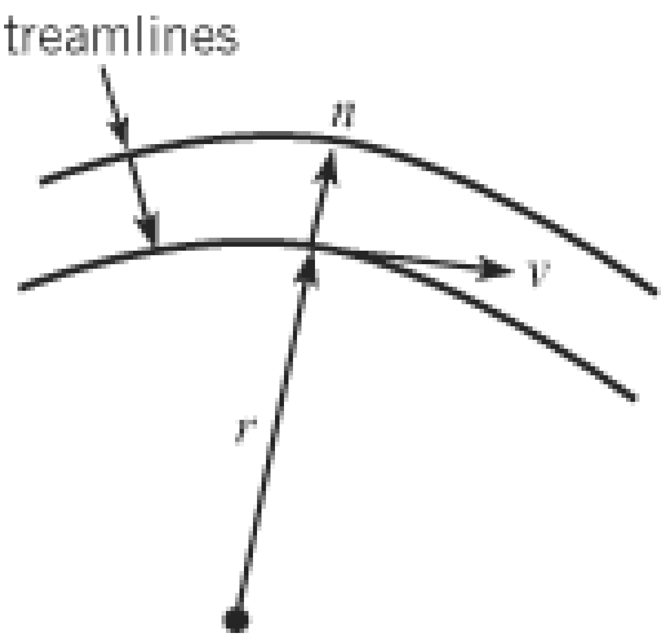


Figure 4.21 *Two adjacent streamlines showing direction in between lines*

Using the irrotationality condition, the acceleration can be written as

$$a_n = -\frac{V^2}{r} = -V\left(\frac{V}{r}\right) = V\frac{dV}{dr} = \frac{d}{dr}\left(\frac{V^2}{2}\right) \qquad (4.37)$$

Also the derivative with respect to *r* can be expressed as a derivative with respect to *n* by

$$\frac{d}{dr}\left(\frac{V^2}{2}\right) = \frac{d}{dn}\left(\frac{V^2}{2}\right)\frac{dn}{dr} = \frac{d}{dn}\left(\frac{V^2}{2}\right)$$

because the direction of *n* is the same as *r* so $dn/dr = 1$. Equation (4.37) can be rewritten as

$$a_n = \frac{d}{dn}\left(\frac{V^2}{2}\right)$$

Substituting the expression for acceleration into Euler's equation, Eq. (4.35), and assuming constant density results in

$$\frac{d}{dn}\left(p+\gamma z+\rho\frac{V^2}{2}\right)=0$$

Recognizing that the derivative of a constant is zero, implies that sum of the terms between the parentheses are zero, hence

$$p+\gamma z+\rho\frac{V^2}{2}=C$$

which is the Bernoulli equation, and *C* is constant in the *n* direction (across streamlines). Thus for an irrotational flow, the constant *C* in the Bernoulli equation is the same across streamlines as well as along streamlines, so it is the same everywhere in the flow field.

Equivalently, the sum of the piezometric head and velocity head

$$\frac{p}{\gamma} + z + \frac{V^2}{2g} = C$$

Thus, for icompressible, inviscid, and irrotaional then between any two points in the flow field,

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

EXAMPLE 4.12 VELOCITY AND PRESSURE DISTRIBUTION IN A FREE VORTEX

A free vortex in air rotates in a horizontal plane and has a velocity of 40 m/s at a radius of 4 km from the vortex center. Find the velocity at 10 km from the center and the pressure difference between the two locations. The air density is 1.2 kg/m^3.

Solution.

The velocity at location 10 km is

$$V = \frac{C}{r}$$

$$\frac{V_{10\text{km}}}{V_{4\text{km}}} = \frac{r_{4\text{km}}}{r_{10\text{km}}} = 0.4$$

$$V_{10\text{km}} = 0.4 \times 40$$

$$= \boxed{16 \text{ m/s}}$$

The pressure difference is obtained by applying the Bernoulli equation for a horizontal plane;

$$p_{4\text{km}} + \rho \frac{V_{4\text{km}}^2}{2} = p_{10\text{km}} + \rho \frac{V_{10\text{km}}^2}{2}$$

$$p_{10\text{km}} - p_{4\text{km}} = \frac{\rho}{2}(V_{4\text{km}}^2 - V_{10\text{km}}^2)$$

$$= \frac{1.2 \text{ kg/m}^3}{2}(40^2 - 16^2) \ (\text{m/s})^2$$

$$= \boxed{806 \text{ Pa}}$$

Pressure Variation in a Cyclonic Storm

A cyclonic storm is characterized by rotating winds with a low-pressure region in the center. Tornadoes and hurricanes are examples of cyclonic storms. A simple model for the flow field in a cyclonic storm is a forced vortex at the center surrounded by a free vortex, as shown in Fig. 4.22. This model is used in several applications of vortex flows. In practice, however, there will be no discontinuity in the slope of the velocity distribution as shown in Fig. 4.22, but rather a smooth transition between the inner forced vortex and the outer free vortex. Still, the model can be used to make reasonable predictions of the pressure field.

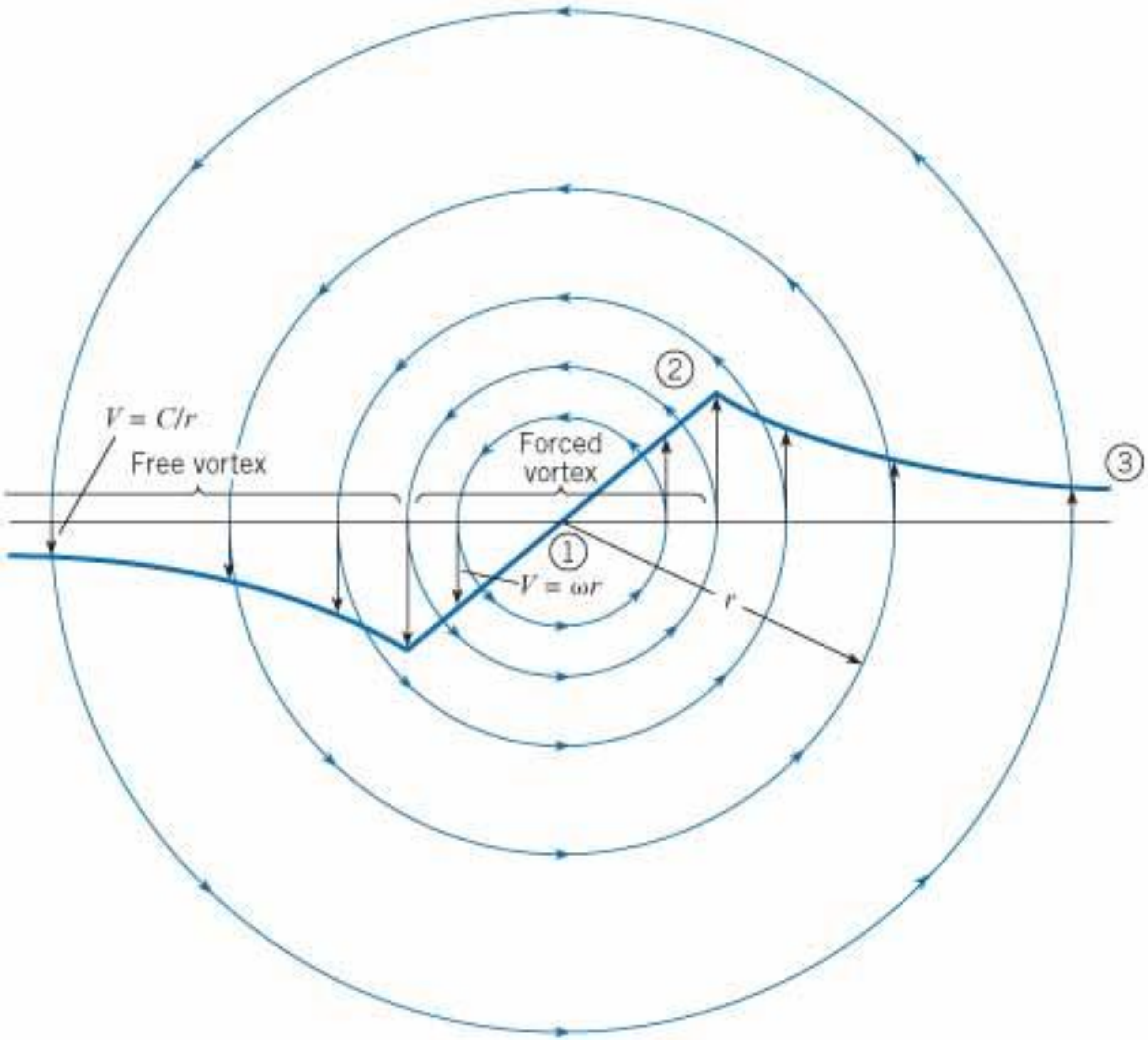


Figure 4.22 *Combination of forced and free vortex to model a cyclonic storm.*

The model for the cyclonic storm is an illustration of where the Bernoulli equation **can and cannot** be used across streamlines. The Bernoulli equation cannot be used across streamlines in the vortex at the center because the flow is rotational. The pressure distribution in the forced vortex is given by the rotating flow equation. The Bernoulli equation can be used across streamlines in the free vortex since the flow is irrotational.

Take point 1 as the center of the forced vortex and point 2 at the junction of the forced and free vortices, where the velocity is maximum. Let point 3 be at the extremity of the free vortex, where the velocity is essentially zero ($V_3=0$) and the pressure is atmospheric pressure p_0. Applying the Bernoulli equation between any arbitrary point in the free vortex and point 3, one can write

$$p+\gamma z+\rho\frac{V^2}{2}=p_3+\gamma z_3+\rho\frac{V_3^2}{2}$$

Neglecting any elevation change, setting $p_0 = p_3$, and taking V_3 as zero gives

$$p - p_0 = -\rho \frac{V^2}{2}$$

which shows that the pressure decreases toward the center. This decreasing pressure provides the centripetal force to keep the flow moving along circular streamlines. The pressure at point 2 is

$$p_2 - p_0 = -\rho \frac{V_{max}^2}{2}$$

Applying the equation for pressure variation in rotating flows in the forced vortex region yields

$$p + \gamma z - \rho \frac{\omega^2 r^2}{2} = p_2 + \gamma z_2 - \rho \frac{\omega^2 r_2^2}{2}$$

Neglect elevation change (air), at point 2, ωr_2 is the maximum speed V_{max}, and ωr is the speed of the fluid in the forced vortex. Solving for the pressure, one finds

$$p = p_2 - \rho \frac{V_{max}^2}{2} + \rho \frac{V^2}{2}$$

Substituting in the expression for p_2 from free vortex region gives,

$$p = p_0 - \rho V_{max}^2 + \rho \frac{V^2}{2}$$

This relates the pressure in the forced region to the outside pressure p_0. The difference between the center of the cyclonic storm where the speed is zero and the outer edge of the storm is

$$p_1 - p_0 = -\rho V_{max}^2$$

The Pressure Coefficient

Describing the pressure distribution is important because pressure gradients influence flow patterns and pressure distributions acting on bodies create resultant forces. A common dimensionless group for describing the pressure distribution is called the *pressure coefficient*:

$$C_p = \frac{p_z - p_{zo}}{\rho V_o^2 / 2} = \frac{h - h_o}{V_o^2 / (2g)}$$

where the subscript ($_0$) is some reference point.

EXAMPLE 4.13 PRESSURE DIFFERENCE IN TORNADO

Assume that a tornado is modeled as the combination of a forced and a free vortex. The maximum wind speed in the tornado is 240 km/h. What is the pressure difference, in centimeters of mercury, between the center and the outer edge of the tornado? The density of the air is 1.2 kg/m^3.

Solution

Using the equation developed earlier for cyclonic storms,

$$p_1 - p_0 = -\rho V_{max}^2$$

The velocity V_{max} = 66.67 m/s, hence

$$p_1 - p_0 = -1.2 \times 66.67^2$$
$$= -5333.3 \text{ Pa}$$

Converting the pressure difference to centimeters of Hg,

$$p_1 - p_0 = \gamma\Delta h, \; \gamma_{hg} = 133.4 \text{ kN/m}^3$$

$$\Delta h = -4.0 \text{ cm of mercury.}$$

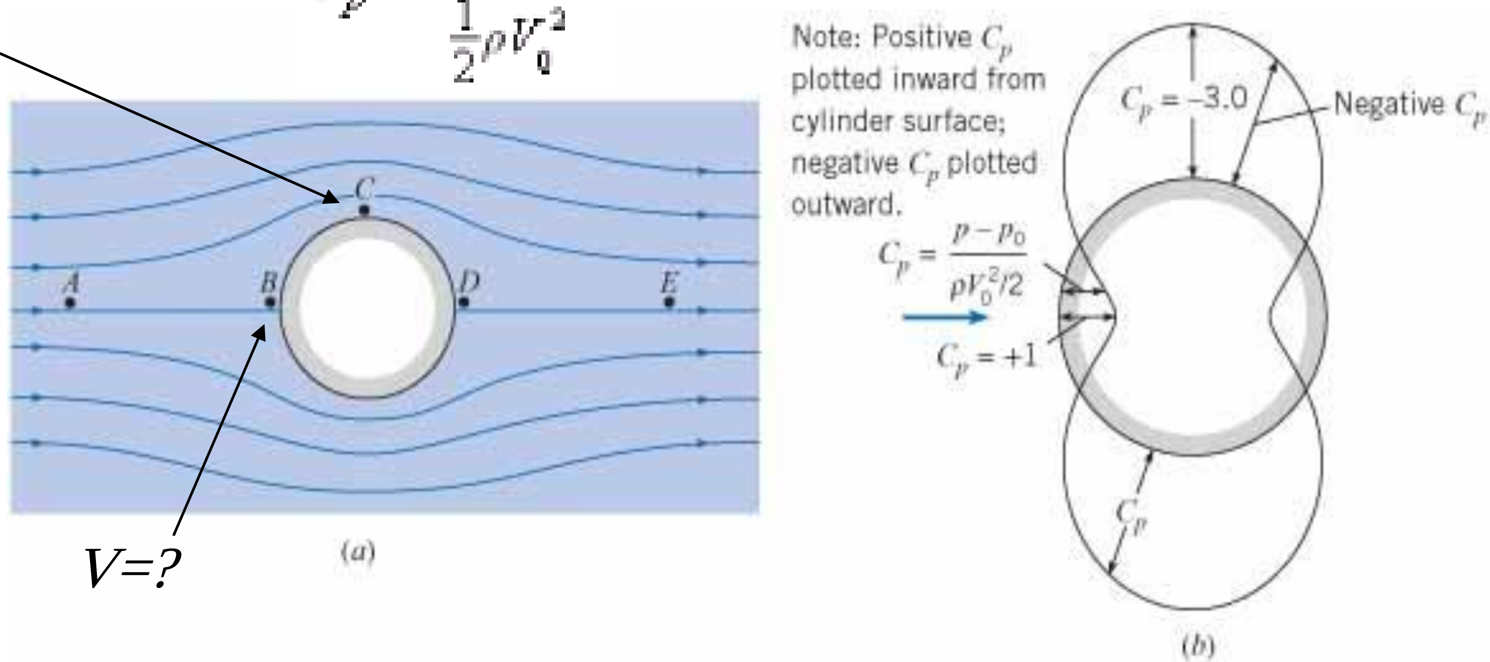
Pressure Distribution around a Circular Cylinder—Ideal Fluid

If a fluid is nonviscous and incompressible (an *ideal fluid*) and if the flow is initially irrotational, then the flow will be irrotational throughout the entire flow field. If the flow is also steady, the Bernoulli equation will apply everywhere because all the restrictions for the Bernoulli equation will have been satisfied. The flow pattern about a circular cylinder with such restrictions is shown in Fig. 4.24a.

Because the flow pattern is symmetrical with either the vertical or the horizontal axis through the center of the cylinder, the pressure distribution on the surface of the cylinder, obtained by application of the Bernoulli equation, is also symmetrical as shown in Fig. 4.24*b*. The pressure coefficient reduces to

$$C_p = \frac{p - p_0}{\frac{1}{2}\rho V_0^2}$$

If $V=2V_0$
C_p?



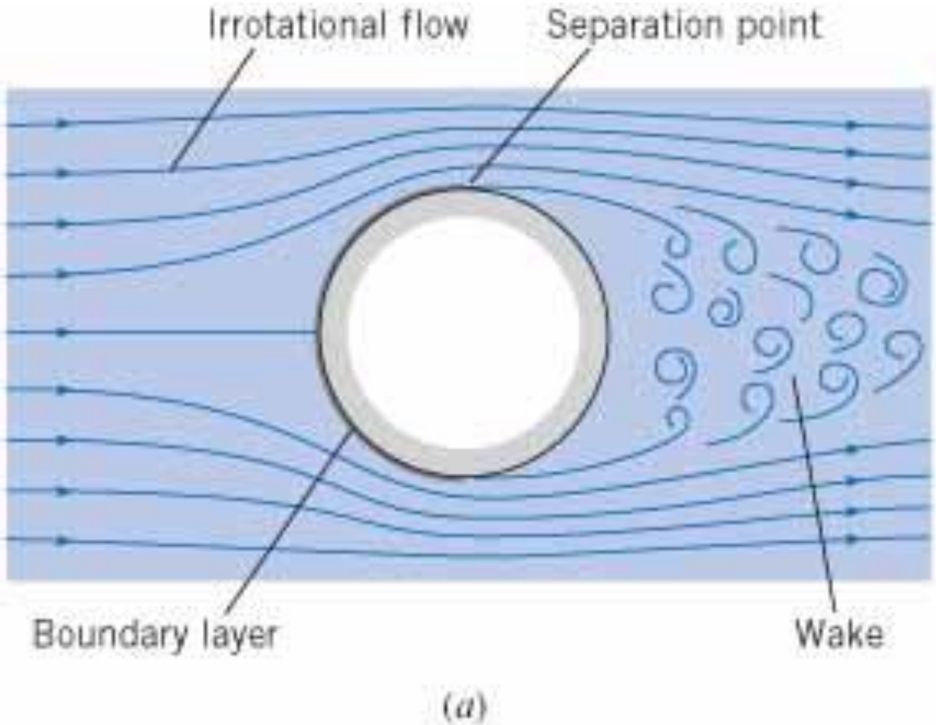
V=?

Figure 4.24
Irrotational flow past a cylinder. (a) Streamline pattern. (b) Pressure distribution.

4.8 Separation

Flow separation occurs when the fluid pathlines adjacent to body deviate from the contour of the body and produce a wake. This flow condition is very common. It tends to increase drag, reduce lift, and produce unsteady forces that can lead to structural failure.

Examples of flow seperation are shown for a cylinder, airfoil and a square rod



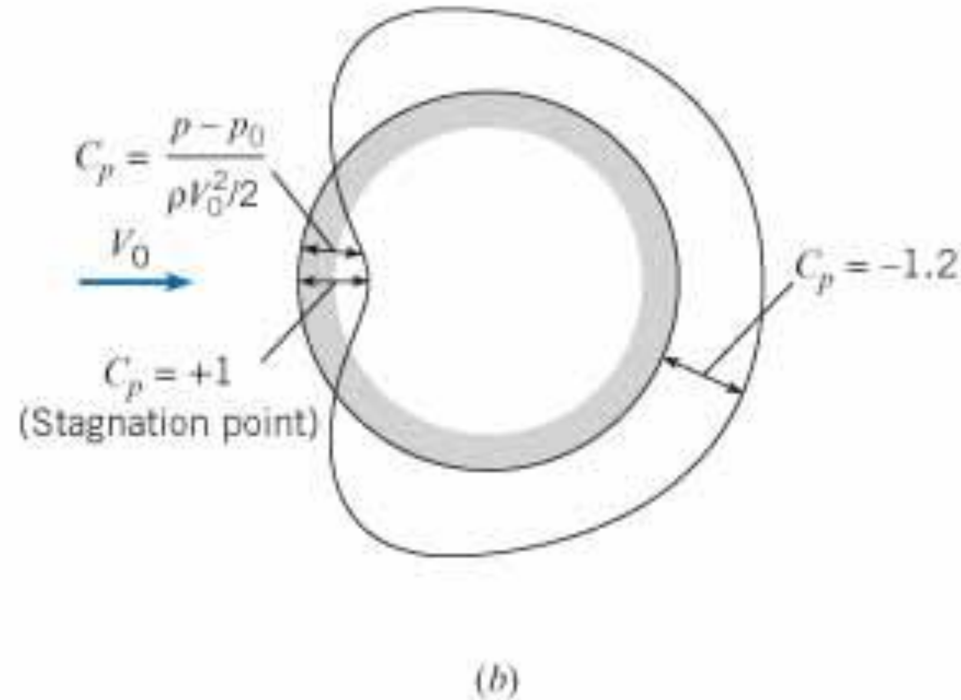


Figure 4.25 *Flow of a real fluid past a circular cylinder.*
(a) Flow pattern.
(b) Pressure distribution

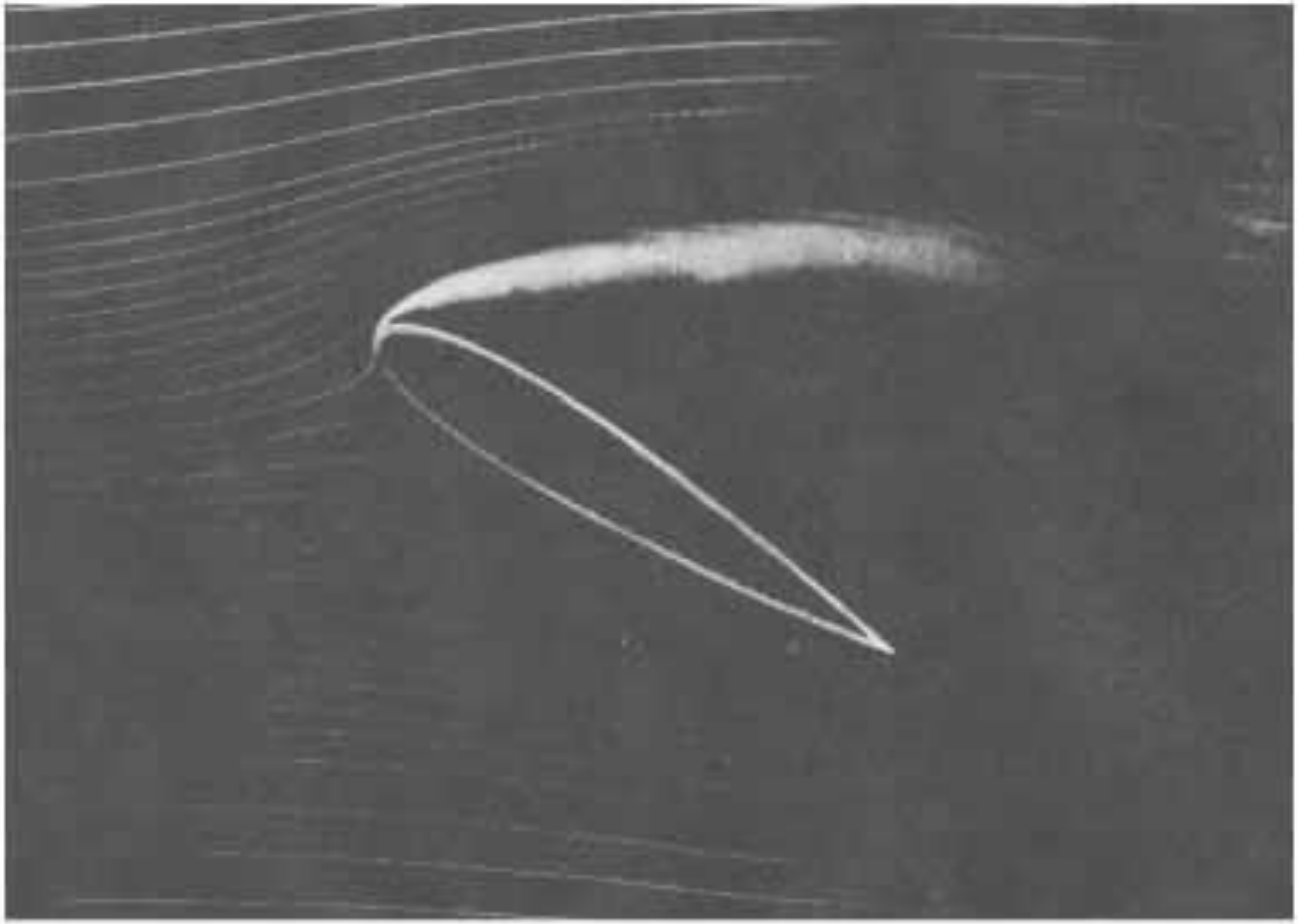


Figure 4.26 *Smoke traces showing separation on an airfoil section at a large angle of attack. (Courtesy of Education Development Center, Inc. Newton, MA)*

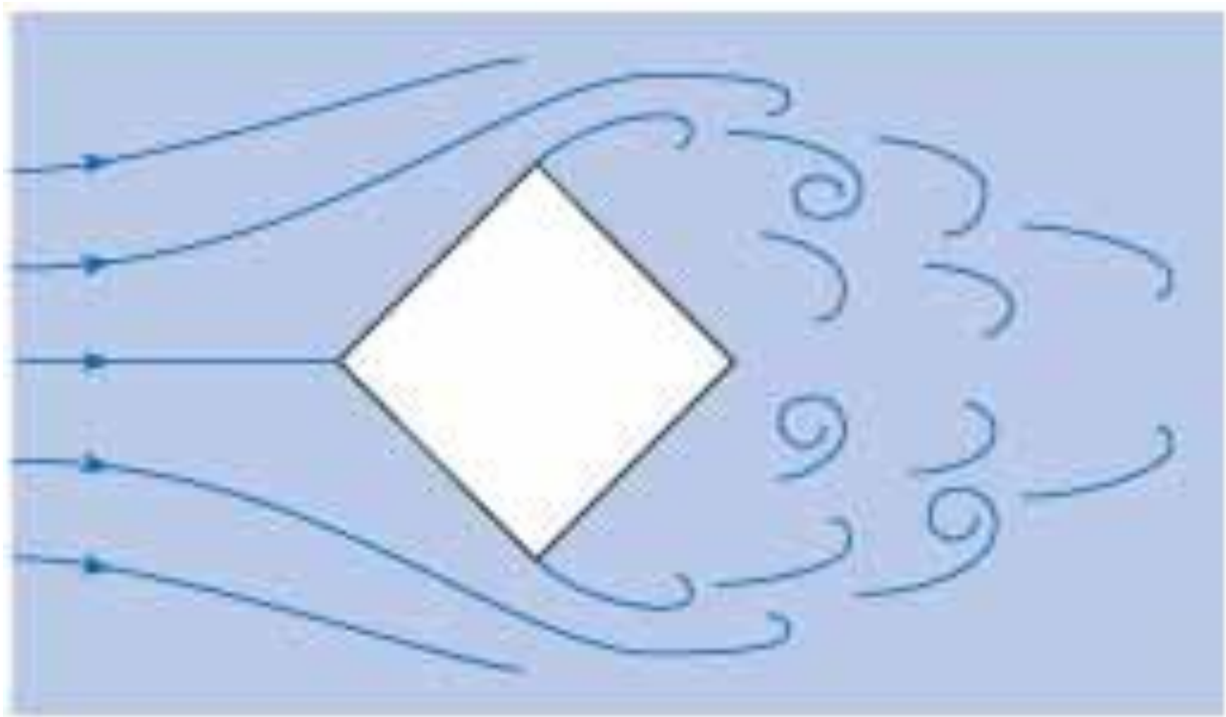
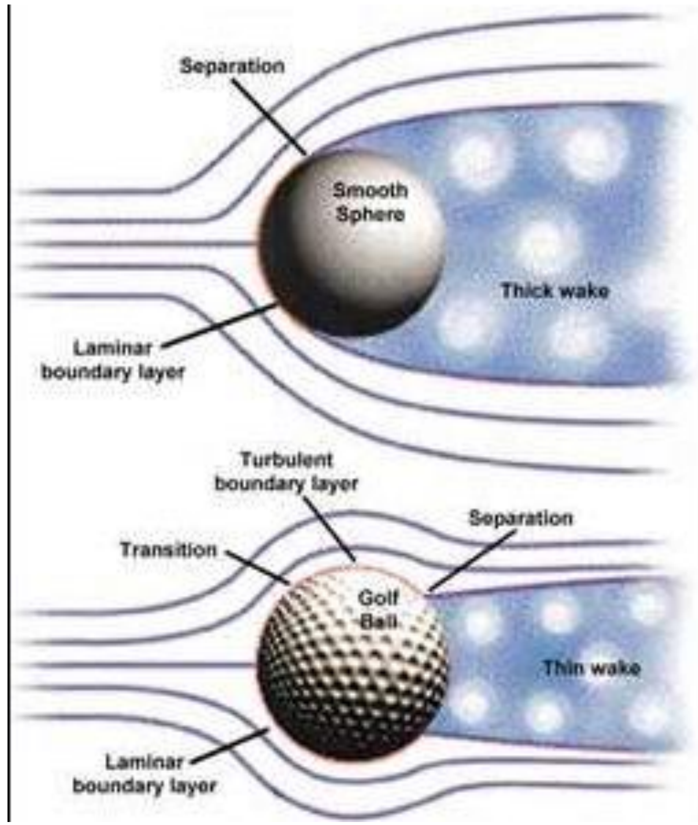


Figure 4.27 *Flow pattern past a square rod illustrating separation at the edges.*


Separation
Smooth
Sphere
Thick wake
Laminar
boundary layer
Turbulent
boundary layer
Separation
Transition
Golf
Ball
Thin wake
Laminar
boundary layer

Fluid Mechanics

Chapter 5
Control Volume Approach and Continuity Equation

Dr. Amer Khalil Ababneh

Approach to Analyses of Fluid flows

The engineer can find flow properties (pressure and velocity) in a flow field in one of two ways. One approach is to generate a series of pathlines or streamlines through the field and determine flow properties at any point along the lines by applying equations like those developed in Chapter 4. This is called the **Lagrangian approach**. The other way is to solve a set of equations for flow properties at any point in the flow field. This is called the **Eulerian approach** (also called control volume approach).

The foundational concepts for the Eulerian approach, or control volume approach, are developed and applied to the conservation of mass. This leads to the **continuity equation**, a fundamental and widely used equation in fluid mechanics.

Control volume is a region in space that allows mass to flow in and out of it.

5.1 Rate of Flow

It is necessary to be able to calculate the flow rates through a control volume. Also, the capability to calculate flow rates is important in analyzing water supply systems, natural gas distribution networks, and river flows.

Discharge

The *discharge*, *Q*, often called the *volume flow rate*, is the volume of fluid that passes through an area per unit time. For example, when filling the gas tank of an automobile, the discharge or volume flow rate would be the gallons per minute flowing through the nozzle. Typical units for discharge are ft^3/s (cfs), ft^3/min (cfm), gpm, m^3/s, and L/s.

To develop an equation for the discharge, consider a fluid flow with a uniform velocity flowing in a pipe, Figure 5.3. The volume of the fluid indicated that passes section A-A is during time interval Δt

$$\Delta\forall = A\,\Delta l = A\,(V\,\Delta t)$$

The volume flow rate is the volume indicated per unit time

$$\frac{\Delta\forall}{\Delta t} = VA$$

Taking the limit as $\Delta t \rightarrow 0$ gives

$$Q = \lim_{\Delta t \longrightarrow 0} \frac{\Delta\forall}{\Delta t} = VA$$

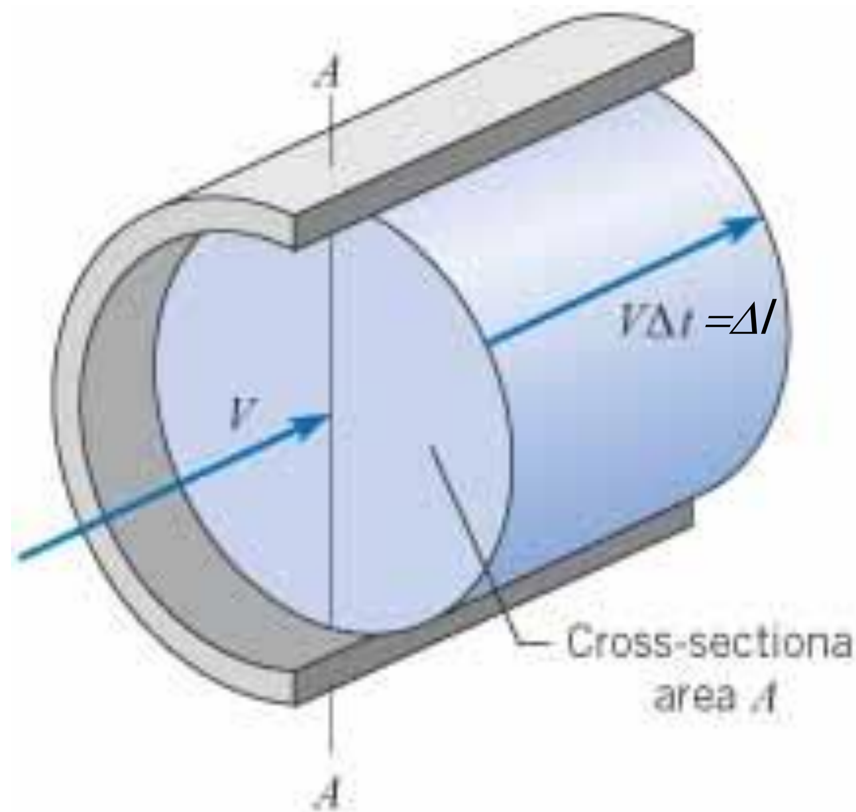


Figure 5.3 *Volume of fluid in flow with uniform velocity distribution that passes section A-A in time t.*

Volume flow rate is referred to as discharge and will be given the symbol *Q*.

In general the velocity is not uniform and in this case the discharge is given by

$$Q = \int_A V \, dA$$

The *mean velocity* is defined as the discharge divided by the cross-sectional area,

$$\bar{V} = \frac{Q}{A}$$

For laminar flows in circular pipes, the velocity profile is parabolic, and the mean velocity is half the centerline velocity. For turbulent pipe flow time-averaged velocity profile is nearly uniformly distributed across the pipe, so the mean velocity is fairly close to the velocity at the pipe center. It is customary to leave the bar off the velocity symbol and simply indicate the mean velocity with *V*.

In Fig. 5.5 the flow velocity vector is not normal to the surface but is oriented at an angle θ with respect to the direction that is normal to the surface. The only component of velocity that contributes to the flow through the differential area *dA* is the component normal to the area, *Vn*. The differential discharge through area *dA* is

$$dQ = V_n dA$$

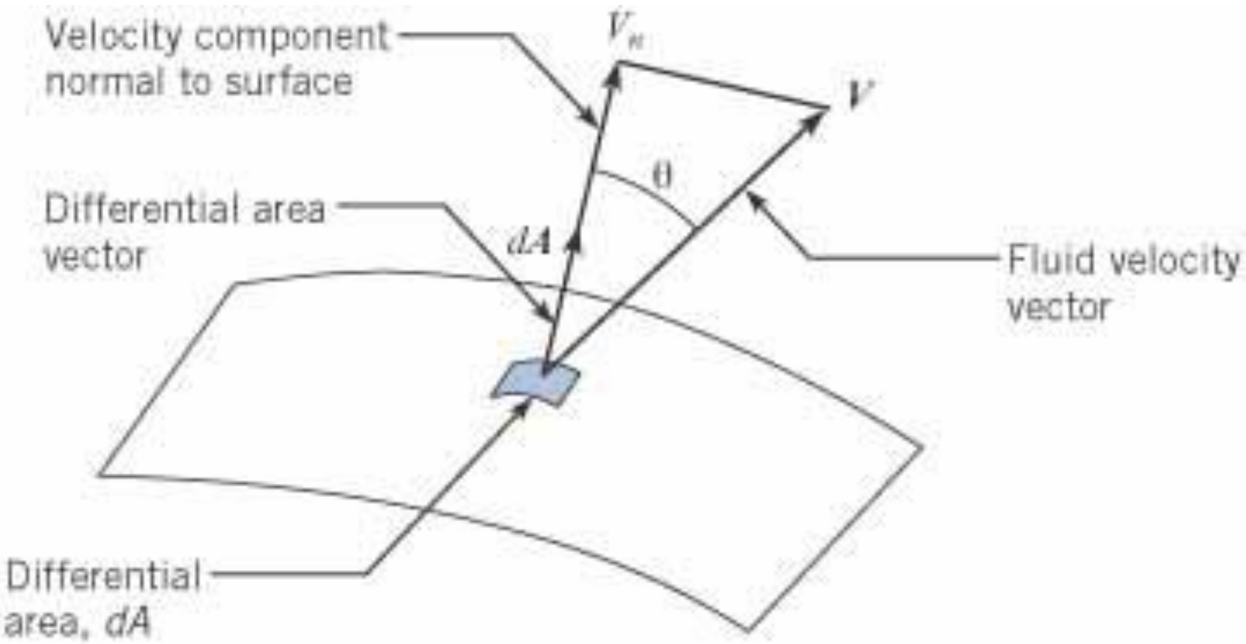


Figure 5.5 *Velocity vector oriented at angle θ with respect to normal.*

Note that the dA is a vector with magnitude and direction normal to the area. But since $V_n\, dA = abs(V) \cos(\theta)\, dA = \mathbf{V.dA}$, then in general, the discharge is

$$Q = \int_A V \cdot dA$$

If the velocity is uniform, the discharge is

$$Q = V \cdot A$$

Note, if the velocity is tangent to the surface area, then the discharge is zero.

Mass Flow Rate

The *mass flow rate*, $\dot{m}$, is the mass of fluid passing through a cross-sectional area per unit time. The common units for mass flow rate are kg/s, lbm/s, and slugs/s. Using the same approach as for volume flow rate, the mass of the fluid in the marked volume in Fig. 5.3 is $\Delta m = \rho \Delta \mathcal{V}$, where ρ is the average density. The *mass flow rate equation* is

$$\dot{m} = \lim_{\Delta t \to 0} \frac{\Delta m}{\Delta t} = \rho \lim_{\Delta t \to 0} \frac{\Delta \mathcal{V}}{\Delta t} = \rho Q$$
$$= \rho A V$$

As before in case of discharge, the general form of mass flow rate is

$$\dot{m} = \int_A \rho \mathbf{V} \cdot d\mathbf{A}$$

The mass flow rate also can be expressed in terms of the mean velocity as,

$$\dot{m} = \rho A \overline{V}$$

EXAMPLE 5.1 VOLUME FLOW RATE AND MEAN VELOCITY

Air that has a mass density of 1.24 kg/m^3 flows in a pipe with a diameter of 30 cm at a mass rate of flow of 3 kg/s. What are the mean velocity and discharge in this pipe?

Solution

1. Discharge:

$$Q = \frac{\dot{m}}{\rho} = \frac{3\ \mathrm{kg/s}}{1.24\ \mathrm{kg/m^3}} = \boxed{2.42\ \mathrm{m^3/s}}$$

2. Mean velocity

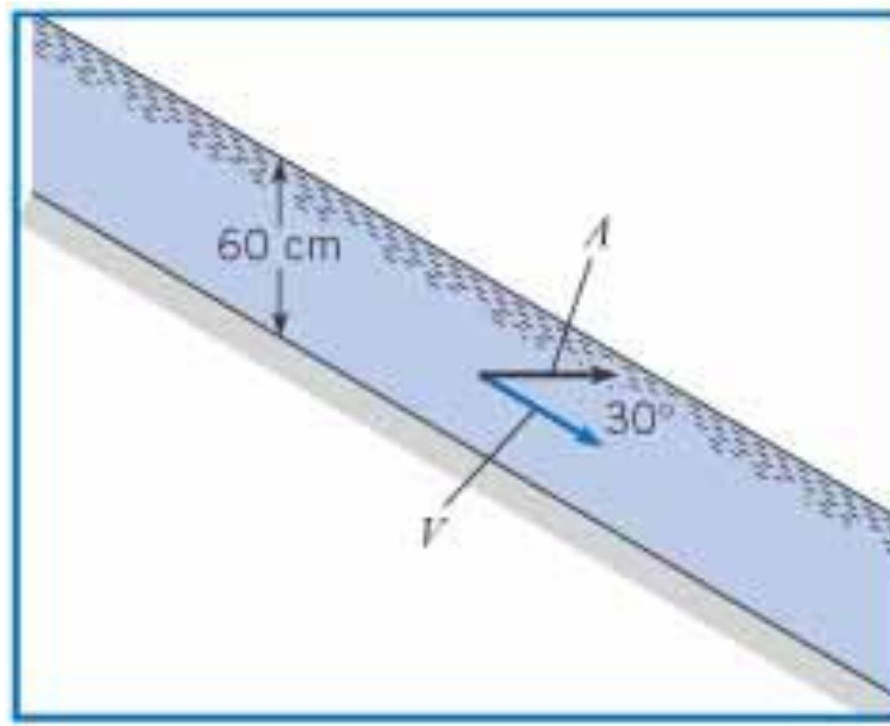
$$V = \frac{Q}{A} = \frac{2.42\ \mathrm{m^3/s}}{(\frac{1}{4}\pi) \times (0.30\ \mathrm{m})^2} = \boxed{34.2\ \mathrm{m/s}}$$

EXAMPLE 5.2 FLOW IN SLOPING CHANNEL

Water flows in a channel that has a slope of 30°. If the velocity is assumed to be constant, 12 m/s, and if a depth of 60 cm is measured along a vertical line, what is the discharge per meter of width of the channel?

Solution

$$
\begin{aligned}
Q &= V \cdot A = V \cos 30 \times A \\
&= 12 \text{ m/s} \times \cos 30^\circ \times 0.6 \text{ m} \\
&= \boxed{6.24 \text{ m}^3/\text{s per meter}}
\end{aligned}
$$



EXAMPLE 5.3 DISCHARGE IN CHANNEL WITH NON-UNIFORM VELOCITY DISTRIBUTION

The water velocity in the channel shown in the accompanying figure has a distribution across the vertical section equal to $u/u_{max} = (y/d)^{1/2}$. What is the discharge in the channel if the water is 2 m deep (d = 2 m), the channel is 5 m wide, and the maximum velocity is 3 m/s?

Solution

The discharge equation,

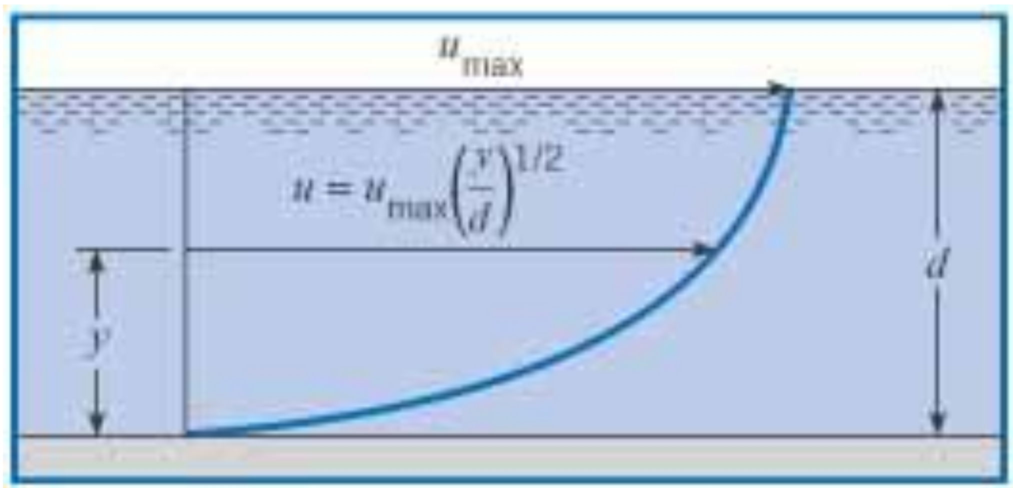
$$Q = \int_0^d u \, dA$$

$$Q = \int_0^2 u_{max}(y/d)^{1/2} 5 \, dy$$

$$= \frac{5u_{max}}{d^{1/2}} \int_0^2 y^{1/2} \, dy$$

$$= \frac{5u_{max}}{d^{1/2}} \frac{2}{3} y^{3/2} \Big|_0^2$$

$$= \frac{5 \times 3}{2^{1/2}} \times \frac{2}{3} \times 2^{3/2} = \boxed{20 \; m^3/s}$$



5.2 Control Volume Approach

The control volume (or Eulerian) approach is the method whereby a volume in the flow field is identified and the governing equations are solved for the flow properties associated with this volume. A scheme is needed that allows one to rewrite the equations for a moving fluid particle in terms of flow through a control volume. Such a scheme is the Reynolds transport theorem.

Definition: System and Control Volume

A *system* is a continuous mass of fluid that always contains the same matter. A system moving through a flow field is shown in Fig. 5.6. The shape of the system may change with time, but the mass is constant since it always consists of the same matter. The fundamental equations, such as Newton's second law and the first law of thermodynamics, apply to a system.

A *control volume* is volume located in **space** and through which **matter can pass**, as shown in Fig. 5.6. The indicated system can pass through the control volume. The selection of the control volume position and shape is problem dependent. The control volume is enclosed by the control surface as shown in Fig. 5.6. Fluid mass enters and leaves the control volume through the control surface. The control volume can deform with time as well as move and rotate in space and the mass in the control volume can change with time.

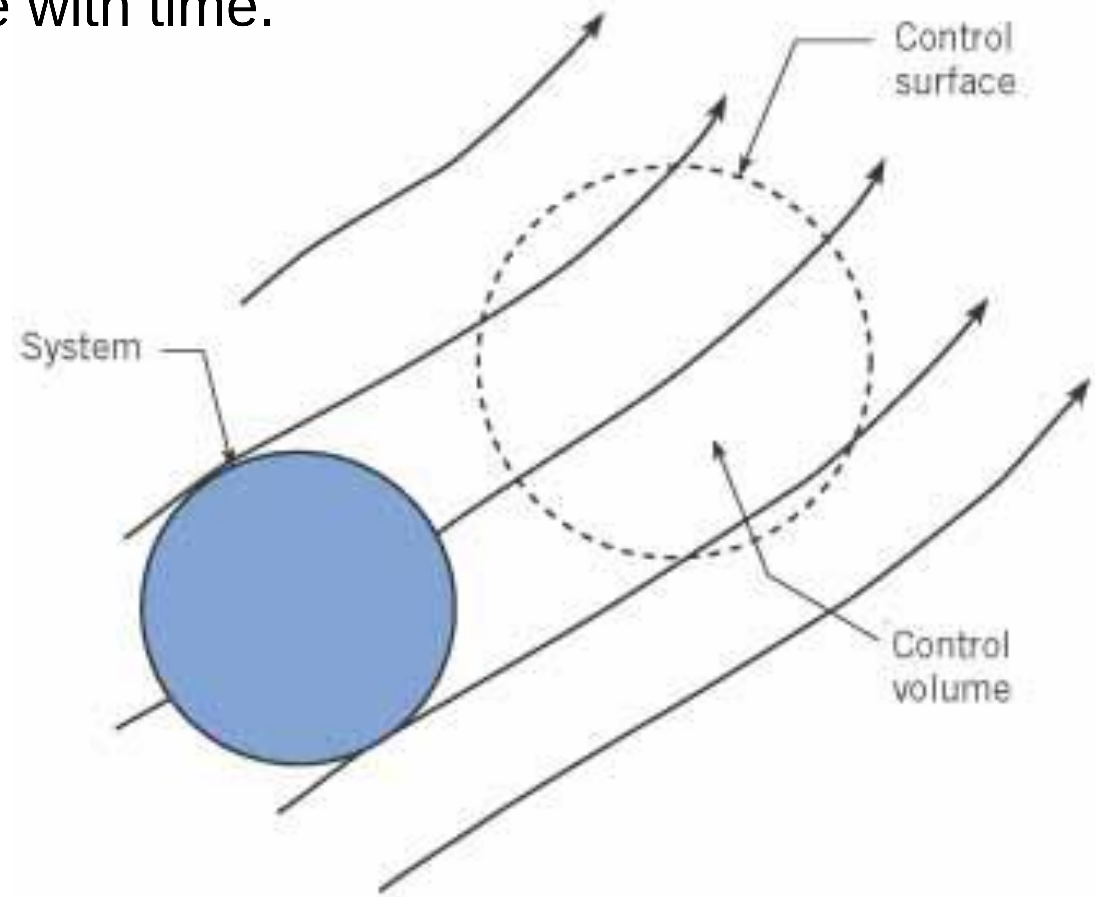


Figure 5.6 *System, control surface, and control volume in a flow field.*

Intensive and Extensive Properties

An ***extensive property*** is any property that depends on the amount of matter present. The extensive properties of a system include mass, m, momentum, $m\mathbf{v}$ (where **v** is velocity), and energy, E. Another example of an extensive property is weight because the weight is mg. An ***intensive property*** is any property that is independent of the amount of matter present. Examples of intensive properties include pressure and temperature. Many intensive properties are obtained by dividing the extensive property by the mass present. The intensive property for momentum is velocity **v**, and for energy is e, the energy per unit mass. The intensive property for weight is g.

In this section an equation for a general extensive property, B, will be developed. The corresponding intensive property will be b. The amount of extensive property B contained in a control volume at a given instant is

For an extensive property B, the corresponding intensive property will be *b;* that is b=B/m, where m is the system mass. The amount of extensive property *B* contained in a control volume at a given instant is

$$B_{cv} = \int_{cv} b\,dm = \int_{cv} b\rho\,d\forall$$

where *dm* and *d*$\forall$ are the differential mass and differential volume, respectively, and the integral is carried out over the control volume.

Property Transport Across the Control Surface

When fluid flows across a control surface, properties such as mass, momentum, and energy are transported with the fluid either into or out of the control volume. Consider the flow through the control volume in the duct in Fig. 5.7. If the velocity is uniformly distributed across the control surface, the mass flow rate through each cross section is given by

$$\dot{m}_1 = \rho_1 A_1 V_1 \qquad \dot{m}_2 = \rho_2 A_2 V_2$$

The net mass flow rate out* of the control volume, that is, the outflow rate minus the inflow rate, is

$$\text{net mass outflow rate} = \dot{m}_2 - \dot{m}_1 = \rho_2 A_2 V_2 - \rho_1 A_1 V_1$$

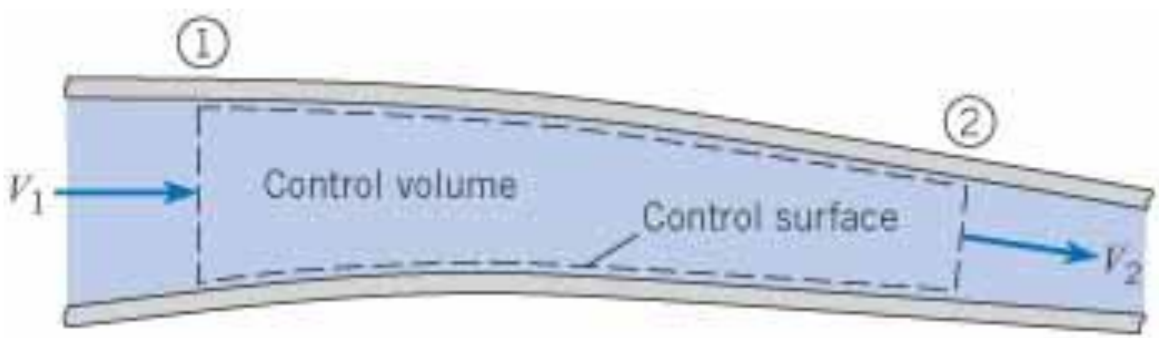
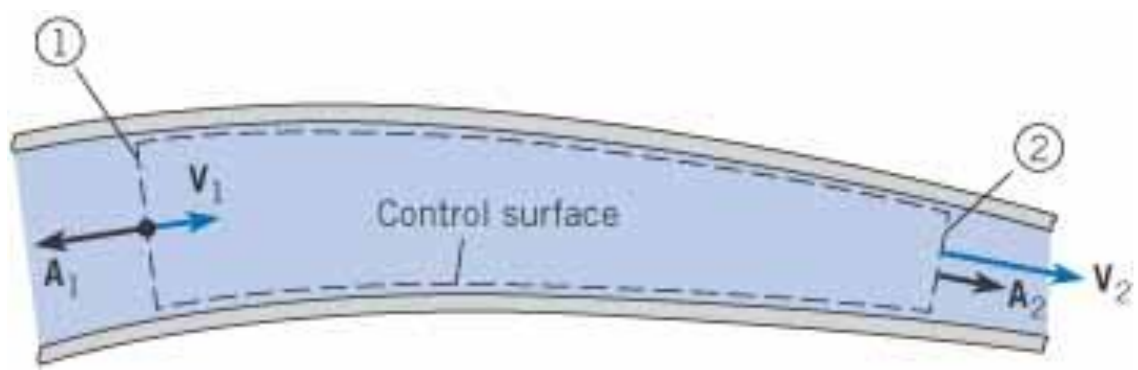


Figure 5.7 *Flow through control volume in a duct.*

The same control volume is shown in Fig. 5.8 with each control surface area represented by a vector, **A**, oriented outward from the Control volume and with magnitude equal to the cross-sectional area. The velocity is represented by a vector, **V**. Taking the dot product of the velocity and area vectors at both stations gives

Figure 5.7 *Flow through control volume in a duct.*



Because at station 1 the velocity and area have the opposite directions while at station 2 the velocity and area vectors are in the same direction. Now the net mass outflow rate can be written a

$$
\begin{aligned}
\text{net mass outflow rate} &= \rho_2 V_2 A_2 - \rho_1 V_1 A_1 \\
&= \rho_2 \mathbf{V}_2 \cdot A_2 + \rho_1 \mathbf{V}_1 \cdot \mathbf{A}_1 \qquad (5.11) \\
&= \sum_{cs} \rho \mathbf{V} \cdot \mathbf{A}
\end{aligned}
$$

Equation (5.11) states that if the dot product $\rho \boldsymbol{V} \cdot \boldsymbol{A}$ is summed for all flows **into and out** of the control volume, the result is the net mass flow rate out of the control volume, or the net mass efflux. If the summation is positive, the net mass flow rate is out of the control volume. If it is negative, the net mass flow rate is into the control volume. If the inflow and outflow rates are equal, then there is no change of mass inside the control volume.

Similarly, to obtain the net rate of flow of an extensive property *B* out of the control volume, the mass flow rate is multiplied by the intensive property *b*:

$$\dot{B}_{net} = \sum_{cs} b \overbrace{\rho V \cdot A}^{\dot{m}} \tag{5.12}$$

Equation (5.12) is applicable for all flows where the properties are uniformly distributed across the area. If the properties vary across a flow section, then it becomes necessary to integrate across the section to obtain the rate of flow. Specifically,

$$\dot{B}_{net} = \int_{cs} b \rho V \cdot dA$$

This is the most general expression for the net rate of flow of an extensive property from a control volume.

Reynolds Transport Theorem

The Reynolds transport theorem relates the Eulerian and Lagrangian approaches. The Reynolds transport theorem is derived by considering the rate of change of an extensive property of a system as it passes through a control volume. A control volume with a system moving through it is shown in Fig. 5.9. The control volume is enclosed by the control surface identified by the dashed line. The system is identified by the darker shaded region. At time *t* the system consists of the material inside the control volume and the material going in, so the property *B* of the system at this time is

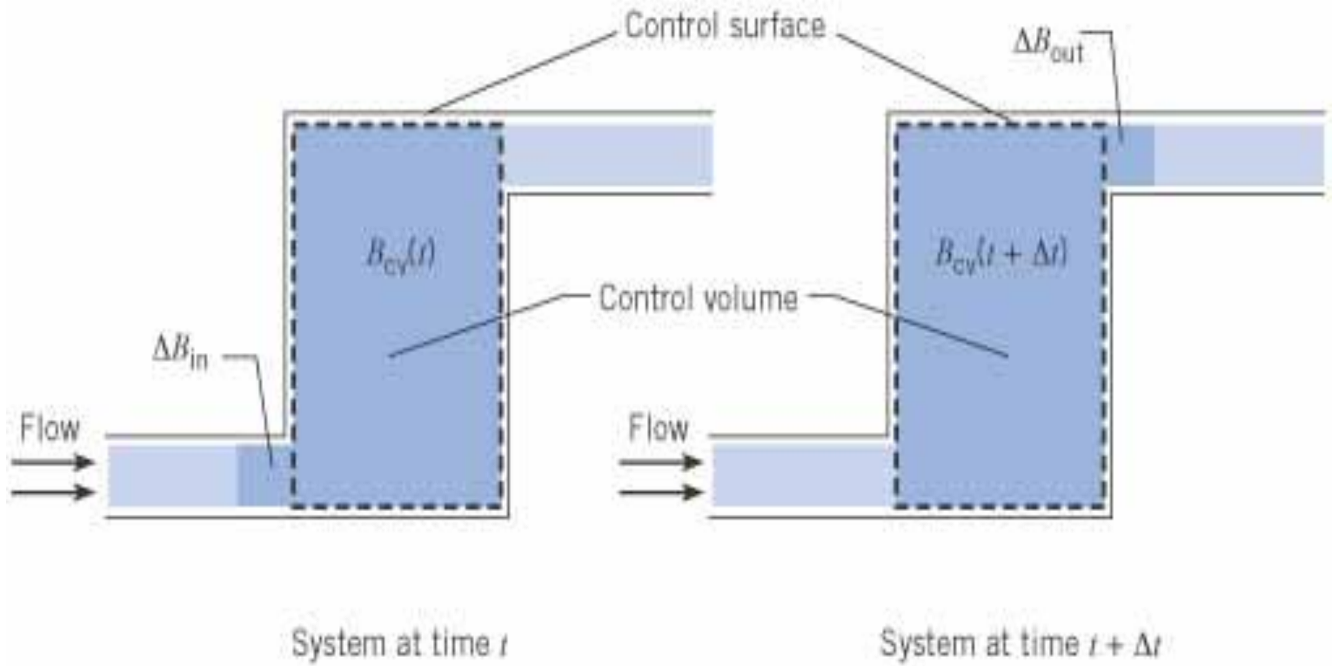


Figure 5.9
Progression of a system through a control volume.

It can be shown that the final form of the Reynolds Transport Theorem is,

$$\underbrace{\frac{dB_{sys}}{dt}}_{\text{Lagrangian}} = \underbrace{\frac{d}{dt}\int_{cv} b\rho d\mathcal{V} + \int_{cs} b\rho V \cdot d\mathbf{A}}_{\text{Eulerian}}$$

In words, the theorem can be stated as follows,

$$\left\{\begin{matrix}\text{Rate of change}\\ \text{of property } B\\ \text{of system}\end{matrix}\right\} = \left\{\begin{matrix}\text{Rate of change}\\ \text{of property } B\\ \text{in control volume}\end{matrix}\right\} + \left\{\begin{matrix}\text{Net outflow}\\ \text{of property } B\\ \text{through control surface}\end{matrix}\right\}$$

5.3 Continuity Equation

The continuity equation derives from the conservation of mass, which, in Lagrangian form, simply states that the mass of the system is constant.

$$m_{\mathrm{sys}} = \mathrm{constant}$$

The Eulerian form is derived by applying the Reynolds transport theorem. In this case the extensive property of the system is its mass, $B_{cv} = m_{sys}$. The corresponding value for b is the mass per unit mass, or simply, unity.

$$b = \frac{m_{\mathrm{sys}}}{m_{\mathrm{sys}}} = 1$$

General Form of the Continuity Equation

The general form of the continuity equation is obtained by substituting the properties for mass into the Reynolds transport theorem, resulting in

$$\frac{dm_{sys}}{dt} = \frac{d}{dt}\int_{cv} \rho d\mathcal{V} + \int_{cs} \rho \mathbf{V} \cdot d\mathbf{A}$$

However, $dm_{sys}/dt = 0$, so the general, or integral, form of the *continuity equation* is

$$\frac{d}{dt}\int_{cv} \rho d\mathcal{V} + \int_{cs} \rho \mathbf{V} d\mathbf{A} = 0$$

This equation can be expressed in words as

$$\left\{\begin{matrix}\text{The accumulation rate}\\ \text{of mass in the}\\ \text{control volume}\end{matrix}\right\} + \left\{\begin{matrix}\text{The net outflow rate}\\ \text{of mass through}\\ \text{the control surface}\end{matrix}\right\} = 0$$

If the mass crosses the control surface through a number of inlet and exit ports, the continuity equation simplifies to

$$\frac{d}{dt}m_{cv} + \sum_{cs}\dot{m}_o - \sum_{cs}\dot{m}_i = 0$$

where m_{cv} is the mass of fluid in the control volume. Note that the two summation terms represent the net mass outflow through the control surface.

EXAMPLE 5.4 MASS ACCUMULATION IN A TANK

A jet of water discharges into an open tank, and water leaves the tank through an orifice in the bottom at a rate of 0.003 m^3/s. If the cross-sectional area of the jet is 0.0025 m^2 where the velocity of water is 7 m/s, at what rate is water accumulating in (or evacuating from) the tank

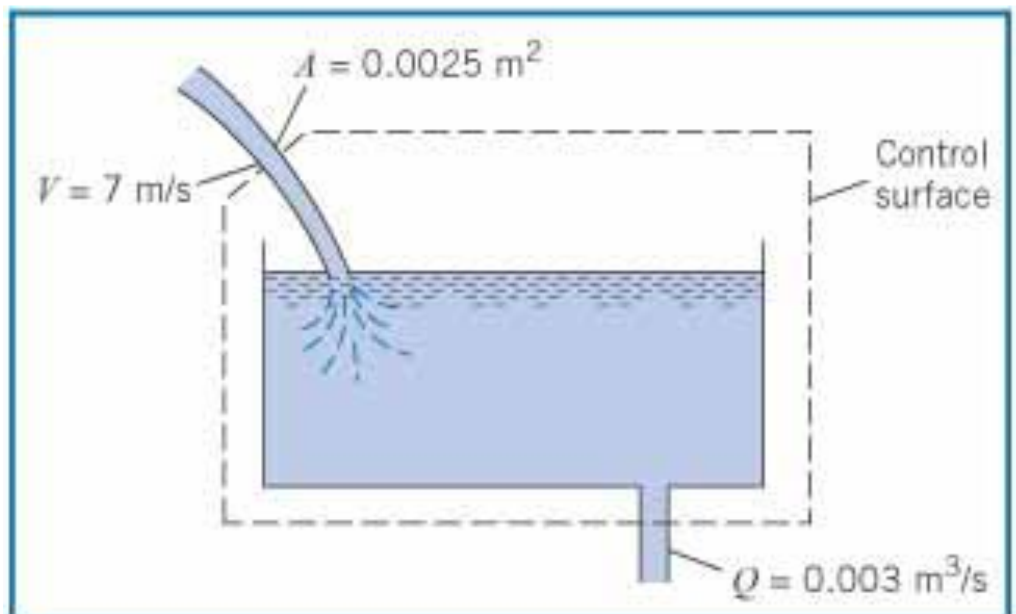
Solution

1. Continuity equation

$$\frac{d}{dt}m_{cv} + \sum_{cs}\dot{m}_o - \sum_{cs}\dot{m}_i = 0$$

Because there is only one inlet and one outlet, the equation reduces to

$$\frac{d}{dt}m_{cv} = \dot{m}_i - \dot{m}_o$$



2. Term-by-term analysis

· The inlet mass flow rate is calculated as follows,

$$\begin{aligned} \dot{m}_i &= \rho V A \\ &= 1000\ \mathrm{kg/m^3} \times 7\ \mathrm{m/s} \times 0.0025\ \mathrm{m^2} \\ &= 17.5\ \mathrm{kg/s} \end{aligned}$$

· Outlet flow rate is

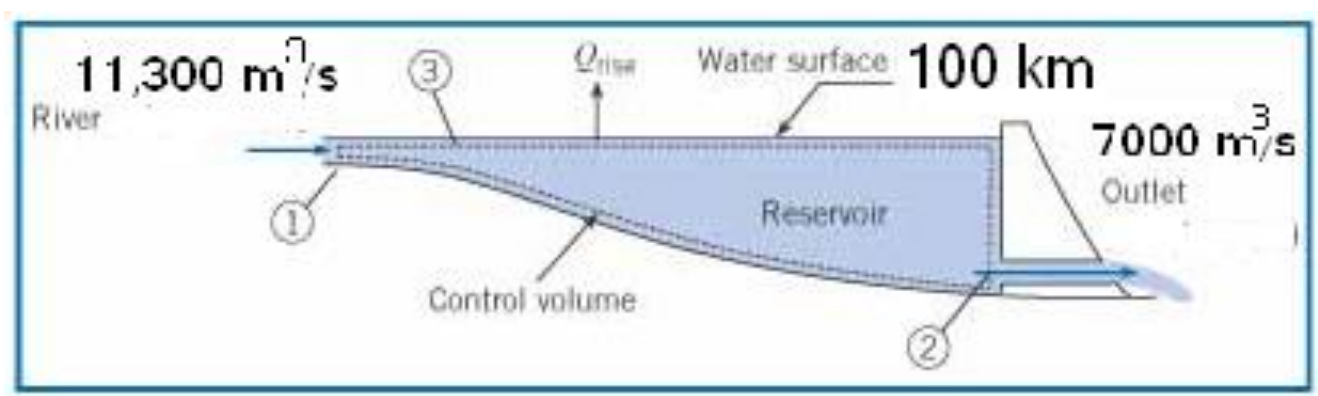
$$\dot{m}_o = \rho Q = 1000\ \mathrm{kg/m^3} \times 0.003\ \mathrm{m^3/s} = 3\ \mathrm{kg/s}$$

3. Accumulation rate:

$$\begin{aligned} \frac{dm_{cv}}{dt} &= 17.5\ \mathrm{kg/s} - 3\ \mathrm{kg/s} \\ &= \boxed{14.5\ \mathrm{kg/s}} \end{aligned}$$

EXAMPLE 5.5 RATE OF WATER RISE IN RESERVOIR

A river discharges into a reservoir at a rate of 11,3,00 m^3/s, and the outflow rate from the reservoir through the flow passages in the dam is 7,000 m^3/s. If the reservoir surface area is 100 km^2, what is the rate of rise of water in the reservoir?



Solution

Continuity equation:

$$\frac{d}{dt}m_{cv} + \sum_{cs}\dot{m}_o - \sum_{cs}\dot{m}_i = 0$$

Considering the control volume is constant, so $dm_{cv}/dt = 0$

At the inlet, the mass flow rate is.

$$\sum_{cs}\dot{m}_i = \rho Q_1$$

There two outlets in this case,

$$\sum_{cs}\dot{m}_o = \rho Q_2 + \rho Q_{rise}.$$

Substitution in the continuity equation,

$$\rho Q_2 + \rho Q_{rise} = \rho Q_1$$

$$Q_{rise} = Q_1 - Q_2$$

The rate of rise simply is velocity,

$$V = \frac{Q_{rise}}{A_1} = \frac{Q_1 - Q_2}{A_1} = \frac{11,300 - 7,000}{100x10^6} = 4.3x10^{-5}\, m/s$$

EXAMPLE 5.6 WATER LEVEL DROP RATE IN DRAINING TANK

A 10 cm jet of water issues from a 1 m diameter tank. Assume that the velocity in the jet is $\sqrt{2gh}$ m/s where h is the elevation of the water surface above the outlet jet. How long will it take for the water surface in the tank to drop from h_0 = 2 m to h_f = 0.50 m?

Solution

1. Continuity equation

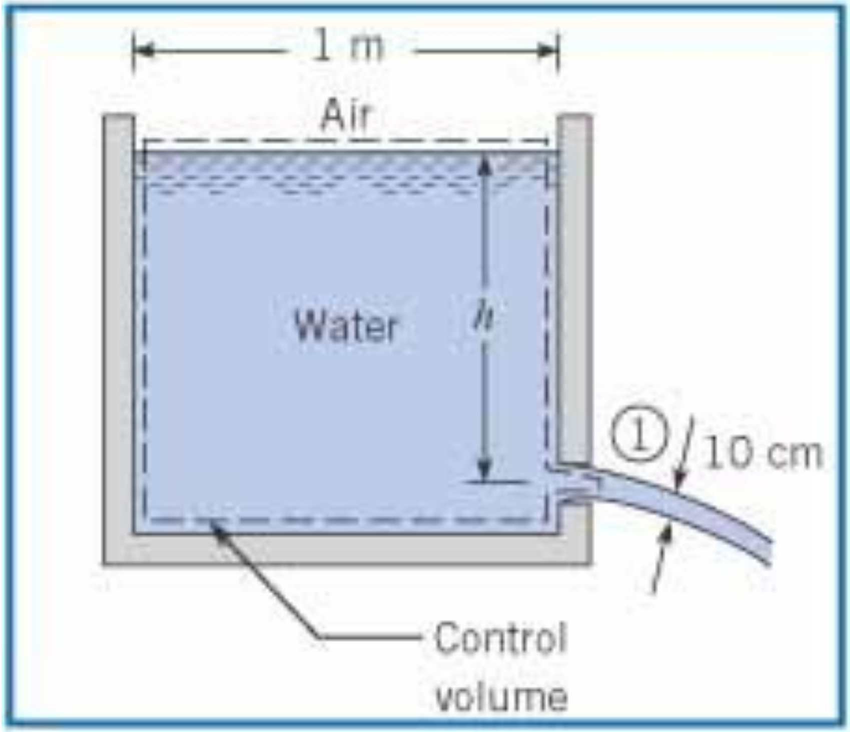
$$\frac{d}{dt}m_{cv} + \sum_{cs}\dot{m}_0 - \sum_{cs}\dot{m}_i = 0$$

Accumulation rate,

$$dm_{cv} = \rho A_T dh$$

$$\frac{dm_{cv}}{dt} = \rho A_T \frac{dh}{dt}$$

where A_T is cross-sectional area of tank.



Inlet mass flow rate with no inflow is

$$\sum_{cs} \dot{m}_i = 0$$

The mass flow rate leaving is

$$\sum_{cs} \dot{m}_o = \rho A_1 V_1 = \rho \sqrt{2gh} A_1$$

Substitution of terms into continuity equation

$$-\rho V_1 A_1 = \frac{d(\rho A_T h)}{dt}$$

$$-\sqrt{2gh} A_1 = A_T \frac{dh}{dt}$$

Equation for elapsed time and separating variables

$$dt = \frac{-A_T}{\sqrt{2g} A_1} \frac{dh}{\sqrt{h}} \quad \text{or} \quad dt = \frac{-A_T}{\sqrt{2g} A_1} h^{-1/2} dh$$

Integrating

$$t = \frac{-2A_T}{\sqrt{2gA_1}} h^{1/2} + C$$

Substituting in initial condition, $h(0) = h_0$, and final condition, $h(t) = h_f$, and solving for time

$$t = \frac{2A_T}{\sqrt{2gA_1}} (h_0^{1/2} - h_f^{1/2})$$

Evaluating the parameters and calculating time

$$A_1 = \frac{\pi}{4}(0.10\text{m})^2 = 0.01\left(\frac{\pi}{4}\right)\text{m}^2$$

$$A_T = \frac{\pi}{4}D^2 = \frac{\pi}{4} \times (1\text{ m})^2 = \frac{\pi}{4}\text{m}^2$$

$$t = \frac{2(\pi/4)\text{ m}^2}{\sqrt{2 \times 9.81\text{ m}/s^2}(\pi/4 \times 0.01\text{ m}^2)} (\sqrt{2\text{m}} - \sqrt{0.5\text{m}})$$

$$= \boxed{31.9\text{ s}}$$

EXAMPLE 5.7 DEPRESSURIZATION OF GAS IN TANK

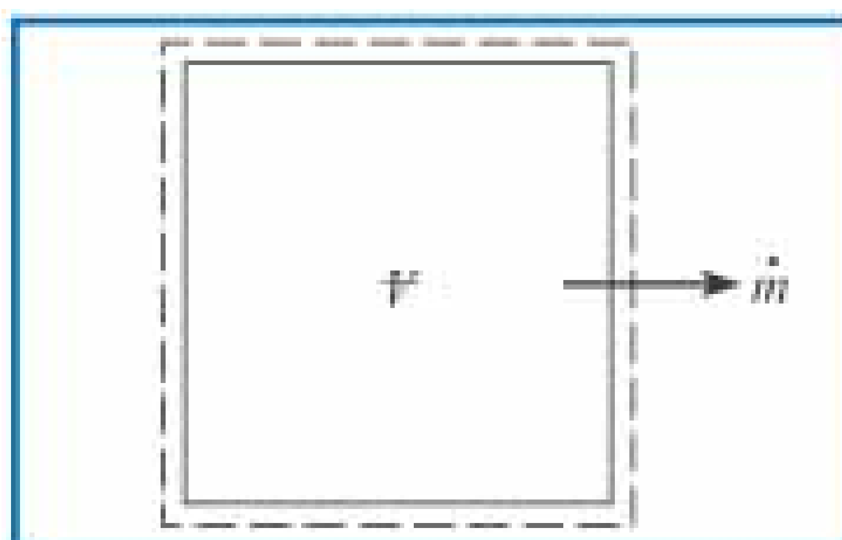
Methane escapes through a small (10^{-7} m^2) hole in a 10 m^3 tank. The methane escapes so slowly that the temperature in the tank remains constant at 23°C. The mass flow rate of methane through the hole is given by the equation below with *p* is the pressure in the tank, *A* is the area of the hole, *R* is the gas constant, and *T* is the temperature in the tank. Calculate the time required for the absolute pressure in the tank to decrease from 500 to 400 kPa.

$$\dot{m} = \frac{0.66\,pA}{\sqrt{RT}}$$

Solution

1. Continuity equation

$$\frac{d}{dt} m_{cv} + \sum_{cs} \dot{m}_o - \sum_{cs} \dot{m}_i = 0$$



Rate of accumulation term. The mass in the control volume is the sum of the mass of the tank shell, m_{shell} and the mass of methane in the tank,

$$m_{cv} = m_{shell} + \rho\forall$$

where V is the internal volume of the tank which is constant. The mass of the tank shell is constant, so

$$\frac{dm_{cv}}{dt} = \forall\frac{d\rho}{dt}$$

There is no mass inflow:

$$\sum_{cs}\dot{m}_i = 0$$

Mass out flow rate is

$$\sum_{cs}\dot{m}_o = 0.66\frac{pA}{\sqrt{RT}}$$

Substituting terms into continuity equation

$$\forall\frac{d\rho}{dt} = -0.66\frac{pA}{\sqrt{RT}}$$

Equation for elapsed time and Using ideal gas law for ρ,

$$V\frac{d}{dt}\left(\frac{p}{RT}\right)=-0.66\frac{pA}{\sqrt{RT}}$$

Because *R*, *T*, *A*, and *V* are constant,

$$\frac{dp}{dt}=-0.66\frac{pA\sqrt{RT}}{V} \quad \text{or} \quad \frac{dp}{p}=-0.66\frac{A\sqrt{RT}\,dt}{V}$$

Integrating equation and substituting limits for initial and final pressure and computing for time,

$$t=\frac{1.52V}{A\sqrt{RT}}\ln\frac{p_0}{p_f}$$

$$t=\frac{1.52(10\ \text{m}^3)}{(10^{-7}\ \text{m}^2)\left(518\frac{\text{J}}{\text{kg}\cdot\text{K}}\times 300\ \text{K}\right)^{1/2}}\ln\frac{500}{400}=\boxed{8.6\times 10^4\ \text{s}}$$

Continuity Equation for Flow in a Pipe

Several simplified forms of the continuity equation are used by engineers for flow in a pipe. Consider a control volume inside a pipe, Fig. 5.10. Mass enters through station 1 and exits through station 2. The control volume is fixed to the pipe walls, and its volume is constant. If the flow is **steady**, then m_{cv} is constant so the mass flow formulation of the continuity equation reduces to

m1 = m2 fix

For flow with a uniform velocity and density distribution, the continuity equation for steady flow in a pipe is

$$\rho_2 A_2 V_2 = \rho_1 A_1 V_1$$

If the flow is incompressible, then

$$A_2 V_2 = A_1 V_1$$

or

$$Q_2 = Q_1$$

This equation is valid for both steady and unsteady incompressible flow.

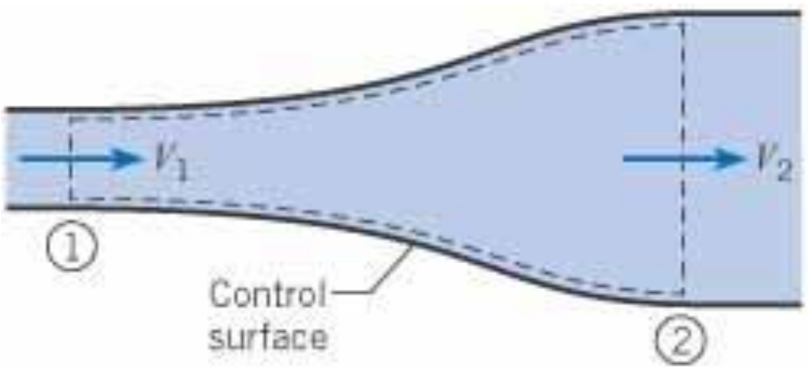


Figure 5.10 *Flow through a pipe section.*

If the flow is not uniformly distributed, the mass flow must be calculated using the integral form of the equation.

If there are more than two ports, then the general form of the continuity equation for steady flow is

$$\sum_{cs} \dot{m}_i = \sum_{cs} \dot{m}_o$$

If the flow is incompressible, then the above can be written in terms of discharge

$$\sum_{cs} Q_i = \sum_{cs} Q_o$$

EXAMPLE 5.8 VELOCITY IN A VARIABLE-AREA PIPE

A 120 cm pipe is in series with a 60 cm pipe. The speed of the water in the 120 cm pipe is 2 m/s.

What is the water speed in the 60 cm pipe?

Solution: incompressible, then

$$V_{60} = V_{120}\frac{A_{120}}{A_{60}} = 2\text{m}/\text{s} \times \frac{(120\ \text{cm})^2}{(60\ \text{cm})^2} = \boxed{8\ \text{m}/\text{s}}$$

EXAMPLE 5.9 WATER FLOW THROUGH A VENTURIMETER

Water with a density of 1000 kg/m^3 flows through a vertical venturimeter as shown. A pressure gage is connected across two taps in the pipe (station 1) and the throat (station 2). The area ratio A_{throat}/A_{pipe} is 0.5. The velocity in the pipe is 10 m/s. Find the pressure difference recorded by the pressure gage. Assume the flow has a uniform velocity distribution and that viscous effects are not important.

Solution:

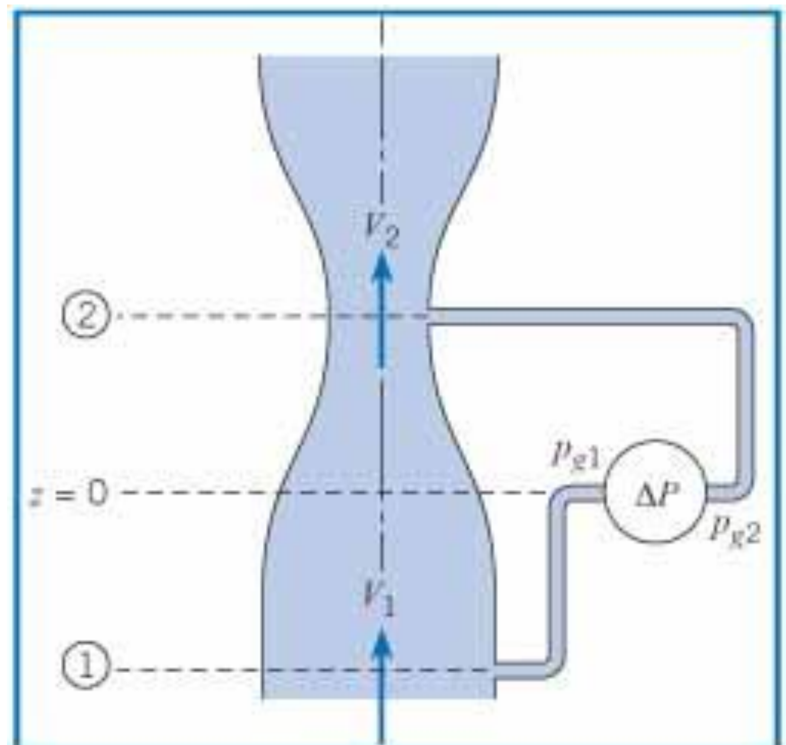
The Bernoulli equation

$$p_1 + \gamma z_1 + \rho \frac{V_1^2}{2} = p_2 + \gamma z_2 + \rho \frac{V_2^2}{2}$$

Rewrite the equation in terms of piezometric pressure.

$$p_{z_1} - p_{z_2} = \frac{\rho}{2}(V_2^2 - V_1^2)$$

$$= \frac{\rho V_1^2}{2}\left(\frac{V_2^2}{V_1^2} - 1\right)$$



Incompressible, then continuity equation $V_2/V_1 = A_1/A_2$

$$p_{z_1} - p_{z_2} = \frac{\rho V_1^2}{2}\left(\frac{A_1^2}{A_2^2} - 1\right)$$

$$= \frac{1000 \text{ kg/m}^3}{2} \times (10 \text{ m/s})^2 \times (2^2 - 1)$$

$$= 150 \text{ kPa}$$

Gage is located at zero elevation. Apply hydrostatic equation through static fluid in gage line between gage attachment point where the pressure is and station 1 where the gage line is tapped into the pipe,

$$p_{z_1} = p_{g_1}$$

But, $p_{z_2} = p_{g_2}$

So, $\Delta p_{gage} = p_{g_1} - p_{g_2} = p_{z_1} - p_{z_2} = \boxed{150 \text{ kPa}}$

5.4 Cavitation

Cavitation is the phenomenon that occurs when the fluid pressure is **reduced to the local vapor pressure and boiling** occurs. Under such conditions vapor bubbles form in the liquid, grow, and then collapse, producing shock waves, noise, and dynamic effects that lead to decreased equipment performance and, frequently, equipment failure. Engineers must design flow systems to avoid potential problems.

However, despite cavitation deleterious effects, cavitation can also be beneficial. Cavitation is responsible for the effectiveness of ultrasonic cleaning

Cavitation typically occurs at locations where the velocity is high. Consider the water flow through the pipe restriction shown in Fig. 5.11. The pipe area decreases, so the velocity increases according to the continuity equation and, in turn, the pressure decreases as dictated by the Bernoulli equation. For low flow rates, there is a relatively small drop in pressure at the restriction, so the water remains well above the vapor pressure and boiling does not occur. However, as the flow rate increases, the pressure at the restriction becomes progressively lower until a flow rate is reached where the pressure is equal to the vapor pressure. At this point, the liquid boils to form bubbles and cavitation ensues. The onset of cavitation can also be affected by the presence of contaminant gases, turbulence and viscosity

Temp	Density	Vapor Pressure
5°C	1000	872
10°C	1000	1,230
15°C	999	1,700
20°C	998	2,340
25°C	997	3,170
30°C	996	4,250
90°C	965	70,100
100°C	958	101,300

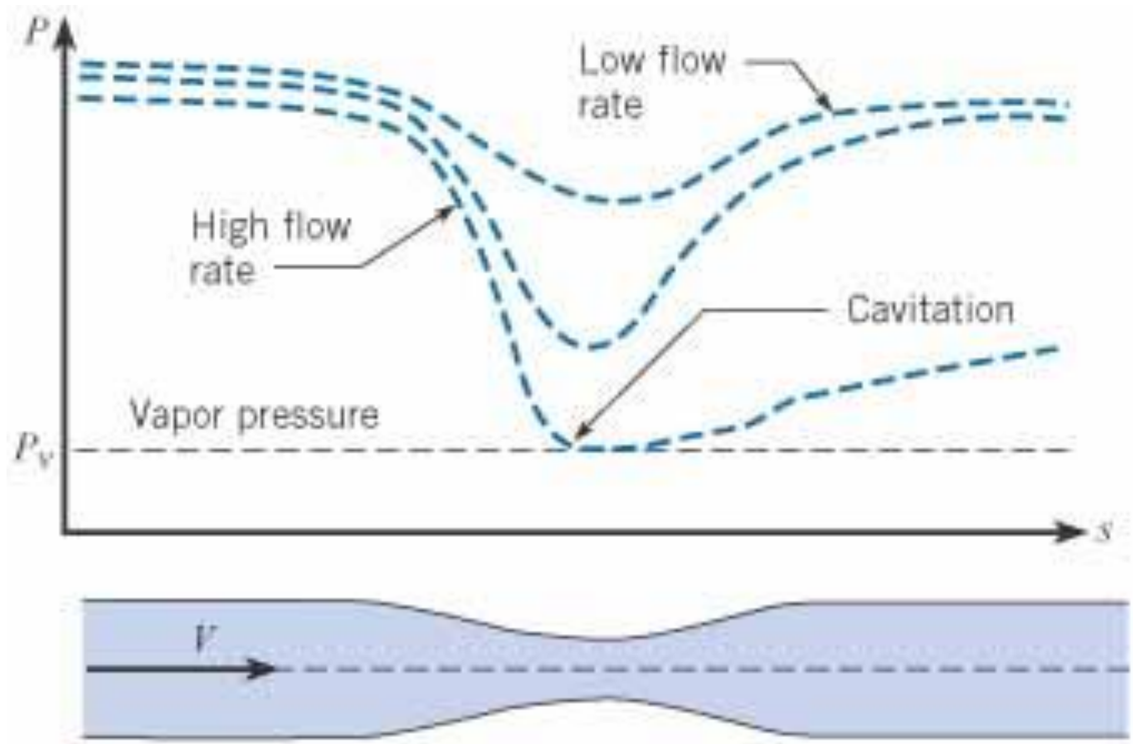


Figure 5.11 *Flow through pipe restriction: variation of pressure for three different flow rates.*

The formation of vapor bubbles at the restriction is shown in Fig. 5.12*a*. The vapor bubbles form and then collapse as they move into a region of higher pressure and are swept downstream with the flow. When the flow velocity is increased further, the minimum pressure is still the local vapor pressure, but the zone of bubble formation is extended as shown in Fig. 5.12*b*. In this case, the entire vapor pocket may intermittently grow and collapse, producing serious vibration problems. Severe damage that occurred on a centrifugal pump impeller is shown in Fig. 5.13. Obviously, cavitation should be avoided or minimized by proper design of equipment and structures and by proper operational procedures.

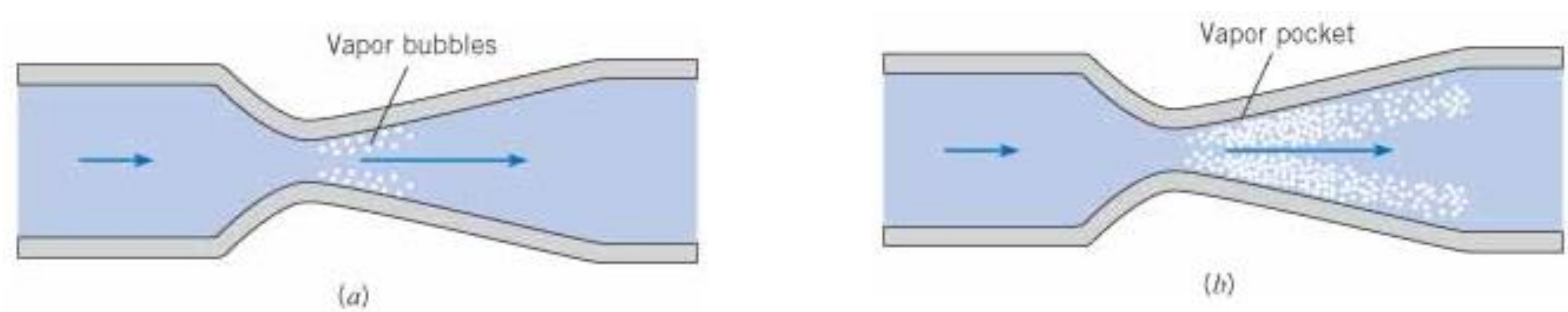


Figure 5.12 *Formation of vapor bubbles in the process of cavitation. (a) Cavitation. (b) Cavitation—higher flow rate.*

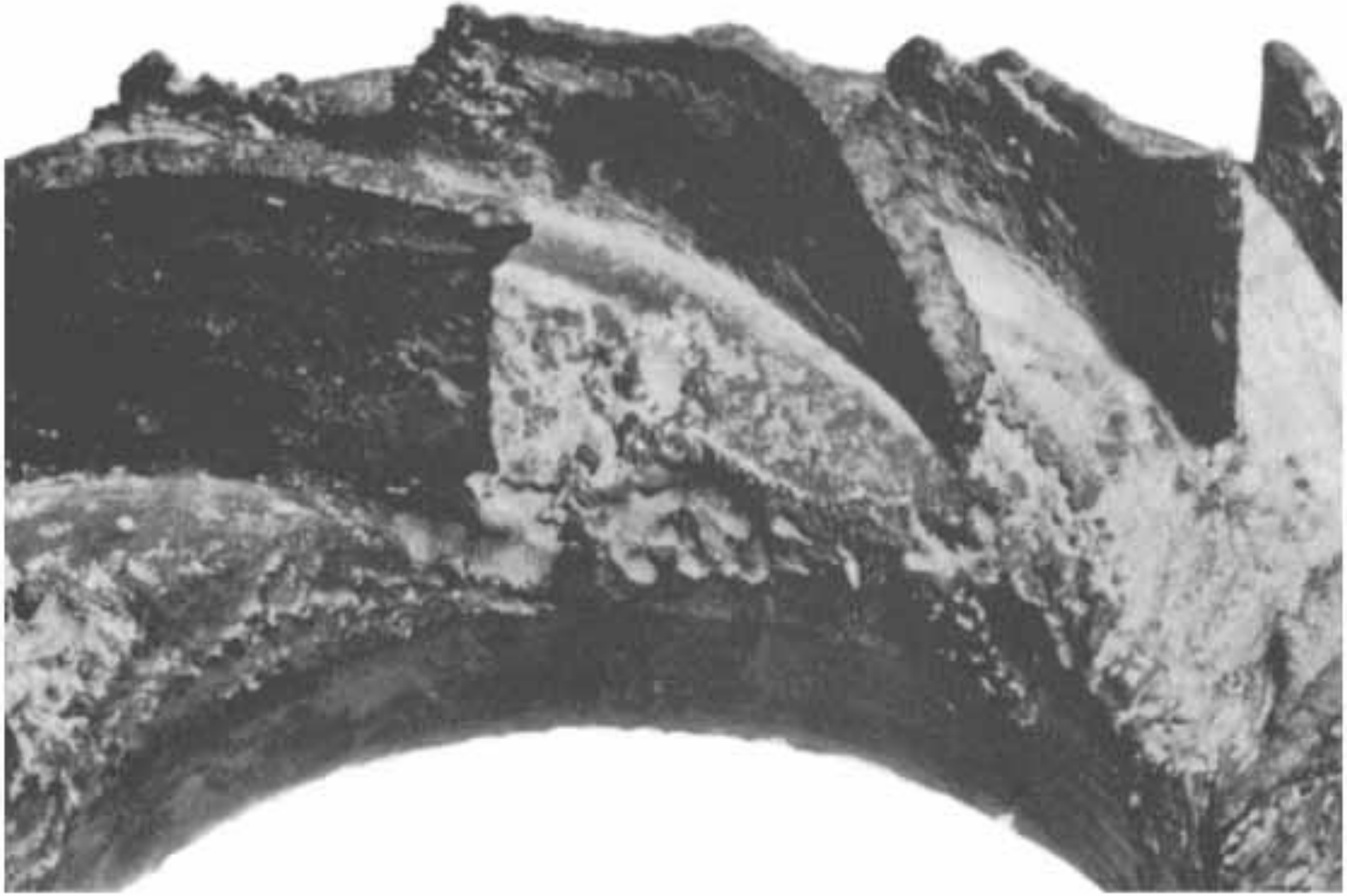


Figure 5.13 *Cavitation damage to impeller of a centrifugal pump.*

5.5 Differential Form of the Continuity Equation

In the analysis of fluid flows and the development of numerical models, one of the fundamental independent equations needed is the differential form of the continuity equation. The derivation is accomplished by applying the integral form of the continuity equation to a small control volume and taking the limit as the volume approaches zero. A small control volume defined by the *x*, *y*, *z* coordinate system is shown in Fig. 5.15.

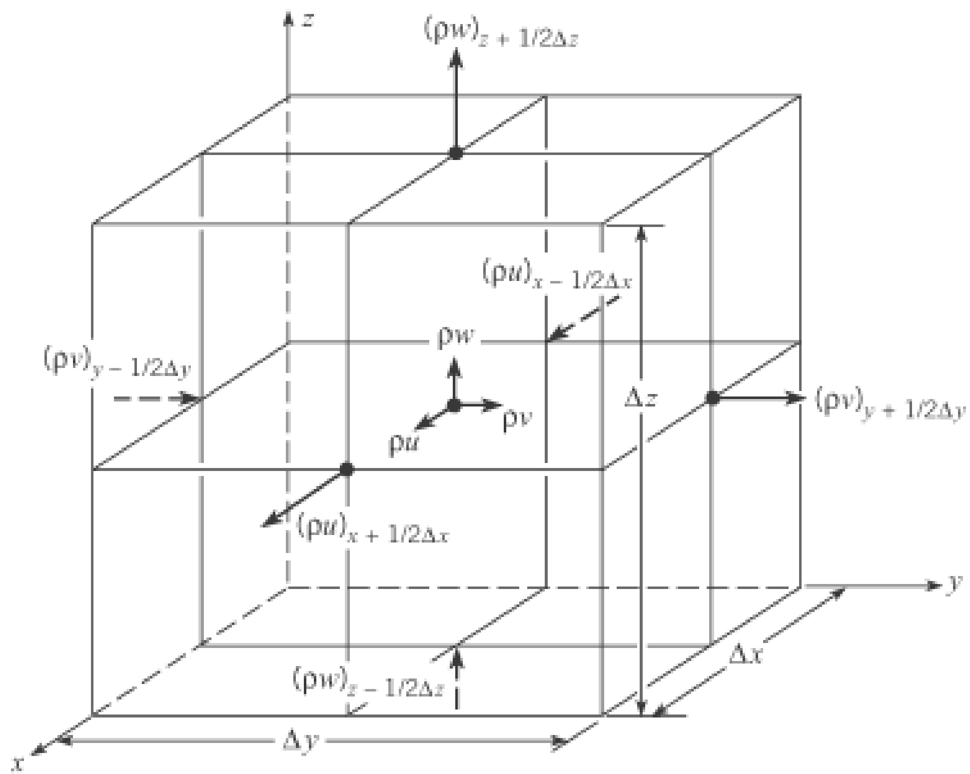


Figure 5.15 *Elemental control volume.*

When applying the integral form of the continuity to the differential control volume one can eventually lead to

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=-\frac{\partial \rho}{\partial t}$$

If the flow is steady, the equation reduces to

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0$$

And if the fluid is incompressible, the equation further simplifies to

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

which is valid for both steady and unsteady flow. In vector notation, Eq. (5.33) is given as

$$\nabla \cdot V=0$$

where ∇ is the del operator, defined as

$$\nabla=\mathbf{i}\frac{\partial}{\partial x}+\mathbf{j}\frac{\partial}{\partial y}+\mathbf{k}\frac{\partial}{\partial z}$$

EXAMPLE 5.10 APPLICATION OF DIFFERENTIAL FORM OF COBTINUITY EQUATION

The expression $\mathbf{V} = 10x\mathbf{i} - 10y\mathbf{j}$ is said to represent the velocity for a two-dimensional (planar) incompressible flow. Check to see if the continuity equation is satisfied.

Solution

Continuity equation for two-dimensional flow:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$u = 10x; \qquad \frac{\partial u}{\partial x} = 10$$

$$v = -10y; \qquad \frac{\partial v}{\partial y} = -10$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 10 - 10 = 0$$

Fluid Mechanics

Chapter 6
Momentum Equation

Dr. Amer Khalil Ababneh

Introduction

The analysis of forces on vanes and pipe bends, the thrust produced by a rocket or turbojet, and torque produced by a hydraulic turbine are all examples of the application of the momentum equation.

In this chapter the Reynolds transport theorem is applied to Newton's second law of motion, $\boldsymbol{F} = m\boldsymbol{a}$, to develop the Eulerian form of the momentum equation.

Application of this equation allows the engineer to analyze forces and moments produced by flowing

6.1 Momentum Equation: Derivation

When forces act on a particle, the particle accelerates according to Newton's second law of motion:

$$\sum \mathbf{F} = m\mathbf{a}$$

Since mass is constant, it follows,

$$\sum \mathbf{F} = \frac{d(m\mathbf{v})}{dt}$$

where $m\mathbf{v}$ is the momentum of the particle. The above equations are for a single particle; however, for a system of particles (e.g. fluid), the law still applies,

$$\sum \mathbf{F} = \frac{d(\mathbf{Mom}_{sys})}{dt}$$

The momentum is the extensive property B for the system, which can be made intensive by dividing by mass; $b = B/m$

Using the Reynolds transport theorem

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{cv} b\rho \, d\forall + \int_{cs} b\rho \, \mathbf{V} \cdot d\mathbf{A}$$

Substituting momentum for B leads to

$$\frac{d(\mathbf{Mom}_{sys})}{dt} = \frac{d}{dt}\int_{cv} \mathbf{v}\rho \, d\forall + \int_{cs} \mathbf{v}\rho V \cdot dA$$

But, rate of momentum change equal to sum of forces acting on system, hence

$$\sum F = \frac{d}{dt}\int_{cv} \mathbf{v}\rho \, d\forall + \int_{cs} \mathbf{v}\rho \, V \cdot dA$$

which is called the integral form of the momentum equation.

In words, it can be stated as,

$$\begin{bmatrix} \text{sum of forces} \\ \text{acting on the matter} \\ \text{in control volume} \end{bmatrix} = \begin{bmatrix} \text{time rate of} \\ \text{change of momentum} \\ \text{in control volume} \end{bmatrix} + \begin{bmatrix} \text{net outflow rate} \\ \text{of momentum} \\ \text{through control surface} \end{bmatrix}$$

It is important to make the following observations: one; the momentum equation is a vector and thus has three components in general, see next slide; two, the equation is based on Newton's second law, thus the momentum per unit mass ($\boldsymbol{v}$) must be with respect to inertial frame of reference, as explained below,

With respect to inertial frame of reference

$$\sum \boldsymbol{F} = \frac{d}{dt}\int_{cv} \boldsymbol{v}\rho \, d\mathcal{V} + \int_{cs} \boldsymbol{v}\rho \, \boldsymbol{V} \cdot d\boldsymbol{A}$$

With respect to control surfaces

If velocity enters and exits the control volume at several ports and occur such that it is uniform at these ports, then the momentum equation is simplified as

$$\sum F = \frac{d}{dt}\int_{cv} \rho v \, d\Psi + \sum_{cs} \dot{m}_O v_O - \sum_{cs} \dot{m}_i v_i$$

The components of the momentum equation

$$x\text{-direction: } \sum F_x = \frac{d}{dt}\int_{cv} v_x \, \rho \, d\Psi + \sum_{cs} \dot{m}_O \, v_{Ox} - \sum_{cs} \dot{m}_i v_{ix}$$

$$y\text{-direction: } \sum F_y = \frac{d}{dt}\int_{cv} v_y \, \rho \, d\Psi + \sum_{cs} \dot{m}_O \, v_{Oy} - \sum_{cs} \dot{m}_i v_{iy}$$

$$z\text{-direction: } \sum F_z = \frac{d}{dt}\int_{cv} v_z \, \rho \, d\Psi + \sum_{cs} \dot{m}_O v_{Oz} - \sum_{cs} \dot{m}_i v_{iz}$$

In applying the momentum equation follow these steps:

1) identify and draw the control volume
2) draw the coordinate system
3) identify where mass enters/leaves the control volume
4) identify the forces acting on the control volume

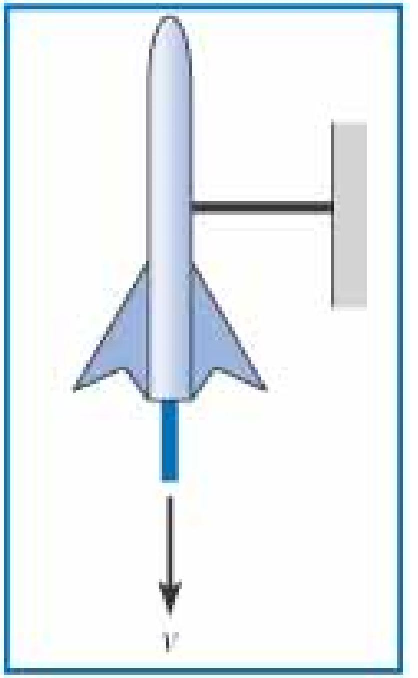
For forces there two kinds: 1) is called body forces like gravity which acts at every element of the body; 2) surface forces, which require a contact with the control volume, thus they act at the surfaces of the control volume.

EXAMPLE 6.1 THRUST OF ROCKET

The sketch below shows a 40 g rocket, of the type used for model rocketry, being fired on a test stand in order to evaluate thrust. The exhaust jet from the rocket motor has a diameter of $d = 1$ cm, a speed of v = 450 m/s, and a density of ρ = 0.5 kg/m3. Assume the pressure in the exhaust jet equals ambient pressure, and neglect any momentum changes inside the rocket motor. Find the force F_b acting on the beam that supports the rocket.

Solution

Apply momentum equation in vertical direction. Observe that there is no momentum accumulation inside the control volume, hence steady. No momentum entering; but only exiting. Also, assume that gases velocity at exit is uniform



Draw the control volume and the frame of reference as indicated.

The forces that are identified are F_b , which the force exerted by the beam and weight. There is no pressure forces since gases exit at atmospheric pressure.

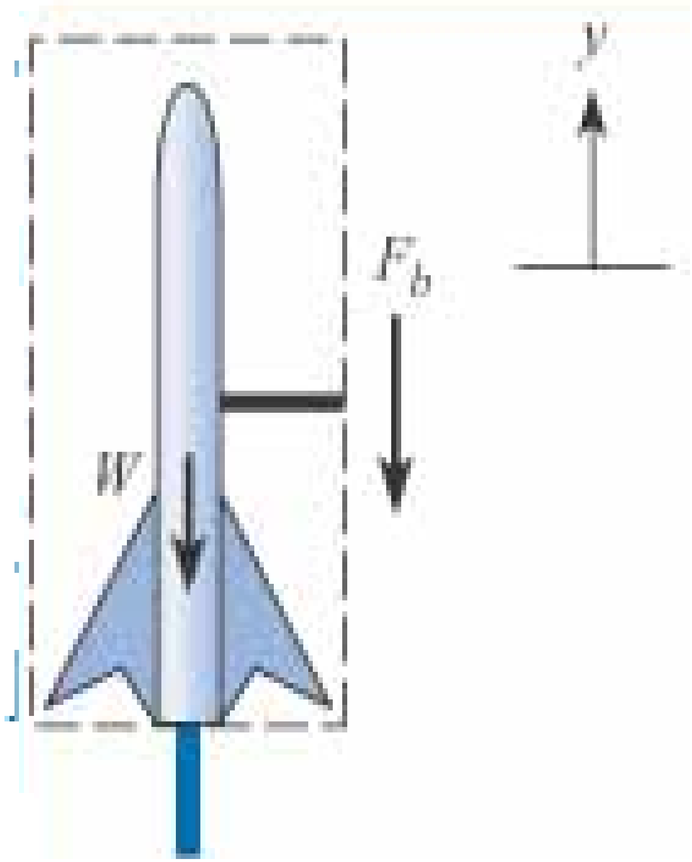
$$-F_b - mg = -\rho A v^2$$

$$F_b = \rho A v^2 - mg$$

$$= (0.5\ \text{kg}/\text{m}^3)(\pi \times 0.01^2\ m^2/4)(450^2\ m^2/s^2)$$

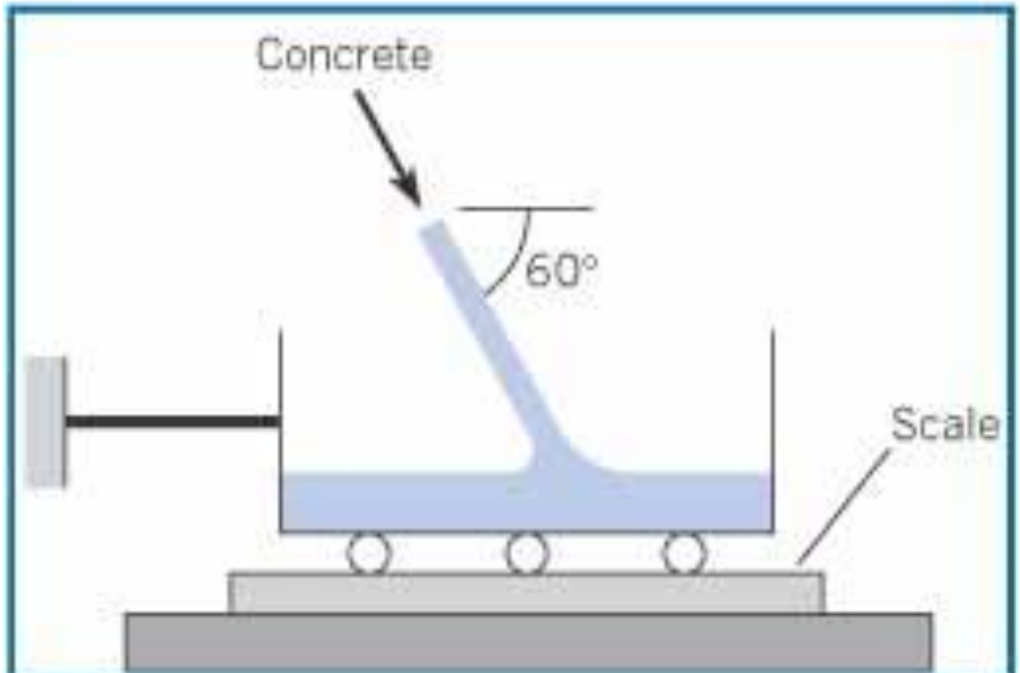
$$-(0.04\ \text{kg})(9.81\ \text{m}/s^2)$$

$$F_b = \boxed{7.56\ \text{N}}$$



EXAMPLE 6.2 CONCRETE FLOWING INTO CART

As shown in the sketch, concrete flows into a cart sitting on a scale. The stream of concrete has a density of ρ = 150 ibm/ft3, an area of A = 1 ft2, and a speed of V = 10 ft/s. At the instant shown, the weight of the cart plus the concrete is 800 lbf. Determine the tension in the cable and the weight recorded by the scale. Assume steady flow.



Solution

The momentum in x and z directs are

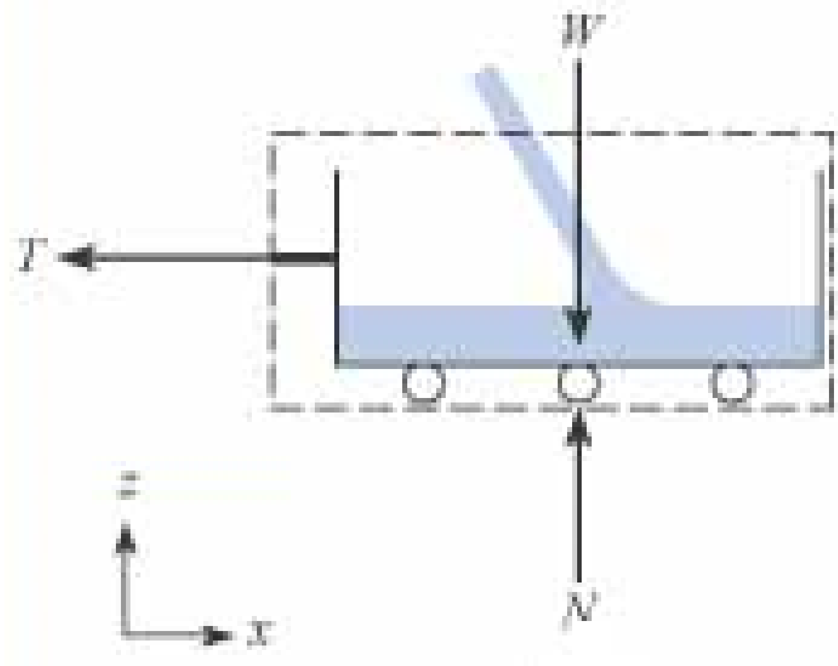
$$\sum F_x = \frac{d}{dt}\int_{cv} v_x \, \rho \, d\mathcal{V} + \sum_{cs} \dot{m}_o v_{ox} - \sum_{cs} \dot{m}_i v_{ix}$$

$$\sum F_z = \frac{d}{dt}\int_{cv} v_z \, \rho \, d\mathcal{V} + \sum_{cs} \dot{m}_o v_{oz} - \sum_{cs} \dot{m}_i v_{iz}$$

The forces acting on the control volume are

$$\sum F_x = -T$$

$$\sum F_z = N - W$$



Observing that the momentum accumulation is zero and no momentum leaving.

The inflow of momentum in the x and z directions are:

$$\sum_{cs} \dot{m}_i v_{ix} = \dot{m}v \cos 60° = \rho A v^2 \cos 60°$$

$$\sum_{cs} \dot{m}_i v_{iz} = \dot{m}(-v \sin 60°) = -\rho A v^2 \sin 60°$$

To evaluate tension, consider the x-direction

$$-T = -\rho A v^2 \cos 60°$$

$$T = (150 \text{ lbm/ft}^3)\left(\frac{\text{slugs}}{32.2 \text{ lbm}}\right)(1 \text{ ft}^2)(10 \text{ ft/s})^2 \cos 60°$$

$$= \boxed{233 \text{ lbf}}$$

To evaluate force on scale, consider the z-direction

$$N - W = -(-\rho A v^2 \sin 60°)$$

$$N = W + \rho A v^2 \sin 60°$$

$$= 800 \text{ lbf} + 403 \text{ lbf} = \boxed{1200 \text{ lbf}}$$

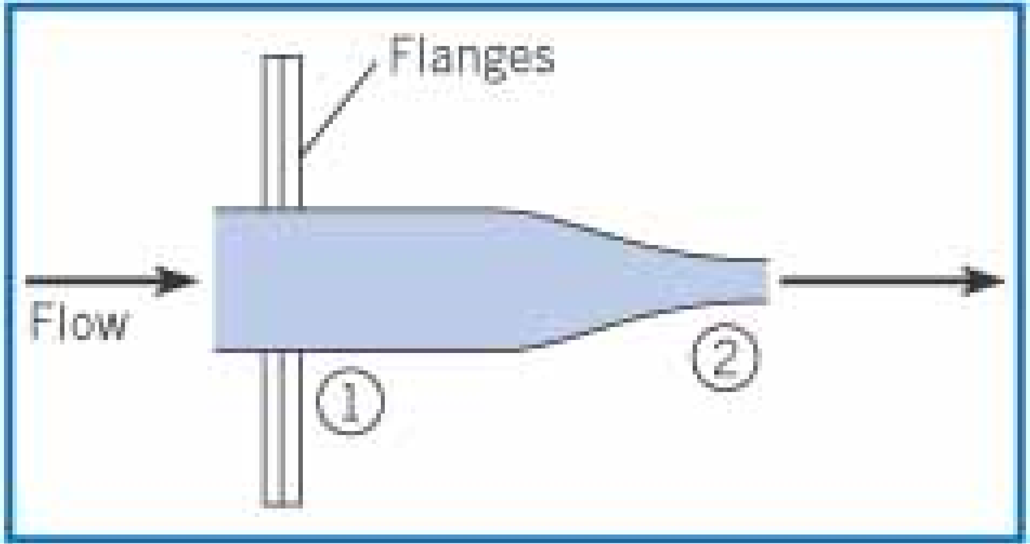
Nozzles

Nozzles are flow devices used to accelerate a fluid stream by reducing the cross-sectional area of the flow. When a fluid flows through a nozzle, it is reasonable to assume the velocity is uniform across inlet and outlet sections. Hence,the momentum flows will have magnitude If the nozzle exhausts into the atmosphere, the pressure at the exit is atmospheric.

In many applications involving finding the force on a nozzle, the Bernoulli equation is used along with the momentum equation.

EXAMPLE 6.3 FORCE ON A NOZZLE

Air flows through a nozzle where the inlet pressure is $p1$ = 105 kPa abs, and the air exhausts into the atmosphere, where the pressure is 101.3 kPa abs. The nozzle has an inlet diameter of 60 mm and an exit diameter of 10 mm, and the nozzle is connected to the supply pipe by flanges. Find the air speed at the exit of the nozzle and the force required to hold the nozzle stationary. Assume the air has a constant density of 1.22 kg/m^3. Neglect the weight of the nozzle.



Solution

1. Select control volume (and control surface). Control volume is stationary Application of the Bernoulli equation between sections 1 and 2

$$p_1 + \gamma z_1 + \frac{1}{2}\rho v_1{}^2 = p_2 + \gamma z_2 + \frac{1}{2}\rho v_2{}^2$$

Set $z_1 = z_2$. thus, the Bernoulli,

$$p_1 + p v_1^2 / 2 = \rho v_2^2 / 2$$

and

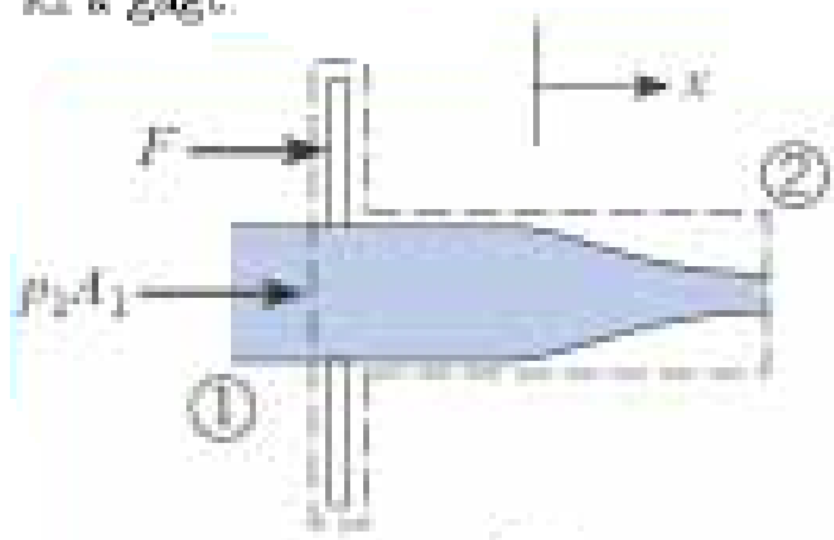
$$p_2 = 0 \text{ kPa gage, and now}$$

$$p_1 = 105 \text{ kPa} - 101.3 \text{ kPa} = 3.7 \text{ kPa gage.}$$

From the continuity equation,

$$v_1 A_1 = v_2 A_2$$

$$v_1 d_1^2 = v_2 d_2^2$$



Substituting the velocity into the Bernoulli, lead to

$$v_2 = \sqrt{\frac{2p_1}{\rho(1-(d_2/d_1)^4)}}$$

$$v_2 = \sqrt{\frac{2 \times 3.7 \times 1000\ \text{pa}}{(1.22\ \text{kg/m}^3)(1-(10/60)^4)}} = \boxed{77.9\ \text{m/s}}$$

Consequently, the inlet velocity

$$v_1 = v_2\left(\frac{d_2}{d_1}\right)^2$$

$$= 77.9\ \text{m/s} \times \left(\frac{1}{6}\right)^2 = 2.16\ \text{m/s}$$

Momentum

$$\sum F_x = \frac{d}{dt}\int_{cv} v_x \rho \, d\Psi + \sum_{cs} \dot{m}_o v_{ox} - \sum_{cs} \dot{m}_i v_{ix}$$

Sum of forces in *x*-direction $\sum F_x = F + p_1 A_1$

Observe that the accumulation term is zero since the the flow is steady. The momentum leaving at section 2 and entering at section 1,

$$\sum_{cs} \dot{m}_o v_{ox} = \dot{m} v_2 \quad \text{and} \quad \sum_{cs} \dot{m}_i v_{ix} = \dot{m} v_1$$

Substituting into the momentum equation,

$$F + p_1 A_1 = \dot{m}(v_2 - v_1)$$

$$F = \rho A_1 v_1 (v_2 - v_1) - p_1 A_1$$

$$= (1.22 \text{ kg/m}^3)\left(\frac{\pi}{4}\right)(0.06 \text{ m})^2(2.16 \text{ m/s}) \times$$

$$(77.9 - 2.16)(\text{m/s}) - 3.7 \times 1000 \text{ N/m}^2 \times \left(\frac{\pi}{4}\right)(0.06 \text{ m})^2$$

$$= 0.564 \text{ N} - 10.46 \text{ N} = \boxed{-9.90 \text{ N}}$$

Vanes

A *vane* is a structural component, typically thin, that is used to turn a fluid jet or is turned by a fluid jet. Examples include a blade in a turbine, a sail on a ship, and a thrust reverser on an aircraft engine. Figure 6.4 shows a flat vane impacted by a jet of fluid. A typical control volume is also shown. In analyzing flow over a vane, it is common to **neglect the pressure change due to elevation difference**. Since the pressure is constant (atmospheric pressure or surrounding pressure), the Bernoulli equation shows the speed is constant $\mathbf{v_1 = v_2 = v_3}$. Another common assumption is that **viscous forces are negligible** compared to pressure forces. Thus when a vane is flat, as in Fig. 6.4, the force needed to hold the vane stationary is normal to the vane.

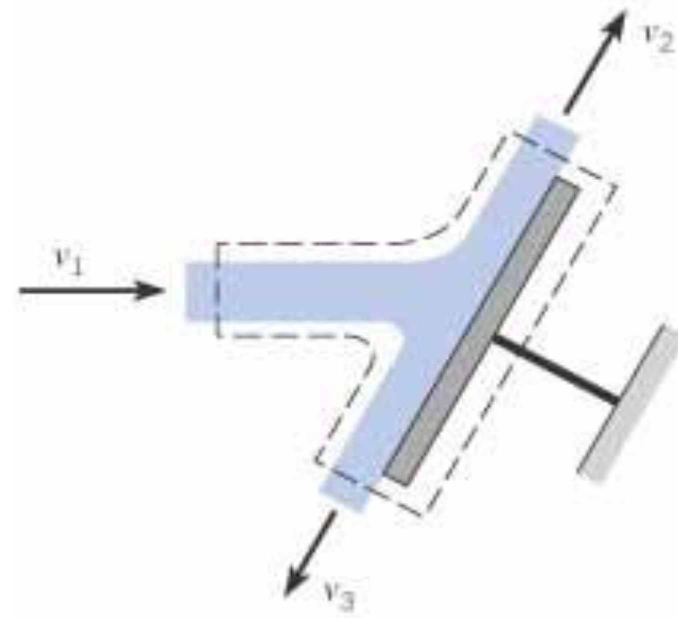


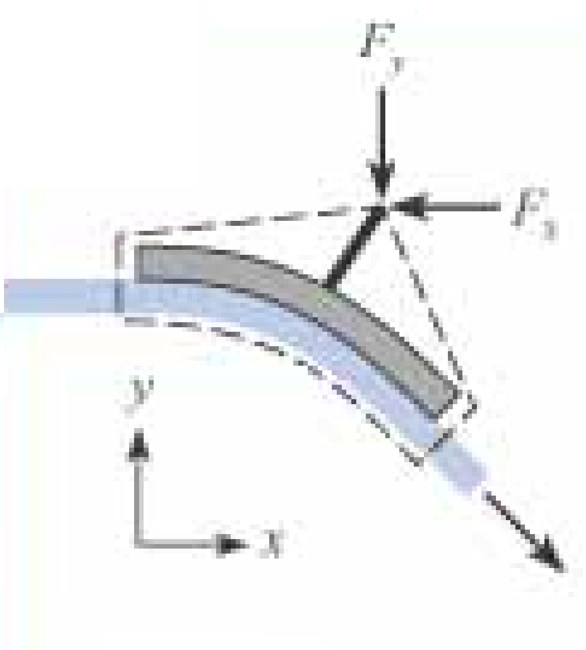
Figure 6.4 *Fluid jet striking a flat vane*.

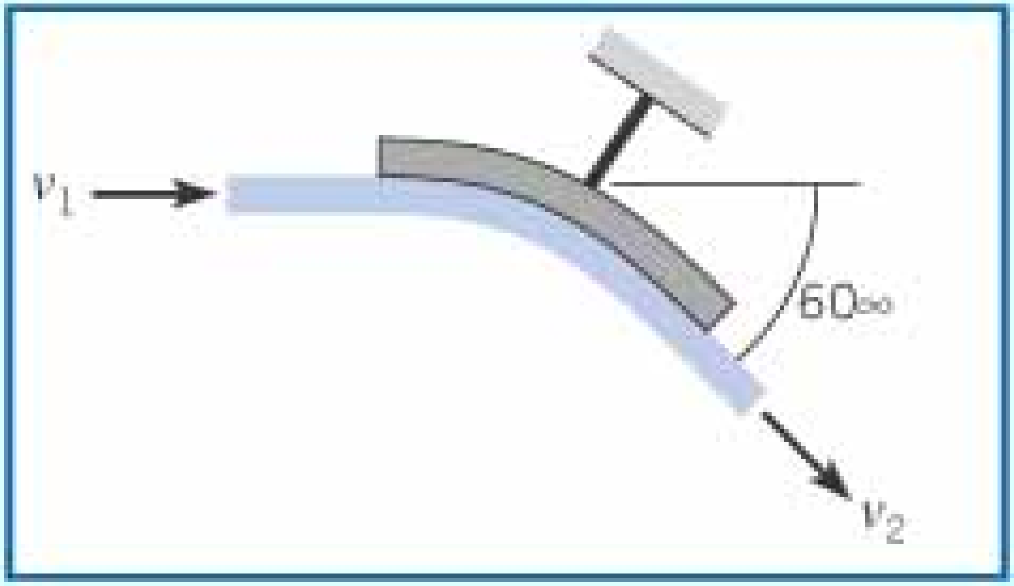
EXAMPLE 6.4 WATER DEFLECTED BY A VANE

A water jet is deflected 60° by a stationary vane as shown in the figure. The incoming jet has a speed of 30 m/s and a diameter of 3 cm. Find the force exerted by the jet on the vane. Neglect the influence of gravity.

Solution

The control volume selected is shown in the sketch to the left. The control volume is stationary.





The momentum:

$$\sum F = \frac{d}{dt}\int_{cv} \rho \mathbf{v}\, d\mathcal{V} + \sum_{cs} \dot{m}_o \mathbf{v}_o - \sum_{cs} \dot{m}_i \mathbf{v}_i$$

The force vector is

$$\sum F = -F_x \mathbf{i} - F_y \mathbf{j}$$

The control volume is stationary and flow is steady leads to the accumulation term equals to zero. The momentum outflow,

$$\sum_{cs} \dot{m}_o \mathbf{v}_o = [(\dot{m}v \cos 60°)\mathbf{i} - (\dot{m}v \sin 60°)\mathbf{j}]\,.$$

The momentum inflow,

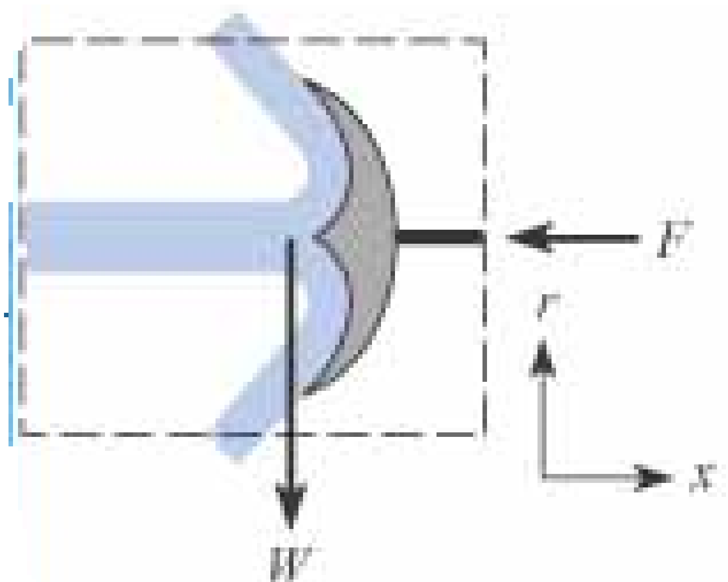
$$\sum_{cs} \dot{m}_i \mathbf{v}_i = \dot{m}v\mathbf{i}$$

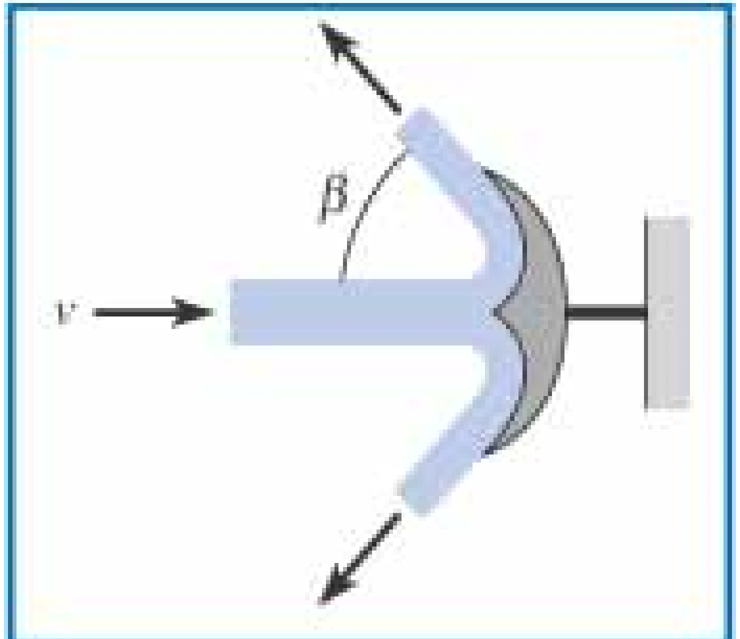
EXAMPLE 6.5 FORCE ON AN AXISYMMETRIC VANE

As shown in the figure, an incident jet of fluid with density ρ, speed v, and area *A* is deflected through an angle β by a stationary, axisymmetric vane. Find the force required to hold the vane stationary. Express the answer using ρ, v, *A*, and β. Neglect the influence of gravity.

Solution

The selected control volume is shown. The control volume is stationary





The momentum equation in *x*-direction.

$$\sum F_x = \frac{d}{dt}\int_{cv} v_x \rho \, d\Psi + \sum_{cs} \dot{m}_o v_{ox} - \sum_{cs} \dot{m}_i v_{ix}$$

The accumulation term is zero since the flow is zero. The sum of forces

$$\sum F_x = -F$$

Momentum outflow is $\sum_{cs} \dot{m}_o v_{ox} = -\dot{m}v \cos\beta$

Momentum inflow is $\sum_{cs} \dot{m}_i v_{ix} = \dot{m}v$

Force on vane

$$-F = -\dot{m}v(1 + \cos\beta)$$

$$F = \dot{m}v(1 + \cos\beta)$$

Apply mass flow rate equation

$$\boxed{F = \rho A v^2 (1 + \cos\beta)}$$

Pipe Bends

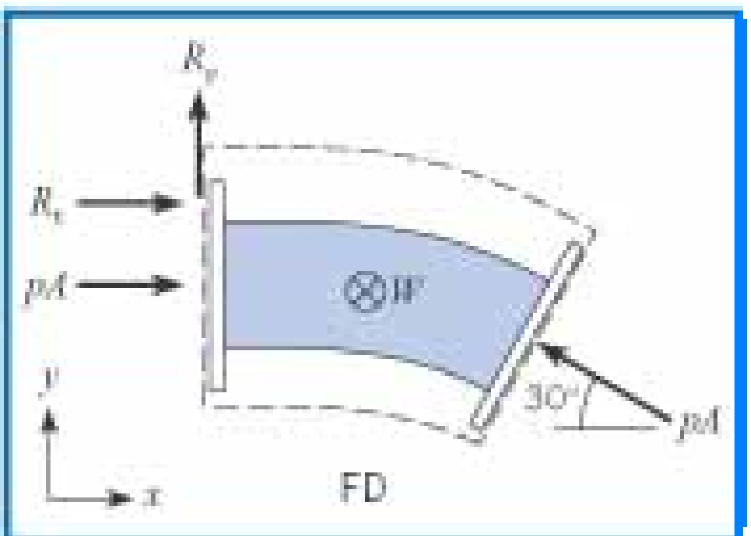
Calculating the force on pipe bends is important in engineering applications using large pipes to design the support system. Because flow in a pipe is usually turbulent, it is common practice to assume that velocity is nearly constant across each cross section of the pipe. Also, the force acting on a pipe cross section is given by pA, where p is the pressure at the centroid of area and A is area.

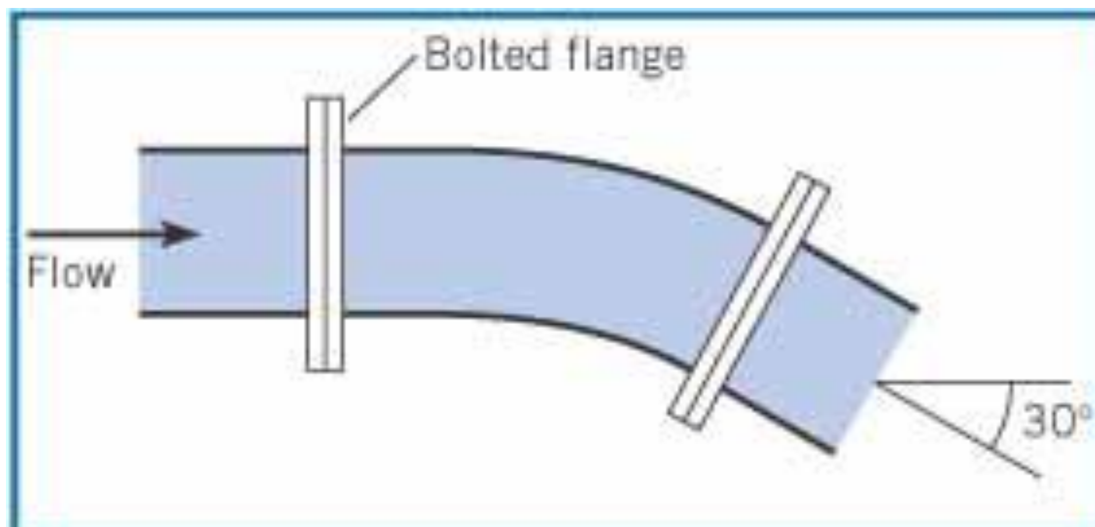
EXAMPLE 6.6 FORCES ACTING ON A PIPE BEND

A 1 m–diameter pipe bend shown in the diagram is carrying crude oil ($S = 0.94$) with a steady flow rate of 2 m^3/s. The bend has an angle of 30° and lies in a horizontal plane. The volume of oil in the bend is 1.2 m^3, and the empty weight of the bend is 4 kN. Assume the pressure along the centerline of the bend is constant with a value of 75 kPa gage. Find the net force required to hold the bend in place.

Solution

The control volume is shown which is stationary.





Solution

The flow is steady (accumulation term is zero) and there is no flow in z-direction, thus

Momentum equation in x-dir:

$$R_x + pA - pA\cos 30^\circ = \dot{m}v\cos 30^\circ - \dot{m}v$$

Momentum equation in the y-dir $R_y + pA\sin 30^\circ = -\dot{m}v\sin 30^\circ$

Momentum equation in the y-dir $R_z - W = 0$

The fluid velocity, pressure force and momentum flux are

$$v = Q/A = \frac{(2\ \text{m}^3/\text{s})}{(\pi \times 0.5^2\ \text{m}^2)} = 2.55\ \text{m/s}$$

$$pA = (75\ \text{kN/m}^2)(\pi \times 0.5^2\ \text{m}^2) = 58.9\ \text{kN}$$

$$\dot{m}v = \rho Q v = (0.94 \times 1000\ \text{kg/m}^3)(2\ \text{m}^3/\text{s})(2.55\ \text{m/s})$$
$$= 4.79\ \text{kN}$$

Therefore, the components of the required force are:

$$R_x = -(pA + \dot{m}v)(1 - \cos 30°)$$

$$= -(58.9 + 4.79)(\text{kN})(1 - \cos 30°) = \boxed{-8.53 \text{ kN}}$$

$$R_y = -(pA + \dot{m}v) \sin 30°$$

$$= -(58.9 + 4.79)(\text{kN})(\sin 30°) = \boxed{-31.8 \text{ kN}}$$

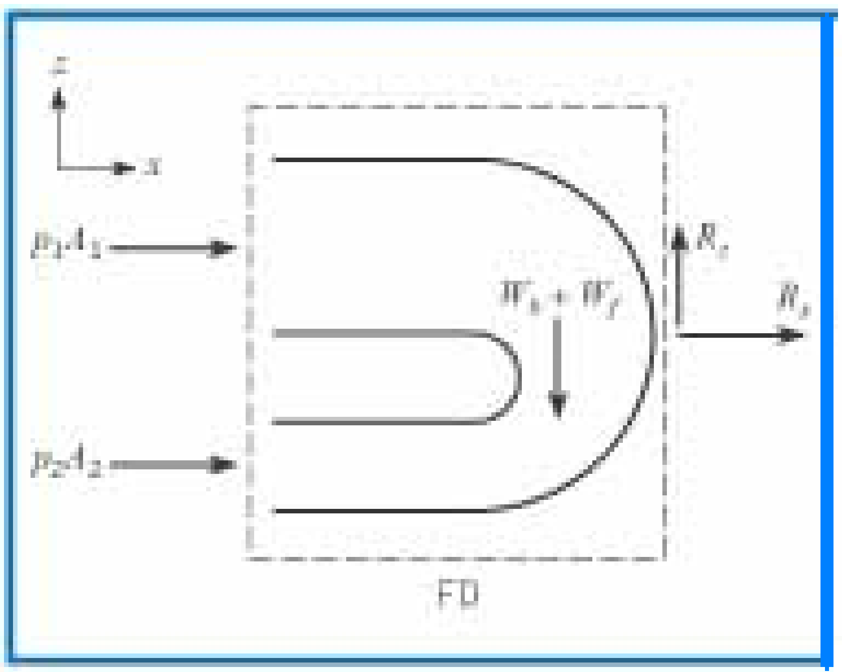
$$W = \gamma \mathcal{V} + 4 \text{ kN}$$

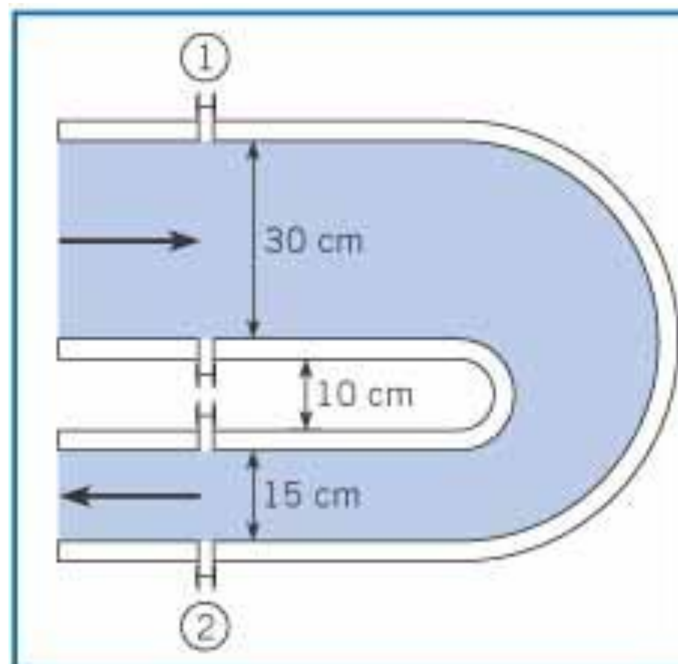
$$= (0.94 \times 9.81 \text{ kN/m}^3)(1.2 \text{ m}^3) + 4 \text{ kN} = \boxed{15.1 \text{ kN}}$$

where $R_z = W$

EXAMPLE 6.7 WATER FLOW THROUGH REDUCING BEND

Water flows through a 180° reducing bend, as shown. The discharge is 0.25 mm/s, and the pressure at the center of the inlet section is 150 kPa gage. If the bend volume is 0.10 m3, and it is assumed that the Bernoulli equation is valid, what force is required to hold the bend in place? The metal in the bend weighs 500 N. The water density is 1000 kg/m^3. The bend is in the vertical plane.





Solution

Momentum equations in *x*- and *z*-directions

$$\sum F_x = \frac{d}{dt}\int_{cv} v_x\,\rho\,dV + \sum_{cs} \dot{m}_O v_{Ox} - \sum_{cs} \dot{m}_I v_{Ix}$$

$$\sum F_z = \frac{d}{dt}\int_{cv} v_z\,\rho\,dV + \sum_{cs} \dot{m}_O v_{Oz} - \sum_{cs} \dot{m}_I v_{Iz}$$

Summation of forces in *x*- and *z*-directions

$$\sum F_x = p_1 A_1 + p_2 A_2 + R_x$$

$$\sum F_z = R_z - W_b - W_f$$

Hence, momentum in x-dir

$$p_1 A_1 + p_2 A_2 + R_x = -\rho Q(v_2 + v_1)$$

$$R_x = -(p_1 A_1 + p_2 A_2) - \rho Q(v_2 + v_1)$$

Hence, momentum in x-dir

$$R_z = W_b + W_f$$

Inlet and outlet speeds

$$v_1 = \frac{Q}{A_1} = \frac{0.25\ \text{m}^3/\text{s}}{\pi/4 \times 0.3^2\ \text{m}^2} = 3.54\ \text{m/s}$$

$$v_2 = \frac{Q}{A_2} = \frac{0.25\ \text{m}^3/\text{s}}{\pi/4 \times 0.15^2\ \text{m}^2} = 14.15\ \text{m/s}$$

Outlet pressure (the Bernoulli equation between sections 1 and 2)

$$p_1 + \frac{\rho v_1^2}{2} + \gamma z_1 = p_2 + \frac{\rho v_2^2}{2} + \gamma z_2$$

From diagram, neglecting pipe wall thickness, $z1 - z2 = 0.325$ m.

$$p_2 = p_1 + \frac{\rho(v_1^2 - v_2^2)}{2} + \gamma(z_1 - z_2)$$

$$= 150\ \text{kPa} + \frac{(1000)(3.54^2 - 14.15^2)\text{Pa}}{2} + (9810)(0.325)\text{Pa}$$

$$= 59.3\ \text{kPa}$$

Simplifying terms,

$$p_1A_1 + p_2A_2 = (150\text{ kPa})(\pi \times 0.3^2 / 4\text{ m}^2) + (59.3\text{ kPa})(\pi \times 0.15^2 / 4\text{ m}^2)$$
$$= 11.6\text{ kN}$$

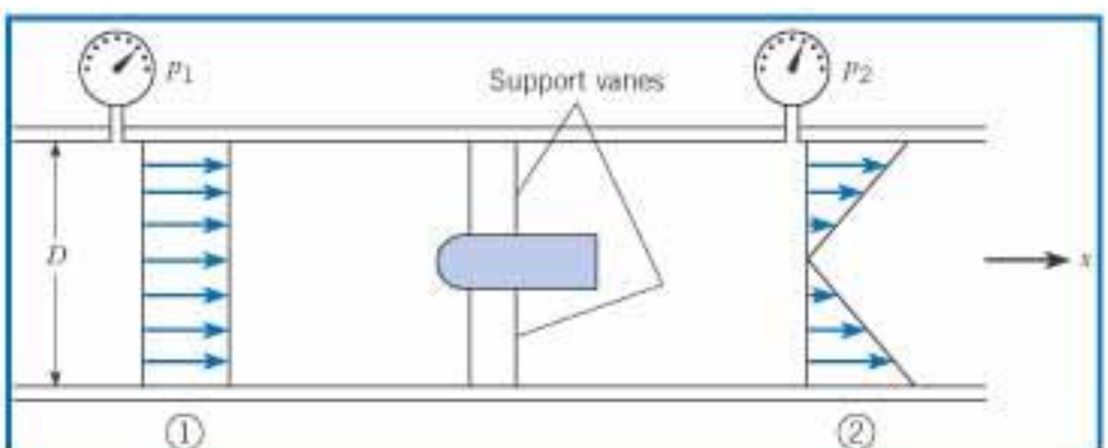
$$\rho Q(v_2 + v_1) = (1000\text{ kg / m}^3)(0.25\text{ m}^3) \times (14.15 + 3.54)(\text{m / s})$$
$$= 4420\text{ N}$$

Hence, the components of the reaction force,

$$R_x = -(11.6\text{ kN}) - (4.42\text{ kN})$$
$$= \boxed{-16.0\text{ kN}}$$
$$R_z = W_b + W_f$$
$$= 500\text{ N} + (9810\text{ N / m}^3)(0.1\text{ m}^3)$$
$$= \boxed{1.48\text{ kN}}$$

EXAMPLE 6.8 DRAG FORCE ON WIND-TUNNEL MODEL

The drag force of a bullet-shaped device may be measured using a wind tunnel. The tunnel is round with a diameter D 1 m, the pressure at section 1 is 1.5 kPa gage, the pressure at section 2 is 1.0 kPa gage, and air density is 1.0 kg/m^3. At the inlet, the velocity is uniform with a magnitude of 30 m/s. At the exit, the velocity varies linearly as shown in the sketch. Determine the drag force on the device and support vanes. Neglect viscous resistance at the wall, and assume pressure is uniform across sections 1 and 2.



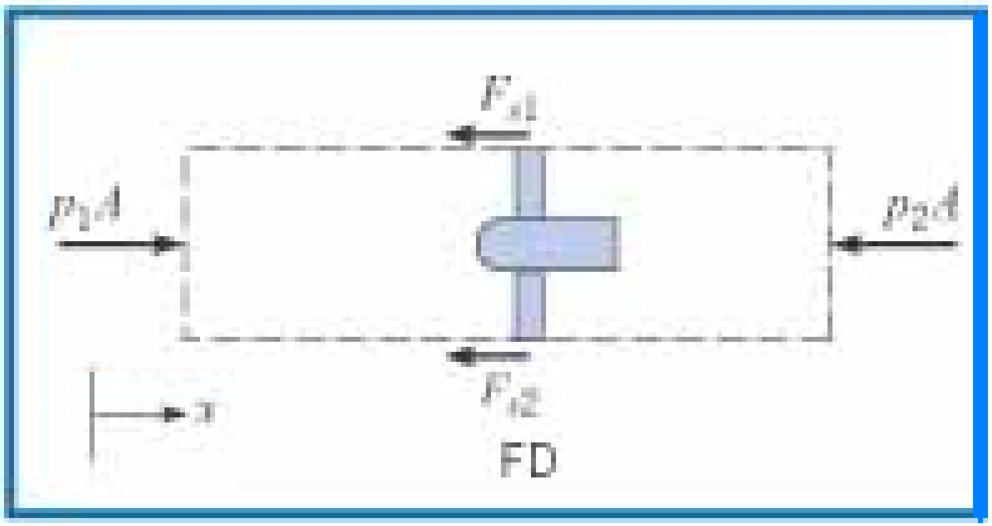
Integral form of momentum equation in *x*-direction

$$\sum F_x = \frac{d}{dt}\int_{cv} \rho v_x dV + \int_{cs} \rho v_x (V \cdot dA)$$

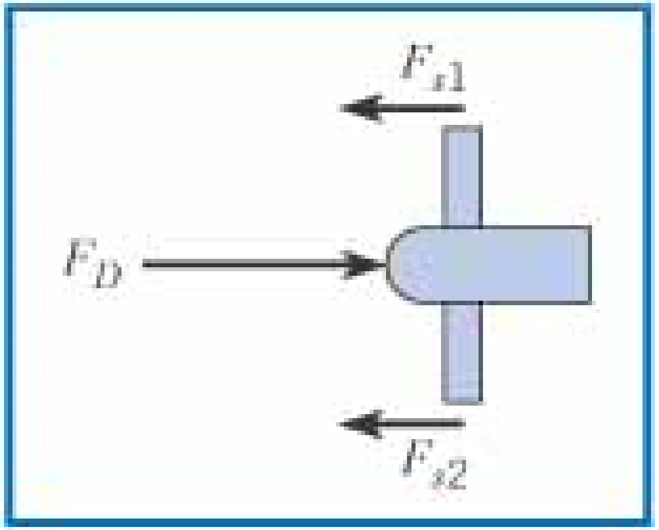
Summation of forces acting on control volume.

$$\sum F_x = p_1 A - p_2 A - (F_{s1} + F_{s2})$$
$$= p_1 A - p_2 A - F_D$$

Note that the F_D is the sum of the two forces (F_{s1}, F_{s2}) and also represents their directions; i.e. forces on the control volume.



Forces on control volume



Forces on model

Need to determine velocity profile at section 2. Velocity is linear in radius, so choose $v_2 = v_1\, K(r/r_o)$, where r_o is the tunnel radius and K is a proportionality factor to be determined as follow,

$$Q_1 = Q_2 \quad (\textit{incompressible flow})$$

$$A_1 v_1 = \int_{A_2} v_2(r)\,dA = \int_0^{r_o} v_1 K(r/r_o) 2\pi r\, dr$$

$$\pi r_o^2 v_1 = 2\pi v_1 K \frac{1}{3} r_o^2$$

$$K = \frac{3}{2}$$

The flow is steady (accumulation term is zero). The momentum flux into the control volume (at section 1) since flow is uniform,

$$\int_1 \rho v_x^2\, dA = \rho v_1^2 A = \dot{m} v_1$$

The momentum flux out of the control volume (at section 2) since flow is non-uniform,

$$\int_2 \rho v_x^2\, dA = \int_0^{r_o} \rho \left[\frac{3}{2} v_1 \left(\frac{r}{r_o}\right)\right]^2 2\pi r dr = \frac{9}{8} \dot{m} v_1$$

Hence, the drag force is found by substituting into the x-dir momentum equation,

$$p_1 A - p_2 A - F_D = \dot{m} v_1 \left(\frac{9}{8} - 1 \right)$$

$$F_D = (p_1 - p_2) A - \frac{1}{8} \rho A v_1^2$$

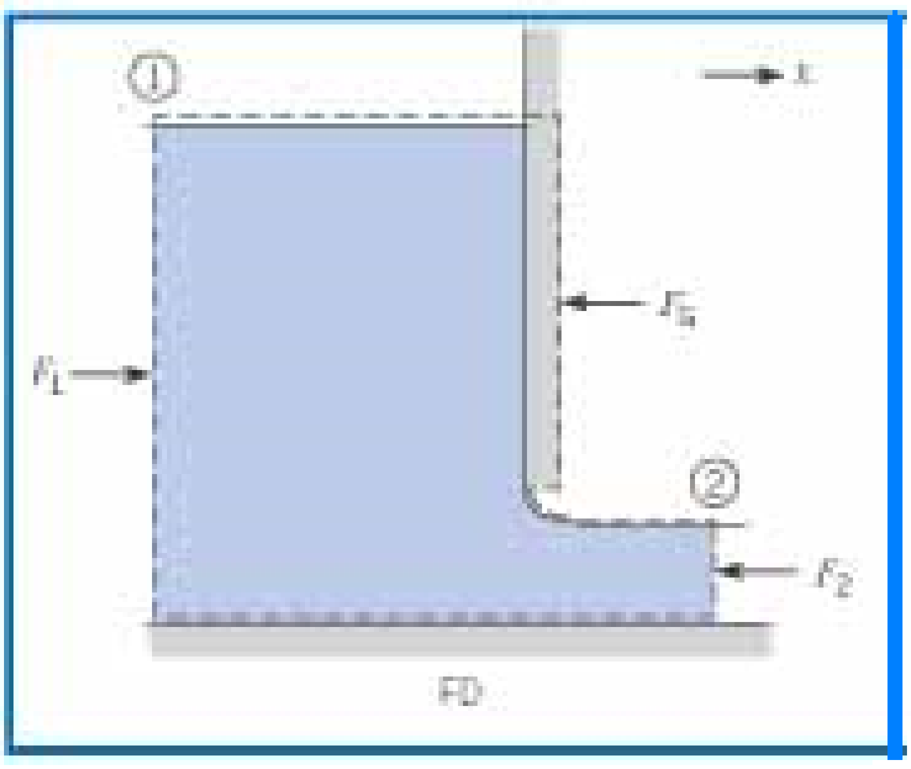
$$= (\pi \times 0.5^2 \text{ m}^2)(1.5 - 1.0)(10^3) \text{ N/m}^2$$

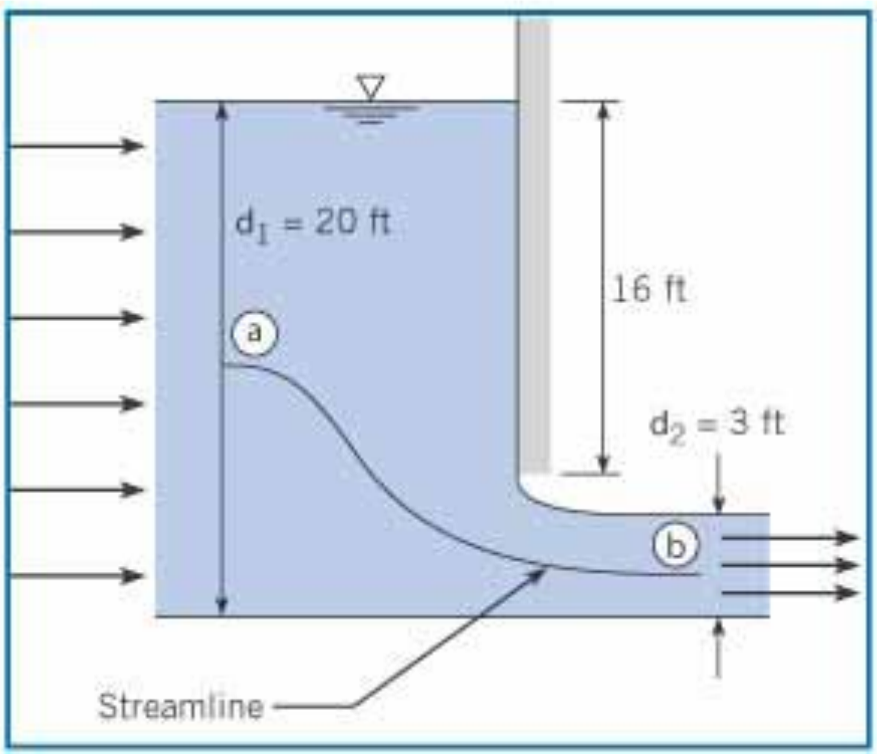
$$- \frac{1}{8} (1 \text{ kg/m}^3)(\pi \times 0.5^2 \text{ m}^2)(30 \text{ m/s})^2$$

$$F_D = \boxed{304 \text{ N}}$$

EXAMPLE 6.9 FORCE ON A SLUICE GATE

A sluice gate is used to control the water flow rate over a dam. The gate is 6 m wide, and the depth of the water above the bottom of the sluice gate is 5 m. The depth of the water upstream of the gate is 6 m, and the depth downstream is 1 m. Estimate the flow rate under the gate and the force on the gate. The water density is 1000 kg/m³.





Momentum in x-dir

$$\sum F_x = \frac{d}{dt}\int_{cv} v_x \rho \, d\forall + \sum_{cs} \dot{m}_o v_{ox} - \sum_{cs} \dot{m}_i v_{ix}$$

The Bernoulli equation

$$\frac{p_a}{\gamma} + z_a + \frac{v_1^2}{2g} = \frac{p_b}{\gamma} + z_b + \frac{v_2^2}{2g}$$

The piezometric pressure is constant across sections 1 and 2, so

$$\frac{p_a}{\gamma} + z_a = d_1 \qquad \text{and} \qquad \frac{p_b}{\gamma} + z_b = d_2$$

From continuity equation, Eq. (5.27), $(v_1 d_1 w) = (v_2 d_2 w)$ where w is the flow width. Combine the Bernoulli and continuity equations.

$$2g(d_1 - d_2) = v_2^2 - v_1^2 = v_2^2\left(1 - \frac{d_2^2}{d_1^2}\right)$$

$$v_2 = \frac{1}{\sqrt{1 - \frac{d_2^2}{d_1^2}}}\sqrt{2g(d_1 - d_2)}$$

Velocity and discharge,

$$v_2 = \frac{1}{\sqrt{1-\left(\frac{3}{20}\right)^2}}\sqrt{2\times 32.2\ \text{ft}/s^2\times(20-3)\ \text{ft}} = 33.5\ \text{ft}/s$$

$$v_1 = \frac{d_2}{d_1}v_2 = 33.5\ \text{ft}/s \times \frac{3\ \text{ft}}{20\ \text{ft}} = 5.02\ \text{ft}/s$$

$$Q = v_2 d_2 w = 33.5\ \text{ft}/s\times 3\ \text{ft}\times 20\ \text{ft} = \boxed{2010\ \text{ft}^3/s}$$

The forces acting on the control volume,

10.045 m/s

$$\sum F_x = F_1 - F_2 - F_{G}$$

The hydrostatic force on planar surface, $F = \overline{p}A$

$$F_1 = \frac{\gamma d_1}{2} d_1 w$$

$$F_2 = \frac{\gamma d_2}{2} d_2 w$$

Accumulation term is zero (steady), and the momentum fluxes at inlet and outlet are:

$$\sum_{cs} \dot{m}_i v_{ix} = \dot{m} v_1 \quad \text{and} \quad \sum_{cs} \dot{m}_o v_{ox} = \dot{m} v_2$$

Hence, the force on the sluice gate is,

$$\frac{\gamma}{2} d_1^2 w - \frac{\gamma}{2} d_2^2 w - F_G = \dot{m}(v_2 - v_1)$$

$$F_G = \frac{\gamma}{2} w (d_1^2 - d_2^2) + \rho Q (v_1 - v_2)$$

$$F_G = \frac{62.4\ \text{lbf/ft}^3}{2} \times 20\ \text{ft} \times (20^2 - 3^2)(\text{ft})^2$$

$$+ 1.94\ \text{slug/ft}^3 \times 2010\ \text{ft}^3/\text{s}$$

$$\times (5.2 - 33.5)(\text{ft/s})$$

$$= 1.33 \times 10^5\ \text{lbf} \times \frac{1\ \text{ton}}{2000\ \text{lbf}} = \boxed{66.5\ \text{tons}}$$

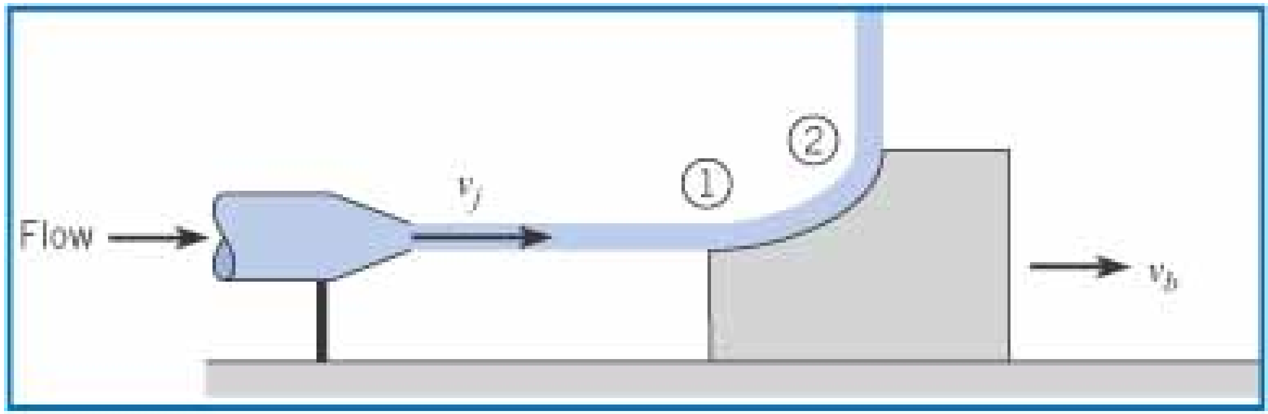
Moving Control Volumes

All the applications of the momentum equation up to this point have involved a stationary control volume. However, in some problems it may be more useful to attach the control volume to a moving body.

As discussed previously, the velocity **v** in the momentum equation must be relative to an inertial reference frame. When applying the momentum equation each mass flow rate is calculated using the velocity with respect to the control surface, but the velocity ***v*** must be evaluated with respect to an inertial reference frame.

EXAMPLE 6.10 JET IMPINGING ON MOVING BLOCK

A stationary nozzle produces a water jet with a speed of 50 m/s and a cross-sectional area of 5 cm^2. The jet strikes a moving block and is deflected 90° relative to the block. The block is sliding with a constant speed of 25 m/s on a surface with friction. The density of the water is 1000 kg/m^3. Find the frictional force *F* acting on the block. Solve the problem using two different inertial reference frames: (a) the moving block and (b) the stationary nozzle.



The selected control volume is shown in the sketch. Observe, the cart is not stationary.

The momentum equation in the x-dir is,

$$\sum F_x = \frac{d}{dt}\int_{cv} v_x \, \rho \, d\Psi + \sum_{cs} \dot{m}_o v_{ox} - \sum_{cs} \dot{m}_i v_{ix}$$

The sum of forces is,

$$\sum F_x = -F$$

The accumulation term is zero (steady). The momentum inflow and outflow are,

Case 1: Frame of ref is attached to control volume

$$\sum_{cs} \dot{m}_i v_{ix} = \dot{m}(v_j - v_b)$$

and $$\sum_{cs} \dot{m}_o v_{ox} = 0$$

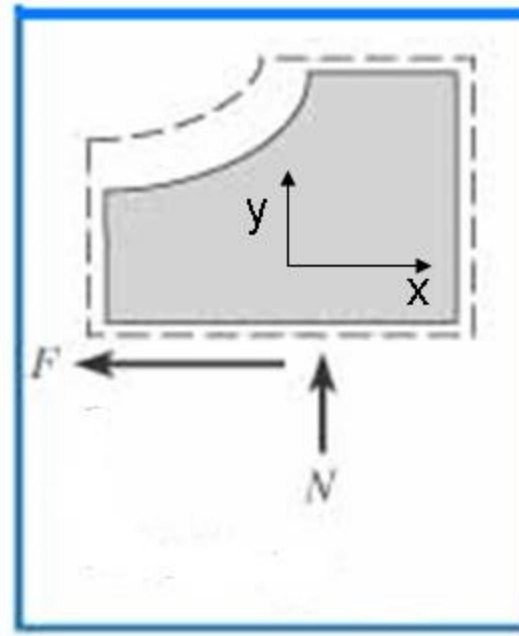
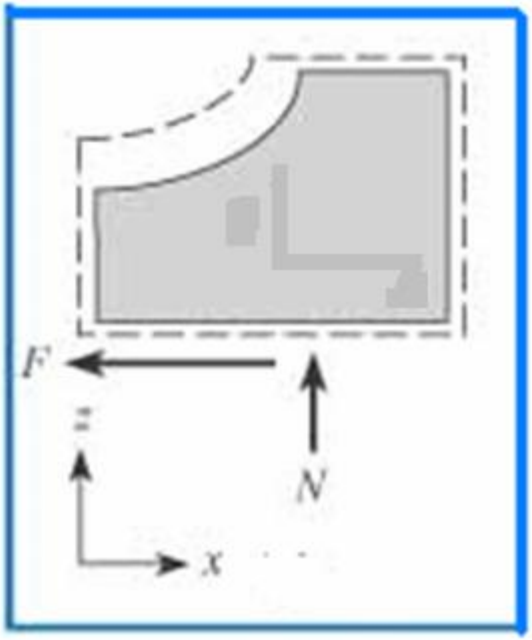


Figure showing a moving control volume

The mass flow rate. Since flow is steady with respect to the block,



$$\dot{m}_1 = \dot{m}_O = \dot{m}$$

$$\dot{m} = \rho A\ (v_j - v_b)$$

To evaluate force Case 1:

$$-F = -\rho A(v_j - v_b)^2$$

$$F = \rho A(v_j - v_b)^2$$

To evaluate force Case 2. Frame of reference is attached to ground

$$-F = \dot{m}v_b - \dot{m}v_j = -\dot{m}(v_j - v_b)$$

$$F = \rho A(v_j - v_b)^2$$

Which both give the same answer, as they should. Evaluate the magnitude of the force,

$$F_x = (1000\ \text{kg}/\text{m}^3)(5\times 10^{-4}\ m^2)(50-25)^2(m/s)^2$$

$$F_x = \boxed{312\ \text{N}}$$

Water Hammer: Physical Description

Whenever a valve is closed in a pipe, a positive **pressure wave** is created upstream of the valve and travels up the pipe at the **speed of sound**. In this context a positive pressure wave is defined as one for which the **pressure is greater than the existing steady-state pressure**. This pressure wave may be great enough to cause **pipe failure**. This process of pressure wave is called *water hammer,* is necessary for the proper design and operation of such systems.

The magnitude of the pressure Δp is

$$\Delta p = \rho V c$$

where ρ is fluid density, V is its velocity and c is the speed of sound in that fluid, which computed as,

$$c = \sqrt{\frac{E_v}{\rho}}$$

where E_v is the bulk modulus of elasticity of the fluid. For water, where E_v = 2.2 GPa, hence c = 1483 m/s.

6.5 Moment-of-Momentum Equation

The moment-of-momentum equation is very useful for situations that involve torques. Examples include analyses of rotating machinery such as pumps, turbines, fans, and blowers.

Torques acting on a control volume are related to **changes in angular momentum** through the moment of momentum equation. Development of this equation parallels the development of the momentum equation as presented previously. When forces act on a system of particles, used to represent a fluid system, Newton's second law of motion can be used to derive an equation for rotational motion:

$$\sum M = \frac{d(H_{sys})}{dt} \qquad (6.24)$$

where ***M*** is a moment and $\boldsymbol{H}_{sys}$ is the total angular momentum of all mass forming the system

Equation (6.24) is a Lagrangian equation, which can be converted to an Eulerian form using the Reynolds transport theorem. The extensive property B_{sys} becomes the angular momentum of the system: $B_{sys} = \boldsymbol{H}_{sys}$. The intensive property b becomes the angular momentum per unit mass. The angular momentum of an element is $\boldsymbol{r} \times m\boldsymbol{v}$, and so $b = \boldsymbol{r} \times \boldsymbol{v}$. Substituting for B_{sys} and b into the Reynolds transport theorem gives

$$\frac{d(\boldsymbol{H}_{sys})}{dt} = \frac{d}{dt}\int_{cv} (\boldsymbol{r} \times \boldsymbol{v})\rho \, d\mathcal{V} + \int_{cs} (\boldsymbol{r} \times \boldsymbol{v})\rho \, \boldsymbol{V} \cdot d\boldsymbol{A}$$

or,

$$\sum \boldsymbol{M} = \frac{d}{dt}\int_{cv} (\boldsymbol{r} \times \boldsymbol{v})\rho \, d\mathcal{V} + \int_{cs} (\boldsymbol{r} \times \boldsymbol{v})\rho \, \boldsymbol{V} \cdot d\boldsymbol{A}$$

If the mass crosses the control surface through a series of inlet and outlet ports with uniformly distributed properties across each port, the moment-of-momentum equation becomes

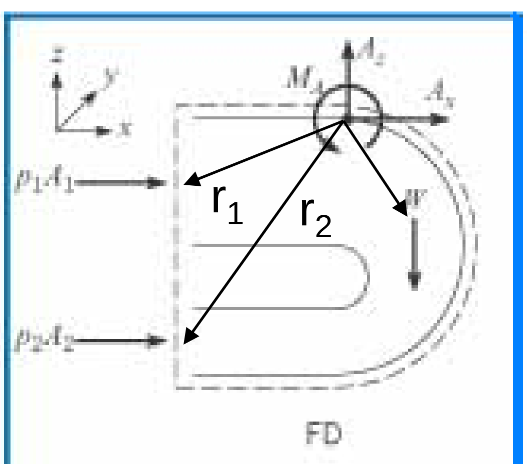
$$\sum \boldsymbol{M} = \frac{d}{dt}\int_{cv} (\boldsymbol{r} \times \boldsymbol{v})\rho \, d\mathcal{V} + \sum_{cs} \boldsymbol{r}_O \times (\dot{m}_O \boldsymbol{v}_O) - \sum_{cs} \boldsymbol{r}_i \times (\dot{m}_i \boldsymbol{v}_i)$$

As the case with momentum equation, the velocities must be with respect to an inertial frame of reference.

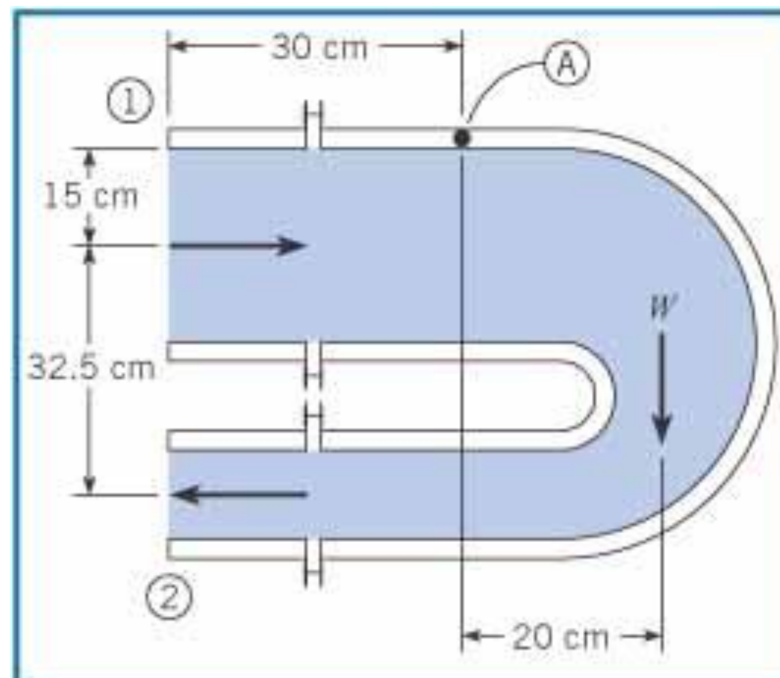
EXAMPLE 6.13 RESISTING MOMENT ON REDUCING BEND

The reducing bend shown in the figure is supported on a horizontal axis through point A. Water flows through the bend at 0.25 m3/s. The inlet pressure at cross-section 1 is 150 kPa gage, and the outlet pressure at section 2 is 59.3 kPa gage. A weight of 1420 N acts 20 cm to the right of point A. Find the moment the support system must resist. The diameters of the inlet and outlet pipes are 30 cm and 10 cm, respectively.

Properties: From Table A.5,
ρ = 998 kg/m3.



Control volume



Apply the moment-of-momentum equation,

$$\sum \mathbf{M}_A = \frac{d}{dt}\int_{cv} (\mathbf{r} \times \mathbf{v})\rho \, dV + \sum_{cs} \mathbf{r}_O \times (\dot{m}_O \mathbf{v}_O) - \sum_{cs} \mathbf{r}_i \times (\dot{m}_i \mathbf{v}_i)$$

Sum of moments (due to external forces, and note the $\mathbf{r}$ vector for each force, also choose clockwise to be positive) about axis A,

$$\sum \mathbf{M}_A = -(M_A + 0.15 p_1 A_1 + 0.475 p_2 A_2 - 0.2W)\mathbf{j}$$

The flow is steady, thus the accumulation is zero. The inflow and outflow of angular momentum are,

$$\sum_{cs} \mathbf{r}_i \times (\dot{m}_i \mathbf{v}_i) = \mathbf{r}_1 \times (\dot{m}\mathbf{v}_1) = -r_1 \dot{m} v_1 \mathbf{j}$$

and

$$\sum_{cs} \mathbf{r}_O \times (\dot{m}_O \mathbf{v}_O) = \mathbf{r}_2 \times (\dot{m}\mathbf{v}_2) = r_2 \dot{m} v_2 \mathbf{j}$$

The resisting moment at *A*

$$M_A = -0.15 p_1 A_1 - 0.475 p_2 A_2 + 0.2W - \dot{m}(r_2 v_2 + r_1 v_1)$$

Compute terms: torque due to pressure

$$0.15 p_1 A_1 = (0.15\ \text{m})(150 \times 1000\ \text{N}/\text{m}^2)(\pi \times 0.3^2/4\ \text{m}^2)$$
$$= 1590\ \text{N} \cdot \text{m}$$
$$0.475 p_2 A_2 = (0.475\ \text{m})(59.3 \times 1000\ \text{N}/\text{m}^2)(\pi \times 0.15^2/4\ \text{m}^2)$$
$$= 498\ \text{N} \cdot \text{m}$$

Net moment-of-momentum flow

$$\dot{m} = \rho Q = (998\ \text{kg}/\text{m}^3)(0.25\ \text{m}^3/s)$$
$$= 250\ \text{kg}/s$$
$$v_1 = \frac{Q}{A_1} = \frac{0.25\ \text{m}^3/s}{\pi \times 0.15^2 \text{m}^2} = 3.54\ \text{m}/s$$
$$v_2 = \frac{Q}{A_2} = \frac{0.25\ \text{m}^3/s}{\pi \times 0.075^2 \text{m}^2} = 14.15\ \text{m}/s$$
$$\dot{m}(r_2 v_2 + r_1 v_1) = (250\ \text{kg}/s) \times (0.475 \times 14.15 + 0.15 \times 3.54)(\text{m}^2/s)$$
$$= 1813\ \text{N} \cdot \text{m}$$

Moment exerted by support

$$M_A = -0.15p_1A_1 - 0.475p_2A_2 + 0.2W - \dot{m}(r_2v_2 + r_1v_1)$$

$$= -(1590\ \text{N}\cdot\text{m}) - (498\ \text{N}\cdot\text{m})$$

$$+ (0.2\ \text{m} \times 1420\ \text{N}) - (1813\ \text{N}\cdot\text{m})$$

$$M_A = \boxed{-3.62\ \text{kN}\cdot\text{m}}$$

Thus, a moment of 3.62 kN **.** m acting in the **j**, or clockwise, direction is needed to hold the bend stationary. Stated differently, the **support system must** be designed to withstand a counterclockwise moment of 3.62 kN **.** m.

6.6 Navier-Stokes Equation

The continuity equation at a point in the flow was derived using a control volume of infinitesimal size (chapter 5). The resulting differential equation is an independent equation in the analysis of fluid flow. The same approach can be applied to the momentum equation, yielding the differential equation for momentum at a point in the flow. For simplicity, the derivation will be restricted to a two-dimensional planar flow, and the extension to three dimensions will be outlined.

Consider the infinitesimal control volume shown in Fig. 6.10*a*. The dimensions of the control volume are Δx and Δy, and the dimension in the third direction (normal to page) is taken as unity. Assume that the center of the control volume is fixed with respect to the coordinate system and that the coordinate system is an inertial reference frame. Also assume that the control surfaces are fixed with respect to the coordinate system. The

In the derivation of the differential form of the momentum equation, one starts with the integral form of the equation and apply it to the small control volume. For example, the forces due to pressure in the x-dir are,

$$F_{x,p} = (p|_{x-\Delta x/2} - p|_{x+\Delta x/2})\Delta y$$

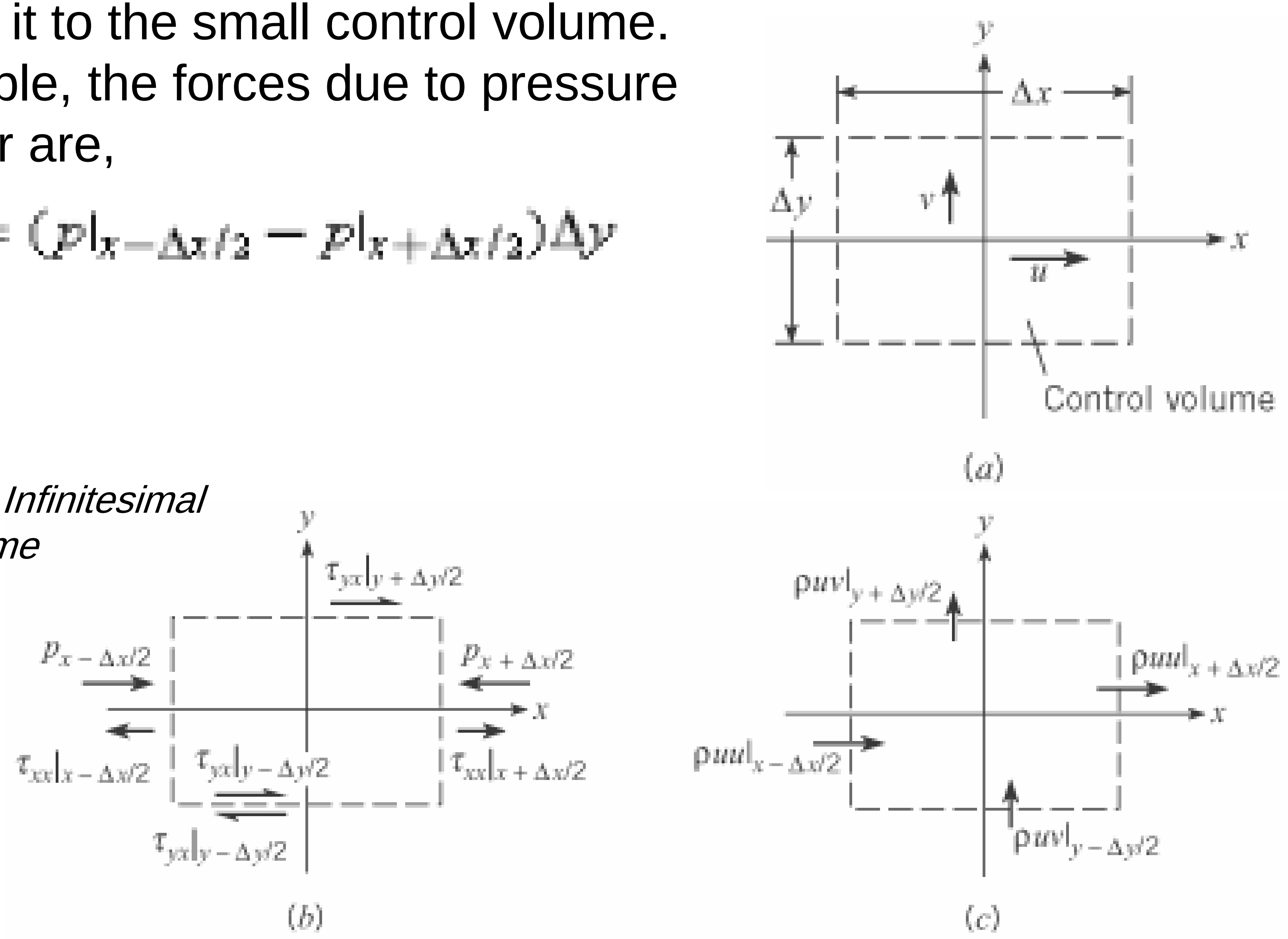


Figure 6.10 *Infinitesimal control volume*

In a similar manner shear force can be dealt with. In the end for incompressible fluid with constant properties, the final form for the momentum differential equation will be,

x-dir: $$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] + \rho g_x$$

y-dir: $$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] + \rho g_y$$

The left terms in both equation represent density fluid acceleration, thus the units are Newton per unit volume. These two terms can be expanded as,

$$\rho \frac{Du}{Dt} = \rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y}$$

$$\rho \frac{Dv}{Dt} = \rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y}$$

The physical meaning for each term in the equation is shown below. Notice this is a form of Newton's second law applied to a fluid particle (the units are per unit volume)

Body (gravity) forces

Net shear forces

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] + \rho g_x$$

Net pressure forces in x-dir

Acceleration times density

Example

In chapter 5 your were given the velocity field for a flow to be

V = 10xi – 10yj

where x and y are given in meters. Let the gravity vector be defined in the y direction. Find the pressure gradient in the x-direction at location x =1 m and y=1 m. Let density be that for water 1000 kg/m^3.

Solution. The flow satisfies the continuity equation as was shown previously. Also, observe if the shear stress is zero upon evaluation of the second partial of the velocity u. Also, the gravity in the x-direction is zero. The flow is also steady since is not a function of time. From the Navier-Stokes equation in the x-dir with no gravity and shear forces, the equation simplifies to

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x}$$

Thus, to compute the pressure gradient at x = 1 m and y = 1 m, only need to compute the acceleration at this location.

The acceleration in the x-direction is

$$\rho\frac{Du}{Dt} = \rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y}$$

The first term on the right hand side is zero since the flow is steady. Also, the third term is zero since the u component is not a function of y. The second term is evaluated as,

$$\frac{\partial u}{\partial x} = 10$$

Hence, $\rho\frac{Du}{Dt} = \rho u\frac{\partial u}{\partial x} = -\frac{\partial p}{\partial x}$ = (1000) x (10 x 1) x 10

= 100 kN/m^3

Fluid Mechanics

Chapter 7
The Energy Equation

Dr. Amer Khalil Ababneh

7.2 Energy Equation: General Form

The energy equation for a system is written as

$$\dot{Q} - \dot{W} = \frac{dE}{dt}$$

In words, this is stated as,

$$\left\{ \begin{matrix} \text{net rate of} \\ \text{thermal energy} \\ \text{enterning system} \end{matrix} \right\} - \left\{ \begin{matrix} \text{net rate at which} \\ \text{system does work} \\ \text{on environment} \end{matrix} \right\} = \left\{ \begin{matrix} \text{rate of change of} \\ \text{energy of the mater} \\ \text{within the system} \end{matrix} \right\}$$

In this case, E is the energy of the system and it is an extensive property. To make it an intensive simply divide by mass; i.e., $e = E/m$. Thus, using the Reynolds transport theorem,

$$\dot{Q} - \dot{W} = \frac{d}{dt} \int_{cv} e\rho d\mathcal{V} + \int_{cs} e\rho \mathbf{V} \cdot d\mathbf{A}$$

This integral form of the energy equation written for a control volume (Eulerian form)

In general, in fluids there are three forms of energy we are interested in; kinetic, potential and internal energies. The kinetic energy per unit mass is $V^2/2$, the potential energy per unit mass is gz, and the internal energy per unit mass is given the symbol u. Thus,

$$e = u + V^2/2 + gz$$

Substituting for e into the integral form of the energy equation leads to

$$\dot{Q} - \dot{W} = \frac{d}{dt}\int_{\mathrm{cv}} \left(\frac{V^2}{2} + gz + u\right)\rho\, d\mathcal{V} + \int_{\mathrm{cs}} \left(\frac{V^2}{2} + gz + u\right)\rho \mathbf{V} \cdot d\mathbf{A} \tag{1}$$

Shaft and Flow Work

Work is classified into two categories:

Work = flow work + shaft work

Each work term involves force acting over a distance. When this force is associated with a pressure distribution, then the work is called ***flow work***. Alternatively, *shaft work* is any work that is not associated with a pressure force. **Shaft work is usually done through a shaft** (from which the term originates) and is commonly associated with devices like **pumps and turbines**. According to the sign convention for work when work is done a system it is negative else when it is done by the system it is positive, therefore for pump work is negative and for turbine work is positive. Thus,

$$\dot{W}_{shaft} = \dot{W}_{turbines} - \dot{W}_{pumps} = \dot{W}_t - \dot{W}_p$$

To derive an equation for flow work, consider Fig. 7.3 which defines a control volume that is situated inside a converging pipe. At section 2, the fluid that is inside the control volume will push on the fluid that is outside of the control volume. The magnitude of the pushing force is p_2A_2. During a time interval Δt, the displacement of the fluid at section 2 is $\Delta x_2 = V_2\Delta t$. Thus, the amount of work is

$$\Delta W_2 = (F_2)(\Delta x_2) = (p_2A_2)(V_2\Delta t)$$

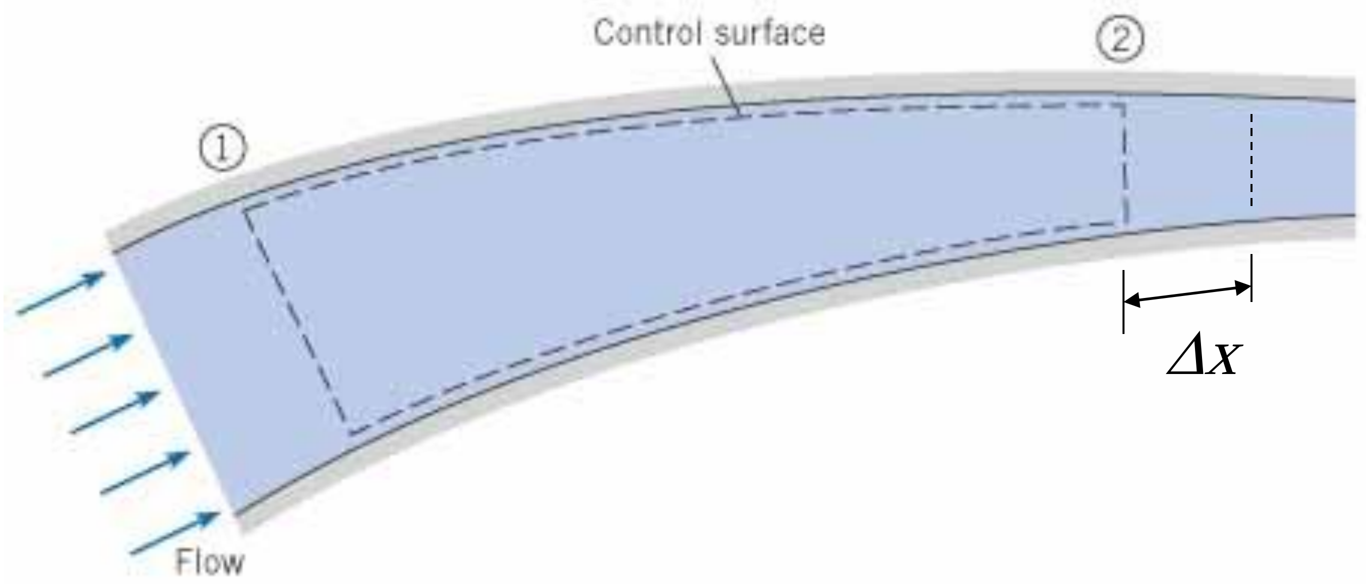


Figure 7.3 *Sketch for deriving flow work.*

Convert the amount of work given by previous equation into a rate of work:

$$\dot{W}_2 = \lim_{\Delta t \longrightarrow 0} \frac{\Delta W_2}{\Delta t} = p_2 A_2 V_2 = \left(\frac{p_2}{\rho}\right)(\rho A_2 V_2) = \dot{m}\left(\frac{p_2}{\rho}\right)$$

This work is positive because the fluid inside the control volume is doing work on the environment. In a similar manner, the flow work at section 1 is negative and is given by

$$\dot{W}_1 = -\dot{m}\left(\frac{p_1}{\rho}\right)$$

Therefore, the net flow work is

$$\dot{W}_{flow} = \dot{W}_2 + \dot{W}_1 = \dot{m}\left(\frac{p_2}{\rho}\right) - \dot{m}\left(\frac{p_1}{\rho}\right)$$

For the generalized situation involving multiple streams of fluid passing across a control surface the net flow work:

$$\dot{W}_{\text{flow}} = \sum_{\text{outlets}} \dot{m}_{\text{out}} \left(\frac{p_{\text{out}}}{\rho} \right) - \sum_{\text{inlets}} \dot{m}_{\text{in}} \left(\frac{p_{\text{in}}}{\rho} \right)$$

To account for velocity and pressure variation across the control surface the integral form is used. Also, use the dot product to account for flow direction. The general equation for flow work is

$$\dot{W}_{\text{flow}} = \int_{\text{cs}} \left(\frac{p}{\rho} \right) \rho \mathbf{V} \cdot d\mathbf{A}$$

Thus, the rate of work is the sum of flow work and shaft work,

$$\dot{W} = \dot{W}_{\text{flow}} + \dot{W}_{\text{shaft}} = \left(\int_{\text{cs}} \left(\frac{p}{\rho} \right) \rho \mathbf{V} \cdot d\mathbf{A} \right) + \dot{W}_{\text{shaft}} \qquad (2)$$

Substitute the work expression, equation (2), into the general energy equation, equation (1), leads to

$$\dot{Q} - \dot{W}_s - \int_{cs} \frac{P}{\rho} \rho \mathbf{V} \cdot d\mathbf{A} = \frac{d}{dt}\int_{cv} \left(\frac{V^2}{2} + gz + u\right)\rho d\forall + \int_{cs} \left(\frac{V^2}{2} + gz + u\right)\rho \mathbf{V} \cdot d\mathbf{A} \quad (3)$$

Combine terms,

$$\dot{Q} - \dot{W}_s = \frac{d}{dt}\int_{cv} \left(\frac{V^2}{2} + gz + u\right)\rho d\forall + \int_{cs} \left(\frac{V^2}{2} + gz + u + \frac{P}{\rho}\right)\rho \mathbf{V} \cdot d\mathbf{A} \quad (4)$$

Realize the definition for enthalpy, $h = u + p/\rho$,

$$\dot{Q} - \dot{W}_s = \frac{d}{dt}\int_{cv} \left(\frac{V^2}{2} + gz + u\right)\rho d\forall + \int_{cs} \left(\frac{V^2}{2} + gz + h\right)\rho \mathbf{V} \cdot d\mathbf{A} \quad (5)$$

For uniform conditions, the equation is simplified to

$$\dot{Q}-\dot{W}_s=\frac{d}{dt}\int_c \left(\frac{V^2}{2}+gz+u\right)\rho d\forall$$

$$+\sum_{cs}\dot{m}_o\left(\frac{V_o^2}{2}+gz_o+h_o\right)-\sum_{cs}\dot{m}_i\left(\frac{V_i^2}{2}+gz_i+h_i\right)$$

7.3 Energy Equation for Pipe Flow

An important application of the energy equation is in pipe flows. However, for this the first step is to develop a way to account for the kinetic energy distribution in the flowing fluid.

Kinetic Energy Correction Factor

Figure 7.4 shows fluid that is pumped through a pipe. At sections 1 and 2, kinetic energy is transported across the control surface by the flowing fluid,

$$\left\{\begin{matrix}\text{Rate of KE}\\ \text{transported}\\ \text{across a section}\end{matrix}\right\} = \int_A \rho V\left(\frac{V^2}{2}\right)dA = \int_A \frac{\rho V^3\,dA}{2}$$

The right hand side is not equal to $\rho \bar{V}^3 A/2$

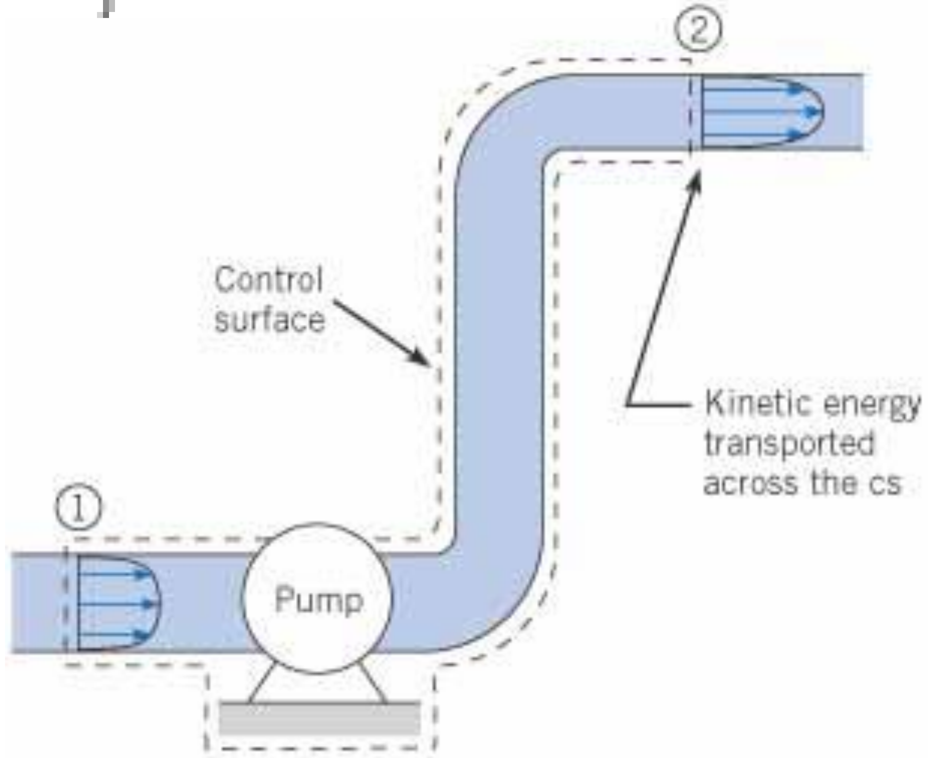


Figure 7.4 *Flow carries kinetic energy into and out of a control surface.*

Define a correction factor as,

$$\alpha = \frac{\text{actual KE / time that crosses a section}}{\text{KE / time by assuming a uniform velocity distribution}} = \frac{\int_A \frac{\rho V^3 dA}{2}}{\frac{\rho \bar{V}^3 A}{2}}$$

For incompressible fluid,

$$\alpha = \frac{1}{A}\int_A \left(\frac{V}{\bar{V}}\right)^3 dA$$

In general, α takes either 1 for turbulent flows, or 2 for laminar flow.

The simplified form of the energy equation is obtained by applying equation (4) to the control volume in Figure 7.4 and assuming steady flow,

$$\dot{Q} - \dot{W}_s + \int_{A_1} \left(\frac{p_1}{\rho} + gz_1 + u_1 \right) \rho V_1 \, dA_1 + \int_{A_1} \frac{\rho V_1^3}{2} \, dA_1$$

$$= \int_{A_2} \left(\frac{p_2}{\rho} + gz_2 + u_2 \right) \rho V_2 \, dA_2 + \int_{A_2} \frac{\rho V_2^3}{2} \, dA_2$$

Recognizing that pressure, elevation and internal energy varies little across the flow in a pipe and using the kinetic correction factor lead to

$$\dot{Q} - \dot{W}_s + \left(\frac{p_1}{\rho} + gz_1 + u_1 + \alpha_1 \frac{\bar{V}_1^2}{2} \right) \dot{m} = \left(\frac{p_2}{\rho} + gz_2 + u_2 + \alpha_2 \frac{\bar{V}_2^2}{2} \right) \dot{m}$$

Substituting for shaft work pump work and turbine work and divide by the mass flow rate simplifies to,

$$\frac{\dot{W}_p}{\dot{m}g} + \frac{p_1}{\gamma} + z_1 + \alpha_1 \frac{\overline{V}_1^2}{2g} = \frac{\dot{W}_t}{\dot{m}g} + \frac{p_2}{\gamma} + z_2 + \alpha_2 \frac{\overline{V}_2^2}{2g} + \frac{u_2 - u_1}{g} - \frac{\dot{Q}}{\dot{m}g}$$

Introduce pump head and turbine head as,

$$pump\ head = h_p = \frac{\dot{W}_p}{\dot{m}g} = \frac{\text{work / time done by pump on flow}}{\text{weight / time of flowing fluid}}$$

$$Turbine\ head = h_t = \frac{\dot{W}_t}{\dot{m}g} = \frac{\text{work / time done by flow on turbine}}{\text{weight / time of flowing fluid}}$$

Substitute these definition into the above equation,

$$\frac{p_1}{\gamma} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 + h_t + \left[\frac{1}{g}(u_2 - u_1) - \frac{\dot{Q}}{\dot{m}g} \right]$$

All terms but the bracketed terms are form of mechanical energies. The bracketed terms on the right hand side are thermal energies and are always positive. This term is called head loss and is represented by h_L. *Head loss* is the conversion of useful mechanical energy to waste thermal energy through viscous action between fluid particles. Therefore, the final form of the simplified energy equation applied to pipe flow,

$$\left(\frac{p_1}{\gamma} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1\right) + h_p = \left(\frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2\right) + h_t + h_L$$

In words,

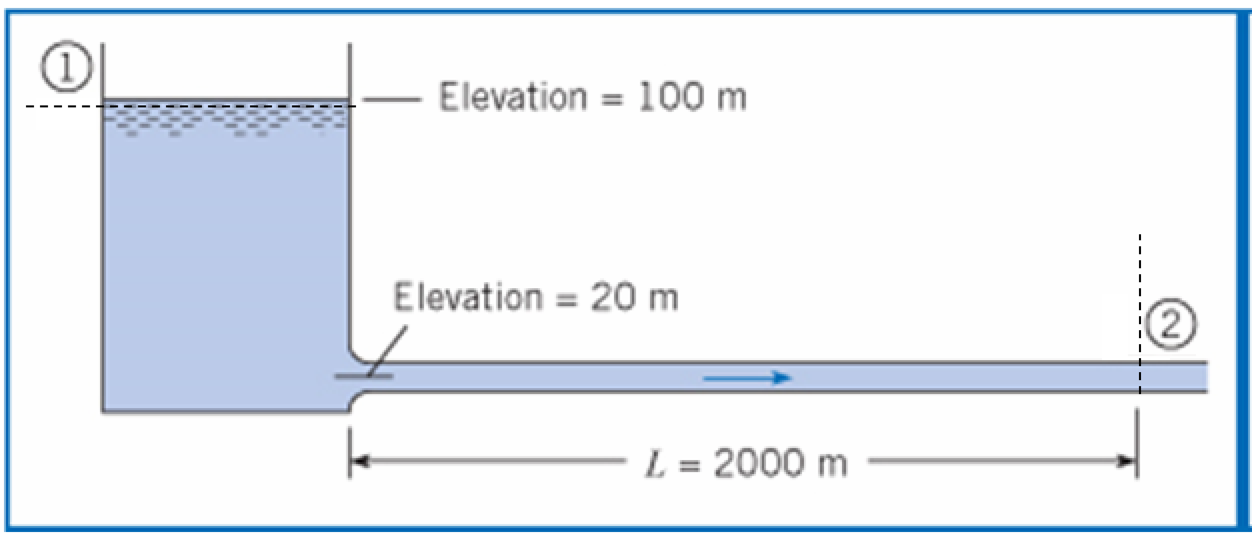
$$\begin{pmatrix}\text{head}\\ \text{caried by}\\ \text{flow into the cv}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{added by}\\ \text{pumps}\end{pmatrix} = \begin{pmatrix}\text{head}\\ \text{carried by}\\ \text{flow out of the cv}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{extracted by}\\ \text{turbines}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{loss due to}\\ \text{viscous effects}\end{pmatrix}$$

EXAMPLE 7.2 PRESSURE IN A PIPE

A horizontal pipe carries cooling water at 10°C for a thermal power plant from a reservoir as shown. The head loss in the pipe is

$$h_L = \frac{0.02(L/D)V^2}{2g}$$

where *L* is the length of the pipe from the reservoir to the point in question, *V* is the mean velocity in the pipe, and *D* is the diameter of the pipe. If the pipe diameter is 20 cm and the rate of flow is 0.06 m3/s, what is the pressure in the pipe at *L* = 2000 m. Assume $\alpha_2 = 1$.



Solution

Apply the energy equation between sections 1 and 2,

$$\frac{p_1}{\gamma}+\alpha_1\frac{\overline{V}_1^2}{2g}+z_1+h_p=\frac{p_2}{\gamma}+\alpha_2\frac{\overline{V}_2^2}{2g}+z_2+h_t+h_L$$

Simplify as,

p_1 = 0 because the pressure at top of a reservoir is p_{atm} = 0 gage.

$V_1 \approx 0$ because the level of the reservoir is constant or changing very slowly.

z_1 = 100 m; z_2 = 20 m.

$h_p = h_t = 0$ because there are no pumps nor turbines in the system

Find V_2 using the flow rate equation

$$V_2=\frac{Q}{A}=\frac{0.06\ \text{m}^3/\text{s}}{(\pi/4)(0.2\ \text{m})^2}=1.910\ \text{m/s}$$

Compute head loss from given relation,

$$h_L = \frac{0.02(L/D)V^2}{2g} = \frac{0.02(2000\text{ m}/0.2\text{ m})(1.910\text{ m/s})^2}{2(9.81\text{ m/s}^2)}$$

$$= 37.2\text{ m}$$

Substitute into the energy equation leads to

$$(z_1 - z_2) = \frac{P_2}{\gamma} + \alpha_2\frac{\bar{V}_2^2}{2g} + h_L$$

$$80\text{ m} = \frac{P_2}{\gamma} + 1.0\frac{(1.910\text{ m/s})^2}{2(9.81\text{ m/s}^2)} + 37.2\text{ m}$$

$$80\text{ m} = \frac{P_2}{\gamma} + (0.186\text{ m}) + (37.2\text{ m})$$

$$P_2 = \gamma(42.6\text{ m}) = (9810\text{ N/m}^3)(42.6\text{ m}) = \boxed{418\text{ KPa}}$$

7.4 Power Equation

to relate head to power and efficiency follows directly from the head definition of the pump head and turbine head,

$$\dot{W}_p = \gamma Q h_p = \dot{m} g h_p$$

$$\dot{W}_t = \gamma Q h_t = \dot{m} g h_t$$

These equations can be generalized to give the power equation,

$$P = \dot{m} g h = \gamma Q h$$

where the h can either be the pump head or turbine head. The efficiency is defined as,

$$\eta \equiv \frac{\text{power output from a machine or system}}{\text{power input to a machine or system}} = \frac{P_{\text{output}}}{P_{\text{input}}}$$

It follows the efficiency for the pump and turbine are,

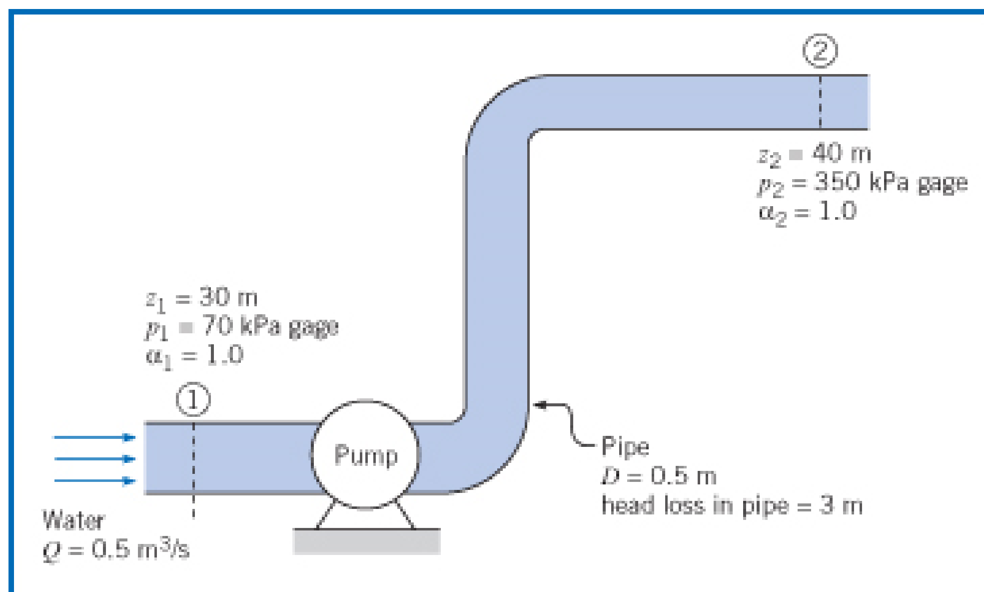
$$\dot{W}_p = \eta_p \dot{W}_s \qquad \text{and} \qquad \dot{W}_s = \eta_t \dot{W}_T$$

Where the subscript s mean shaft while p and T indicate energy to or from fluid.

EXAMPLE 7.3 POWER NEEDED BY A PUMP

A pipe 50 cm in diameter carries water (10°C) at a rate of 0.5 m^3/s. A pump in the pipe is used to move the water from an elevation of 30 m to 40 m. The pressure at section 1 is 70 kPa gage and the pressure at section 2 is 350 kPa gage.

What power in kilowatts and in horsepower must be supplied to the flow by the pump? Assume h_L = 3 m of water and $\alpha_1 = \alpha_2 = 1$.



Solution. Apply the energy equation between sections 1 and 2,

$$\frac{p_1}{\gamma} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 + h_t + h_L$$

Simplify terms,

Velocity head cancels because $V_1 = V_2$.

$h_t = 0$ because there are no turbines in the system.

All other head terms are given.

Inserting terms into the general equation gives

$$\frac{p_1}{\gamma} + z_1 + h_p = \frac{p_2}{\gamma} + z_2 + h_L$$

Consequently,

$$h_p = \left(\frac{p_2 - p_1}{\gamma}\right) + (z_2 - z_1) + h_L$$

$$= \left(\frac{(350.000 - 70.000)\ \text{N/m}^2}{9810\ \text{N/m}^3}\right) + (10\ \text{m}) + (3\ \text{m})$$

$$= (28.5\ \text{m}) + (10\ \text{m}) + (3\ \text{m}) = 41.5\ \text{m}$$

Power consumption,

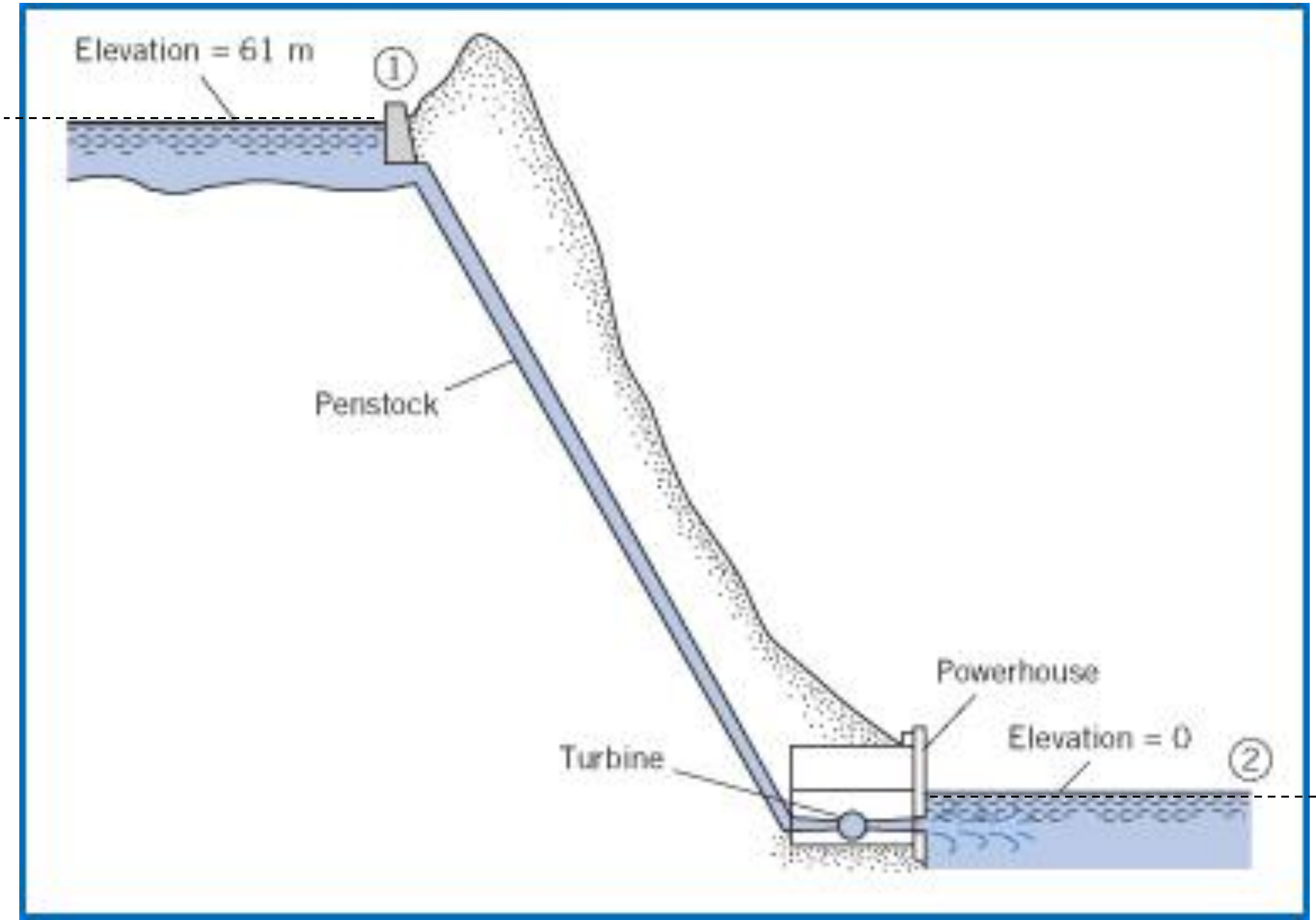
$$P = \gamma Q h_p$$

$$= (9810 \text{ N/m}^3)(0.5 \text{ m}^3/\text{s})(41.5 \text{ m})$$

$$= \boxed{204 \text{ kW}} = (204 \ \cancel{\text{kW}})\left(\frac{1.0 \text{ hp}}{0.746 \ \cancel{\text{kW}}}\right) = \boxed{273 \text{ hp}}$$

EXAMPLE 7.4 POWER PRODUCED BY A TURBINE

At the maximum rate of power generation, a small hydroelectric power plant takes a discharge of 14.1 m^3/s through an elevation drop of 61 m. The head loss through the intakes, penstock, and outlet works is 1.5 m. The combined efficiency of the turbine and electrical generator is 87%. What is the rate of power generation?



Solution. Apply the energy equation between sections 1 and 2,

$$\frac{p_1}{\gamma} = \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 + h_t + h_L$$

Simplify terms,

Velocity heads are negligible because $V_1 \approx 0$ and $V_2 \approx 0$.

Pressure heads are zero because $p_1 = p_2 = 0$ gage.

$h_p = 0$ because there is no pump in the system.

Elevation head terms are given.

Substitute into energy equation,

$$\begin{aligned} h_t &= (z_1 = z_2) h_L \\ &= (61 \text{ m}) - (1.5 \text{ m}) = 59.5 \text{ m} \end{aligned}$$

Power: $$\begin{aligned} P_{\text{input to turbin}} &= \gamma Q h_t = (9810 \text{ N} / \text{m}^3)(14.1 \text{ m}^3 / \text{s})(59.5 \text{ m}) \\ &= 8.23 \text{ MW} \end{aligned}$$

Efficiency: $$\begin{aligned} P_{\text{output from generator}} &= \eta P_{\text{input to turbine}} = 0.87(8.23 \text{ MW}) \\ &= \boxed{7.16 \text{ MW}} \end{aligned}$$

7.5 Comparing the Bernoulli Equation and the Energy Equation

While the Bernoulli equation and the energy equation have a similar form and several terms in common, they are not the same equation. The difference between the two is important to understand for conceptual understanding of these two very important equations.

The Bernoulli equation and the energy equation are derived in different ways. The Bernoulli equation was derived by applying Newton's second law to a particle and then integrating the resulting equation along a streamline. The energy equation was derived by starting with the first law of thermodynamics and then using the Reynolds transport theorem. Consequently, the Bernoulli equation involves only mechanical energy, whereas the energy equation includes both mechanical and thermal energy.

The two equations have different methods of application. The Bernoulli equation is applied by selecting two points on a streamline and then equating terms at these points. In addition, these two points can be anywhere in the flow field for the special case of irrotational flow

The energy equation is applied by selecting an inlet section and an outlet section in a pipe and then equating terms as hey apply to the pipe. The two equations have different assumptions. The Bernoulli equation applies to steady, incompressible, and inviscid flow. The energy equation applies to steady, viscous, incompressible flow in a pipe with additional energy being added through a pump or extracted through a turbine.

Under special circumstances the energy equation can be reduced to the Bernoulli equation. If the flow is inviscid, there is no head loss. If the flow in the pipe is uniform, then $\alpha = 1$. There is no pump or turbine along a streamline. In this case the energy equation is identical to the Bernoulli equation.

Note that the energy equation cannot be developed starting with the Bernoulli equation.

In summary, the energy equation is *not* the Bernoulli equation.

7.6 Transitions

In the analyses of these components energy, momentum, and continuity equations are used together to analyze (a) head loss for an abrupt expansion and (b) forces on transitions.

Abrupt Expansion

An *abrupt or sudden expansion* in a pipe or duct is a change from a smaller section area to a larger section area as shown in Fig. 7.6. Notice that a confined jet of fluid from the smaller pipe discharges into the larger pipe and creates a zone of separated flow. Because the streamlines in the jet are initially straight and parallel, the piezometric pressure distribution across the jet at section 1 will be uniform. This same uniform pressure distribution will also occur in the zone of separated flow. Apply the energy equation between sections 1 and 2:

$$\frac{p_1}{\gamma} + \alpha_1 \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_L \qquad (6)$$

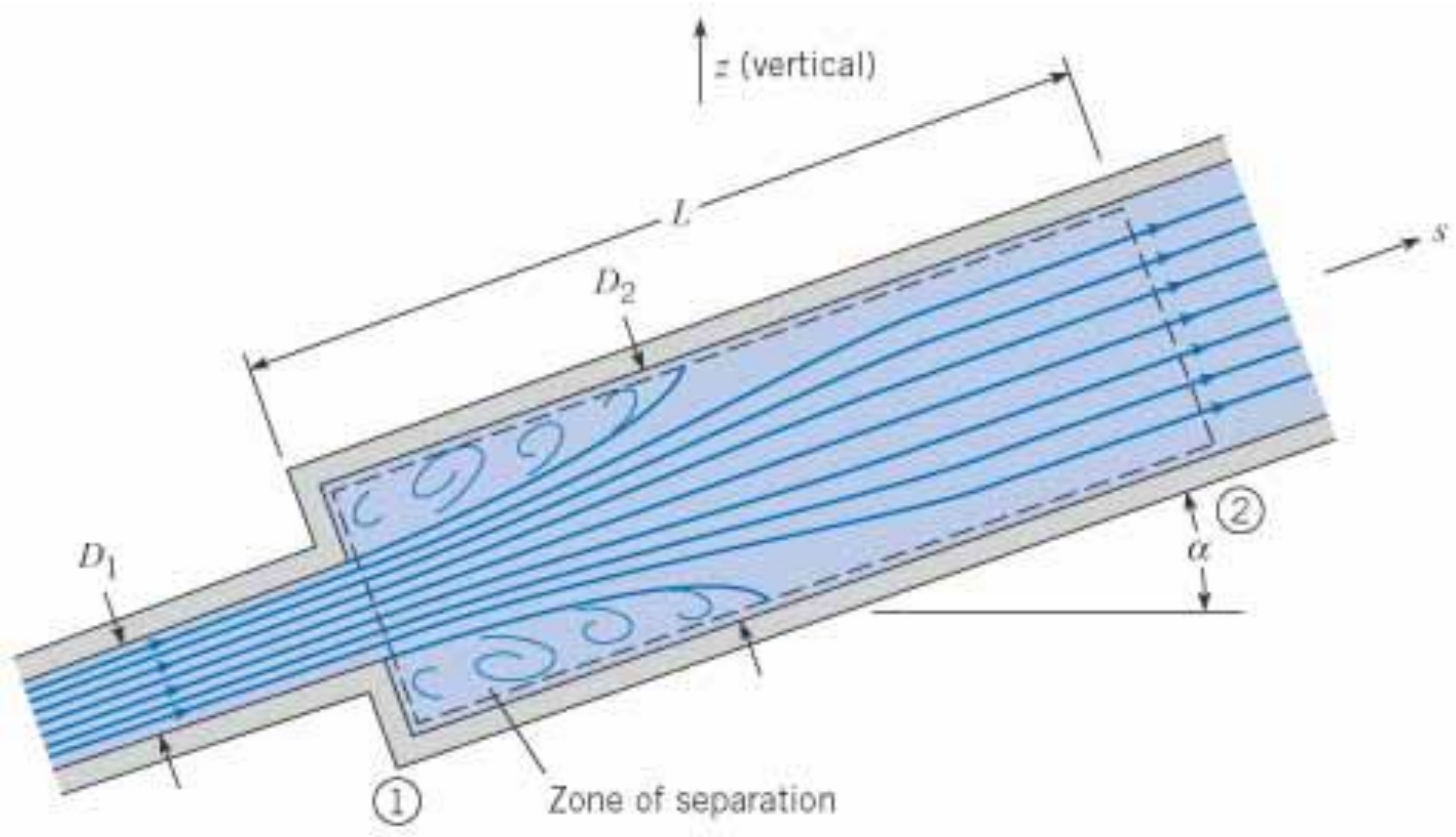


Figure 7.6 *Flow through an abrupt expansion.*

Assume turbulent flow conditions so $\alpha_1 = \alpha_2 \approx 1$. The momentum equation for the fluid in the large pipe between section 1 and section 2, written for the *s* direction, is

$$\sum F_s = \dot{m}V_2 - \dot{m}V_1$$

Neglect the force due to shear stress to give,

$$p_1 A_2 - p_2 A_2 - \gamma A_2 L \sin\alpha = \rho V_2^2 A_2 - \rho V_1^2 A_1$$

or

$$\frac{p_1}{\gamma} - \frac{p_2}{\gamma} - (z_2 - z_1) = \frac{V_2^2}{g} - \frac{V_1^2}{g}\frac{A_1}{A_2} \qquad (7)$$

The continuity equation simplifies to

$$V_1 A_1 = V_2 A_2 \qquad (8)$$

Combining equations (6), (7), and (8)and solving for the head loss h_L caused by a sudden expansion

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

If a pipe discharges fluid into a reservoir, then V_2 = 0 and the sudden-expansion head loss simplifies to

$$h_L = \frac{V^2}{2g}$$

which is the velocity head of the liquid in the pipe. This energy is dissipated by the viscous action of the fluid in the reservoir.

Forces on Transitions

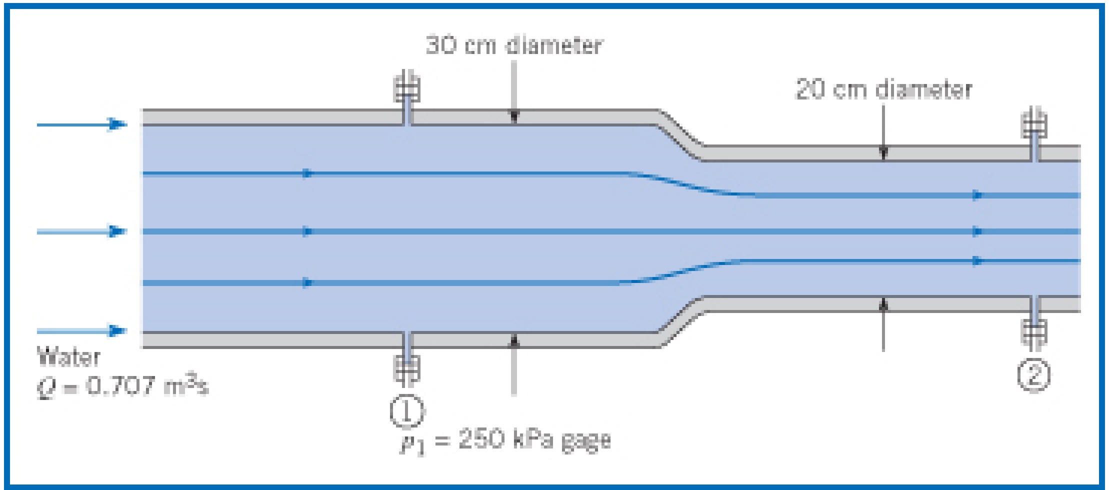
To find forces on transitions in pipes, apply the momentum equation in combination with the energy equation, the flow rate equations, and head loss equations.

EXAMPLE 7.5 FORCE O A COTRACTION IN A PIPE

A pipe 30 cm in diameter carries water (10°C, 250 kPa) at a rate of 0.707 m^3/s. The pipe contracts to a diameter of 20 cm. The head loss through the contraction is given by

$$h_L = 0.1\frac{V_2^2}{2g}$$

where V_2 is the velocity in the 20 cm pipe. What horizontal force is required to hold the transition in place? Assume $\alpha_1 = \alpha_2 = 1$.



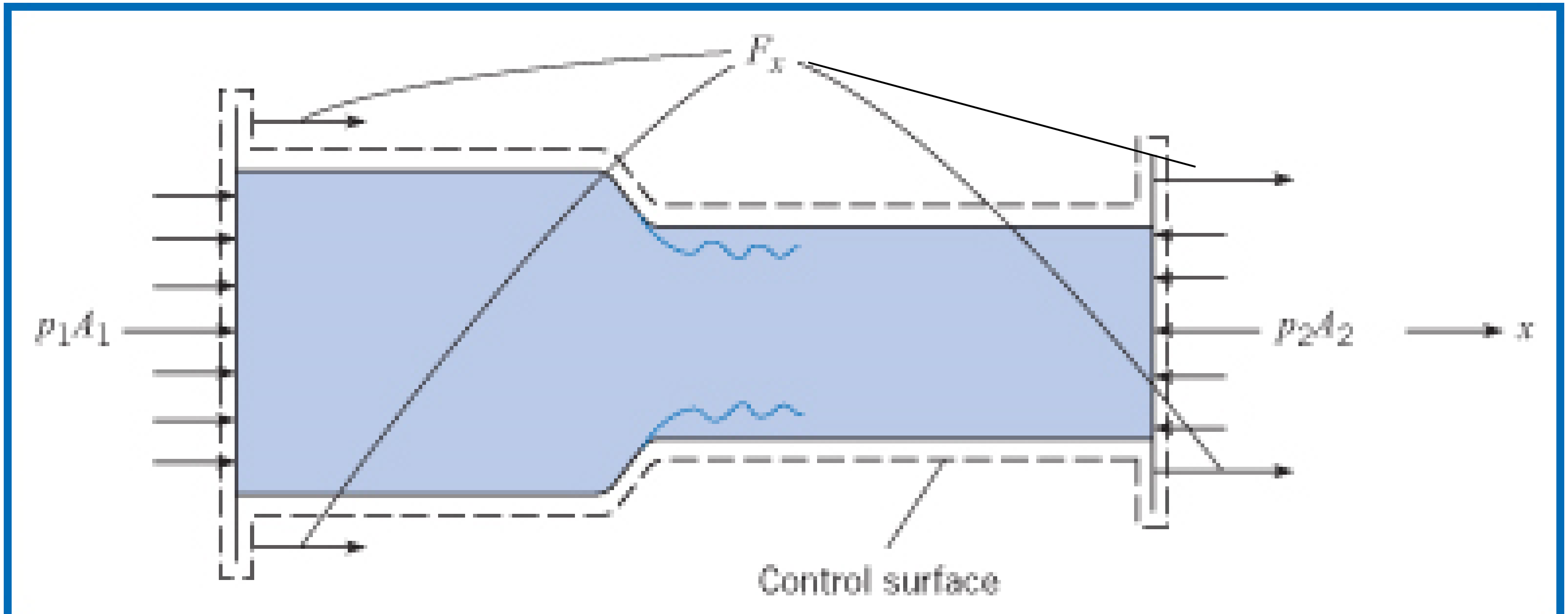
Solution. Momentum equation (horizontal direction)

$$\sum \mathbf{F} = \frac{d}{dt}\int_{cv} \rho \mathbf{v} d\mathcal{V} + \sum_{cs} \dot{m}_O \mathbf{v}_O - \sum_{cs} \dot{m}_i \mathbf{v}_i$$

The forces acting on the control volume are: $\sum F_x = p_1 A_1 + F_x - p_2 A_2.$

Momentum accumulation is zero because flow is steady.

Momentum efflux is $\sum_{cs} \dot{m}_O \mathbf{v}_O - \sum_{cs} \dot{m}_i \mathbf{v}_i = (\dot{m} V_2 - \dot{m} V_1)\hat{i}$



Substitute force and momentum terms into the momentum equation

$$p_1 A_1 - p_2 A_2 + F_x = \dot{m} V_2 - \dot{m} V_1$$

$$F_x = \rho Q (V_2 - V_1) + p_2 A_2 - p_1 A_1$$

Energy equation (from section 1 to section 2). Since $\alpha_1 = \alpha_2 = 1$, $z_1 = z_2$, and $h_p = h_t = 0$, simplifies to

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + h_L$$

Pressure at section: $$p_2 = p_1 - \gamma \left(\frac{V_2^2}{2g} - \frac{V_1^2}{2g} + h_L \right)$$

Find velocities using the flow rate equation

$$V_1 = \frac{Q}{A_1} = \frac{0.707 \text{ m}^3/\text{s}}{(\pi/4) \times (0.3 \text{ m})^2} = 10 \text{ m/s}$$

$$V_2 = \frac{Q}{A_2} = \frac{0.707 \text{ m}^3/\text{s}}{(\pi/4) \times (0.2 \text{ m})^2} = 22.5 \text{ m/s}$$

Calculate head loss.

$$h_L = \frac{0.1\, V_2^2}{2g} = \frac{0.1 \times (22.5\ \text{m/s})^2}{2 \times (9.81\ \text{m/s}^2)} = 2.58\ \text{m}$$

Calculate pressure.

$$\begin{aligned} p_2 &= p_1 - \gamma\left(\frac{V_2^2}{2g}\right) - \frac{V_1^2}{2g} + h_L \\ &= 250\ \text{kpa} - 9.81\ \text{kN/m}^3 \\ &\quad \times \left(\frac{(22.5\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} - \frac{(10\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} + 2.58\ \text{m}\right) \\ &= 21.6\ \text{kpa} \end{aligned}$$

Calculate *Fx*.

$$F_x = \rho Q(V_2 - V_1) + p_2 A_2 - p_1 A_1$$
$$= (1000\ \text{kg}/\text{m}^3)(0.707\ \text{m}^3/\text{s})(22.5 - 10)(\text{m}/\text{s})$$
$$+ (21{,}600\ \text{Pa})\left(\frac{\pi(0.2\ \text{m})^2}{4}\right) - (250{,}000\ \text{Pa})$$
$$\times \left(\frac{\pi(0.3\ \text{m})^2}{4}\right)$$
$$= (8837 + 677 - 17{,}670)\text{N} = -8.16\ \text{kN}$$

$$\boxed{F_x = 8{,}16\ \text{kN applied in the negative } x \text{ direction}}$$

7.7 Hydraulic and Energy Grade Lines

The hydraulic grade line (HGL) and the energy grade line (EGL), which are graphical representations that show head in a system. The *EGL*, shown in Fig. 7.7, is a line that indicates the total head at each location in a system.

$$\frac{p_1}{\gamma} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 + h_t + h_L$$

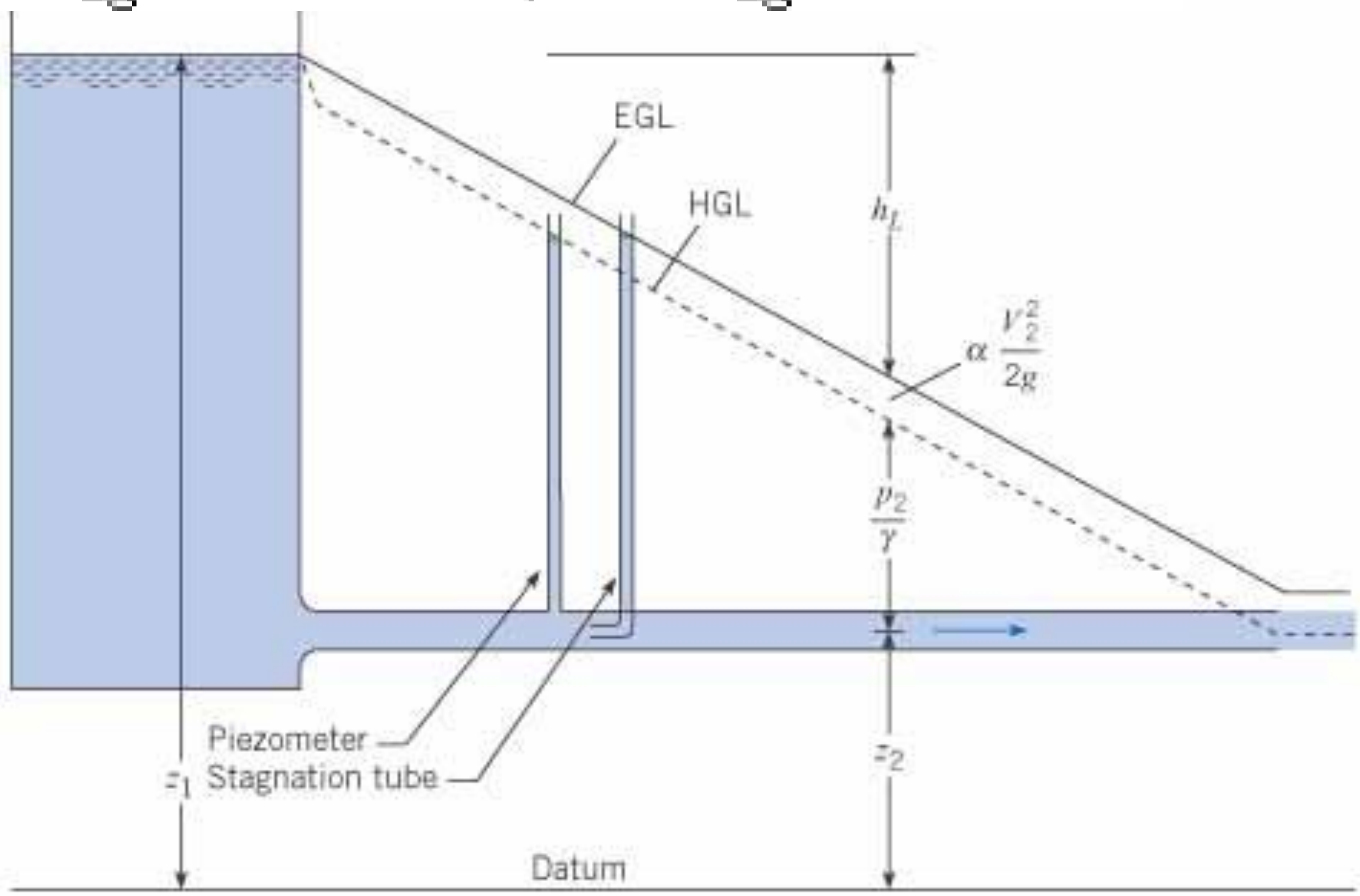


Figure 7.7 *EGL and HGL in a straight pipe.*

The *EGL* is related to terms in the energy equation by

$$\text{EGL} = \begin{pmatrix}\text{velocity}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{pressure}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{elevation}\\ \text{head}\end{pmatrix} = \alpha\frac{V^2}{2g} + \frac{p}{\gamma} + z = \begin{pmatrix}\text{total}\\ \text{head}\end{pmatrix}$$

The *HGL,* shown in Fig. 7.7, is a line that indicates the piezometric head at each location in a system:

$$\text{HGL} = \begin{pmatrix}\text{pressure}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{elevation}\\ \text{head}\end{pmatrix} = \frac{p}{\gamma} + z = \begin{pmatrix}\text{piezometric}\\ \text{head}\end{pmatrix}$$

Since the HGL gives piezometric head, the HGL will be coincident with the liquid surface in a piezometer as shown in Fig.7.7. Similarly, the EGL will be coincident with the liquid surface in a stagnation tube.

Examples on EGL and HGL

For this case, note how the EGL decreases along the flow direction to the left of the pump due to head loss in the pipe. The EGL increases by h_p that receives from the pump. Because the diameter is constant, the increase in EGL is equal to HGL.

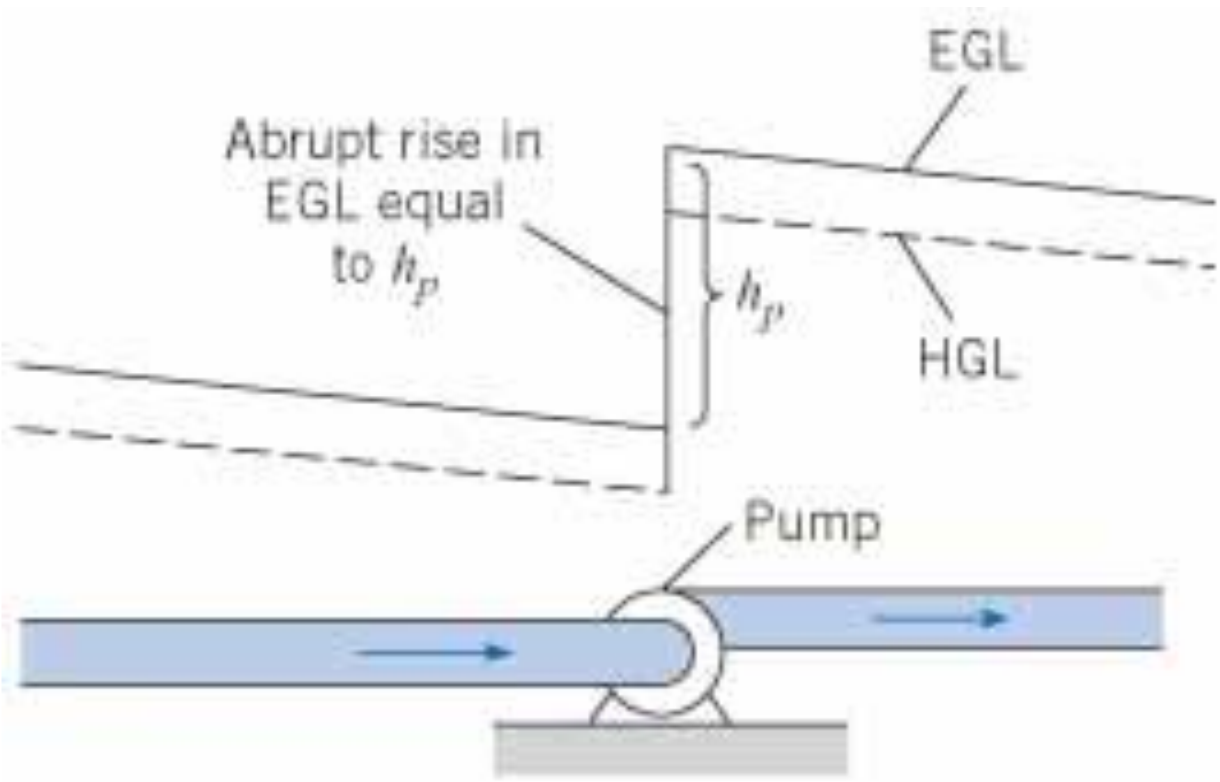


Figure 7.8 *Rise in EGL and HGL due to Pump.*

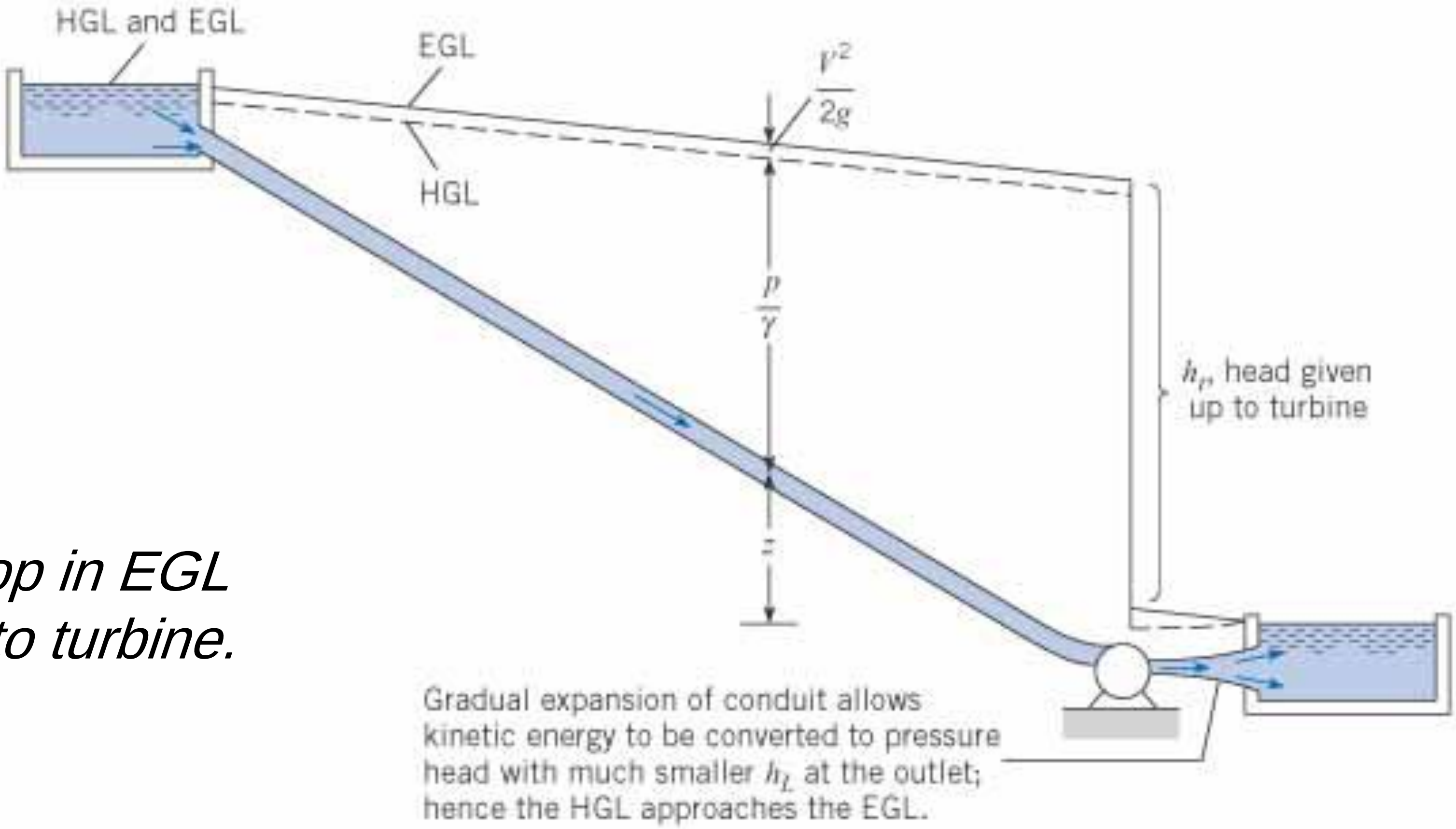
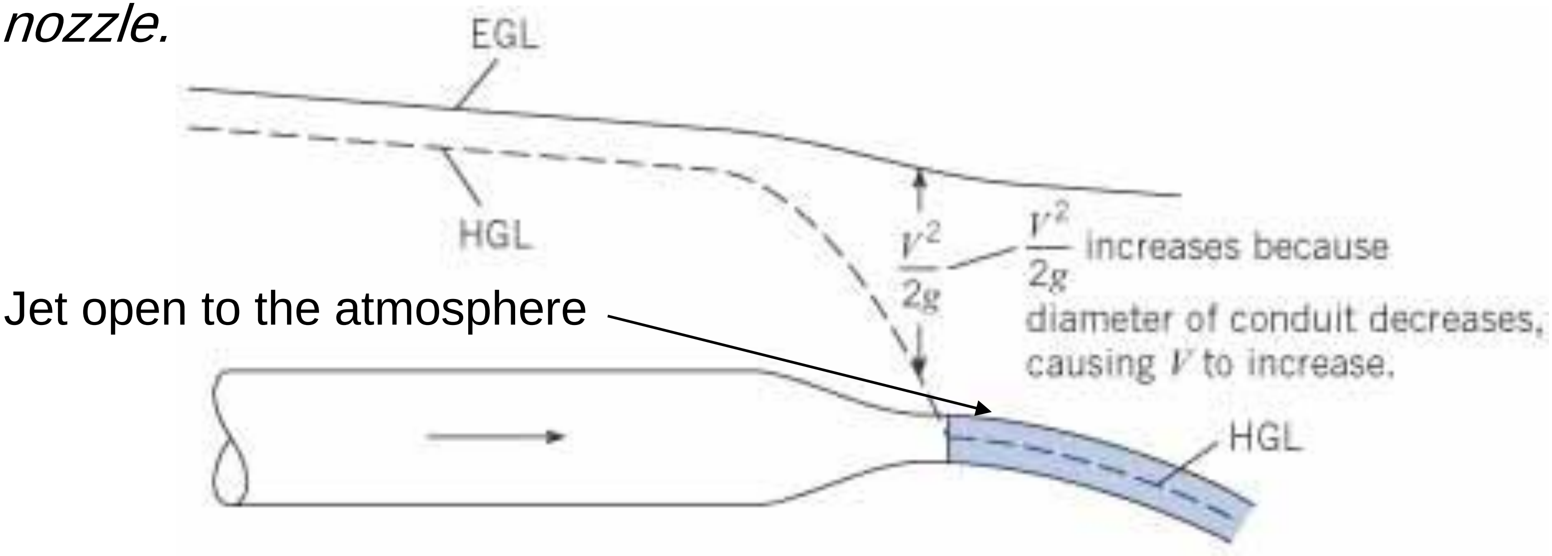


Figure 7.9 *Drop in EGL and HGL due to turbine.*

Figure 7.10 *Change in HGL and EGL due to flow through a nozzle.*



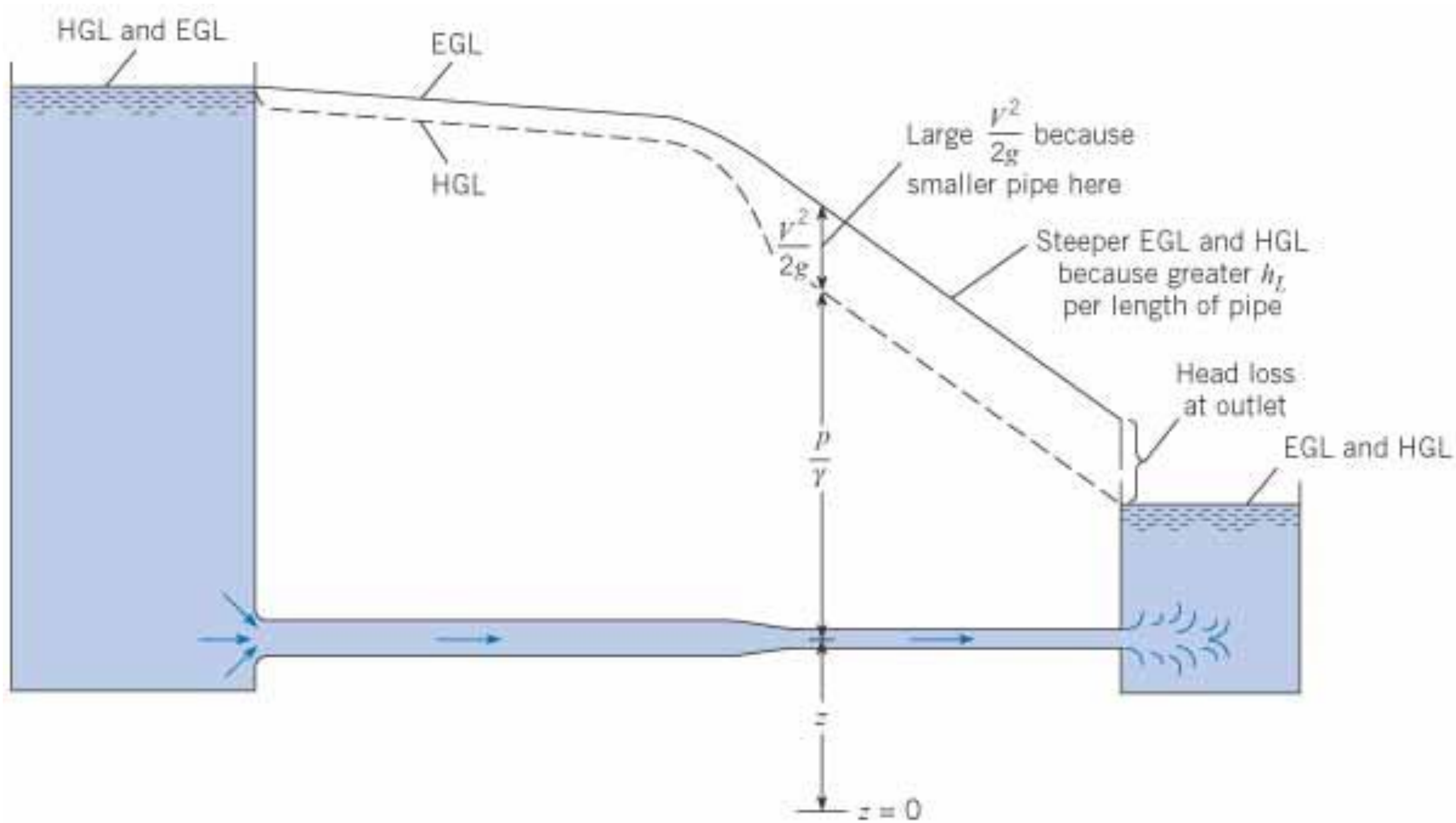


Figure 7.11 *Change in EGL and HGL due to change in diameter of pipe.*

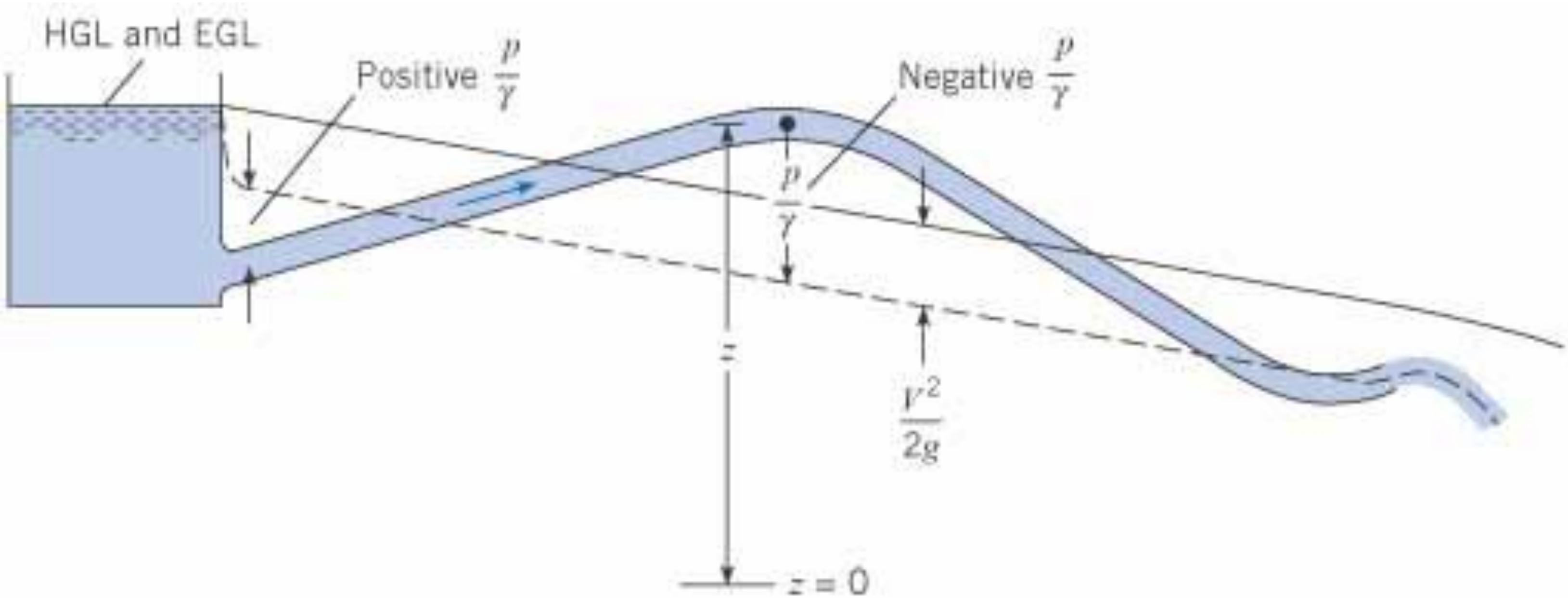


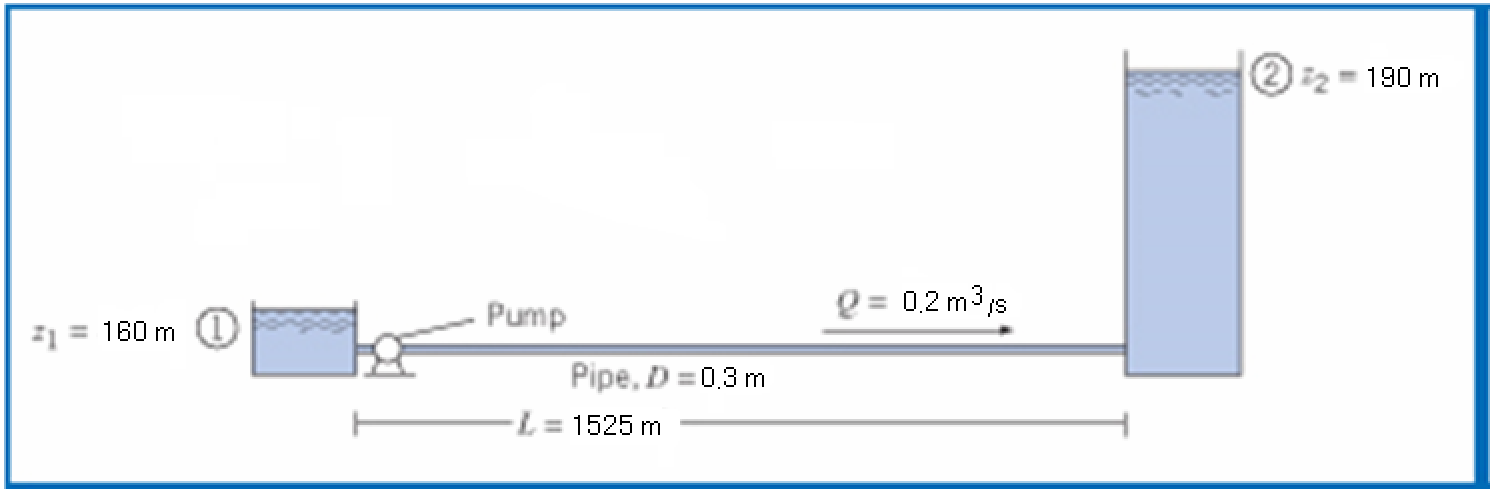
Figure 7.12 *Subatmospheric pressure when pipe is above HGL.*

EXAMPLE 7.6 EGL AND HGL FOR A SYSTEM

A pump draws water (10°C) from a reservoir, where the water-surface elevation is 160 m, and forces the water through a pipe 1525 m long and 0.3 m in diameter. This pipe then discharges the water into a reservoir with water-surface elevation of 190 m. The flow rate is 0.2 m^3/s, and the head loss in the pipe is given by

$$h_L = 0.01\left(\frac{L}{D}\right)\left(\frac{V^2}{2g}\right)$$

Determine the head supplied by the pump, *hp*, and the power supplied to the flow, and draw the HGL and EGL for the system. Assume that the pipe is horizontal and is 155 m in elevation.



Solution

Energy equation (general form)

$$\frac{p_1}{\gamma} + \alpha_1 \frac{\bar{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\bar{V}_2^2}{2g} + z_2 + h_t + h_L$$

Velocity heads are negligible because $V_1 \approx 0$ and $V_2 \approx 0$.

· Pressure heads are zero because $p_1 = p_2 = 0$ gage.

· $h_t = 0$ because there are no turbines in the system.

$$h_p = (z_2 - z_1) + h_L$$

Calculate V using the flow rate equation

$$V = \frac{Q}{A} \frac{0.2 m^3/s}{(\pi/4) 0.3^2 m^2} = 2.83\, m/s$$

Calculate head loss.

$$h_L = 0.01 \left(\frac{1525}{0.3}\right)\left(\frac{2.83^2}{2}\right)$$

$$= 20.75\, m$$

Calculate h_p.

$$h_p = (z_2 - z_1) + h_L = (190 - 160) + 20.75 = 50.75\,m$$

Power
$$\dot{W}_p = \gamma Q h_p = (9.81)(0.2)(50.75)$$
$$= 99.5\,kW$$

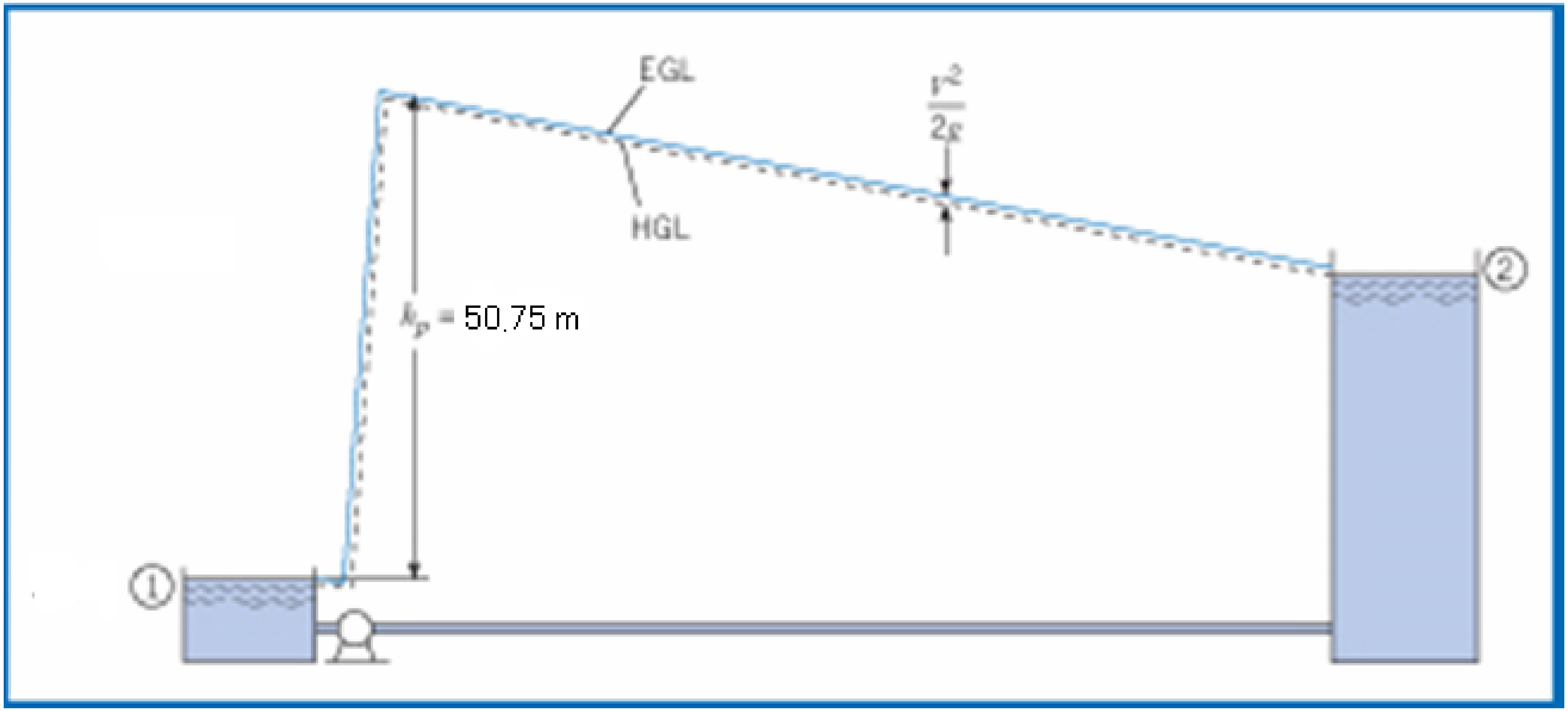


Figure. EGL and HGL lines.

Fluid Mechanics

Chapter 8
Dimensional Analysis and Similitude

Dr. Amer Khalil Ababneh

Introduction

Because of the complexity of fluid mechanics, the design of many fluid systems relies heavily on experimental results. Tests are typically carried out on a **subscale model**, and the results are extrapolated to the **full-scale system (prototype).** The principles underlying the correspondence between the model and the prototype are addressed in this chapter.

Dimensional analysis is the process of grouping of variables into significant dimensionless groups, thus reducing problem complexity.

Similitude (Similarity) is the process by which geometric and dynamic parameters are selected for the subscale model so that meaningful correspondence can be made to the full size prototype.

8.2 Buckingham Π Theorem

In 1915 Buckingham showed that the number of independent dimensionless groups (dimensionless parameters) can be reduced from a set of variables in a given process is *n* - *m*, where *n* is the number of variables involved and *m* is the number of basic dimensions included in the variables.

Buckingham referred to the dimensionless groups as Π, which is the reason the theorem is called the Π theorem. Henceforth dimensionless groups will be referred to as π-*groups*. If the equation describing a physical system has *n* dimensional variables and is expressed as

$$y_1 = f(y_2, y_3, \ldots y_n)$$

then it can be rearranged and expressed in terms of (*n* - *m*) π-groups as

$$\pi_1 = \vartheta(\pi_2, \pi_3, \ldots, \pi_{n-m})$$

Example

If there are five variables (*F*, *V*, ρ, μ, and *D*) to describe the drag on a sphere and three basic dimensions (*L*, *M*, and *T*) are involved. By the Buckingham Π theorem there will be 5 - 3 = 2 π-groups that can be used to correlate experimental results in the form

$$F = f(V, \rho, \mu, D)$$

$$\pi_1 = \varphi(\pi_2)$$

8.3 Dimensional Analysis

Dimensional analysis is the process used to obtain the π-groups. There are two methods: the step-by-step method and the exponent method.

The Step-by-Step Method

The method is best describe by an example

EXAMPLE 8.1 Π-GROUPS FOR BODY FALLING IN A VACUUM

There are three significant variables for a body falling in a vacuum (no viscous effects): the velocity V; the acceleration due to gravity, g; and the distance through which the body falls, h. Find the π-groups using the step-by-step method.

Solution

The variables involved in this example are *n=3, V, g, and h*. The dimension of these variables in terms of the basic dimensions are:

$$[V] = L / T$$

$$[g] = L / T^2$$

$$[h] = L$$

►Note the notation used: brackets means dimension of the variable contained between the brackets.

►The number of basic dimension that appear in the dimension of the variable is 2

►Hence, the number of π-groups is $n - m = 3 - 2 = 1$

►Construct the following table by listing the variable along with their dimensions in terms of the basic dimensions.

The steps involved in this table are

Variable	[]	Variable	[]	Variable	[]
V	L/T	V/h	1/T	$(V/h)/(g/h)^{1/2}$	0
g	L/T^2	g/h	$1/T^2$		
h	L	-	-		

1- List the variable along with their dimensions in the first and second columns.
2- Choose a variable to combine new variables with and to eliminate a dimension from the new formed variables. List these in the second and third column
3- If the resulting dimensions is 0 (dimensionless) stop
4- Else, choose another variable from the third column to form new variables and to eliminate that dimension.
5- Repeat as necessary to arrive at dimensionless groups.
The final result as expected one p-group,

$$\pi = \frac{V}{\sqrt{gh}}$$

Consequently, the functional form is

$$\frac{V}{\sqrt{gh}} = C$$

EXAMPLE 8.2 Π-GROUPS FOR DRAG ON A SPHERE USING STEP-BY-STEP METHOD

The drag F_D of a sphere in a fluid flowing past the sphere is a function of the viscosity μ, the mass density ρ, the velocity of flow *V*, and the diameter of the sphere *D*. Use the step-by-step method to find the π-groups.

Solution

Given $F_D = f(V, \rho, \mu\ D)$.

Dimensions of significant variables,

$$F = \frac{ML}{T^2},\ V = \frac{L}{T},\ \rho = \frac{M}{L^3},\ \mu = \frac{M}{LT},\ D = L$$

Follow the same steps as in previous example and construct table as shown on next slide.

Variable	[ ]	Variable	[ ]	Variable	[ ]	Variable	[ ]
F_D	ML/T^2	F_D/D	$M/T2$	$F_D/(\rho D^4)$	$1/T^2$	$F_D/(\rho V^2 D^2)$	0
V	L/T	V/D	$1/T$	V/D	$1/T$		
ρ	M/L^3	ρD^3	M	-	-		
μ	$M/(LT)$	μD	M/T	$\mu/(\rho D^2)$	$1/T$	$\mu/(\rho V D)$	0
D	L	-					

The final π-groups are

$$\pi_1 = \frac{\rho F_D}{\mu^2} \text{ and } \pi_2 = \frac{\mu}{\rho V D}$$

Thus, the final functional form is

$$\pi_1 = \phi(\pi_2)$$

The Exponent Method

An alternative method for finding the π-groups is the exponent method. This method involves solving a set of algebraic equations to satisfy dimensional homogeneity. The procedural steps for the exponent method is illustrated in the following example.

EXAMPLE 8.3 Π-GROUPS FOR DRAG ON A SPHERE USING EXPONENT METHOD

The drag of a sphere, F_D, in a flowing fluid is a function of the velocity V, the fluid density ρ the fluid viscosity μ and the sphere diameter D. Find the π-groups using the exponent method.

Solution

Given $F_D = f(V, \rho, \mu\ D)$.

Dimensions of significant variables are

$$[F] = \frac{ML}{T^2},\quad [V] = \frac{L}{T},\quad [\rho] = \frac{M}{L^3},\quad [\mu] = \frac{M}{LT},\quad [D] = L$$

Number of π-groups is 5 - 3 = 2.
Choose repeating variables equal to the number of basic dimensions, m = 3. The repeating variables are typically ρ, V, D, Form product with the remaining dimensions at a time. Start with dimensions of force as follow,

$$\frac{ML}{T^2} \times \left[\frac{L}{T}\right]^a \times \left[\frac{M}{L^3}\right]^b \times [L]^c = 0$$

Dimensional homogeneity. Equate powers of dimensions on each side. For M, the equation is

M: 1 + *b = 0, implies b = -1*

T: -2 – a = 0, implies a = -2

L: 1 + a -3b + c = 0, implies c = -2

Thus, the resulting π-groups is: $\pi_1 = \frac{F}{\rho V^2 D^2}$

Repeat the same procedure with the viscosity as below,

$$\frac{M}{LT} \times \left[\frac{L}{T}\right]^a \times \left[\frac{M}{L^3}\right]^b \times [L]^c = 0$$

Solving for the exponents lead to the second π-groups,

$$\pi_2 = \frac{\rho VD}{\mu}$$

The final functional form: $\frac{F}{\rho V^2 D^2} = f\left(\frac{\rho VD}{\mu}\right)$

8.4 Common π-Groups

The most common π-groups can be found by applying dimensional analysis to all the variables that might be significant in a general flow situation.

Variables that have significance in a general flow field are the velocity *V*, the density ρ, the viscosity μ, and the acceleration due to gravity *g*. In addition, if fluid compressibility were likely, then the bulk modulus of elasticity, E_v, should be included. If there is a liquid-gas interface, the surface tension effects may also be significant. Finally the flow field will be affected by a general length, *L*, such as the width of a building or the diameter of a pipe. These variables will be regarded as the **independent** variables. The primary dimensions of the significant independent variables are:

$$[V] = L/T \qquad [\rho] = M/L^3 \qquad [\mu] = M/LT$$

$$[g] = L/T^2 \qquad [E_v] = M/LT^2 \qquad [\sigma] = M/T^2 \qquad [L] = L$$

There are several other independent variables that could be identified for thermal effects, such as temperature, specific heat, and thermal conductivity. Inclusion of these variables is beyond the scope of this text.

Typically one is interested in pressure distributions (p), shear stress distributions (τ), and forces on surfaces and objects (F) in the flow field. These will be identified as the **dependent** variables. The primary dimensions of the dependent variables are

$$[p] = M / LT^2 \qquad [\tau] = [\Delta p] = M / LT^2 \qquad [F] = (ML) / T^2$$

There are 10 significant variables, which, by application of the Buckingham Π theorem, means there are seven π-groups. Utilizing either the step-by-step method or the exponent method yields و

$$\frac{p}{\rho V^2} \qquad \frac{\tau}{\rho V^2} \qquad \frac{F}{\rho V^2 L^2}$$

$$\frac{\rho V L}{\mu} \qquad \frac{V}{\sqrt{E_v / \rho}} \qquad \frac{\rho L V^2}{\sigma} \qquad \frac{V^2}{gL}$$

The first three groups, the dependent π-groups, are identified by **specific names**. For these groups it is common practice to use the kinetic pressure, $\rho V^2/2$, instead of ρV^2. In most applications one is concerned with a pressure difference, so the pressure π-group is expressed as

$$C_p = \frac{p - p_0}{\frac{1}{2}\rho V^2}$$

where C_p is called the pressure coefficient and p_0 is a reference pressure. The π-group associated with shear stress is called the shear-stress coefficient and defined as

$$c_f = \frac{\tau}{\frac{1}{2}\rho V^2}$$

where the subscript f denotes "friction." The π-group associated with force is referred to, here, as a force coefficient and defined as

$$C_F = \frac{F}{\frac{1}{2}\rho V^2 L^2}$$

The independent π-groups are named after earlier contributors to fluid mechanics. The π-group $VL\rho/\mu$ is called the Reynolds number, after Osborne Reynolds, and designated by Re. The group $V/\sqrt{E_v/\rho}$ is rewritten as (V/c), since is the speed of sound, c. This π-group is called the Mach number and designated by M. The π-group $\rho L V^2/\sigma$ is called the Weber number and designated by We. The remaining π-group is usually expressed as $V/\sqrt{gL}$ and identified as the Froude (rhymes with "food") number * and written as Fr.

The general functional form for all the π-groups is

$$C_p, c_f, C_F = f\ (\mathrm{Re}, \mathrm{M}, \mathrm{We}, \mathrm{Fr})$$

which means that either of the three dependent π-groups are functions of the four independent π-groups; that is, the pressure coefficient, the shear-stress coefficient, or the force coefficient are functions of the Reynolds number, Mach number, Weber number, and Froude number.

Each independent π-group has an important interpretation as indicated by the ratio column in Table 8.3 (see textbook). The Reynolds number can be viewed as the ratio of kinetic to viscous forces. The kinetic forces are the forces associated with fluid motion. The Bernoulli equation indicates that the pressure difference required to bring a moving fluid to rest is the kinetic pressure, $\rho V^2/2$, so the kinetic forces, F_k should be proportional to

$$F_k \propto \rho V^2 L^2$$

The shear force due to viscous effects, F_v, is proportional to the shear stress and area

$$F_v \propto \tau A \propto \tau L^2$$

and the shear stress is proportional to

$$\tau = \mu \frac{dV}{dy} \propto \frac{\mu V}{L}$$

so $F_v \sim \mu VL$. Taking the ratio of the kinetic to the viscous forces

$$\frac{F_k}{F_v} \propto \frac{\rho VL}{\mu} = \text{Re}$$

yields the Reynolds number. The magnitude of the Reynolds number provides important information about the flow. A low Reynolds number implies viscous effects are important; a high Reynolds number implies kinetic forces predominate. The Reynolds number is one of the most widely used π-groups in fluid mechanics. It is also often written using kinematic viscosity, $\text{Re} = \rho VL/\mu = VL/\nu$.

The other p-groups are also given physical interpretation:

►The Mach number is an indicator of how important compressibility effects are in a fluid flow

►The Froude number is important when gravitational force influences the pattern of flow, such as in flow over a spillway

►The Weber number is important in liquid atomization where surface tension of the liquid at the droplet’s surface is responsible for maintaining the droplet's shape

8.5 Similitude

Scope of Similitude

Similitude is the theory and art of predicting prototype performance from model observations. Experiments are performed to obtain information that cannot be obtained by analytical means alone. The rules of similitude must be applied to select parameters for the model. The theory of similitude involves the application of π-groups, such as the Reynolds number or the Froude number, to predict prototype performance from model tests. Present engineering practice makes use of model tests more frequently than most people realize. For example, whenever a new airplane is being designed, tests are made not only on the general scale model of the prototype airplane but also on various components of the plane. Numerous tests are made on individual wing sections as well as on the engine pods and tail sections.

Other examples include models for automobile, fast trains, dams, rivers, flood-control structure, etc.

Two conditions must be satisfied for similitude between model and prototype: 1) Geometric similitude, 2) Dynamic Similitude

Geometric Similitude

Geometric similitude means that the model is an exact geometric replica of the prototype. Consequently, if a 1:10 scale model is specified, all linear dimensions of the model must be 1 / 10 of those of the prototype. In Fig. 8.4 if the model and prototype are geometrically similar, the following equalities hold:

$$\frac{\ell_m}{\ell_p} = \frac{w_m}{w_p} = \frac{c_m}{c_p} = L_r$$

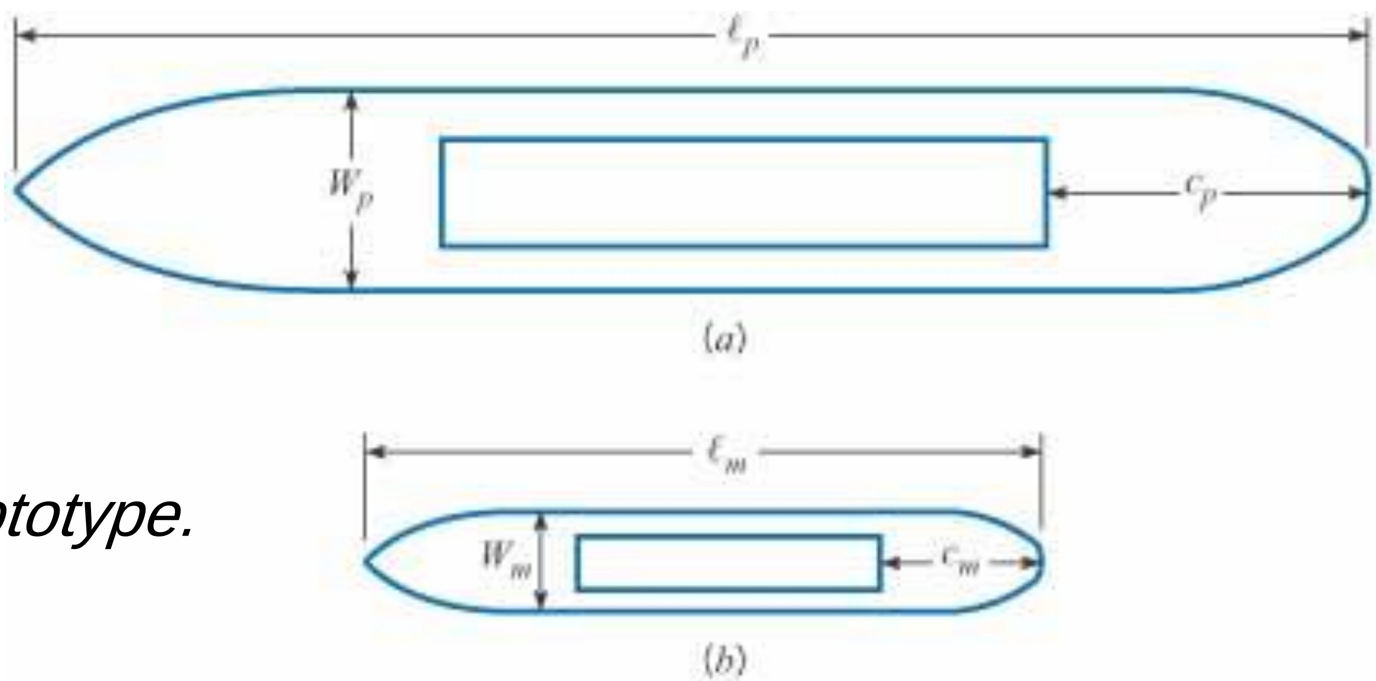


Figure 8.4 *(a) Prototype. (b) Model.*

Here ℓ, w, and c are specific linear dimensions associated with the model and prototype, and L_r is the scale ratio between model and prototype. It follows that the ratio of corresponding areas between model and prototype will be the square of the length ratio: $A_r = L_r^2$. Similarly, the ratio of corresponding volumes will be given by $Vol)_m/Vol)_p = L_r^3$.

Dynamic Similitude

Dynamic similitude means that the forces that act on corresponding masses in the model and prototype are in the same ratio (F_m/F_p = constant) throughout the entire flow field. For example, the ratio of the kinetic to viscous forces must be the same for the model and the prototype. Since the forces acting on the fluid elements control the motion of those elements, it follows that dynamic similarity will yield similarity of **flow patterns**. Consequently, the flow patterns for the model and the prototype will be the same if geometric similitude is satisfied and if the relative forces acting on the fluid are the same in the model as in the prototype. This latter condition requires that the appropriate π-groups be the same for the model and prototype, because these π-groups are indicators of relative forces within the fluid.

A more physical interpretation of the force ratios can be illustrated by considering the flow over the spillway shown in Fig. 8.5*a*. Here corresponding masses of fluid in the model and prototype are acted on by corresponding forces. These forces are the force of gravity F_g, the pressure force F_p, and the viscous resistance force F_v. These forces add vectorially to yield a resultant force F_R, which will in turn produce an acceleration of the volume of fluid in accordance with Newton's second law of motion. Hence, because the force polygons in the prototype and model are similar, the magnitudes of the forces in the prototype and model will be in the same ratio as the magnitude of the vectors representing mass times acceleration

$$\frac{m_m a_m}{m_p a_p} = \frac{F_{gm}}{F_{gp}}$$

or

$$\frac{\rho_m L_m^3 (V_m / t_m)}{\rho_p L_p^3 (V_p / t_p)} = \frac{\gamma_m L_m^3}{\gamma_p L_p^3}$$

which reduces to,

$$\frac{V_m}{g_m t_m} = \frac{V_p}{g_p t_p}$$

But,

$$\frac{t_m}{t_p} = \frac{L_m / V_m}{L_p / V_p}$$

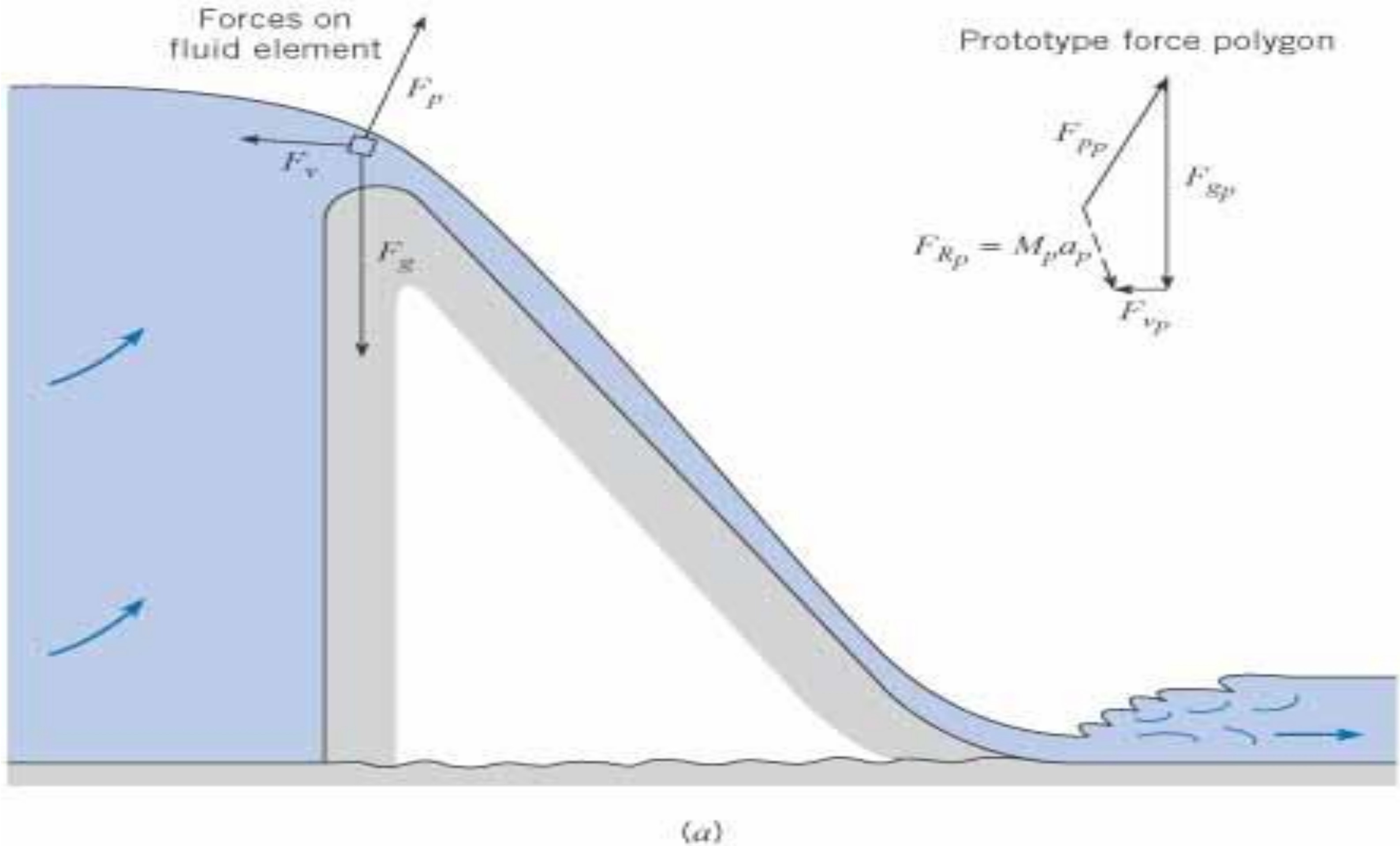
so,

$$\frac{V_m^2}{g_m L_m} = \frac{V_p^2}{g_p L_p}$$

Taking the square root of each side gives

$$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}} \text{ or } \mathrm{Fr}_m = \mathrm{Fr}_p$$

Thus the Froude number for the model must be equal to the Froude number for the prototype to have the same ratio of forces on the model and the prototype.



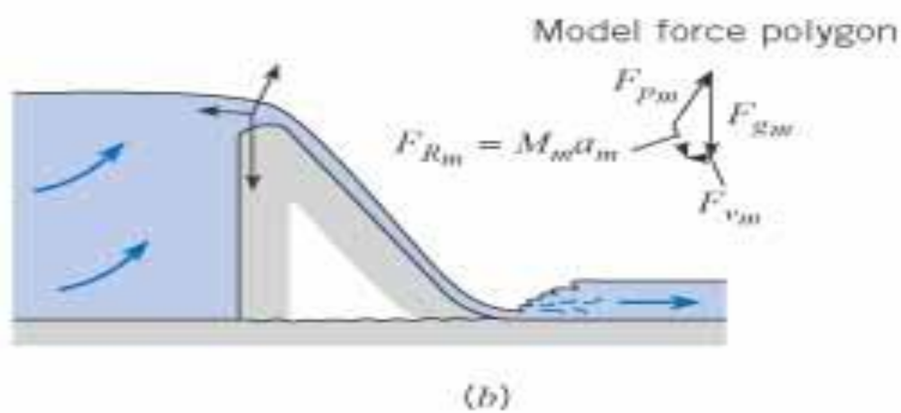


Figure 8.5 *Model-prototype relations: (a) prototype view; and (b) model view*

Equating the ratio of the forces producing acceleration to the ratio of viscous forces,

$$\frac{m_m a_m}{m_p a_p} = \frac{F_{vm}}{F_{vp}}$$

where $F_v \sim \mu VL$ leads to

$$\mathrm{Re}_m = \mathrm{Re}_p$$

Referring back to the general functional relationship

$$C_p,\ c_f,\ C_F = f\ (\mathrm{Re},\ \mathrm{M},\ \mathrm{We},\ \mathrm{Fr})$$

if the independent π-groups are the same for the model and the prototype, then dependent π-groups must also be equal so

$$C_{p,m} = C_{p,p} \qquad c_{f,m} = c_{f,p} \qquad C_{F,m} = C_{F,p}$$

To have complete similitude between the model and the prototype, it is necessary to have both geometric and dynamic similitude.

8.6 Model Studies for Flows without Free-Surface Effects

Free-surface effects are absent in the flow of liquids or gases in closed conduits, including control devices such as valves, or in the flow about bodies (e.g., aircraft) that travel through air or are deeply submerged in a liquid such as water (submarines). Free-surface effects are also absent where a structure such as a building is stationary and wind flows past it. In all these cases, fluids reasonably may be assumed incompressible, the Reynolds-number criterion is the most significant for dynamic similarity. **That is, the Reynolds number for the model must equal the Reynolds number for the prototype**.

EXAMPLE 8.4 REYOLDS NUMBER SIMILITUDE

The drag characteristics of a blimp 5 m in diameter and 60 m long are to be studied in a wind tunnel. If the speed of the blimp through still air is 10 m/s, and if a 1/10 scale model is to be tested, what airspeed in the wind tunnel is needed **for dynamically similar conditions**? Assume the same air pressure and temperature for both model and prototype

Solution

Reynolds-number similitude

$$\mathrm{Re}_m = \mathrm{Re}_p$$

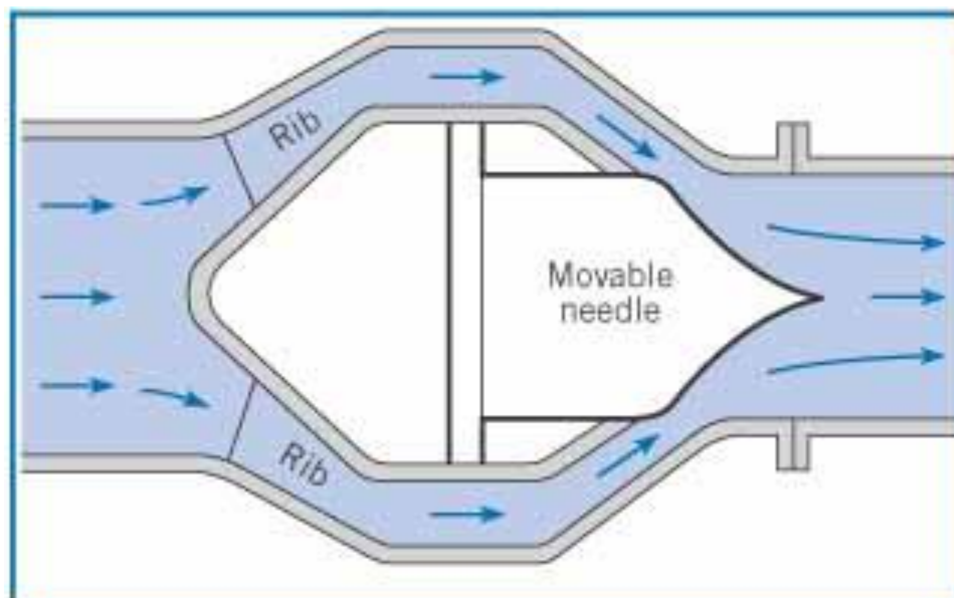
$$\frac{V_m L_m}{\nu_m} = \frac{V_p L_p}{\nu_p}$$

Model velocity

$$V_m = V_p \frac{L_p}{L_m} \frac{\nu_m}{\nu_p} = 10 \text{ m/s} \times 10 \times 1 = \boxed{100 \text{ m/s}}$$

EXAMPLE 8.5 REYOLDS NUMBER SIMILITUDE OF A VALVE

The valve shown is the type used in the control of water in large conduits. Model tests are to be done, using water as the fluid, to determine how the valve will operate under wide-open conditions. The prototype size is 6 ft in diameter at the inlet. What flow rate is required for the model if the prototype flow is 700 cfs? Assume that the temperature for model and prototype is 60°F and that the model inlet diameter is 1 ft.



Solution

Reynolds-number similitude

$$\mathrm{Re}_m = \mathrm{Re}_p$$

$$\frac{V_m L_m}{\nu_m} = \frac{V_p L_p}{\nu_p}$$

Velocity ratio

$$\frac{V_m}{V_p} = \frac{L_p}{L_m}\frac{\nu_m}{\nu_p}$$

Since $\nu_p = \nu_m$,

$$\frac{V_m}{V_p} = \frac{L_p}{L_m}$$

Discharge

$$\frac{Q_m}{Q_p} = \frac{V_m}{V_p}\frac{A_m}{A_p} = \frac{L_p}{L_m}\left(\frac{L_m}{L_p}\right)^2 = \frac{L_m}{L_p}$$

$$Q_m = 700 \text{ cfs} \times \frac{1}{6} = \boxed{117 \text{ cfs}}$$

8.7 Model-Prototype Performance

Geometric (scale model) and dynamic (same π-groups) similitude mean that the dependent π-groups are the same for both the model and the prototype. For this reason, measurements made with the model can be applied directly to the prototype. Such correspondence is illustrated in this section. Recall,

$$C_p, c_f, C_F = f\,(\mathrm{Re}, \mathrm{M}, \mathrm{We}, \mathrm{Fr})$$

EXAMPLE 8.6 APPLICATION OF PRESSURE COEFFICIENT

A 1/10 scale model of a blimp is tested in a wind tunnel under dynamically similar conditions. The speed of the blimp through still air is 10 m/s. A 17.8 kPa pressure difference is measured between two points on the model. What will be the pressure difference between the two corresponding points on the prototype? The temperature and pressure in the wind tunnel is the same as the prototype.

Solution

Reynolds-number similitude

$$\mathrm{Re}_m = \mathrm{Re}_p$$

$$\frac{V_m L_m}{\nu_m} = \frac{V_p L_p}{\nu_p}$$

$$\frac{V_p}{V_m} = \frac{L_m}{L_p} = \frac{1}{10}$$

Pressure coefficient correspondence

$$\frac{\Delta p_m}{\frac{1}{2}\rho_m V_m^2} = \frac{\Delta p_p}{\frac{1}{2}\rho_p V_p^2}$$

$$\frac{\Delta p_p}{\Delta p_m} = \left(\frac{V_p}{V_m}\right)^2 = \left(\frac{L_m}{L_p}\right)^2 = \frac{1}{100}$$

Pressure difference on prototype

$$\Delta p_p = \frac{\Delta p_m}{100} = \frac{17.8\ \mathrm{kPa}}{100} = \boxed{178\ \mathrm{Pa}}$$

EXAMPLE 8.7 DRAG FORCE FROM WIND TUNNEL TESTING

A 1/10 scale of a blimp is tested in a wind tunnel under dynamically similar conditions. If the drag force on the model blimp is measured to be 1530 N, what corresponding force could be expected on the prototype? The air pressure and temperature are the same for both model and prototype.

Solution

Reynolds-number similitude

$$\mathrm{Re}_m = \mathrm{Re}_p$$

$$\frac{V_m L_m}{\nu_m} = \frac{V_p L_p}{\nu_p}$$

$$\frac{V_p}{V_m} = \frac{L_m}{L_p} = \frac{1}{10}$$

Force coefficient correspondence

$$\frac{F_p}{\frac{1}{2}\rho_p V_p^2 L_p^2} = \frac{F_m}{\frac{1}{2}\rho_m V_m^2 L_m^2}$$

$$\frac{F_p}{F_m} = \frac{V_p^2}{V_m^2}\frac{L_p^2}{L_m^2} = \frac{L_m^2}{L_p^2}\frac{L_p^2}{L_m^2} = 1$$

Therefore

$$F_p = 1530\mathrm{N}$$

8.8 Approximate Similitude at High Reynolds Numbers

Consider the following dimensionless functional relationship,

$$C_p = f(Re)$$

In this case, the pressure coefficient is a function of the Reynolds number which is an important parameter for setting dynamical similarity between model and prototype for situations of incompressible fluids and free-surface effects are absent.

There are situations where the functional relationship approaches an asymptotic value with increasing Re, For example consider determining the pressure coefficient in a venturi as shown in Fig 8.6. The pressure difference is measured as shown and the velocity is at the throat.

It is seen from the figure that for Re greater than 3000, the C_p becomes constant; in other words, same C_p is obtained whether test is made at Re $50{,}000$ or $100{,}1000$.

This is what is meant by approximate similitude. That is, if the Re for the flow over the prototype is high, The Re for testing on the model may not be necessarily of the same large magnitude.

To understand the physics involved, recall the meaning of Re which is the ratio of kinetic (inertia) forces to viscous forces. For low Reynolds numbers, the viscous forces are significant, so pressure changes are a result of fluid acceleration and viscous resistance. However, for high Re, the viscous forces are of less importance and the pressure changes are due to kinetic forces, thus the Cp does not change; i.,e., the ratio of pressure difference to kinetic forces remain constant.

Two examples are given on approximate similitude.

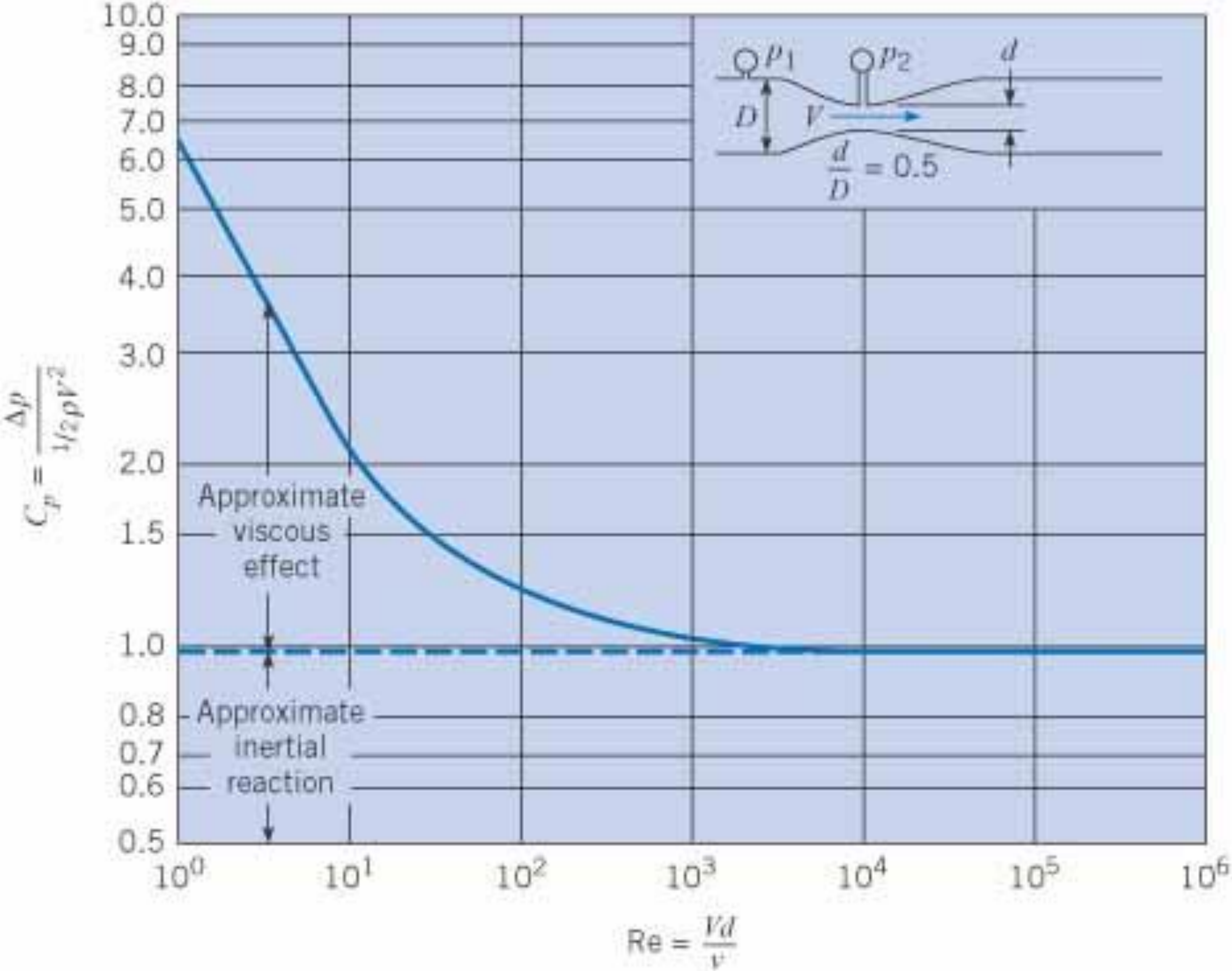


Figure 8.6 *C_p for a venturi meter as a function of the Reynolds number*

EXAMPLE 8.8 MEASURING HEAD LOSS IN NOZZLE IN REVERSE FLOW

Tests are to be performed to determine the head loss in a nozzle under a reverse-flow situation. The prototype operates with water at 50°F and with a nominal reverse-flow velocity of 5 ft/s. The diameter of the prototype is 3 ft. The tests are done in a 1/12 scale model facility with water at 60°F. A head loss (pressure drop) of 1 psid is measured with a velocity of 20 ft/s on the model. What will be the head loss in the actual nozzle? Assume approximate similitude to exist for Re>3000.

Solution

Determine the Re for model and prototype,

$$\mathrm{Re}_m = \frac{VD}{\nu} = \frac{20\ \mathrm{ft/s} \times (3/12\ \mathrm{ft})}{1.22 \times 10^{-5}\ \mathrm{ft^2/s}} = 4.10 \times 10^5$$

$$\mathrm{Re}_p = \frac{5\ \mathrm{ft/s} \times 3\ \mathrm{ft}}{1.41 \times 10^{-5}\ \mathrm{ft^2/s}} = 1.06 \times 10^6$$

Because both Re for model and prototype exceed 3000, then by approximate similitude we have,

$$C_p)_p = C_p)_m$$

Thus, compute $C_p)_m$,

$$C_{p,m} = \frac{\Delta p}{\frac{1}{2}\rho V^2} = \frac{1 \text{ lbf/in}^2 \times 144 \text{ in}^2/\text{ft}^2}{\frac{1}{2} \times 1.94 \text{ slug/ft}^3 \times (20 \text{ ft/s})^2} = 0.371$$

Equate this value with $C_p)_p$ for the prototype and then find Δp و

$$\Delta p_p = 0.371 \times \frac{1}{2}\rho V^2 = 0.371 \times 0.5 \times 1.94 \text{ slug/ft}^3 \times (5 \text{ ft/s})^2$$

$$= 9.0 \text{ lbf/ft}^2 = \boxed{0.0625 \text{ psid}}$$

EXAMPLE 8.9 MODEL TESTS FOR DRAG FORCE ON AN AUTOMOBILE

A 1/10 scale of an automobile is tested in a wind tunnel with air at atmospheric pressure and 20°C. The automobile is 4 m long and travels at a velocity of 100 km/hr in air at the same conditions. What should the wind-tunnel speed be such that the measured drag can be related to the drag of the prototype? Experience shows that the dependent π-groups are independent of Reynolds numbers exceeding 10^5. The speed of sound is 1235 km/hr.

Solution

Compute Re for the actual automobile,

$$\mathrm{Re}_p = \frac{VL\rho}{\mu} = \frac{100 \text{ km/hr} \times 0.278 \text{ (m/s)/(km/hr)} \times 4 \text{ m}}{1.51 \times 10^{-5} \text{ m}^2\text{/s}}$$

$$= 7.4 \times 10^6$$

Since prototype Re exceeds 10^5, then the model testing can be conducted at at least Re = 10^5. Thus the wind speed on the model,

$$V_m \geq \text{Re}_m \frac{\nu_m}{L_m} = 10^5 \times \frac{1.51 \times 10^{-5}\ \text{m}^2/\text{s}}{0.4\ \text{m}}$$

$$\geq \boxed{3.8\ \text{m/s}}$$

Thus, for wind velocities greater than 3.8 m/s, dynamic similarity holds between the model and prototype, therefore, correlation of results can be made between model and prototype.

8.9 Free-Surface Model Studies Spillway Models

The flow over a spillway is a classic case of a free-surface flow. The major influence, besides the spillway geometry itself, on the flow of water over a spillway is the action of gravity. Hence the Froude-number similarity criterion is used for such model studies.

Typically, these flows have high Re (why?), thus viscous effect are insignificant. However, if the model is made too small, the viscous forces as well as the surface-tension forces would have a larger relative effect on the flow in the model than in the prototype. Therefore, in practice, spillway models are made large enough so that the viscous effects have about the same relative effect in the model as in the prototype

It is not uncommon to design and construct model spillway sections that are 2 m or 3 m high. Figures 8.7 and 8.8 show a comprehensive model and spillway model for Hell's Canyon Dam in Idaho.

EXAMPLE 8.10 MODELING FLOOD DISCHARGE OVER A SPILLWAY

A 1/49 scale model of a proposed dam is used to predict prototype flow conditions. If the design flood discharge over the spillway is 15,000 m^3/s, what water flow rate should be established in the model to simulate this flow? If a velocity of 1.2 m/s is measured at a point in the model, what is the velocity at a corresponding point in the prototype?

Solution

This is a free-surface problem; i.e., there is a free surface in the flow and gravity has effects on the flow pattern. Therefore, for dynamic similarity, Froude no must be the same,

$$\mathrm{Fr}_m = \mathrm{Fr}_p$$

$$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}}$$ which leads to $$\frac{V_m}{V_p} = \sqrt{\frac{L_m}{L_p}}$$

The question is about flow rate which can be obtained by multiplying velocity and area (L^2), thus

$$\frac{Q_m}{Q_p} = \frac{A_m}{A_p}\frac{V_m}{V_p} = \frac{L_m^2}{L_p^2}\sqrt{\frac{L_m}{L_p}} = \left(\frac{L_m}{L_p}\right)^{5/2}$$

Thus, the discharge for the model,

$$Q_m = Q_p\left(\frac{1}{49}\right)^{5/2} = 15{,}000\frac{\text{m}^3}{\text{s}} \times \frac{1}{16{,}800} = \boxed{0.89\ \text{m}^3/\text{s}}$$

The velocity for the corresponding point on the prototype is,

$$\frac{V_p}{V_m} = \sqrt{\frac{L_p}{L_m}}$$

$$V_p = \sqrt{49} \times 1.2\ \text{m/s} = \boxed{8.4\ \text{m/s}}$$

Fluid Mechanics

Chapter 9
Surface Resistance

Dr. Amer Khalil Ababneh

Wind tunnel used for testing flow over models.

Introduction

Resistances exerted by surfaces are a result of viscous stresses which create resistance (or drag) to motion as a body travels through a fluid. Aeronautical engineers and naval architects are vitally interested in the drag on an airplane or the surface.

The phenomena responsible for shear stress are viscosity, and velocity gradients presented in Chapter 2.

In addition the concepts of the boundary layer and separation, introduced in Chapter 4, will be further expanded.

Will consider surface resistance in two flow situations:

1) uniform flows

2) Non-uniform flows; i.e., boundary-layer flows

- Laminar boundary-layer flows
- Turbulent boundary-layer flows

9.1 Surface Resistance with Uniform Laminar Flow

A one-dimensional laminar flow with parallel streamlines occurs for example between two plates: one is stationary and the other moving and also between two stationary parallel plates. These flows are uniform and steady. These flows illustrate the connections between velocity gradient and shear stress.

Differential Equation for Uniform Laminar flow

Consider the control volume shown in Fig. 9.1, which is aligned with the flow direction *s*. The streamlines are inclined at an angle θ wi respect to the horizontal plane. The control volume has dimensions $\Delta s \times \Delta y \times$ unity; that is, the control volume has a unit length into the page. By application of the momentum equation, the sum of the forces acting in the *s*-direction is equal to the net outflow of momentum from the control volume. The flow is unifor so the outflow of momentum is equal to the inflow and the momentum equation reduces to

$$\sum F_s = 0$$

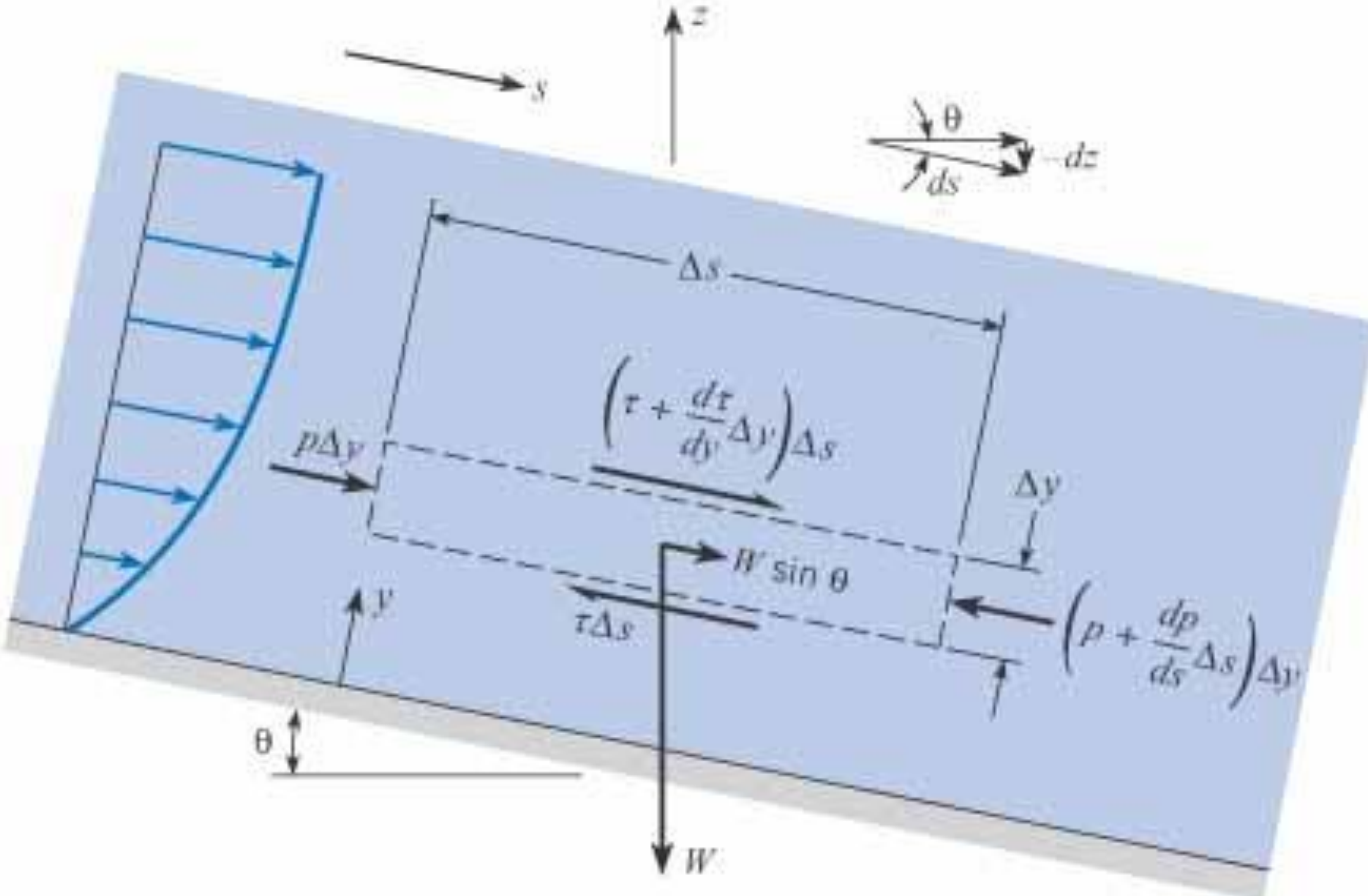


Figure 9.1 *Control volume for analysis of uniform flow with parallel streamlines.*

There are three types of forces that act on the control volume:

- Pressure forces
- Shear forces
- Weight

The pressure forces in s direction,

$$p\Delta y - \left(p + \frac{dp}{ds}\Delta s\right)\Delta y = -\frac{dp}{ds}\Delta s\Delta y$$

The net force due to shear stress is,

$$\left(\tau + \frac{d\tau}{dy}\Delta y\right)\Delta s - \tau\Delta s = \frac{d\tau}{dy}\Delta y\Delta s$$

The weight along the flow direction is,

$$\rho g\Delta s\Delta y \ \sin\theta = -\gamma\Delta y\Delta s\frac{dz}{ds}$$

Summing the above forces and dividing by ΔsΔy lead to,

$$\frac{d\tau}{dy} = \frac{d}{ds}(p + \gamma z)$$

Substituting for shear stress $\tau = \mu\, du/dy$, leads to

$$\frac{d^2u}{dy^2} = \frac{1}{\mu}\frac{d}{ds}(p + \gamma z) \qquad (9.3)$$

where μ is constant. This equation is now applied to a flow between two plates; one is moving and the other is stationary.

Flow Produced by a Moving Plate (Couette Flow)

Consider the flow between the two plates shown in Fig. 9.2. The lower plate is fixed, and the upper plate is moving with a speed U. The plates are separated by a distance L. In this problem there is no pressure gradient in the flow direction ($dp/ds = 0$), and the streamlines are in the horizontal direction ($dz/ds = 0$),so Eq. (9.3) reduces to

$$\frac{d^2u}{dy^2} = 0$$

along with the two boundary conditions:

$$u = 0 \quad \text{at} \quad y = 0$$
$$u = U \quad \text{at} \quad y = L$$

Integrating this equation twice gives

$$u = C_1 y + C_2$$

Applying the boundary conditions results in

$$u = \frac{y}{L} U$$

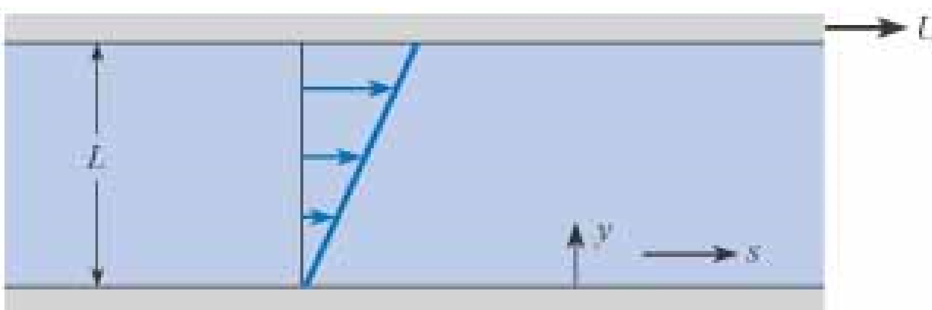


Figure 9.2 *Flow generated by a moving plate (Couette flow).*

The shear stress is constant and equal to

$$\tau = \mu \frac{du}{dy} = \mu \frac{U}{L}$$

This flow is known as a *Couette flow* after a French scientist, M. Couette, who did pioneering work on the flow between parallel plates and rotating cylinders. It has application in the design of **lubrication systems**.

EXAMPLE 9.1 SHEAR STRESS I, COUETTE FLOW

SAE 30 lubricating oil at *T* = 38°C flows between two parallel plates, one fixed and the other moving at 1.0 m/s. Plates are spaced 0.3 mm apart. What is the shear stress on the plates?

Solution:

$$\tau = \mu \frac{du}{dy} = \mu \frac{U}{L}$$

$$= (1.0 \times 10^{-1}\ \mathrm{N \cdot s/m^2})(1.0\ \mathrm{m/s}) / (3 \times 10^{-4}\ \mathrm{m})$$

$$\tau = \boxed{333\ \mathrm{N/m^2}}$$

This is significant.

Non-Uniform Flows: Boundary-Layer Flows

9.2 Qualitative Description of the Boundary Layer

The *boundary layer* is the region adjacent to a surface over which the velocity changes from the free- stream value (with respect to the object) to zero at the surface. This region, which is generally very thin, occurs because of the viscosity of the fluid. The velocity gradient at the surface is responsible for the viscous shear stress and surface resistance.

The boundary-layer development for flow past a thin plate oriented parallel to the flow direction shown in Fig. 9.4*a*. The thickness of the boundary layer, δ, is defined as the distance from the surface to the point where the velocity is 99% of the free-stream velocity. The actual thickness of a boundary layer may be 2%–3% of the plate length, so the boundary-layer thickness shown in Fig. 9.4*a* is exaggerated at least by a factor of five to show details of the flow field. Fluid passes over the top and underneath the plate, so two boundary layers are depicted (one above and one below the plate). For convenience, the surface is assumed to be stationary, and the free-stream fluid is moving at a velocity U_o.

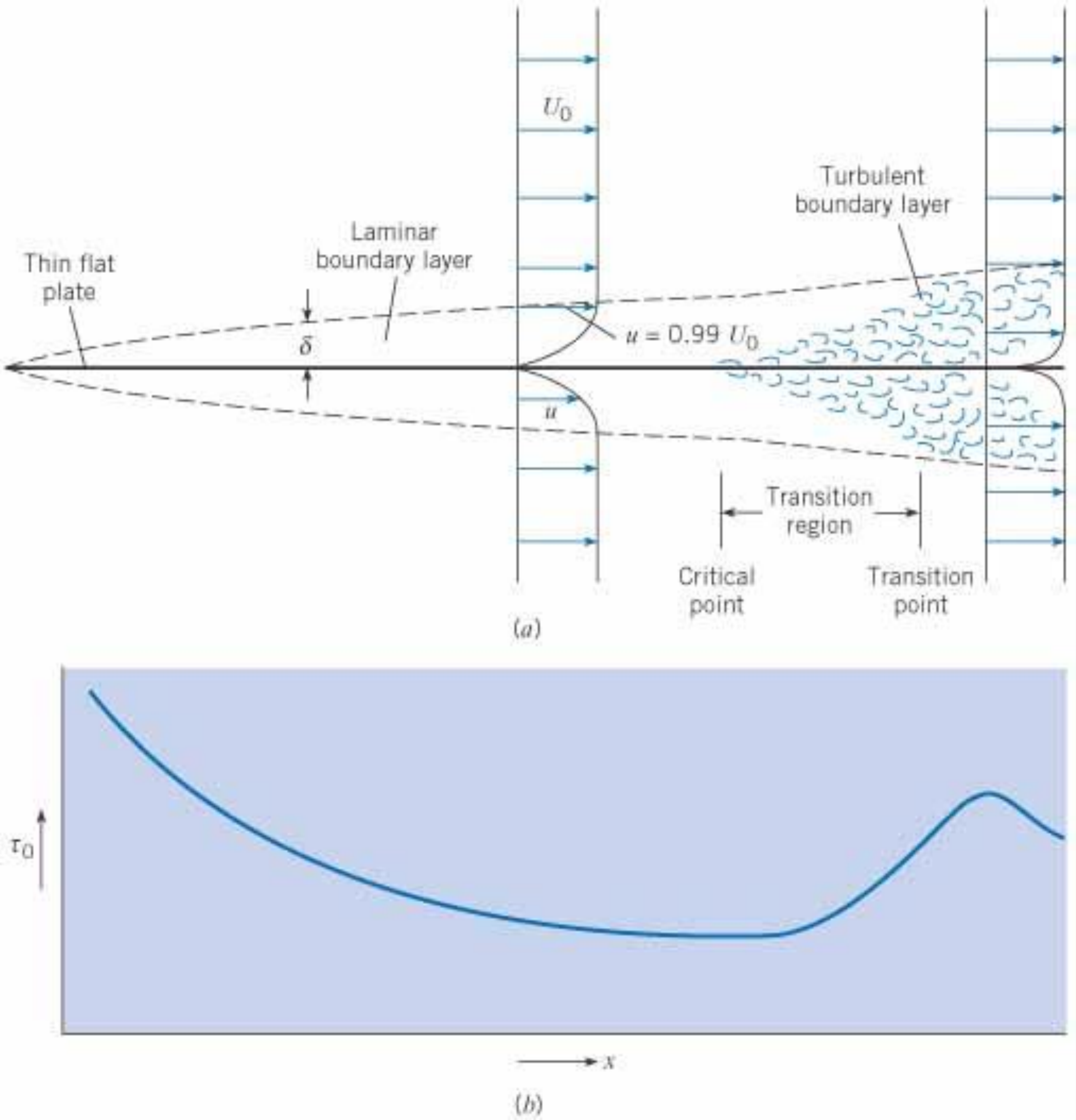


Figure 9.4
Development of boundary layer and shear stress along a thin, flat plate.
(a) Flow pattern above and below the plate.
(b) Shear-stress distribution on either side of plate.

The development and growth of the boundary layer occurs because of the “no-slip” condition at the surface; that is, the fluid velocity at the surface must be zero. As the fluid particles next to the plate pass close to the leading edge of the plate, a retarding force (from the shear stress) begins to act on the particles to slow them down. As these particles progress farther downstream, they continue to be subjected to shear stress from the plate, so they continue to decelerate. In addition, these particles (because of their lower velocity) retard other particles adjacent to them but farther out from the plate. Thus the boundary layer becomes thicker, or “grows,” in the downstream direction. The broken line in Fig. 9.4*a* identifies the outer limit of the boundary layer. As the boundary layer becomes thicker, the velocity gradient at the wall becomes smaller and the local shear stress is reduced.

The initial section of the boundary layer is the laminar boundary layer. In this region the flow is smooth and steady. Thickening of the laminar boundary layer continues smoothly in the downstream direction until a point is reached where the boundary layer becomes unstable. Beyond this point, the critical point, small disturbances in the flow will grow and spread, leading to turbulence. The boundary becomes fully turbulent at the transition point. The region between the critical point and the transition point is called the transition region.

The turbulent boundary layer is characterized by intense cross-stream mixing as turbulent eddies transport high-velocity fluid from the boundary layer edge to the region close to the wall. This cross-stream mixing gives rise to a high effective viscosity, which can be three orders of magnitude higher than the actual viscosity of the fluid itself. The effective viscosity, due to turbulent mixing is not a property of the fluid but rather a property of the flow, namely, the mixing process. Because of this intense mixing, the velocity profile is much “fuller” than the laminar-flow velocity profile as shown in Fig. 9.4*a*. This situation leads to an increased velocity gradient at the surface and a larger shear stress.

The shear-stress distribution along the plate is shown in Fig. 9.4*b*. It is easy to visualize that the shear stress must be relatively large near the leading edge of the plate where the velocity gradient is steep, and that it becomes progressively smaller as the boundary layer thickens in the downstream direction. At the point where the boundary layer becomes turbulent, the shear stress at the boundary increases because the velocity profile changes producing a steeper gradient at the surface.

9.3 Laminar Boundary Layer - Boundary-Layer Equations

In 1904 Prandtl, 1 first stated the essence of the boundary-layer hypothesis, which is that viscous effects are concentrated in a thin layer of fluid (the boundary layer) next to solid boundaries.

In 1908, Blasius, one of Prandtl's students, obtained a solution for the flow in a laminar boundary layer on a flat plate with a constant free-stream velocity. One of Blasius's key assumptions was that the shape of the nondimensional velocity distribution did not vary from section to section along the plate. That is, he assumed that a plot of the relative velocity, u/U_0, versus the relative distance from the boundary, y/δ, would be the same at each section. With this assumption and with Prandtl's equations of motion for boundary layers, Blasius obtained a numerical solution for the relative velocity distribution, shown in Fig. 9.5. In this plot, x is the distance from the leading edge of the plate, and Re_x is the Reynolds number based on the free-stream velocity and the length along the plate ($\mathrm{Re}_x = U_0 x/\nu$). In Fig. 9.5 the outer limit of the boundary layer ($u/U_0 = 0.99$) occurs at approximately,

$$\frac{y}{x}\mathrm{Re}_x^{1/2} = 5$$

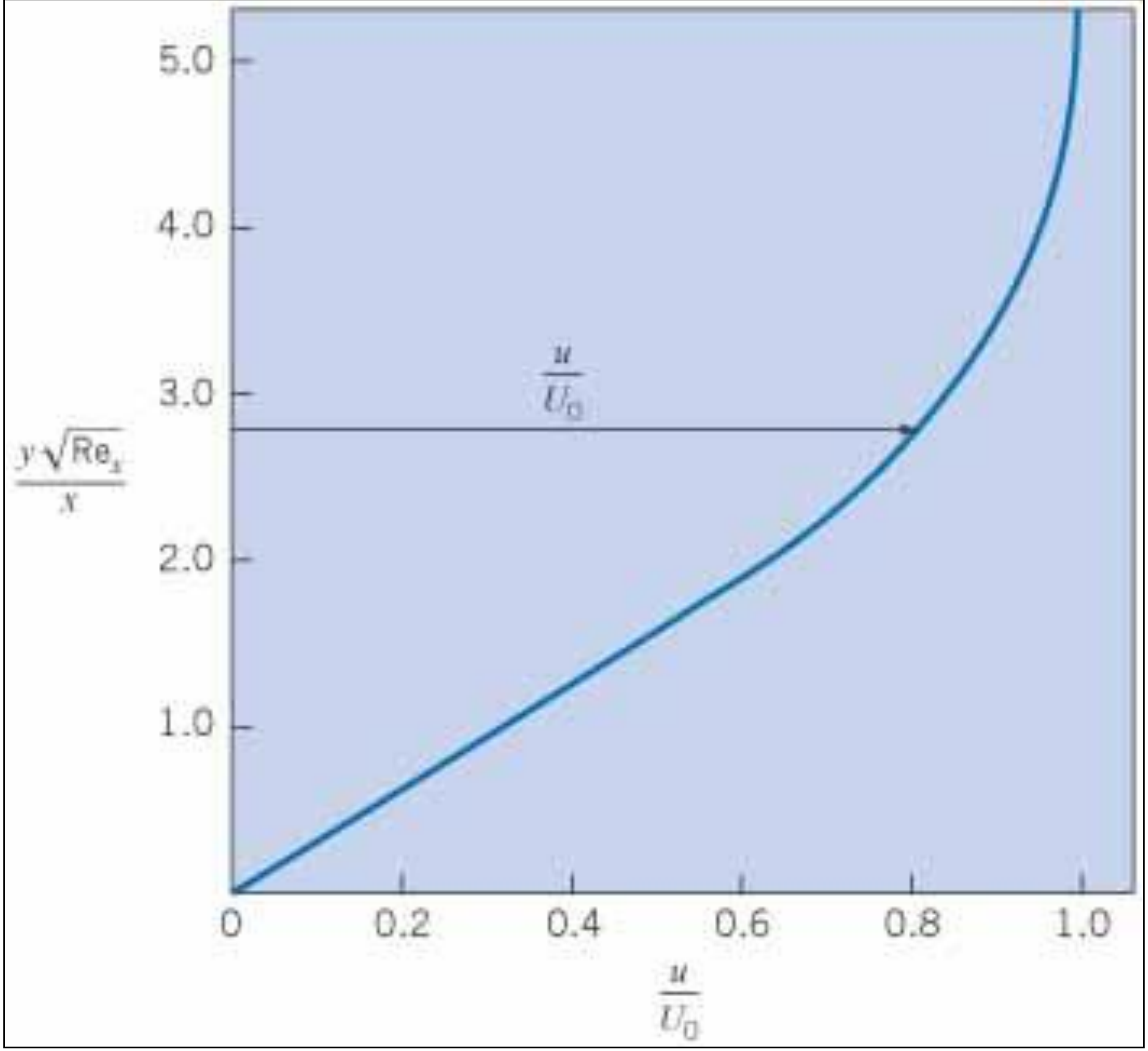


Figure 9.5 *Velocity distribution in laminar boundary layer.*

Since $y = \delta$ at this point, the following relationship is derived for the *boundary-layer thickness* in laminar flow on a flat plate

$$\frac{\delta}{x}\mathrm{Re}_x^{1/2} = 5 \quad \text{or} \quad \delta = \frac{5x}{\mathrm{Re}_x^{1/2}}$$

The Blasius solution also showed that

$$\left.\frac{d(u / U_0)}{d[(y / x)\mathrm{Re}_x^{1/2}]}\right|_{y=0} = 0.332$$

which can be used to find the shear stress at the surface. The velocity gradient at the boundary becomes (at a section; x =constant)

$$\left.\frac{du}{dy}\right|_{y=0} = 0.332\frac{U_0}{x}\mathrm{Re}_x^{1/2}$$

$$\left.\frac{du}{dy}\right|_{y=0} = 0.332\frac{U_0^{3/2}}{x^{1/2}\nu^{1/2}}$$

Shear Stress

The shear stress at the boundary is obtained from

$$\tau_0 = \mu \frac{du}{dy}\bigg|_{y=0} = 0.332\mu \frac{U_0}{x} \mathrm{Re}_x^{1/2}$$

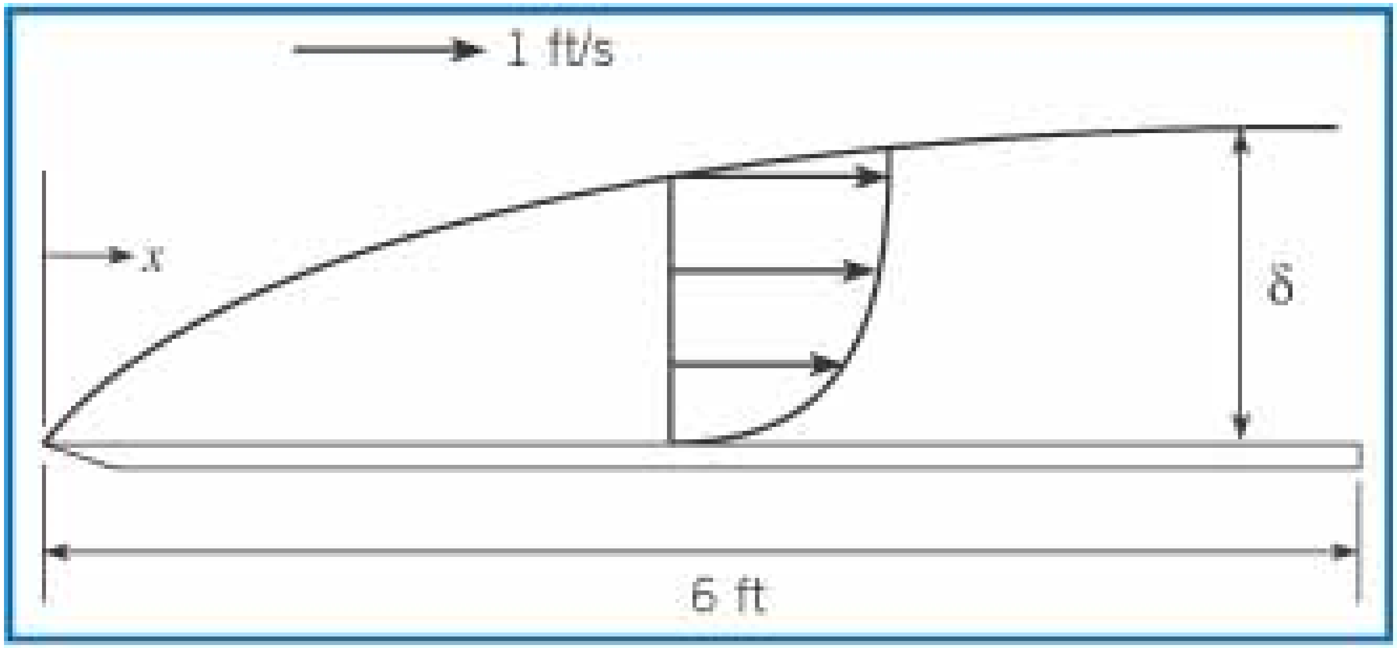
Surface Resistance

Because the shear stress at the boundary, τ_o, varies along the plate, it is necessary to integrate this stress over the entire surface to obtain the total surface resistance, F_s. For one side of the plate,

$$F_s = \int_0^L \tau_0 B \, dx \qquad (9.13)$$

EXAMPLE 9.3 LAMINAR BOUNDARY-LAYER THICKNESS AND SHEAR STRESS

Crude oil at 70°F(ν = 10-4 ft2/s, S = 0.86) with a free-stream velocity of 1 ft/s flows past a thin, flat plate that is 4 ft wide and 6 ft long in a direction parallel to the flow. The flow is laminar. Determine and plot the boundary-layer thickness and the shear stress distribution along the plate.



Solution

Reynolds-number variation with distance

$$\mathrm{Re}_x = \frac{U_0 x}{\nu} = \frac{1 \times x}{10^{-4}} = 10^4 x$$

Boundary-layer thickness

$$\delta = \frac{5x}{\mathrm{Re}_x^{1/2}} = \frac{5x}{10^2 x^{1/2}} = 5 \times 10^{-2} x^{1/2}\ \mathrm{ft}$$

Shear-stress distribution

$$\tau_0 = 0.332\mu \frac{U_0}{x} \mathrm{Re}_x^{1/2}$$

$$\mu = \rho\nu = 1.94\ \mathrm{slugs/ft^3} \times 0.86 \times 10^{-4}\ \mathrm{ft^3/s}$$

$$= 1.67 \times 10^{-4}\ \mathrm{lbf\text{-}s/ft^2}$$

$$\tau_0 = 0.332(1.67 \times 10^{-4}) \frac{1}{x} (10^2 x^{1/2}) = \frac{5.54 \times 10^{-3}}{x^{1/2}}\mathrm{psf}$$

The results for Example 9.3 are plotted in the accompanying figure and listed in Table 9.1.

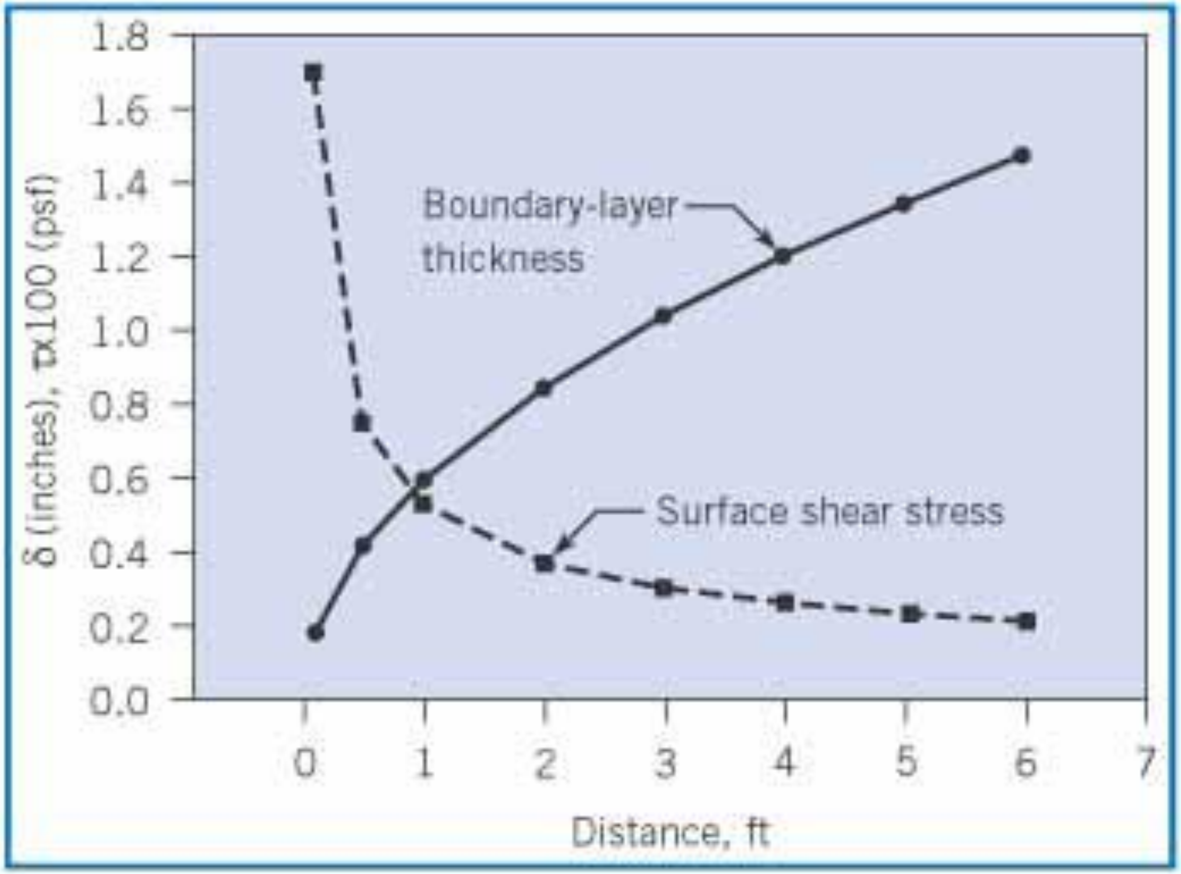


Table 9.1 RESULT— δ AND τo FOR DIFFERENT VALUES OF *x*

	x = 0.1 ft	**x = 1.0 ft**	**x = 2 ft**	**x = 4 ft**	**x = 6 ft**
$x^{1/2}$	0.316	1.00	1.414	2.00	2.45
τ_0, psf	0.018	0.0055	0.0037	0.0028	0.0023
δ, ft	0.016	0.050	0.071	0.10	0.122
δ, in	0.190	0.600	0.848	1.200	1.470

To find the resistive force, substitute in Eq. (9.13) for τ_0 and integrate, giving

$$F_s = \int_0^L 0.332 B\mu \frac{U_0 U_0^{1/2} x^{1/2}}{x \nu^{1/2}} dx$$

$$= 0.664 B\mu U_0 \frac{U_0^{1/2} L^{1/2}}{\nu^{1/2}} \qquad (9.14)$$

$$= 0.664 B\mu U_0 \mathrm{Re}_L^{1/2}$$

Shear-Stress Coefficients

It is convenient to express the shear stress at the boundary, τ_0, and the total shearing force F_s in terms of π-groups involving the kinetic pressure of the free stream, . The *local shear-stress coefficient*, c_f, is defined as

$$c_f = \frac{\tau_0}{\rho U_0^2 / 2}$$

Substituting the value for τ_0 in the above gives c_f as a function of Reynolds number based on the distance from the leading edge.

$$c_f = \frac{0.664}{\mathrm{Re}_x^{1/2}} \quad \text{where} \quad Re_x = \frac{Ux}{\nu}$$

The total shearing force, given by Eq. (9.13), can also be expressed as a π-group

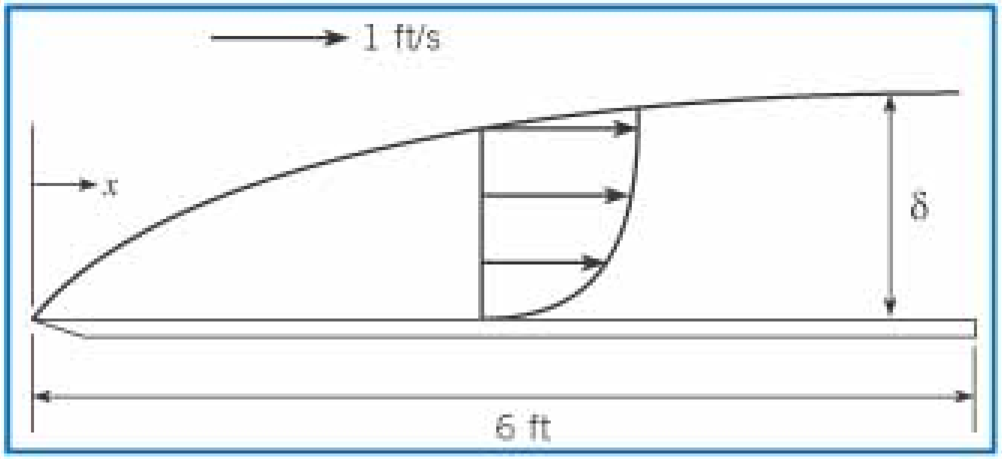
$$C_f = \frac{F_s}{(\rho U_0^2 / 2)A}$$

where *A* is the plate area. This π-group is called the average shear-stress coefficient. Substituting Eq. (9.14) into the definition of C_f, gives

$$C_f = \frac{1.33}{\mathrm{Re}_L^{1/2}} \quad \text{where} \quad Re_L = \frac{UL}{\nu}$$

EXAMPLE 9.4 RESISTANCE CALCULATION FOR LAMINAR BOU#DARY LAYER ON A FLAT PLATE

Crude oil at 70°F (v = 10-4 ft2 /s, S = 0.86.) with a free-stream velocity of 1 ft/s flows past a thin, flat plate that is 4 ft wide and 6 ft long in a direction parallel to the flow. The flow is laminar. Determine the resistance on one side of the plate.



Solution

Reynolds number.

$$\mathrm{Re} = \frac{U_0 L}{\nu} = \frac{1\ \mathrm{ft/s} \times 6\ \mathrm{ft}}{10^{-4}\ \mathrm{ft}^2/s} = 6 \times 10^4$$

Value for C_f

$$C_f = \frac{1.33}{\mathrm{Re}_L^{1/2}} = \frac{1.33}{(6 \times 10^4)^{1/2}} = 0.0054$$

Total shear force.

$$\begin{aligned} F_s &= \frac{C_f B L \rho U_0^2}{2} \\ &= 0.0054 \times 4\ \mathrm{ft} \times 6\ \mathrm{ft} \times 0.86 \\ &\quad \times 1.94\ \mathrm{slugs/ft^3} \times \frac{1^2 \mathrm{ft}/s^2}{2} = \boxed{0.108\ \mathrm{lbf}} \end{aligned}$$

9.4 Boundary Layer Transition

Transition is the zone where the laminar boundary layer changes into a turbulent boundary layer as shown in Fig. 9.4*a*. As the laminar boundary layer continues to grow, the viscous stresses are less capable of damping disturbances in the flow. A point is then reached where disturbances occurring in the flow are amplified, leading to turbulence. The critical point occurs at a Reynolds number of about 10^5(Re_{cr} 10^5) based on the distance from the leading edge. Vortices created near the wall grow and mutually interact, ultimately leading to a fully turbulent boundary layer at the transition point, which nominally occurs at a Reynolds number of 3×10^6(Re_{tr} 3×10^6). For purposes of simplicity in this text, it will be assumed that the boundary layer changes from laminar to turbulent flow at a Reynolds number 500,000.

Transition to a turbulent boundary layer can be influenced by several other flow conditions, such as free-stream turbulence, pressure gradient, wall roughness, wall heating, and wall cooling. With appropriate roughness elements at the leading edge, the boundary layer can become turbulent at the very beginning of the plate. In this case it is said that the boundary layer is “tripped” at the leading edge.

9.5 Turbulent Boundary Layer

In the majority of practical problems the boundary layer is turbulent which is primarily responsible for surface shear force, or surface resistance.

Velocity Distribution

The velocity distribution in the turbulent boundary layer is more complicated than the laminar boundary layer. The turbulent boundary has **three zones** of flow that require different equations for the velocity distribution in each zone, as opposed to the single relationship of the laminar boundary layer. Figure 9.6 shows a portion of a turbulent boundary layer in which the three different zones of flow are identified. The zone adjacent to the wall is the viscous sublayer; the zone immediately above the viscous sublayer is the logarithmic region; and, finally, beyond that region is the velocity defect region. Each of these velocity zones will be discussed separately. **Figure 9.6** *Sketch of zones in turbulent boundary layer.* Viscous Sublayer The zone

The three zones in a turbulent boundary layer.

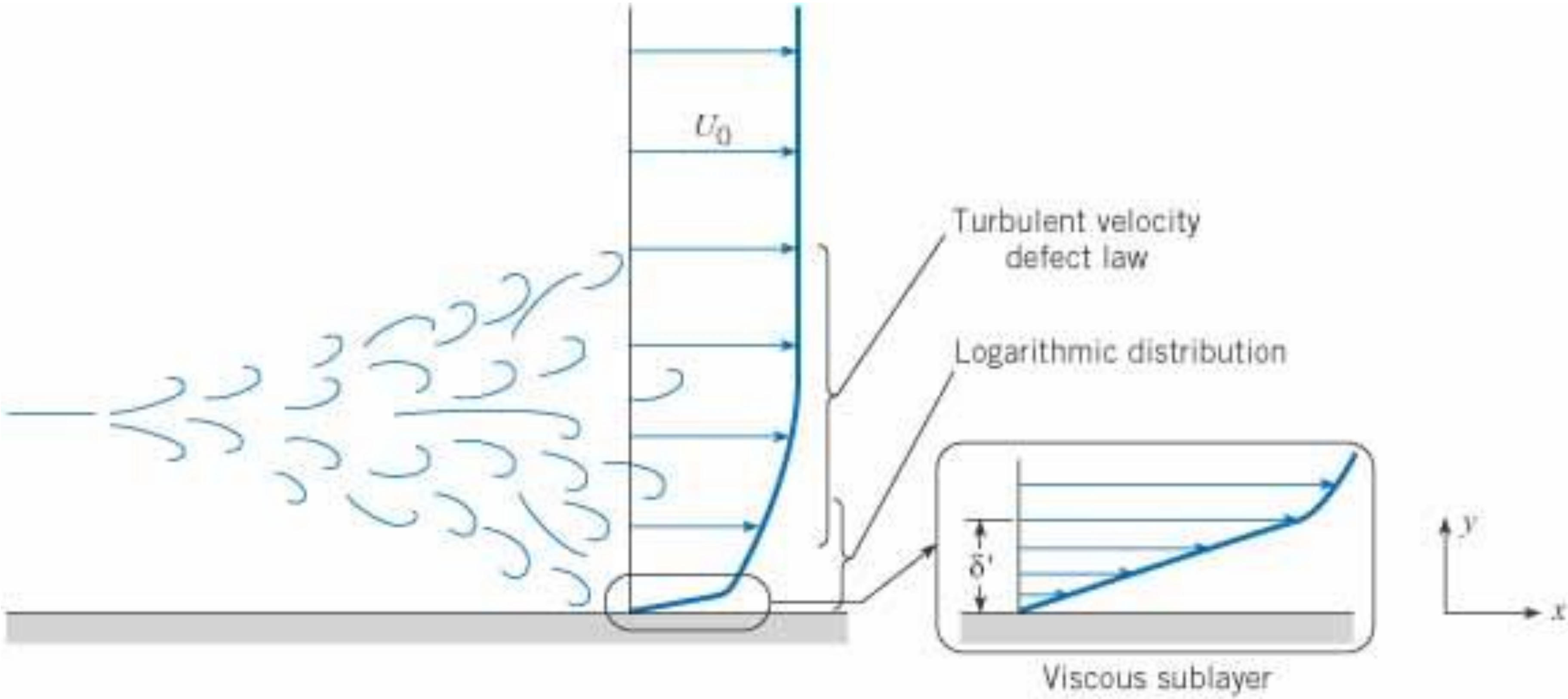


Figure 9.6 *Sketch of zones in turbulent boundary layer.*

Boundary-Layer Thickness and Shear-Stress Correlations

Unlike the laminar boundary layer, there is no analytically derived equation for the thickness of the turbulent boundary layer. It can be shown that the thickness of the turbulent boundary layer is

$$\delta = \frac{0.16x}{\mathrm{Re}_x^{1/7}}$$

where x is the distance from the leading edge of the plate and $\mathrm{Re}x$ is U_0x/ν.

Many empirical expressions have been proposed for the local shear-stress distribution for the turbulent boundary layer on a flat plate. One of the simplest correlations is

$$c_f = \frac{\tau_0}{\rho U_0^2 / 2} = \frac{0.027}{\mathrm{Re}_x^{1/7}}$$

The corresponding average shear-stress coefficient is

$$C_f = \frac{0.523}{\ln^2(0.06\mathrm{Re}_L)}$$

The boundary layer, however, usually consists of laminar and turbulent. To account for this combined effect, it can be shown that the average shear-stress coefficient for this case is,

$$C_f = \frac{0.523}{\ln^2(0.06\mathrm{Re}_L)} - \frac{1520}{\mathrm{Re}_L}$$

The variation of C_f with Reynolds number is shown by the solid line in Fig. 9.12. This curve corresponds to a boundary layer that begins as a laminar boundary layer and then changes to a turbulent boundary layer after the transition Reynolds number. This is the normal condition for a flat-plate boundary layer. Table 9.3 summarizes the equations for boundary-layer-thickness, and for local shear-stress and average shear-stress coefficients for the boundary layer on a flat plate.

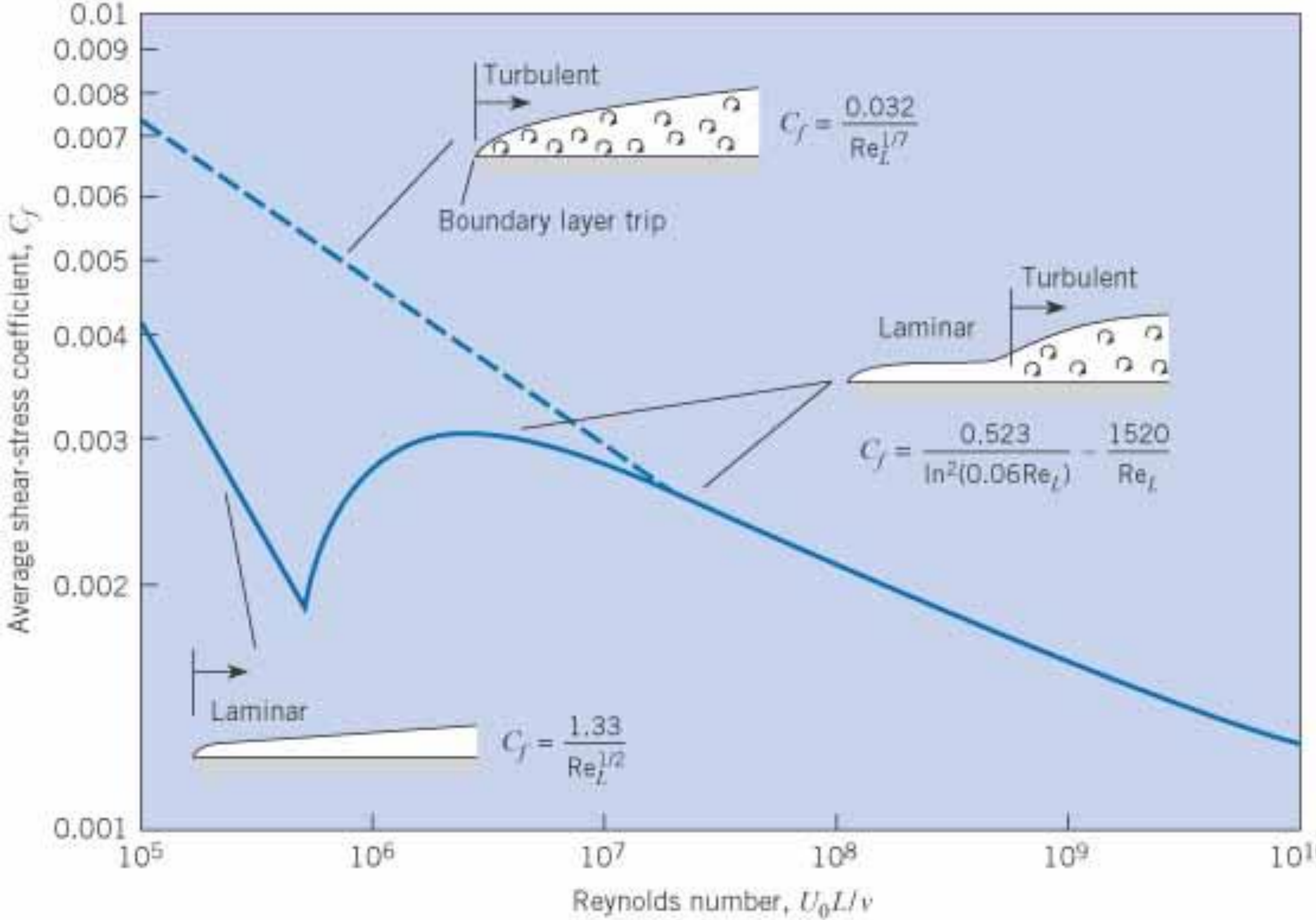


Figure 9.12 *Average shear-stress coefficients.*

Table 9.3 SUMMARY OF EQUATIONS FOR BOU"DARY LAYER ON A FLAT PLATE

	Laminar Flow Re*x*, Re*L* < 5 × 10^5	Turbulent Flow Re*x*, Re*L* ≥ 5 × 10^5
Boundary-Layer Thickness, δ	$\delta = \dfrac{5x}{\mathrm{Re}_x^{1/2}}$	$\delta = \dfrac{0.16x}{\mathrm{Re}_x^{1/7}}$
Local Shear-Stress Coefficient, c_f	$c_f = \dfrac{0.664}{\mathrm{Re}_x^{1/2}}$	$c_f = \dfrac{0.455}{\ln^2(0.06\mathrm{Re}_x)}$
Average Shear-Stress Coefficient, C_f	$C_f = \dfrac{1.33}{\mathrm{Re}_L^{1/2}}$	$C_f = \dfrac{0.523}{\ln^2(0.06\mathrm{Re}_L)} - \dfrac{1520}{\mathrm{Re}_L}$

If the boundary layer is “tripped” by some roughness or leading-edge disturbance (such as a wire across the leading edge), the boundary layer is turbulent from the leading edge. This is shown by the dashed line in Fig. 9.12. For this condition the boundary layer thickness, local shear-stress coefficient, and average shear-stress coefficient are fit reasonably well by the following equations,

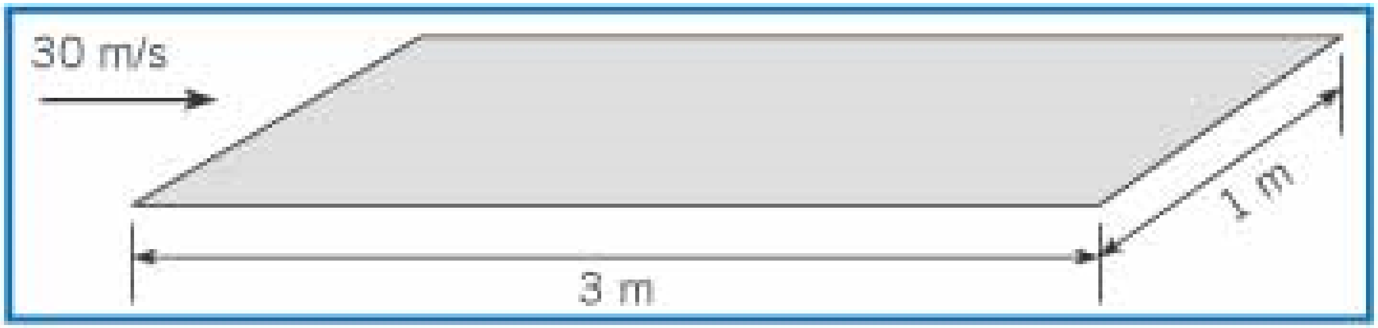
$$\delta = \frac{0.16x}{\mathrm{Re}_x^{1/7}} \quad c_f = \frac{0.027}{\mathrm{Re}_x^{1/7}} \quad C_f = \frac{0.032}{\mathrm{Re}_L^{1/7}}$$

which are valid up to a Reynolds number of 10^7.

EXAMPLE 9.6 LAMINAR/TURBULENT BOUNDARY LAYER ON FLAT PLATE

Assume that air 20°C and normal atmospheric pressure flows over a smooth, flat plate with a velocity of 30 m/s. The initial boundary layer is laminar and then becomes turbulent at a transitional Reynolds number of 5×10^5. The plate is 3 m long and 1 m wide. What will be the average resistance coefficient C_f for the plate? Also, what is the total shearing resistance of one side of the plate, and what will be the resistance due to the turbulent part and the laminar part of the boundary layer?

Air properties at 20 °C: $\rho = 1.2 \text{kg}/\text{m}^3, \nu = 1.51 \times 10^{-5} \text{ m}^2/\text{s}.$



Solution

Reynolds number based on plate length

$$\mathrm{Re}_L = \frac{30\ \mathrm{m/s} \times 3\ \mathrm{m}}{1.51 \times 10^{-5}\ \mathrm{m}^2/\mathrm{s}} = 5.96 \times 10^6$$

Average shear-stress coefficient

$$C_f = \frac{0.523}{\ln^2(0.06\mathrm{Re}_L)} - \frac{1520}{\mathrm{Re}_L} = \boxed{0.00294}$$

Total shear force

$$F_s = C_f B L \rho (U_0^2/2)$$

$$= 0.00294 \times 1\ \mathrm{m} \times 3\ \mathrm{m} \times 1.2\ \mathrm{kg/m}^3 \times \frac{(30\ \mathrm{m/s})^2}{2} = \boxed{4.76\ \mathrm{N}}$$

Transition point

$$\frac{U x_{\mathrm{tr}}}{\nu} = 500{,}000$$

$$x_{\mathrm{tr}} = \frac{500{,}000 \times 1.51 \times 10^{-5}}{30} = 0.252\ \mathrm{m}$$

Laminar average shear-stress coefficient

$$C_f = \frac{1.33}{\mathrm{Re}_{\mathrm{tr}}^{1/2}} = 0.00188$$

Laminar shear force

$$F_{s,\mathrm{lam}} = 0.00188 \times 1\ \mathrm{m} \times 0.252\ \mathrm{m} \times 1.2\ \mathrm{kg/m^3} \times \frac{(30\ \mathrm{m/s})^2}{2}$$

$$= \boxed{0.256\ \mathrm{N}}$$

Turbulent shear force

$$F_{s,\mathrm{turb}} = 4.76\ \mathrm{N} - 0.26\ \mathrm{N} = \boxed{4.50\ \mathrm{N}}$$

EXAMPLE 9.7 RESISTANCE FORCE WITH TRIPPED BOUNDARY LAYER

Air at 20°C flows past a smooth, thin plate with a free-stream velocity of 20 m/s. Plate is 3 m wide and 6 m long in the direction of flow and boundary layer is tripped at the leading edge.

Solution

Reynolds number

$$\mathrm{Re}_L = \frac{\rho U L}{\mu} = \frac{1.2 \times 20 \times 6}{1.81 \times 10^{-5}} = 7.96 \times 10^6$$

Reynolds number is less than 10^7. Average shear-stress coefficient

$$C_f = \frac{0.032}{\mathrm{Re}_L^{1/7}}$$

$$= \frac{0.032}{(7.96 \times 10^6)^{1/7}} = 0.0033$$

Resistance force

$$F_s = 2 \times C_f A \frac{\rho U_0^2}{2}$$

$$= 0.0033 \times 3\ \text{m} \times 6\ \text{m} \times 1.2\ \text{kg/m}^3 \times (20\ \text{m/s})^2$$

$$= \boxed{28.5\ \text{N}}$$

Fluid Mechanics

Chapter 10
Flow in Conduits

Dr. Amer Khalil Ababneh

The Alaskan pipeline, a significant accomplishment of the engineering profession, transports oil 1286 km across the state of Alaska. The pipe diameter is 1.2 m, and the 44 pumps are used to drive the flow

A *conduit* is any pipe, tube, or duct that is completely filled with a flowing fluid. Examples include a pipeline transporting liquefied natural gas, a microchannel transporting hydrogen in a fuel cell, and a duct transporting air for heating of a building. A pipe that is partially filled with a flowing fluid, for example a drainage pipe, is classified as an open-channel flow.

The main goal of this chapter is to describe how to predict head loss. Predicting head loss involves classifying flow as laminar or turbulent and then using equations to calculate head losses in pipes and components.

10.1 Classifying Flow

The flow in a conduit may be classified as: (a) whether the flow is laminar or turbulent, and (b) whether the flow is developing or fully developed.

Laminar Flow and Turbulent Flow

Flow in a conduit is classified as being either laminar or turbulent, depending on the magnitude of the Reynolds number. The original research involved visualizing flow in a glass tube as shown in Fig. 10.1*a*. Reynolds 1 in the 1880s injected dye into the center of the tube and observed the following:

- When the velocity was low, the streak of dye flowed down the tube with little expansion, as shown in Fig. 10.1*b*. However, if the water in the tank was disturbed, the streak would shift about in the tube.

- If velocity was increased, at some point in the tube, the dye would all at once mix with the water as shown in Fig. 10.1*c*.

- When the dye exhibited rapid mixing (Fig. 10.1*c*), illumination with an electric spark revealed eddies in the mixed fluid as shown in Fig. 10.1*d*.

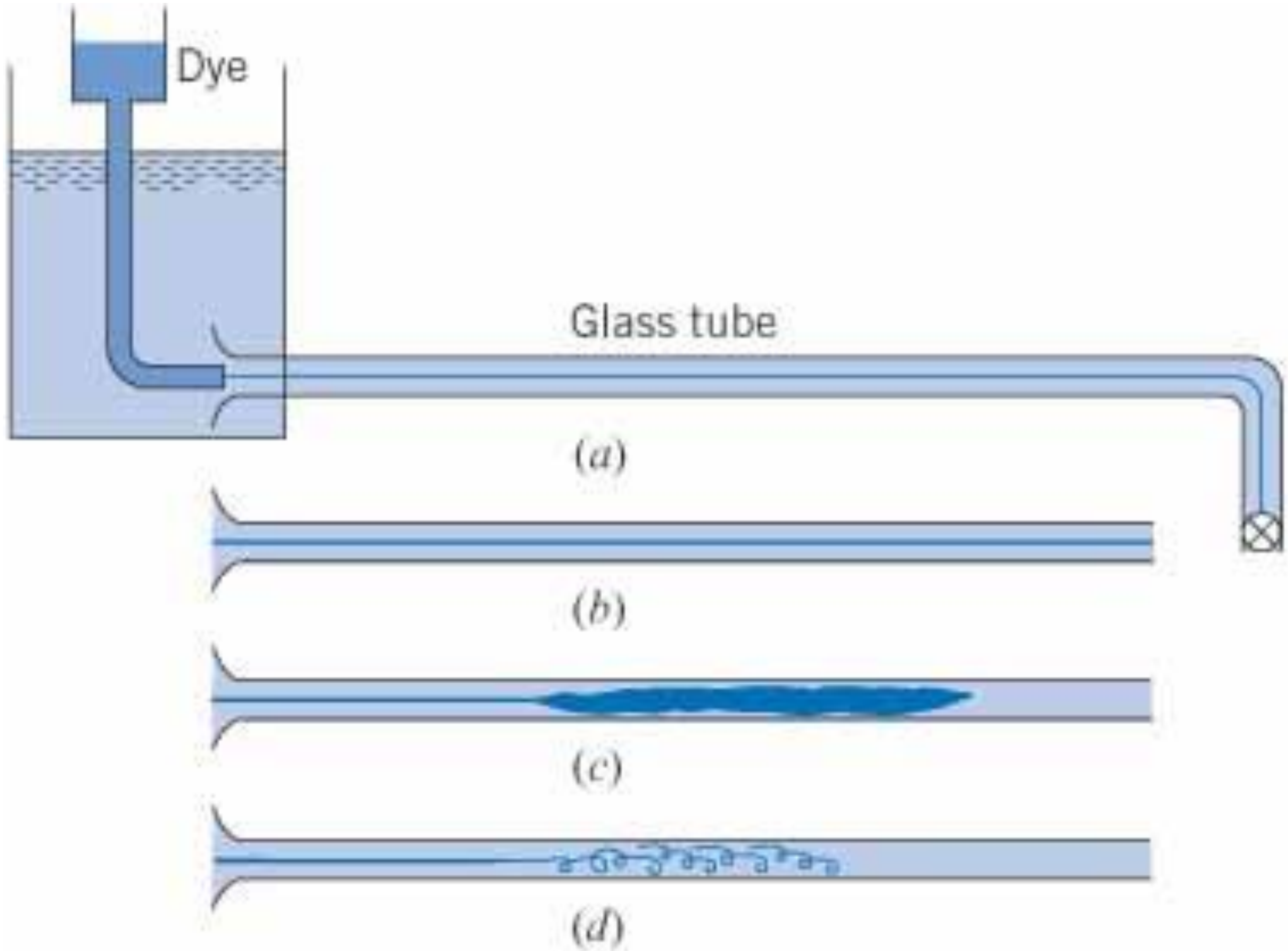


Figure 10.1 *Reynolds' experiment.*
(a) Apparatus.
(b) Laminar flow of dye in tube.
(c) Turbulent flow of dye in tube.
(d) Eddies in turbulent flow.

Reynolds showed that the onset of turbulence was related to a π-group that is now called the Reynolds number ($Re = \rho VD/\mu$) in honor of Reynolds' pioneering work. Reynolds discovered that if the fluid in the upstream reservoir was not completely still or if the pipe had some vibrations, then the change from laminar to turbulent flow occurred at Re ~ 2100. However, if conditions were ideal, it was possible to reach a much higher Reynolds number before the flow became turbulent. Reynolds also found that, when going from high velocity to low velocity, the change back to laminar flow occurred at Re ~ 2000. Based on Reynolds' experiments, engineers use guidelines to establish whether or not flow in a conduit will be laminar or turbulent. The guidelines used in this text are as follows:

$$
\begin{aligned}
&\mathrm{Re} \leq 2000 && \text{laminar flow} \\
&2000 \leq \mathrm{Re} \leq 3000 && \text{unpredictable} \\
&\mathrm{Re} \geq 3000 && \text{turbulent flow}
\end{aligned}
\qquad (10.1)
$$

The range (2000 ≤ Re ≤ 3000) corresponds to a the type of flow that is unpredictable because it can changes back and forth between laminar and turbulent states.

Recognize that precise values of Reynolds number versus flow regime do not exist. Thus, the guidelines given in Eq. (10.1) are approximate and other references may give slightly different values. For example, some references use Re = 2300 as the criteria for turbulence.

There are several equations for calculating Reynolds number in a pipe

$$\mathrm{Re} = \frac{VD}{\nu} = \frac{\rho VD}{\mu} = \frac{4Q}{\pi D\nu} = \frac{4\dot{m}}{\pi D\mu}$$

Developing Flow and Fully Developed Flow

Flow in a conduit is classified as being developing flow or fully developed flow. For example, consider laminar fluid entering a pipe from a reservoir as shown in Fig. 10.2. As the fluid moves down the pipe, the velocity distribution changes in the streamwise direction as viscous effects cause the plug-type profile to gradually change into a parabolic profile. This region of changing velocity profile is called *developing flow*. After the parabolic distribution is achieved, the flow profile remains unchanged in the streamwise direction, and flow is called *fully developed flow*.

The distance required for flow to develop is called the *entrance length* (*Le*). Correlations for entry length are:

$$\frac{L_e}{D} = 0.05\ \mathrm{Re} \qquad \text{(laminar flow: Re} \le 2000)$$

$$\frac{L_e}{D} = 50 \qquad \text{(turbulent flow: Re} \ge 3000)$$

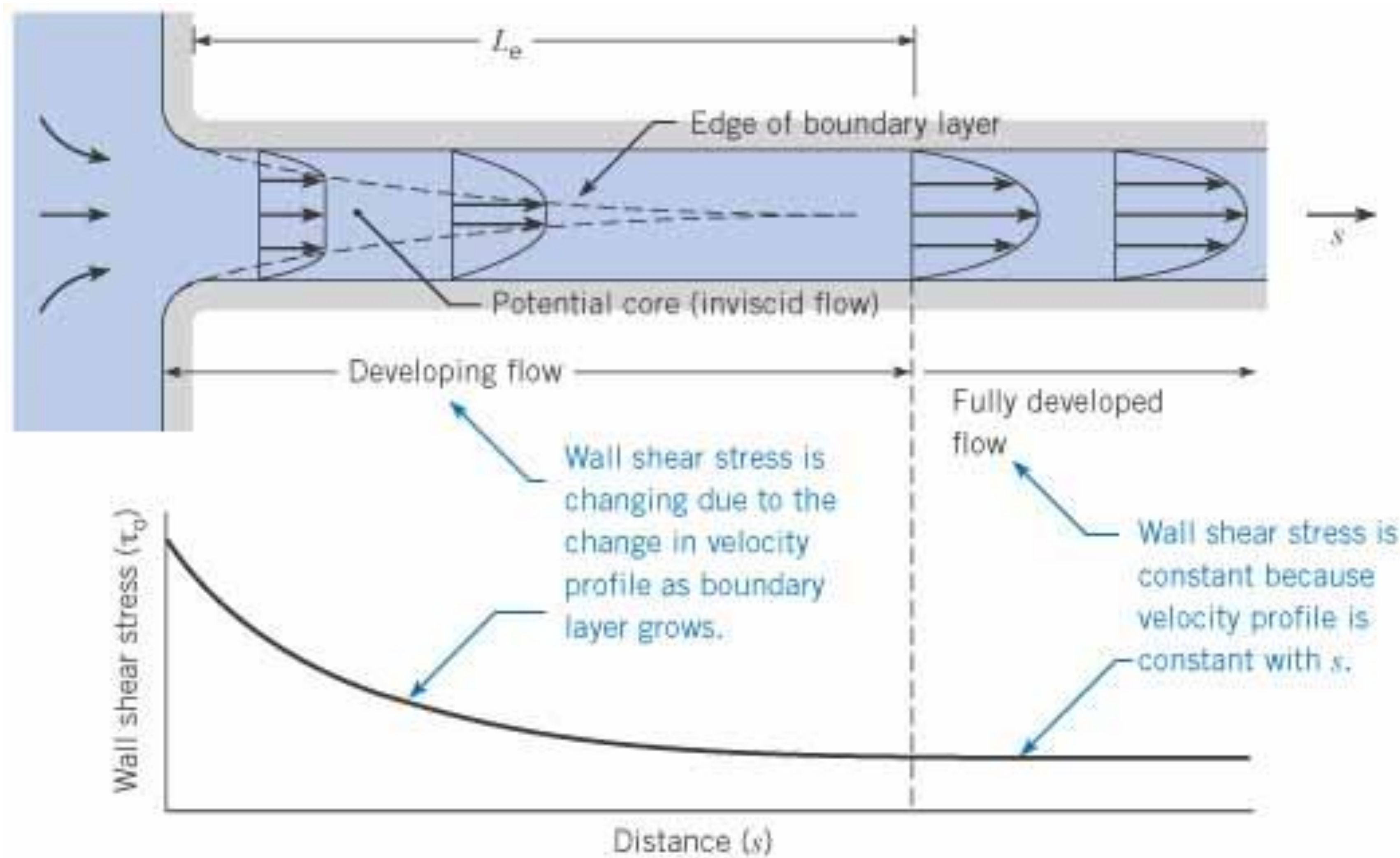


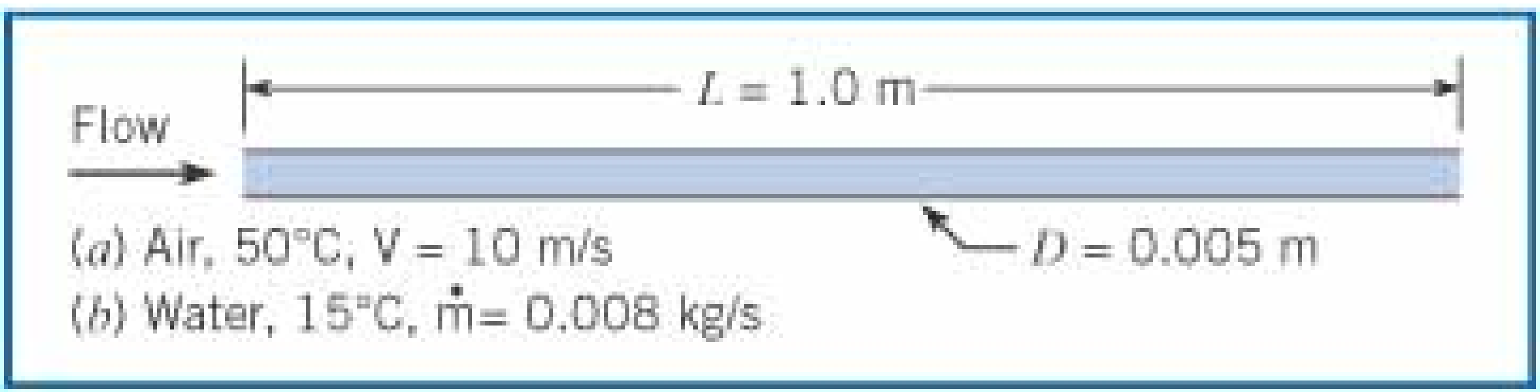
Figure 10.2 *In developing flow, the wall shear stress is changing. In fully developed flow, the wall shear stress is constant.*

EXAMPLE 10.1 CLASSIFYING FLOW IN CONDUITS

Consider fluid flowing in a round tube of length 1 m and diameter 5 mm. Classify the flow as laminar or turbulent and calculate the entrance length for (a) air (50°C) with a speed of 12 m/s and (b) water (15°C) with a mass flow rate of 8 gm/s,

Properties:

1. Air (50°C), Table A.3, $\nu = 1.79 \times 10^{-5}$ m²/s.
2. Water (15°C), Table A.5, $\mu = 1.14 \times 10^{-3}$ N · s/m².



Solution

(a) Air

$$\mathrm{Re} = \frac{VD}{\nu} = \frac{(12\ \mathrm{m/s})(0.005\,\mathrm{m})}{1.79 \times 10^{-5}\ \mathrm{m^2/s}} = 3350$$

Since Re > 3000, the flow is turbulent

$$L_e = 50D = 50(0.005\ \mathrm{m}) = \boxed{0.25\ \mathrm{m}}$$

(b) Water

$$\mathrm{Re} = \frac{4\dot{m}}{\pi D \mu} = \frac{4(0.008\ \mathrm{kg/s})}{\pi(0.005\,\mathrm{m})(1.14 \times 10^{-3}\ \ \mathrm{N \cdot s/m^2})}$$

$$= 1787$$

Since Re < 2000, the flow is laminar

$$L_e = 0.05\,\mathrm{Re}D = 0.05(1787)(0.005\ \mathrm{m}) = \boxed{0.447\ \mathrm{m}}$$

10.3 Pipe Head Loss

The Darcy-Weisbach equation is used for calculating head loss in a straight pipe. This equation is one of the most useful equations in fluid mechanics.

Combined (Total) Head Loss

Pipe head loss is one type of head loss; the other type is called component head loss. All head losses are classified using these two categories:

$$(\text{Total head loss}) = (\text{Pipe head loss}) + (\text{Component head loss})$$

Component head loss is associated with flow through devices such as valves, bends, and tees. *Pipe head loss* is associated with fully developed flow in conduits, and it is caused by shear stresses that act on the flowing fluid. Note that pipe head loss is sometimes called major head loss, and component head loss is sometimes called minor head loss.

Derivation of the Darcy-Weisbach Equation

To derive the Darcy-Weisbach equation, consider Fig. 10.4. Assume fully developed and steady flow in a round tube of constant diameter *D*. Situate a cylindrical control volume of diameter *D* and length ΔL inside the pipe. Define a coordinate system with an axial coordinate in the streamwise direction (*s* direction) and a radial coordinate in the *r* direction.

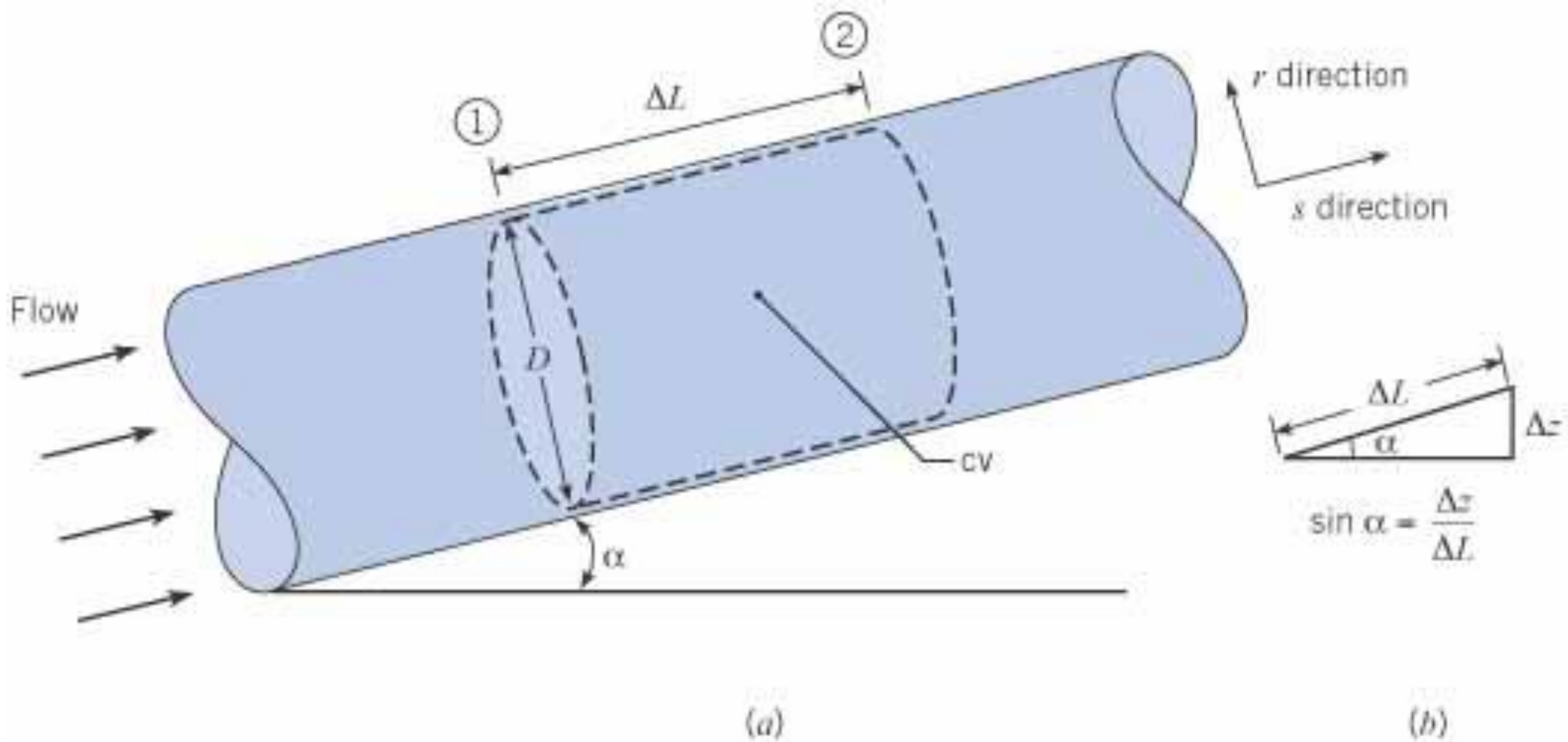


Figure 10.4 *Initial situation for the derivation of the Darcy-Weisbach equation.*

Apply the momentum equation to the control volume shown in Fig. 10.4.

$$\sum \mathbf{F} = \frac{d}{dt}\int_{cv} \mathbf{v}\rho d\mathcal{V} + \int_{cs} \mathbf{v}\rho \mathbf{V}\cdot d\mathbf{A} \qquad (10.5)$$

(Net forces) = (Momentum accumulation rate) + (Net efflux of momentum)

The net efflux of momentum is zero because the velocity distribution at section 2 is identical to the velocity distribution at section 1. The momentum accumulation term is also zero because the flow is steady. Thus, Eq. (10.5) simplifies to $\Sigma \boldsymbol{F} = 0$. Forces are shown in Fig. 10.5. Summing forces in the streamwise direction gives

$$F_{\text{pressure}} + F_{\text{shear}} + F_{\text{weight}} = 0$$

$$(p_1 - p_2)\left(\frac{\pi D^2}{4}\right) - \tau_0(\pi D \Delta L) - \gamma\left[\left(\frac{\pi D^2}{4}\right)\Delta L\right]\sin\alpha = 0$$

Since, $\sin\alpha = (\Delta z/\Delta L)$, the equation becomes,

$$(p_1 + \gamma z_1) - (p_2 + \gamma z_2) = \frac{4\Delta L \tau_0}{D}$$

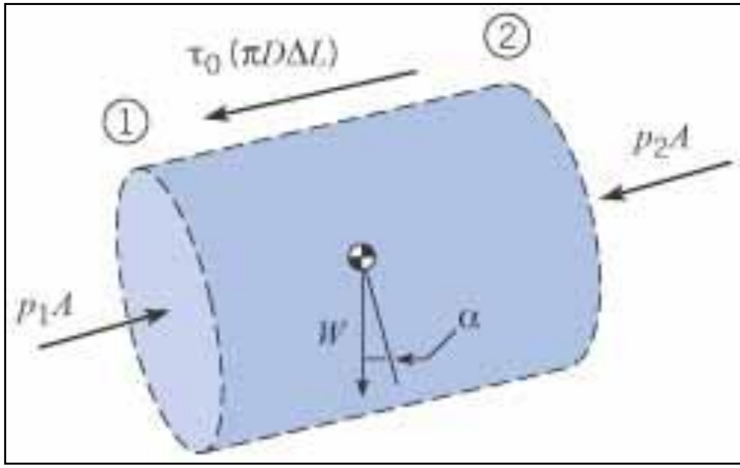


Figure 10.5 *Force diagram.*

Next, apply the energy equation to the control volume shown in Fig. 10.4. Recognize that $h_p = h_t = 0$, $V_1 = V_2$, and α1 = α2. Thus, the energy equation reduces to

$$\frac{p_1}{\gamma} + z_1 = \frac{p_2}{\gamma} + z_2 + h_L$$

$$(p_1 + \gamma z_1) - (p_2 + \gamma z_2) = \gamma h_L$$

Combine the equation from the momentum and the above (form the energy) and replace ΔL by L. Also, introduce a new symbol h_f to represent head loss in pipe.

$$h_f = \begin{pmatrix} \text{head loss} \\ \text{in a pipe} \end{pmatrix} = \frac{4L\tau_0}{D\gamma}$$

Rearrange the right side of Eq. (10.9).

$$h_f = \left(\frac{L}{D}\right)\left\{\frac{4\tau_0}{\rho V^2/2}\right\}\left\{\frac{\rho V^2/2}{\gamma}\right\} = \left\{\frac{4\tau_0}{\rho V^2/2}\right\}\left(\frac{L}{D}\right)\left\{\frac{V^2}{2g}\right\}$$

Define a new π-group called the *friction factor* f that gives the ratio of wall shear stress (τ_o) to kinetic pressure ($\rho V^2/2$):

$$f \equiv \frac{(4 \cdot \tau_0)}{(\rho V^2 / 2)} \approx \frac{\text{shear stress acting at the wall}}{\text{kinetic pressure}}$$

In the technical literature, the friction factor is identified by several different labels that are synonymous: friction factor, Darcy friction factor, Darcy-Weisbach friction factor, and the resistance coefficient. There is also another coefficient called the Fanning friction factor, often used by chemical engineers, which is related to the Darcy-Weisbach friction factor by a factor of 4.

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

This text uses only the Darcy-Weisbach friction factor. Combining the previous equations, gives the Darcy-Weisbach equation:

$$h_f = f \frac{L}{D} \frac{V^2}{2g}$$

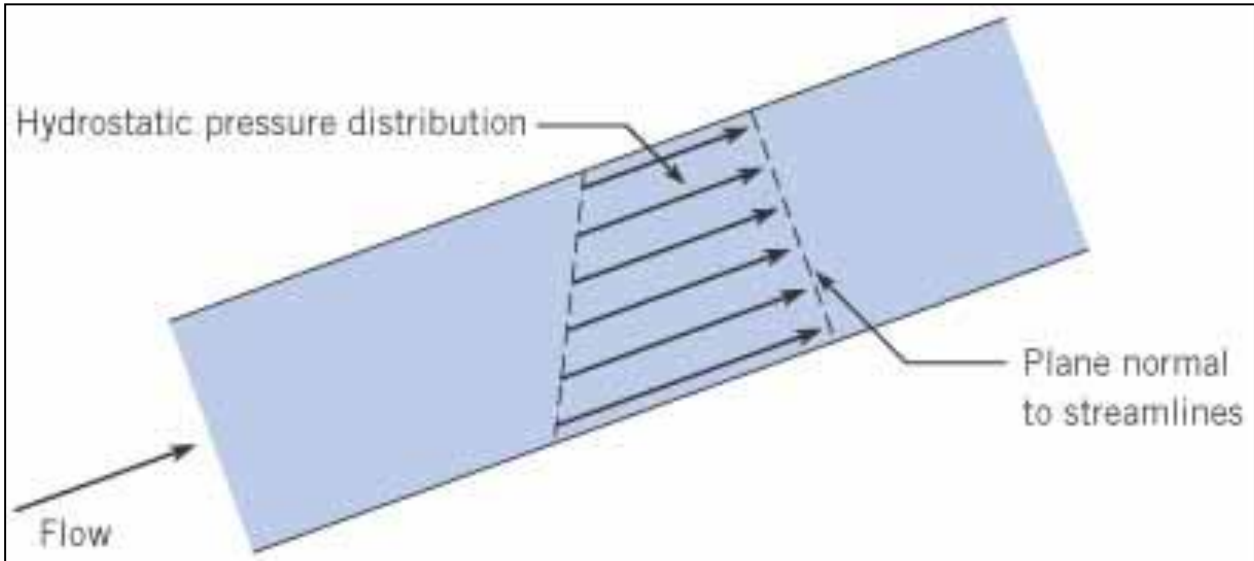
- To use the Darcy-Weisbach equation, the flow should be fully developed and steady. The Darcy-Weisbach equation is used for either laminar flow or turbulent flow and for either round pipes or nonround conduits such as a rectangular duct.
- The Darcy-Weisbach equation shows that head loss depends on the friction factor, the pipe-length-to-diameter ratio, and the mean velocity squared.
- The key to using the Darcy-Weisbach equation is calculating a value of the friction factor f.

10.4 Stress Distributions in Pipe Flow

This section derives equations for the stress distributions on a plane that is oriented normal to stream lines. These equations, which apply to both laminar and turbulent flow, provide insights about the nature of the flow.

In pipe flow the pressure acting on a plane that is normal to the direction of flow is hydrostatic. This means that the pressure distribution varies linearly as shown in Fig. 10.6. What is the reason that the pressure distribution is hydrostatic?

Figure 10.6 *For fully developed flow in a pipe, the pressure distribution on an area normal to streamlines is hydrostatic.*



To derive an equation for the shear-stress variation, consider flow of a Newtonian fluid in a round tube that is inclined at an angle α with respect to the horizontal as shown in Fig. 10.7. Assume that the flow is fully developed, steady, and laminar. Define a cylindrical control volume of length ΔL and radius r.

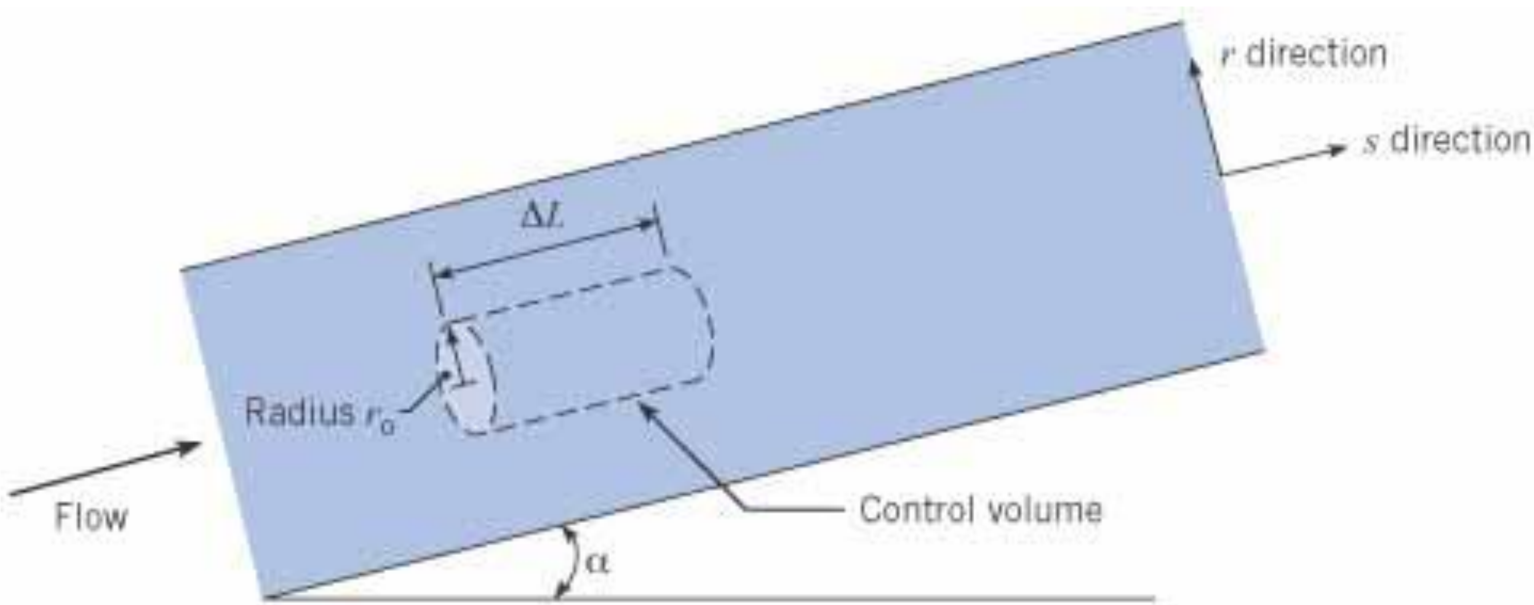


Figure 10.7 *Sketch for derivation of an equation for shear stress.*

Apply the momentum equation in the *s* direction. The net momentum efflux is zero because the flow is fully developed; that is, the velocity distribution at the inlet is the same as the velocity distribution at the exit. The momentum accumulation is also zero because the flow is steady. The momentum equation (6.5) simplifies to force equilibrium.

$$\sum F_s = F_{\text{pressure}} + F_{\text{weight}} + F_{\text{shear}} = 0$$

Analyze each term using the force diagram shown in Fig. 10.8:

$$pA - \left(p + \frac{dp}{ds}\Delta L\right)A - W \sin \alpha - \tau(2\pi r)\Delta L = 0$$

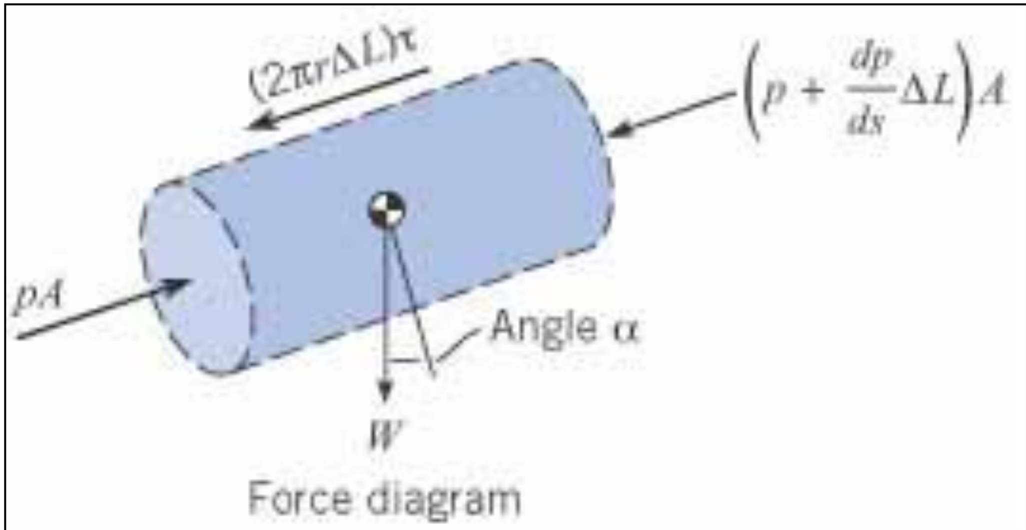


Figure 10.8 *Force diagram corresponding to the control volume defined in Fig. 10.6.*

Since $W = \gamma A \Delta L$, also $\sin \alpha = \Delta z / \Delta L$. Next, divide the previous equation by $A\Delta L$:

$$\tau = \frac{r}{2}\left[-\frac{d}{ds}(p + \gamma z)\right] \tag{10.15}$$

Equation (10.15) shows that the shear-stress distribution varies linearly with r as shown in Fig. 10.9. Notice that the shear stress is zero at the centerline, it reaches a maximum value of τ_0 at the wall, and the variation is linear in between. This linear shear stress variation applies to both laminar and turbulent flow.

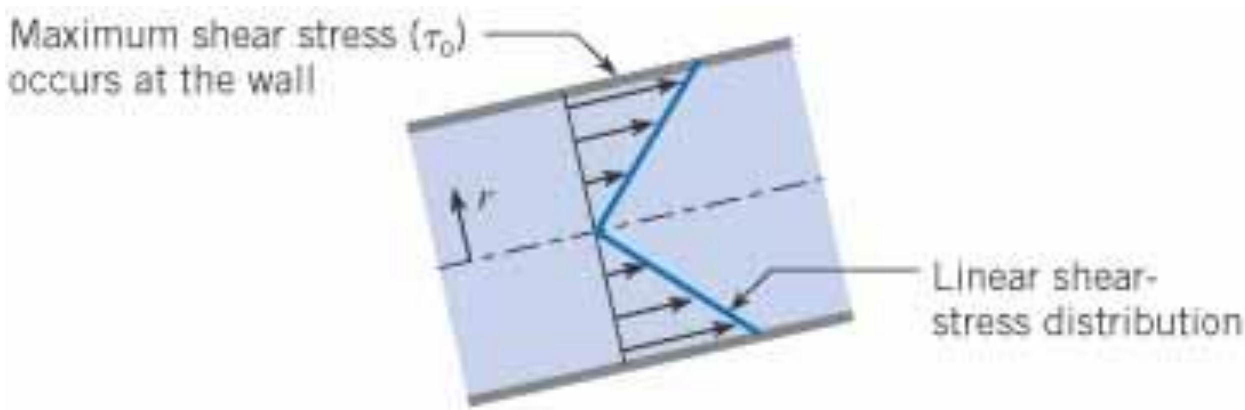


Figure 10.9 *In fully developed flow (laminar or turbulent), the shear-stress distribution on an area that is normal to streamlines is linear.*

10.5 Laminar Flow in a Round Tube

Laminar flow is important for flow in small conduits called micro-channels, for lubrication flow, and for analyzing other flows in which viscous forces are dominant. Also, knowledge of laminar flow provides a foundation for the study of advanced topics. *Laminar flow* is a flow regime in which fluid motion is smooth, the flow occurs in layers (laminae), and the mixing between layers occurs by molecular diffusion, a process that is much slower than turbulent mixing.

Laminar flow occurs when Re ≤ 2000. Laminar flow in a round tube is called *Poiseuille flow* or *Hagen-Poiseuille flow* in honor of pioneering researchers who studied low-speed flows in the 1840s.

Velocity Profile

To derive an equation for the velocity profile in laminar flow, start by relating stress to rate-of-strain

$$\tau = \mu \frac{dV}{dy}$$

In pipe flow, velocity is expressed as a function of r while y is the distance from the wall; r and y are related by $y = r_0 - r$, thus the derivative of the velocity becomes

$$\tau = \mu\left(\frac{dV}{dy}\right) = \mu\left(\frac{dV}{dr}\right)\left(\frac{dr}{dy}\right) = -\left(\mu\frac{dV}{dr}\right)$$

Substitute for shear stress from the stress distribution,

$$-\left(\frac{2\mu}{r}\right)\left(\frac{dV}{dr}\right) = \frac{d}{ds}(p + \gamma z)$$

Observing that the left hand side (velocity) is a function of r while the right hand side is differentiated with s; this can only be true if the two sides equal to a constant.

Another way is to remember the flow is fully developed, which means that is not accelerating, hence the difference in the piezometric pressure must be constant.

Therefore, when integrating the right hand side is constant, hence

$$V = -\left(\frac{r^2}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right) + C$$

where $\gamma\Delta h = \Delta(p + \gamma z)$. To evaluate C, apply the no-slip condition,

$$V(r = r_0) = 0$$

$$0 = -\frac{r_0^2}{4\mu}\left(\frac{\gamma \Delta h}{\Delta L}\right) + C$$

Solving for C, substituting and re-arranging lead to,

$$V = \frac{r_0^2 - r^2}{4\mu}\left[-\frac{d}{ds}(p + \gamma z)\right] = -\left(\frac{r_0^2 - r^2}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)$$

The maximum velocity occurring at $r = 0$, hence

$$V_{\max} = -\left(\frac{r_0^2}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)$$

This expression can be substituted into the velocity expression to give,

$$V(r) = -\left(\frac{r_0^2 - r^2}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right) = V_{\max}\left(1 - \left(\frac{r}{r_0}\right)^2\right)$$

This indicates that the velocity distribution is parabolic which is shown in Figure 10.10

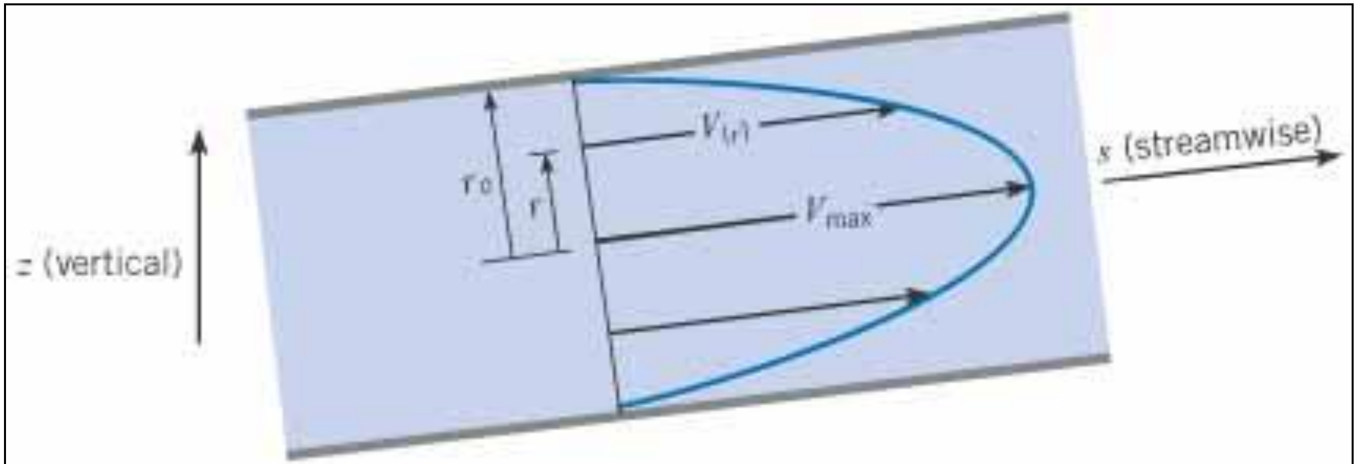


Figure 10.10 *The velocity profile in Poiseuille flow is parabolic.*

Discharge and Mean Velocity *V*

The discharge is easily obtained form,

$$Q = \int V \, dA$$

$$= -\int_0^{r_0} \frac{(r_0^2 - r^2)}{4\mu} \left(\frac{\gamma \Delta h}{\Delta L}\right)(2\pi r \, dr)$$

which upon integration leads to

$$Q = -\left(\frac{\pi}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)\frac{(r^2 - r_0^2)^2}{2}\Bigg|_0^{r_0} = -\left(\frac{\pi r_0^4}{8\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)$$

The mean velocity is obtained from,

$Q = \bar{V} A$ which leads to $\bar{V} = -\left(\frac{r_0^2}{8\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)$

Notice that the mean and maximum velocities are related,

$$\bar{V} = -\left(\frac{D^2}{32\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right) = \frac{V_{max}}{2}$$

Head Loss and Friction Factor *f*

The equation for head loss in a round tube, assume fully developed flow in the pipe shown in Fig.10.11, is obtained by going back to the derived equation,

$$h_f = \begin{pmatrix}\text{head loss} \\ \text{in a pipe}\end{pmatrix} = \frac{4L\tau_0}{D\gamma}$$

Given the velocity profile one can find the shear at the wall and then substituting into the above equation with re-arranging gives,

$$h_f = \frac{32\mu L\bar{V}}{\gamma D^2}$$

Equating the last expression with the Darcy-Weisbach,

$$h_f = \frac{32\mu LV}{\gamma D^2} = f\frac{L}{D}\frac{V^2}{2g}$$

After some manipulation give,

$$\text{or } f = \left(\frac{32\mu LV}{\gamma D^2}\right)\left(\frac{D}{L}\right)\left(\frac{2g}{V^2}\right) = \frac{64\mu}{\rho DV} = \frac{64}{\text{Re}}$$

EXAMPLE 10.2 HEAD LOSS FOR LAMI)AR FLOW

Oil (S = 0.85) with a kinematic viscosity of 6 × 10-4 m^2/s flows in a 15 cm pipe at a rate of 0.020 m^3/s. What is the head loss per 100 m length of pipe?

Solution

Determine the flow nature by computing Reynolds number. For the Re, the mean velocity is required,

$$V = \frac{Q}{A} = \frac{0.020\ \text{m}^3/\text{s}}{(\pi D^2)/4} = \frac{0.020\ \text{m}^3/\text{s}}{\pi((0.15\ \text{m})^2/4)} = 1.13\ \text{m/s}$$

Thus,

$$\text{Re} = \frac{VD}{\nu} = \frac{(1.13\ \text{m/s})(0.15\ \text{m})}{6 \times 10^{-4}\ \text{m}^2/\text{s}} = 283$$

Since Re < 2000, flow is laminar, hence the head loss,

$$f = \frac{64}{\text{Re}} = \frac{64}{283} = 0.226$$

$$h_f = f\left(\frac{L}{D}\right)\left(\frac{V^2}{2g}\right) = 0.226\left(\frac{100\ \text{m}}{0.15\ \text{m}}\right)\left(\frac{(1.13\ \text{m/s})}{2 \times 9.81\ \text{m/s}^2}\right)^2$$

$$= 9.83\ \text{m}$$

10.6 Turbulent Flow and the Moody Diagram

This section presents equations for calculating the friction factor *f*, and presents a famous graph called the Moody diagram.

Qualitative Description of Turbulent Flow

Turbulent flow is a flow regime in which the movement of fluid particles is chaotic, eddying, and unsteady, with significant movement of particles in directions transverse to the flow direction. Because of the chaotic motion of fluid particles, turbulent flow produces high levels of mixing and has a velocity profile that is more uniform or flatter than the corresponding laminar velocity profile. Turbulent flow normally occurs when Re $\geq$ 3000. Engineers and scientists model turbulent flow by using an empirical approach. This is because the complex nature of turbulent flow has prevented researchers from establishing a mathematical solution of general utility. Over the years, researchers have proposed many equations for shear stress and head loss in turbulent pipe flow. The empirical equations that have proven to be the most reliable and accurate for engineering use are presented in the next section.

Equations for the Velocity Distribution

The time-average velocity distribution is often described using an equation called the power-law formula.

$$\frac{u(r)}{u_{max}} = \left(\frac{r_0 - r}{r_0}\right)^m \qquad (10.35)$$

where u_{max} is velocity in the center of the pipe, r_0 is the pipe radius, and m is an empirically determined variable that depends on Re as shown in Table 10.2.

Table 10.2 EXPONENTS FOR POWER-LAW EQUATION AND RATIO OF MEAN TO MAXIMUM VELOCITY

Re	4×10^3	2.3×10^4	1.1×10^5	1.1×10^6	3.2×10^6
m	$\frac{1}{6.0}$	$\frac{1}{6.6}$	$\frac{1}{7.0}$	$\frac{1}{8.8}$	$\frac{1}{10.0}$
u_{max}/V	1.26	1.24	1.22	1.18	1.16

Notice in Table 10.2 that the velocity in the center of the pipe is typically about 20% higher than the mean velocity *V*. While Eq. (10.35) provides an accurate representation of the velocity profile, it does not predict an accurate value of wall shear stress.

An alternative approach to Eq. (10.35) is to use the turbulent boundary-layer equations presented in Chapter 9. The most significant of these equations, called the logarithmic velocity distribution,

$$\frac{u(r)}{u*} = 2.44 \ln \frac{u*(r_0 - r)}{\nu} + 5.56 \qquad (10.36)$$

where *u** the shear velocity, is given by $u* = \sqrt{\tau_0 / \rho}$

Equations for the Friction Factor, *f*

To derive an equation for *f in turbulent flow*, substitute the log law in Eq. (10.36) into the definition of mean velocity given

$$V = \frac{Q}{A} = \left(\frac{1}{\pi r_0^2}\right)\int_0^{r_0} u(r)2\pi r dr = \left(\frac{1}{\pi r_0^2}\right)\int_0^{r_0} u_*\left[2.44 \ln \frac{u_*(r_0 - r)}{\nu} + 5.56\right]2\pi r dr$$

After some manipulations,

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}(\mathrm{Re}\sqrt{f}) - 0.8 \qquad (10.37)$$

Equation (10.37) gives the resistance coefficient for turbulent flow in tubes that have **smooth walls**. To determine the influence of roughness on the walls, Nikuradse, one of Prandtl's graduate students, glued uniform-sized grains of sand to the inner walls of a tube and then measured pressure drops and flow rates.

Nikuradse's data, Fig. 10.12, shows the friction factor f plotted as function of Reynolds number for various sizes of sand grains. To characterize the size of sand grains, Nikuradse used a variable called the ***sand roughness height*** with the symbol k_s. The π-group, k_s/D is given the name ***relative roughness***.

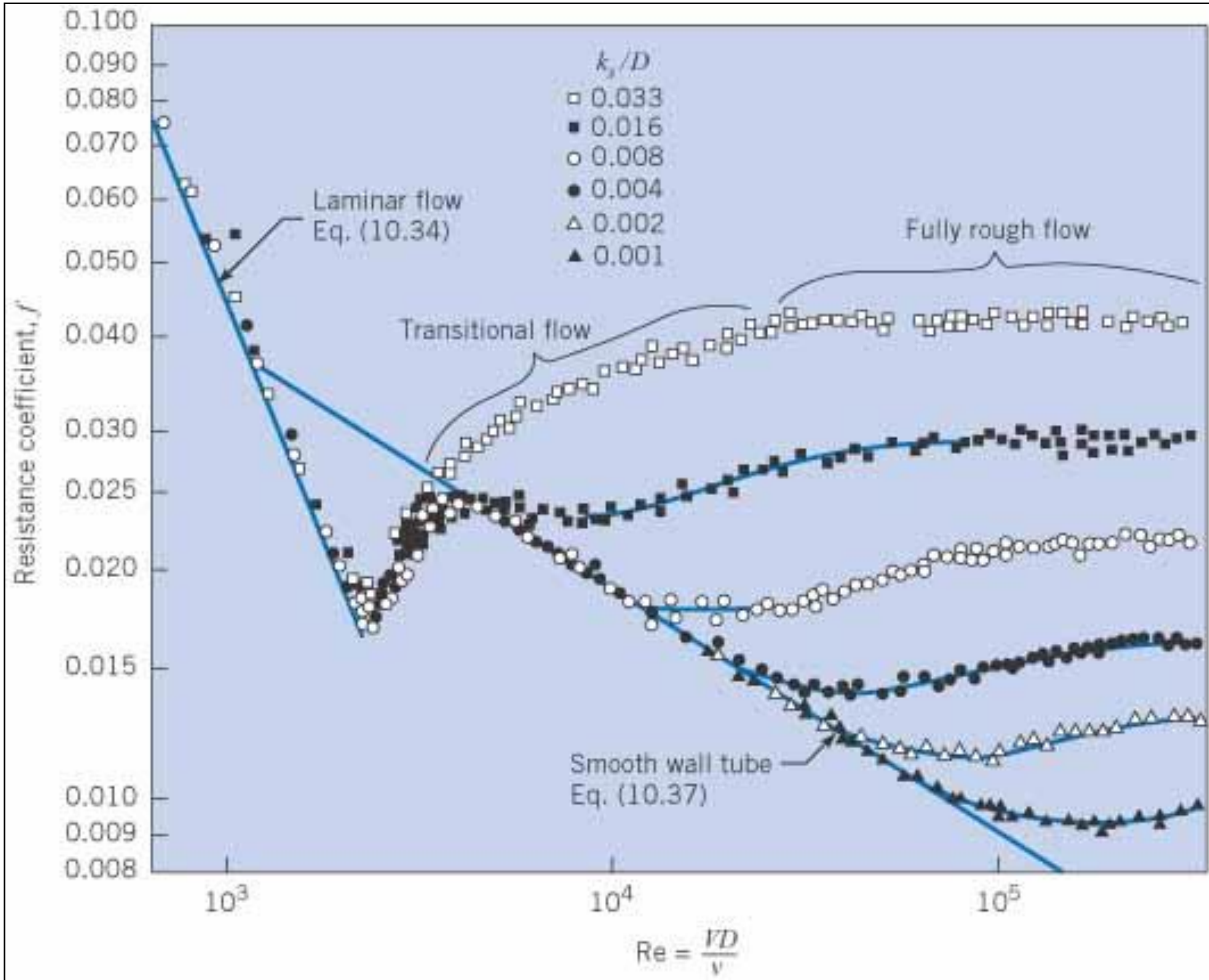


Figure 10.12
Resistance coefficient f versus Re for sand-roughened pipe. [After ikuradse 4].

In laminar flow, the data in Fig. 10.12 show that wall roughness does not influence f. In particular, notice how the data corresponding to various values of k_s/D collapse into a single line that is labeled “laminar flow.”

In turbulent flow, the data in Fig. 10.12 show that wall roughness has a major impact on f. When $k_s/D = 0.033$, then values of f are about 0.04. As the relative roughness drops to 0.002, values of f decrease by a factor of about 3. Eventually wall roughness does not matter, and the value of f can be predicted by assuming that the tube has a smooth wall. This latter case corresponds to the curve that is labeled “smooth wall tube.” The effects of roughness are summarized by White 5 and presented in Table 10.3. These regions are also labeled in Fig. 10.12.

Table 10.3 EFFECTS OF WALL ROUGHNESS

Type of Flow	Parameter Ranges		Influence of Parameters on f
Laminar Flow	Re < 2000	NA	f depends on Reynolds number f is independent of wall roughness (k_s/D)
Turbulent Flow, Smooth Tube	Re > 3000	$\left(\frac{k_s}{D}\right)\text{Re} < 10$	f depends on Reynolds number f is independent of wall roughness (k_s/D)
Transitional Turbulent Flow	Re > 3000	$10 < \left(\frac{k_s}{D}\right)\text{Re} < 1000$	f depends on Reynolds number f depends on wall roughness (k_s/D)
Fully Rough Turbulent Flow	Re > 3000	$\left(\frac{k_s}{D}\right)\text{Re} > 1000$	f is independent of Reynolds number f depends on wall roughness (k_s/D)

Moody Diagram

Colebrook advanced Nikarudse's work by acquiring data for commercial pipes and then developing an empirical equation, called the Colebrook-White formula, for the friction factor. Moody used the Colebrook-White formula to generate a design chart similar to that shown in Fig. 10.13. This chart is now known as the *Moody diagram* for commercial pipes.

In the Moody diagram, Fig. 10.13, the variable k_s denotes the *equivalent sand roughness.* That is, a pipe that has the same resistance characteristics at high Re values as a sand-roughened pipe is said to have a roughness equivalent to that of the sand-roughened pipe.

Table 10.4 gives the equivalent sand roughness for various kinds of pipes. This table can be used to calculate the relative roughness for a given pipe diameter, which, in turn, is used in the Moody diagram to find the friction factor.

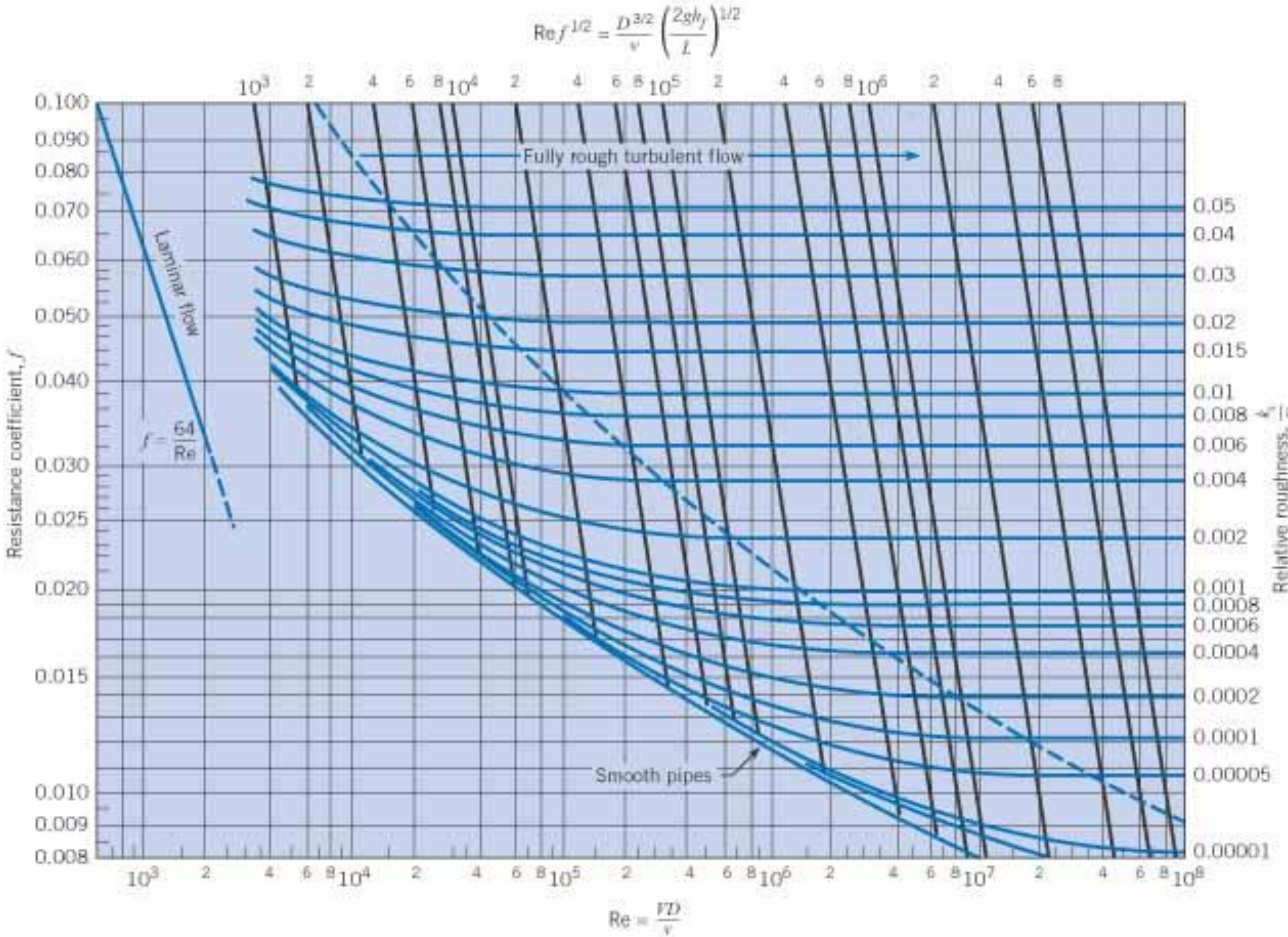


Figure 10.13 *Resistance coefficient f versus Re. Reprinted with minor variations. [After Moody 3.*

Table 10.4 EQUIVALENT NA'D-GRAIN ROUGHNESS, (k_s), FOR VARIOUS PIPE MATERIAL

Boundary Material	k_s, Millimeters	k_s, Inches
Glass, plastic	Smooth	Smooth
Copper or brass tubing	0.0015	6×10^{-5}
Wrought iron, steel	0.046	0.002
Asphalted cast iron	0.12	0.005
Galvanized iron	0.15	0.006
Cast iron	0.26	0.010
Concrete	0.3 to 3.0	0.012–0.12
Riveted steel	0.9–9	0.035–0.35
Rubber pipe (straight)	0.025	0.001

EXAMPLE 10.3 HEAD LOSS IN A PIPE (Direct application)

Water (T = 20°C) flows at a rate of 0.05 m³/s in a 20 cm asphalted cast-iron pipe. What is the head loss per kilometer of pipe?

Solution

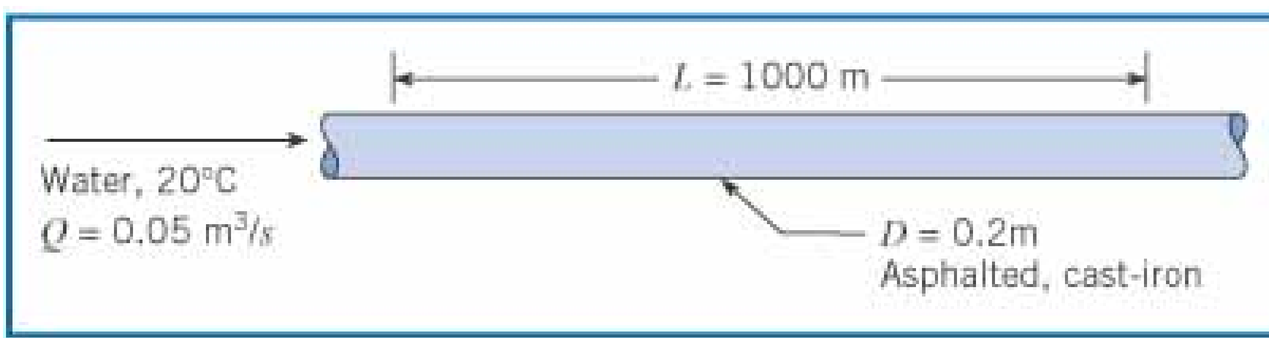
Mean velocity, $$V = \frac{Q}{A} = \frac{0.05\ \mathrm{m}^3/\mathrm{s}}{(\pi/4)(0.02\ \mathrm{m})^2} = 1.59\ \mathrm{m/s}$$

Reynolds number $$\mathrm{Re} = \frac{VD}{\nu} = \frac{(1.59\ \mathrm{m/s})(0.20\ \mathrm{m})}{10^{-6}\ \mathrm{m}^2/\mathrm{s}} = 3.18 \times 10^5$$

Flow is turbulent. Equivalent sand roughness (Table 10.4): k_s = 0.12 mm. Hence, the relative roughness:

$$k_s/D = (0.00012\ \mathrm{m})/(0.2\ \mathrm{m}) = 0.0006$$

Look up f on the Moody diagram for Re = 3.18 × 105 and k_s/D = 0.0006:



Darcy-Weisbach equation

$$h_f = f\left(\frac{L}{D}\right)\left(\frac{V^2}{2g}\right) = 0.019\left(\frac{1000\ \text{m}}{0.20\ \text{m}}\right)\left(\frac{1.59^2\ \text{m}^2/s^2}{2(9.81\ \text{m}/s^2)}\right)$$

$$= \boxed{12.2\ \text{m}}$$

10.8 Combined Head Loss

This section describes how to calculate head loss in components.

The Minor Loss Coefficient, *K*

When fluid flows through a component such as a partially open value or a bend in a pipe, viscous effects cause the flowing fluid to lose mechanical energy. For example, Fig. 10.14 shows flow through a "generic component." At section 2, the fluid head of the flow will be less than at section 1. To characterize component head loss, engineers use a π-group called the *minor loss coefficient K*

$$K = \frac{(\Delta h)}{(V^2/2g)} = \frac{(\Delta p)}{(\rho V^2/2)}$$

where Δh is drop in piezometric head that is caused by a component, Δp is the pressure drop that is caused by the component, and V is mean velocity.

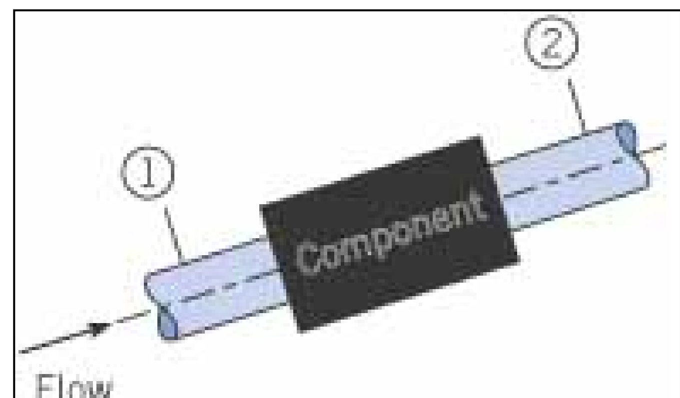


Figure 10.14 *Flow through a generic component.*

Data for the Minor Loss Coefficient

Pipe inlet. Near the entrance to a pipe when the entrance is rounded, flow is developing as shown in Fig. 10.1 and the wall shear stress is higher than that found in fully developed flow. Alternatively, if the pipe inlet is abrupt, or sharp-edged, as in Fig. 10.15, separation occurs just downstream of the entrance. Hence the streamlines converge and then diverge with consequent turbulence and relatively high head loss. The loss coefficient for the abrupt inlet is approximately 0.5. Other values of head loss are summarized in Table 10.5.

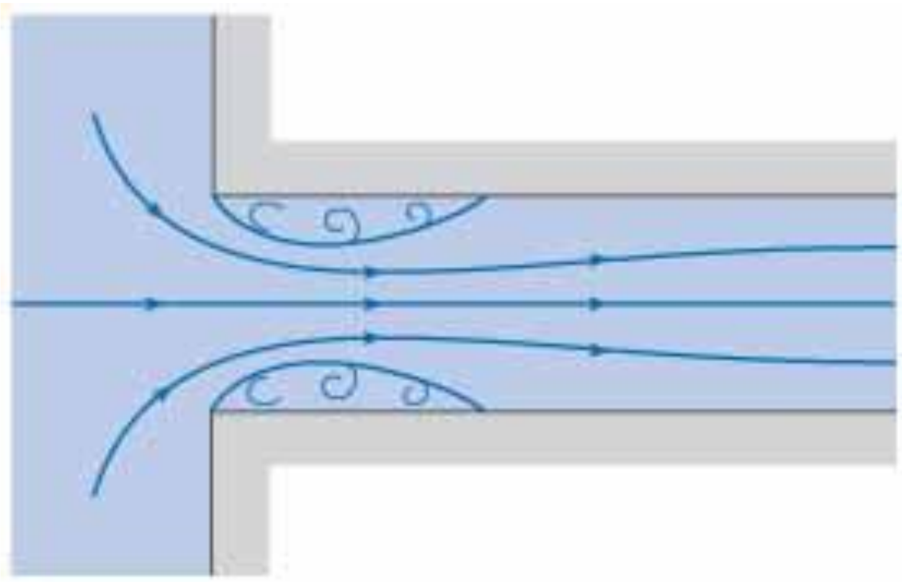
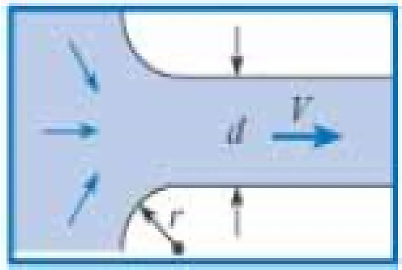
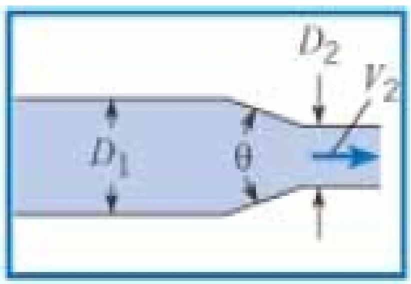
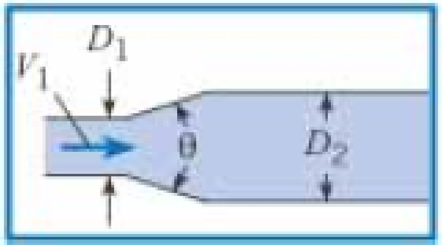


Figure 10.15 *Flow at a sharp-edged inlet.*

Table 10.5 LOSS COEFFICIENTS FOR VARIOUS TRANSITIONS AND FITTINGS

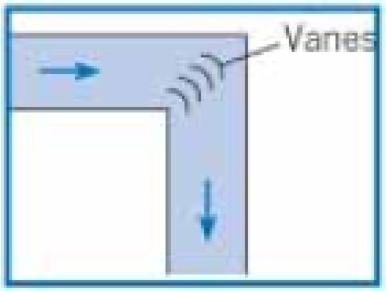
Description	Sketch	Additional Data		K
Pipe entrance		r/d		K_e
$h_L = K_e V^2/2g$		0.0		0.50
		0.1		0.12
		>0.2		0.03
Contraction			K_C	K_C
$h_L = K_C V_2^2 / 2g$		D_2/D_1	$\theta = 60°$	$\theta = 180°$
		0.00	0.08	0.50
		0.20	0.08	0.49
		0.40	0.07	0.42
		0.60	0.06	0.27
		0.80	0.06	0.20
		0.90	0.06	0.10

Expansion

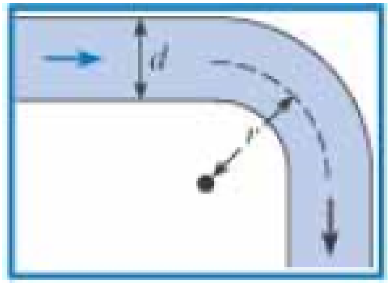


$h_L = K_E V_1^2 / 2g$

D_1/D_2	K_E θ = 20°	K_E θ = 180°
0.00		1.00
0.20	0.30	0.87
0.40	0.25	0.70
0.60	0.15	0.41
0.80	0.10	0.15

Description	**Sketch**	**Additional Data**	***K***
90° miter bend		Without vanes	$K_b = 1.1$
		With vanes	$K_b = 0.2$

■

		With vanes	$K_b = 0.2$
90° smooth bend		r/d	
		1	$K_b = 0.35$
		2	0.19
		4	0.16
		6	0.21
		8	0.28
		10	0.32
Threaded pipe fittings	Globe valve—wide open		$K_v = 10.0$
	Angle valve—wide open		$K_v = 5.0$
	Gate valve—wide open		$K_v = 0.2$
	Gate valve—half open		$K_v = 5.6$
	Return bend		$K_b = 2.2$
	Tee		
	Straight-through flow		$K_t = 0.4$
	Side-outlet flow		$K_t = 1.8$
	90° elbow		$K_b = 0.9$
	45° elbow		$K_b = 0.4$

Flow in an Elbow. In an elbow (90° smooth bend), considerable head loss is produced by secondary flows and by separation that occurs near the inside of the bend and downstream of the midsection as shown in Fig. 10.16.

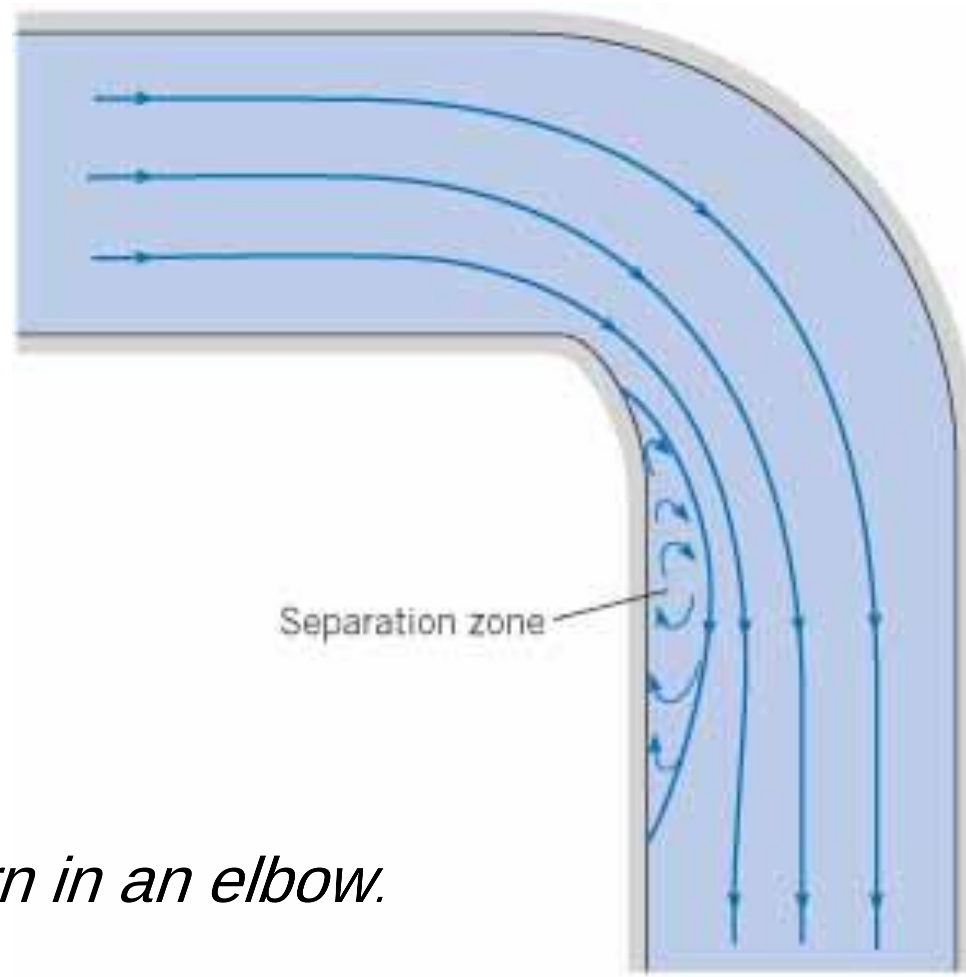


Figure 10.16 *Flow pattern in an elbow.*

Combined Head Loss Equation

The total head loss is,

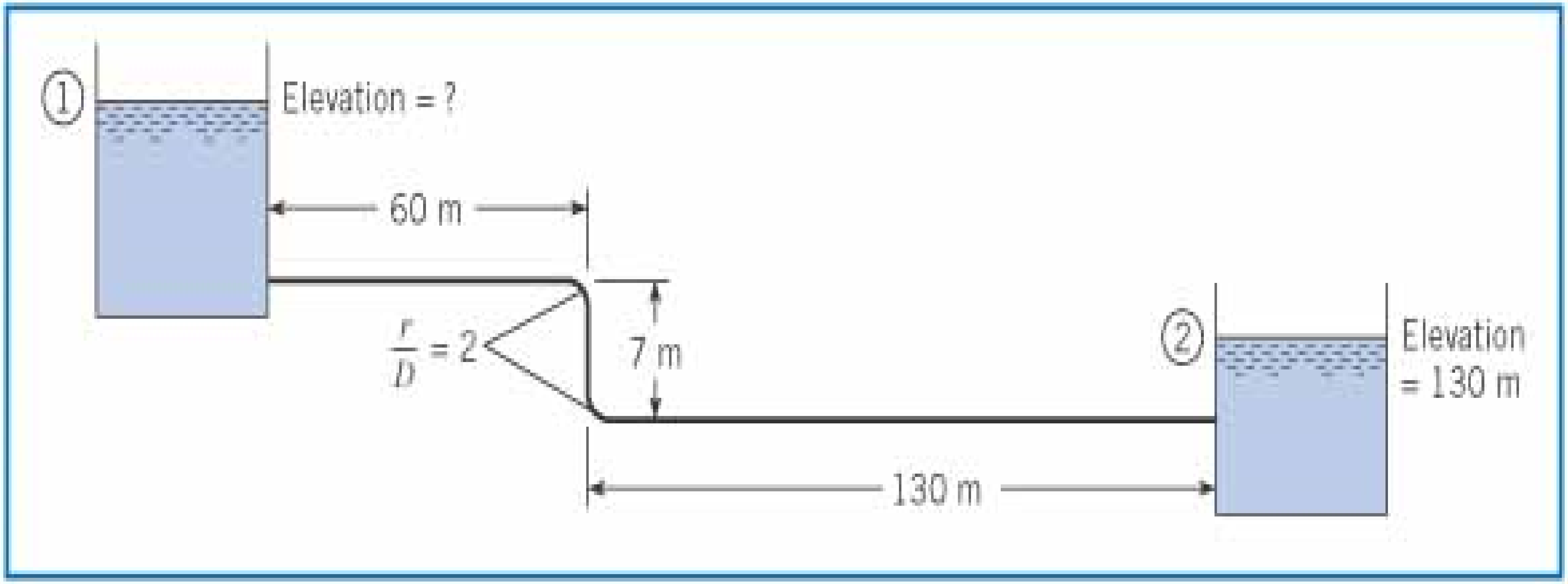
$$(\text{Total head loss}) = (\text{Pipe head loss}) + (\text{Component head loss})$$

An equation for the combined head loss,

$$h_L = \sum_{\text{pipes}} f \frac{L}{D} \frac{V^2}{2g} + \sum_{\text{components}} K \frac{V^2}{2g} = \frac{V^2}{2g}\left[\sum_{\text{pipes}} f \frac{L}{D} + \sum_{\text{components}} K\right]$$

EXAMPLE 10.7 PIPE SYSTEM WITH COMBI&ED HEAD LOSS

If oil ($v = 4 \times 10\text{-}5\ m^2/s$; $S = 0.9$) flows from the upper to the lower reservoir at a rate of 0.028 m^3/s in the 15 cm smooth pipe, what is the elevation of the oil surface in the upper reservoir?



Solution

Energy equation and term-by-term analysis

$$\frac{p_1}{\gamma} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 + h_t + h_L$$

$$0 + 0 + z_1 + 0 = 0 + 0 + z_2 + 0 + h_L$$

$$z_1 = z_2 + h_L$$

Interpretation: Change in elevation head is balanced by the total head loss. The combined head loss equation

$$h_L = \sum_{\text{pipes}} f \frac{L}{D} \frac{V^2}{2g} + \sum_{\text{components}} K \frac{V^2}{2g}$$

$$h_L = f \frac{L}{D} \frac{V^2}{2g} + \left(2K_b \frac{V^2}{2g} + K_e \frac{V^2}{2g} + K_g \frac{V^2}{2g} \right)$$

$$= \frac{V^2}{2g} \left(f \frac{L}{D} + 2K_b + K_e + K_g \right)$$

Substitute into the energy equation

$$z_1 = z_2 + \frac{V^2}{2g}\left(f\frac{L}{D} + 2K_b + K_e + K_E\right)$$

Resistance coefficient. Flow rate equation,

$$V = \frac{Q}{A} = \frac{(0.028\ \mathrm{m}^3/\mathrm{s})}{(\pi/4)(0.15\ \mathrm{m})^2} = 1.58\ \mathrm{m/s}$$

The Reynolds number

$$\mathrm{Re} = \frac{VD}{\nu} = \frac{1.58\ \mathrm{m/s}(0.15\ \mathrm{m})}{4\times10^{-5}\ \mathrm{m}^2/\mathrm{s}} = 5.93\times10^3$$

Thus, flow is turbulent. Use Moody diagram or Swamee-Jain equation

$$f = \frac{0.25}{\left[\log_{10}\left(\frac{k_s}{3.7D} + \frac{5.74}{\mathrm{Re}^{0.9}}\right)\right]^2} = \frac{0.25}{\left[\log_{10}\left(0 + \frac{5.74}{5930^{0.9}}\right)\right]^2} = 0.036$$

Calculate z_1:

$$z_1 = (130\text{ m}) + \frac{(1.58\text{ m/s})^2}{2(9.81)\text{m/s}^2}\left(0.036\frac{(197\text{ m})}{(0.15\text{ m})} + 2(0.19) + 0.5 + 1.0\right)$$

$$\boxed{z_1 = 136\text{ m}}$$