CHAPTER 5

MASS, BERNOULLI, AND ENERGY EQUATIONS

This chapter deals with three equations commonly used in fluid mechanics: the mass, Bernoulli, and energy equations. The *mass equation* is an expression of the conservation of mass principle. The *Bernoulli equation* is concerned with the conservation of kinetic, potential, and flow energies of a fluid stream and their conversion to each other in regions of flow where net viscous forces are negligible and where other restrictive conditions apply. The *energy equation* is a statement of the conservation of energy principle. In fluid mechanics, it is found convenient to separate *mechanical energy* from *thermal energy* and to consider the conversion of mechanical energy to thermal energy as a result of frictional effects as *mechanical energy loss*. Then the energy equation becomes the *mechanical energy balance*.

We start this chapter with an overview of conservation principles and the conservation of mass relation. This is followed by a discussion of various forms of mechanical energy and the efficiency of mechanical work devices such as pumps and turbines. Then we derive the Bernoulli equation by applying Newton's second law to a fluid element along a streamline and demonstrate its use in a variety of applications. We continue with the development of the energy equation in a form suitable for use in fluid mechanics and introduce the concept of *head loss*. Finally, we apply the energy equation to various engineering systems.

OBJECTIVES

When you finish reading this chapter, you should be able to

- Apply the mass equation to balance the incoming and outgoing flow rates in a flow system
- Recognize various forms of mechanical energy, and work with energy conversion efficiencies
- Understand the use and limitations of the Bernoulli equation, and apply it to solve a variety of fluid flow problems
- Work with the energy equation expressed in terms of heads, and use it to determine turbine power output and pumping power requirements

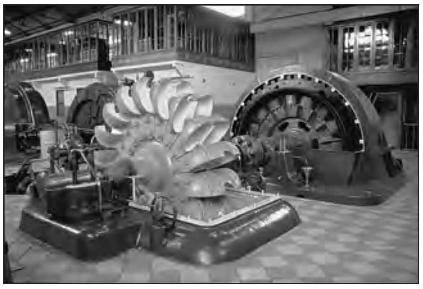


FIGURE 5–1
Many fluid flow devices such as this Pelton wheel hydraulic turbine are analyzed by applying the conservation of mass, momentum, and energy principles.
Courtesy of Hydro Tasmania, www.hydro.com.au. Used by permission.

5–1 ▪ INTRODUCTION

You are already familiar with numerous **conservation laws** such as the laws of conservation of mass, conservation of energy, and conservation of momentum. Historically, the conservation laws are first applied to a fixed quantity of matter called a *closed system* or just a *system*, and then extended to regions in space called *control volumes*. The conservation relations are also called *balance equations* since any conserved quantity must balance during a process. We now give a brief description of the conservation of mass, momentum, and energy relations (Fig. 5–1).

Conservation of Mass

The conservation of mass relation for a closed system undergoing a change is expressed as $m_{sys} =$ *constant* or $dm_{sys}/dt = 0$, which is a statement of the obvious that the mass of the system remains constant during a process. For a control volume (CV), mass balance is expressed in the rate form as

Conservation of mass:
$$\dot{m}_{in} - \dot{m}_{out} = \frac{dm_{CV}}{dt} \quad \text{(5–1)}$$

where $\dot{m}_{in}$ and $\dot{m}_{out}$ are the total rates of mass flow into and out of the control volume, respectively, and dm_{CV}/dt is the rate of change of mass within the control volume boundaries. In fluid mechanics, the conservation of mass relation written for a differential control volume is usually called the *continuity equation*. Conservation of mass is discussed in Section 5–2.

Conservation of Momentum

The product of the mass and the velocity of a body is called the *linear momentum* or just the *momentum* of the body, and the momentum of a rigid body of mass m moving with a velocity $\vec{V}$ is $m\vec{V}$. Newton's second law states that the acceleration of a body is proportional to the net force acting on it and is inversely proportional to its mass, and that the rate of change of the momentum of a body is equal to the net force acting on the body. Therefore, the momentum of a system remains constant when the net force acting on it is zero, and thus the momentum of such systems is conserved. This is known as the *conservation of momentum principle*. In fluid mechanics, Newton's second law is usually referred to as the *linear momentum equation*, which is discussed in Chap. 6 together with the *angular momentum equation*.

Conservation of Energy

Energy can be transferred to or from a closed system by heat or work, and the conservation of energy principle requires that the net energy transfer to or from a system during a process be equal to the change in the energy content of the system. Control volumes involve energy transfer via mass flow also, and the *conservation of energy principle*, also called the *energy balance*, is expressed as

Conservation of energy:
$$\dot{E}_{in} - \dot{E}_{out} = \frac{dE_{CV}}{dt} \quad \text{(5–2)}$$

where $\dot{E}_{in}$ and $\dot{E}_{out}$ are the total rates of energy transfer into and out of the control volume, respectively, and dE_{CV}/dt is the rate of change of energy within the control volume boundaries. In fluid mechanics, we usually limit our consideration to mechanical forms of energy only. Conservation of energy is discussed in Section 5–6.

5–2 ▪ CONSERVATION OF MASS

The conservation of mass principle is one of the most fundamental principles in nature. We are all familiar with this principle, and it is not difficult to understand. As the saying goes, You cannot have your cake and eat it too! A person does not have to be a scientist to figure out how much vinegar-and-oil dressing will be obtained by mixing 100 g of oil with 25 g of vinegar. Even chemical equations are balanced on the basis of the conservation of mass principle. When 16 kg of oxygen reacts with 2 kg of hydrogen, 18 kg of water is formed (Fig. 5–2). In an electrolysis process, the water will separate back to 2 kg of hydrogen and 16 kg of oxygen.

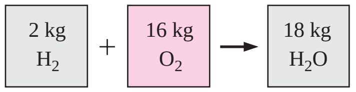


FIGURE 5–2
Mass is conserved even during chemical reactions.

Mass, like energy, is a conserved property, and it cannot be created or destroyed during a process. However, mass m and energy E can be converted to each other according to the well-known formula proposed by Albert Einstein (1879–1955):

$$E = mc^2 \qquad \text{(5–3)}$$

where c is the speed of light in a vacuum, which is $c = 2.9979 \times 10^8$ m/s. This equation suggests that the mass of a system changes when its energy changes. However, for all energy interactions encountered in practice, with the exception of nuclear reactions, the change in mass is extremely small and cannot be detected by even the most sensitive devices. For example, when 1 kg of water is formed from oxygen and hydrogen, the amount of energy released is 15,879 kJ, which corresponds to a mass of 1.76×10^{-10} kg. A mass of this magnitude is beyond the accuracy required by practically all engineering calculations and thus can be disregarded.

For *closed systems,* the conservation of mass principle is implicitly used by requiring that the mass of the system remain constant during a process. For *control volumes,* however, mass can cross the boundaries, and so we must keep track of the amount of mass entering and leaving the control volume.

Mass and Volume Flow Rates

The amount of mass flowing through a cross section per unit time is called the **mass flow rate** and is denoted by $\dot{m}$. The dot over a symbol is used to indicate *time rate of change*.

A fluid flows into or out of a control volume, usually through pipes or ducts. The differential mass flow rate of fluid flowing across a small area element dA_c in a cross section of the pipe is proportional to dA_c itself, the fluid density ρ, and the component of the flow velocity normal to dA_c, which we denote as V_n, and is expressed as (Fig. 5–3)

$$\delta\dot{m} = \rho V_n \, dA_c \qquad \text{(5–4)}$$

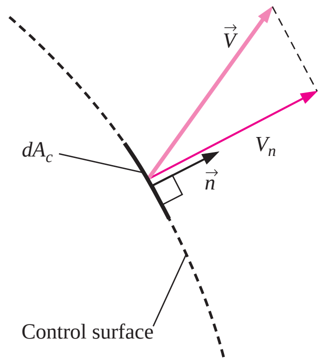


FIGURE 5–3
The normal velocity V_n for a surface is the component of velocity perpendicular to the surface.

Note that both δ and d are used to indicate differential quantities, but δ is typically used for quantities (such as heat, work, and mass transfer) that are *path functions* and have *inexact differentials,* while d is used for quantities (such as properties) that are *point functions* and have *exact differentials.* For flow through an annulus of inner radius r_1 and outer radius r_2, for example,

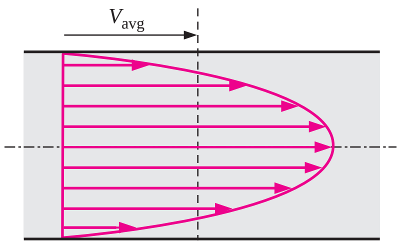


FIGURE 5–4
The average velocity V_{avg} is defined as the average speed through a cross section.

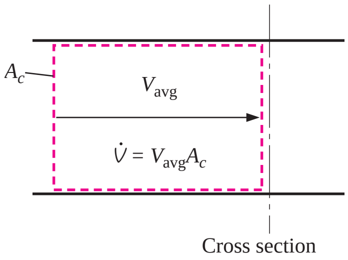


FIGURE 5–5
The volume flow rate is the volume of fluid flowing through a cross section per unit time.

$\int_1^2 dA_c = A_{c2} - A_{c1} = \pi(r_2^2 - r_1^2)$ but $\int_1^2 \delta\dot{m} = \dot{m}_{total}$ (total mass flow rate through the annulus), not $\dot{m}_2 - \dot{m}_1$. For specified values of r_1 and r_2, the value of the integral of dA_c is fixed (thus the names point function and exact differential), but this is not the case for the integral of $\delta\dot{m}$ (thus the names path function and inexact differential).

The mass flow rate through the entire cross-sectional area of a pipe or duct is obtained by integration:

$$\dot{m} = \int_{A_c} \delta\dot{m} = \int_{A_c} \rho V_n \, dA_c \qquad \text{(kg/s)} \tag{5–5}$$

While Eq. 5–5 is always valid (in fact it is *exact*), it is not always practical for engineering analyses because of the integral. We would like instead to express mass flow rate in terms of average values over a cross section of the pipe. In a general compressible flow, both ρ and V_n vary across the pipe. In many practical applications, however, the density is essentially uniform over the pipe cross section, and we can take ρ outside the integral of Eq. 5–5. Velocity, however, is *never* uniform over a cross section of a pipe because of the no-slip condition at the walls. Rather, the velocity varies from zero at the walls to some maximum value at or near the centerline of the pipe. We define the **average velocity** V_{avg} as the average value of V_n across the entire cross section of the pipe (Fig. 5–4),

Average velocity:
$$V_{avg} = \frac{1}{A_c}\int_{A_c} V_n \, dA_c \tag{5–6}$$

where A_c is the area of the cross section normal to the flow direction. Note that if the speed were V_{avg} all through the cross section, the mass flow rate would be identical to that obtained by integrating the actual velocity profile. Thus for incompressible flow or even for compressible flow where ρ is uniform across A_c, Eq. 5–5 becomes

$$\dot{m} = \rho V_{avg} A_c \qquad \text{(kg/s)} \tag{5–7}$$

For compressible flow, we can think of ρ as the bulk average density over the cross section, and then Eq. 5–7 can still be used as a reasonable approximation. For simplicity, we drop the subscript on the average velocity. Unless otherwise stated, V denotes the average velocity in the flow direction. Also, A_c denotes the cross-sectional area normal to the flow direction.

The volume of the fluid flowing through a cross section per unit time is called the **volume flow rate** $\dot{V}$ (Fig. 5–5) and is given by

$$\dot{V} = \int_{A_c} V_n \, dA_c = V_{avg} A_c = V A_c \qquad (\text{m}^3/\text{s}) \tag{5–8}$$

An early form of Eq. 5–8 was published in 1628 by the Italian monk Benedetto Castelli (circa 1577–1644). Note that many fluid mechanics textbooks use Q instead of $\dot{V}$ for volume flow rate. We use $\dot{V}$ to avoid confusion with heat transfer.

The mass and volume flow rates are related by

$$\dot{m} = \rho\dot{V} = \frac{\dot{V}}{v} \tag{5–9}$$

where v is the specific volume. This relation is analogous to $m = \rho V = V/v$, which is the relation between the mass and the volume of a fluid in a container.

Conservation of Mass Principle

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

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

or

$$m_{in} - m_{out} = \Delta m_{CV} \qquad \text{(kg)} \tag{5–10}$$

where $\Delta m_{CV} = m_{final} - m_{initial}$ is the change in the mass of the control volume during the process (Fig. 5–6). It can also be expressed in *rate form* as

$$\dot{m}_{in} - \dot{m}_{out} = dm_{CV}/dt \qquad \text{(kg/s)} \tag{5–11}$$

where $\dot{m}_{in}$ and $\dot{m}_{out}$ are the total rates of mass flow into and out of the control volume, and dm_{CV}/dt is the rate of change of mass within the control volume boundaries. Equations 5–10 and 5–11 are often referred to as the **mass balance** and are applicable to any control volume undergoing any kind of process.

Consider a control volume of arbitrary shape, as shown in Fig. 5–7. The mass of a differential volume dV within the control volume is $dm = \rho\, dV$. The total mass within the control volume at any instant in time t is determined by integration to be

Total mass within the CV:
$$m_{CV} = \int_{CV} \rho\, dV \tag{5–12}$$

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

Rate of change of mass within the CV:
$$\frac{dm_{CV}}{dt} = \frac{d}{dt}\int_{CV} \rho\, dV \tag{5–13}$$

For the special case of no mass crossing the control surface (i.e., the control volume resembles a closed system), the conservation of mass principle reduces to that of a system that can be expressed as $dm_{CV}/dt = 0$. This relation is valid whether the control volume is fixed, moving, or deforming.

Now consider mass flow into or out of the control volume through a differential area dA on the control surface of a fixed control volume. Let $\vec{n}$ be the outward unit vector of dA normal to dA and $\vec{V}$ be the flow velocity at dA relative to a fixed coordinate system, as shown in Fig. 5–7. In general, the velocity may cross dA at an angle θ off the normal of dA, and the mass flow rate is proportional to the normal component of velocity $\vec{V}_n = \vec{V}\cos\theta$ ranging from a maximum outflow of $\vec{V}$ for $\theta = 0$ (flow is normal to dA) to a minimum of zero for $\theta = 90°$ (flow is tangent to dA) to a maximum *inflow* of $\vec{V}$ for $\theta = 180°$ (flow is normal to dA but in the opposite direction). Making

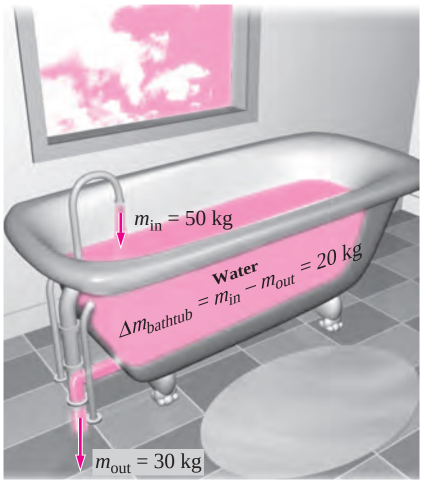


FIGURE 5–6
Conservation of mass principle for an ordinary bathtub.

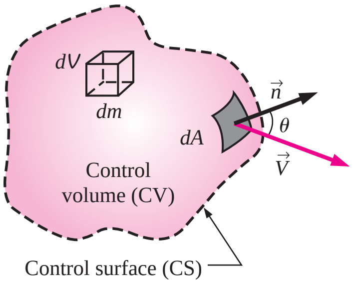


FIGURE 5–7
The differential control volume dV and the differential control surface dA used in the derivation of the conservation of mass relation.

use of the concept of dot product of two vectors, the magnitude of the normal component of velocity can be expressed as

Normal component of velocity:
$$V_n = V\cos\theta = \vec{V}\cdot\vec{n} \quad \textbf{(5–14)}$$

The mass flow rate through dA is proportional to the fluid density ρ, normal velocity V_n, and the flow area dA, and can be expressed as

Differential mass flow rate:
$$\delta\dot{m} = \rho V_n\,dA = \rho(V\cos\theta)\,dA = \rho(\vec{V}\cdot\vec{n})\,dA \quad \textbf{(5–15)}$$

The net flow rate into or out of the control volume through the entire control surface is obtained by integrating $\delta\dot{m}$ over the entire control surface,

Net mass flow rate:
$$\dot{m}_{\text{net}} = \int_{CS}\delta\dot{m} = \int_{CS}\rho V_n\,dA = \int_{CS}\rho(\vec{V}\cdot\vec{n})\,dA \quad \textbf{(5–16)}$$

Note that $\vec{V}\cdot\vec{n} = V\cos\theta$ is positive for $\theta < 90°$ (outflow) and negative for $\theta > 90°$ (inflow). Therefore, the direction of flow is automatically accounted for, and the surface integral in Eq. 5–16 directly gives the *net* mass flow rate. A positive value for $\dot{m}_{\text{net}}$ indicates net outflow, and a negative value indicates a net inflow of mass.

Rearranging Eq. 5–11 as $dm_{CV}/dt + \dot{m}_{\text{out}} - \dot{m}_{\text{in}} = 0$, the conservation of mass relation for a fixed control volume can then be expressed as

General conservation of mass:
$$\frac{d}{dt}\int_{CV}\rho\,dV + \int_{CS}\rho(\vec{V}\cdot\vec{n})\,dA = 0 \quad \textbf{(5–17)}$$

It states that *the time rate of change of mass within the control volume plus the net mass flow rate through the control surface is equal to zero.*

The general conservation of mass relation for a control volume can also be derived using the Reynolds transport theorem (RTT) by taking the property B to be the mass m (Chap. 4). Then we have $b = 1$ since dividing the mass by mass to get the property per unit mass gives unity. Also, the mass of a system is constant, and thus its time derivative is zero. That is, $dm_{sys}/dt = 0$. Then the Reynolds transport equation reduces immediately to Eq. 5–17, as shown in Fig. 5–8, and thus illustrates that the Reynolds transport theorem is a very powerful tool indeed. In Chap. 6 we apply the RTT to obtain the linear and angular momentum equations for control volumes.

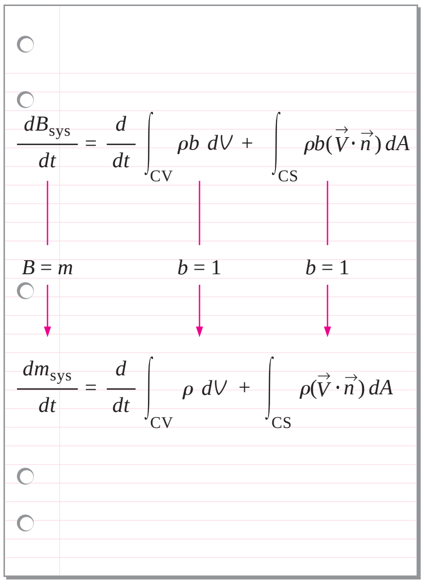


FIGURE 5–8
The conservation of mass equation is obtained by replacing B in the Reynolds transport theorem by mass m, and b by 1 (m per unit mass $= m/m = 1$).

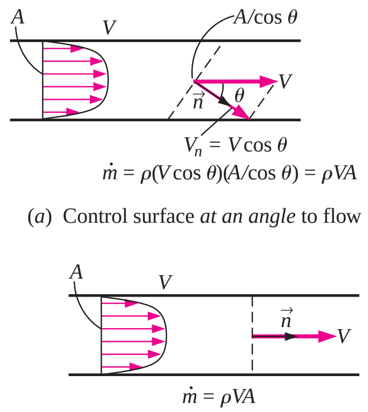
Splitting the surface integral in Eq. 5–17 into two parts—one for the outgoing flow streams (positive) and one for the incoming streams (negative)—the general conservation of mass relation can also be expressed as

$$\frac{d}{dt}\int_{CV}\rho\,dV + \sum_{\text{out}}\int_A \rho V_n\,dA - \sum_{\text{in}}\int_A \rho V_n\,dA = 0 \quad \textbf{(5–18)}$$

where A represents the area for an inlet or outlet, and the summation signs are used to emphasize that *all* the inlets and outlets are to be considered. Using the definition of mass flow rate, Eq. 5–18 can also be expressed as

$$\frac{d}{dt}\int_{CV}\rho\,dV = \sum_{\text{in}}\dot{m} - \sum_{\text{out}}\dot{m} \quad \text{or} \quad \frac{dm_{CV}}{dt} = \sum_{\text{in}}\dot{m} - \sum_{\text{out}}\dot{m} \quad \textbf{(5–19)}$$

There is considerable flexibility in the selection of a control volume when solving a problem. Several control volume choices may be correct, but some are more convenient to work with. A control volume should not introduce any unnecessary complications. The proper choice of a control volume can make the solution of a seemingly complicated problem rather easy. A simple rule in selecting a control volume is to make the control surface *normal to flow* at all locations where it crosses fluid flow, whenever possible. This way the dot product $\vec{V} \cdot \vec{n}$ simply becomes the magnitude of the velocity, and the integral $\int_A \rho(\vec{V} \cdot \vec{n})\, dA$ becomes simply ρVA (Fig. 5–9).



(*a*) Control surface *at an angle* to flow

(*b*) Control surface *normal* to flow

FIGURE 5–9
A control surface should always be selected *normal to flow* at all locations where it crosses the fluid flow to avoid complications, even though the result is the same.

Moving or Deforming Control Volumes

Equations 5–17 and 5–18 are also valid for moving or deforming control volumes provided that the *absolute velocity* $\vec{V}$ is replaced by the *relative velocity* $\vec{V}_r$, which is the fluid velocity relative to the control surface (Chap. 4). In the case of a nondeforming control volume, relative velocity is the fluid velocity observed by a person moving with the control volume and is expressed as $\vec{V}_r = \vec{V} - \vec{V}_{CV}$, where $\vec{V}$ is the fluid velocity and $\vec{V}_{CV}$ is the velocity of the control volume, both relative to a fixed point outside. Again note that this is a *vector* subtraction.

Some practical problems (such as the injection of medication through the needle of a syringe by the forced motion of the plunger) involve deforming control volumes. The conservation of mass relations developed can still be used for such deforming control volumes provided that the velocity of the fluid crossing a deforming part of the control surface is expressed relative to the control surface (that is, the fluid velocity should be expressed relative to a reference frame attached to the deforming part of the control surface). The relative velocity in this case at any point on the control surface is expressed as $\vec{V}_r = \vec{V} - \vec{V}_{CS}$, where $\vec{V}_{CS}$ is the local velocity of the control surface at that point relative to a fixed point outside the control volume.

Mass Balance for Steady-Flow Processes

During a steady-flow process, the total amount of mass contained within a control volume does not change with time (m_{CV} = constant). Then the conservation of mass principle requires that the total amount of mass entering a control volume equal the total amount of mass leaving it. For a garden hose nozzle in steady operation, for example, the amount of water entering the nozzle per unit time is equal to the amount of water leaving it per unit time.

When dealing with steady-flow processes, we are not interested in the amount of mass that flows in or out of a device over time; instead, we are interested in the amount of mass flowing per unit time, that is, *the mass flow rate* $\dot{m}$. *The conservation of mass principle* for a general steady-flow system with multiple inlets and outlets can be expressed in rate form as (Fig. 5–10)

Steady flow:
$$\sum_{\text{in}} \dot{m} = \sum_{\text{out}} \dot{m} \qquad \text{(kg/s)} \tag{5–20}$$

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

Many engineering devices such as nozzles, diffusers, turbines, compressors, and pumps involve a single stream (only one inlet and one outlet). For these cases, we denote the inlet state by the subscript 1 and the outlet state by the subscript 2, and drop the summation signs. Then Eq. 5–20 reduces, for *single-stream steady-flow systems*, to

Steady flow (single stream):
$$\dot{m}_1 = \dot{m}_2 \quad \rightarrow \quad \rho_1 V_1 A_1 = \rho_2 V_2 A_2 \tag{5–21}$$

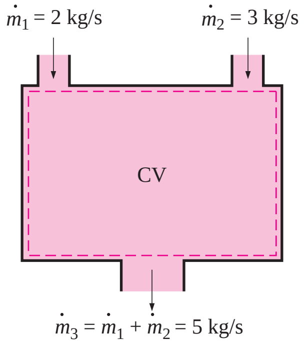


FIGURE 5–10
Conservation of mass principle for a two-inlet–one-outlet steady-flow system.

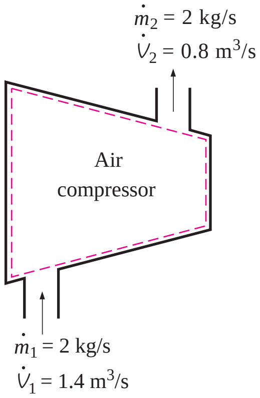


FIGURE 5–11
During a steady-flow process, volume flow rates are not necessarily conserved although mass flow rates are.

Special Case: Incompressible Flow

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

Steady, incompressible flow:
$$\sum_{\text{in}} \dot{V} = \sum_{\text{out}} \dot{V} \qquad (\text{m}^3/\text{s}) \tag{5–22}$$

For single-stream steady-flow systems it becomes

Steady, incompressible flow (single stream):
$$\dot{V}_1 = \dot{V}_2 \rightarrow V_1A_1 = V_2A_2 \tag{5–23}$$

It should always be kept in mind that there is no such thing as a "conservation of volume" principle. Therefore, the volume flow rates into and out of a steady-flow device may be different. The volume flow rate at the outlet of an air compressor is much less than that at the inlet even though the mass flow rate of air through the compressor is constant (Fig. 5–11). This is due to the higher density of air at the compressor exit. For steady flow of liquids, however, the volume flow rates, as well as the mass flow rates, remain constant since liquids are essentially incompressible (constant-density) substances. Water flow through the nozzle of a garden hose is an example of the latter case.

The conservation of mass principle is based on experimental observations and requires every bit of mass to be accounted for during a process. If you can balance your checkbook (by keeping track of deposits and withdrawals, or by simply observing the "conservation of money" principle), you should have no difficulty applying the conservation of mass principle to engineering systems.

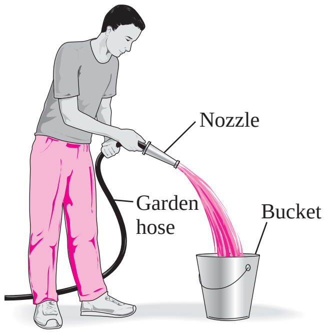


FIGURE 5–12
Schematic for Example 5–1.

EXAMPLE 5–1 Water Flow through a Garden Hose Nozzle

A garden hose attached with a nozzle is used to fill a 10-gal bucket. The inner diameter of the hose is 2 cm, and it reduces to 0.8 cm at the nozzle exit (Fig. 5–12). If it takes 50 s to fill the bucket with water, determine (*a*) the volume and mass flow rates of water through the hose, and (*b*) the average velocity of water at the nozzle exit.

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

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

Properties We take the density of water to be 1000 kg/m^3 = 1 kg/L.

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

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

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

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

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

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

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

Discussion It can be shown that the average velocity in the hose is 2.4 m/s. Therefore, the nozzle increases the water velocity by over six times.

EXAMPLE 5–2 Discharge of Water from a Tank

A 4-ft-high, 3-ft-diameter cylindrical water tank whose top is open to the atmosphere is initially filled with water. Now the discharge plug near the bottom of the tank is pulled out, and a water jet whose diameter is 0.5 in streams out (Fig. 5–13). The average velocity of the jet is given by $V = \sqrt{2gh}$, where h is the height of water in the tank measured from the center of the hole (a variable) and g is the gravitational acceleration. Determine how long it will take for the water level in the tank to drop to 2 ft from the bottom.

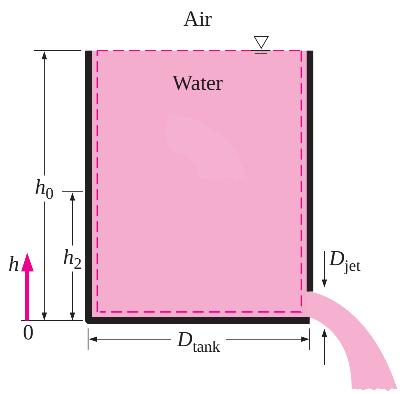


FIGURE 5–13
Schematic for Example 5–2.

SOLUTION The plug near the bottom of a water tank is pulled out. The time it will take for half of the water in the tank to empty is to be determined.

Assumptions **1** Water is an incompressible substance. **2** The distance between the bottom of the tank and the center of the hole is negligible compared to the total water height. **3** The gravitational acceleration is 32.2 ft/s².

Analysis We take the volume occupied by water as the control volume. The size of the control volume decreases in this case as the water level drops, and thus this is a variable control volume. (We could also treat this as a fixed control volume that consists of the interior volume of the tank by disregarding the air that replaces the space vacated by the water.) This is obviously an unsteady-flow problem since the properties (such as the amount of mass) within the control volume change with time.

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

$$\dot{m}_{in} - \dot{m}_{out} = \frac{dm_{CV}}{dt} \quad \textbf{(1)}$$

During this process no mass enters the control volume ($\dot{m}_{in} = 0$), and the mass flow rate of discharged water can be expressed as

$$\dot{m}_{out} = (\rho V A)_{out} = \rho\sqrt{2gh}A_{jet} \quad \textbf{(2)}$$

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

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

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

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

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

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

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

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

Substituting, the time of discharge is determined to be

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

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

Discussion Using the same relation with $h_2 = 0$ gives $t = 43.1$ min for the discharge of the entire amount of water in the tank. Therefore, emptying the bottom half of the tank takes much longer than emptying the top half. This is due to the decrease in the average discharge velocity of water with decreasing h.

5–3 ▪ MECHANICAL ENERGY AND EFFICIENCY

Many fluid systems are designed to transport a fluid from one location to another at a specified flow rate, velocity, and elevation difference, and the system may generate mechanical work in a turbine or it may consume mechanical work in a pump or fan during this process. These systems do not involve the conversion of nuclear, chemical, or thermal energy to mechanical energy. Also, they do not involve any heat transfer in any significant amount, and they operate essentially at constant temperature. Such systems can be analyzed conveniently by considering the *mechanical forms of energy* only and the frictional effects that cause the mechanical energy to be lost (i.e., to be converted to thermal energy that usually cannot be used for any useful purpose).

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

A pump transfers mechanical energy to a fluid by raising its pressure, and a turbine extracts mechanical energy from a fluid by dropping its pressure. Therefore, the pressure of a flowing fluid is also associated with its mechanical energy. In fact, the pressure unit Pa is equivalent to Pa = N/m^2 = N · m/m^3 = J/m^3, which is energy per unit volume, and the product $P\text{v}$ or its equivalent P/ρ has the unit J/kg, which is energy per unit mass. Note that pressure itself is not a form of energy. But a pressure force acting on a fluid through a distance produces work, called *flow work*, in the amount of P/ρ per unit mass. Flow work is expressed in terms of fluid properties, and it is convenient to view it as part of the energy of a flowing fluid and call it *flow* *energy*. Therefore, the mechanical energy of a flowing fluid can be expressed on a unit-mass basis as (Fig. 5–14).

$$e_{\text{mech}} = \frac{P}{\rho} + \frac{V^2}{2} + gz$$

where P/ρ is the *flow energy*, $V^2/2$ is the *kinetic energy*, and gz is the *potential energy* of the fluid, all per unit mass. Then the mechanical energy change of a fluid during incompressible flow becomes

$$\Delta e_{\text{mech}} = \frac{P_2 - P_1}{\rho} + \frac{V_2^2 - V_1^2}{2} + g(z_2 - z_1) \quad \text{(kJ/kg)} \tag{5–24}$$

Therefore, the mechanical energy of a fluid does not change during flow if its pressure, density, velocity, and elevation remain constant. In the absence of any losses, the mechanical energy change represents the mechanical work supplied to the fluid (if $\Delta e_{\text{mech}} > 0$) or extracted from the fluid (if $\Delta e_{\text{mech}} < 0$).

Consider a container of height h filled with water, as shown in Fig. 5–15, with the reference level selected at the bottom surface. The gage pressure and the potential energy per unit mass are, respectively, $P_A = 0$ and $\text{pe}_A = gh$ at point A at the free surface, and $P_B = \rho gh$ and $\text{pe}_B = 0$ at point B at the bottom of the container. An ideal hydraulic turbine would produce the same work per unit mass $w_{\text{turbine}} = gh$ whether it receives water (or any other fluid with constant density) from the top or from the bottom of the container. Note that we are also assuming ideal flow (no irreversible losses) through the pipe leading from the tank to the turbine. Therefore, the total mechanical energy of water at the bottom is equivalent to that at the top.

The transfer of mechanical energy is usually accomplished by a rotating shaft, and thus mechanical work is often referred to as *shaft work*. A pump or a fan receives shaft work (usually from an electric motor) and transfers it to the fluid as mechanical energy (less frictional losses). A turbine, on the other hand, converts the mechanical energy of a fluid to shaft work. In the absence of any irreversibilities such as friction, mechanical energy can be converted entirely from one mechanical form to another, and the **mechanical efficiency** of a device or process can be defined as (Fig. 5–16)

$$\eta_{\text{mech}} = \frac{\text{Mechanical energy output}}{\text{Mechanical energy input}} = \frac{E_{\text{mech, out}}}{E_{\text{mech, in}}} = 1 - \frac{E_{\text{mech, loss}}}{E_{\text{mech, in}}} \tag{5–25}$$

A conversion efficiency of less than 100 percent indicates that conversion is less than perfect and some losses have occurred during conversion. A

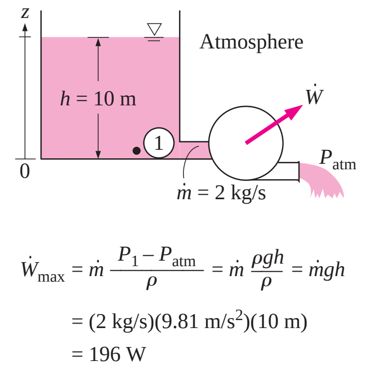


FIGURE 5–14
In the absence of any changes in flow velocity and elevation, the power produced by an ideal hydraulic turbine is proportional to the pressure drop of water across the turbine.

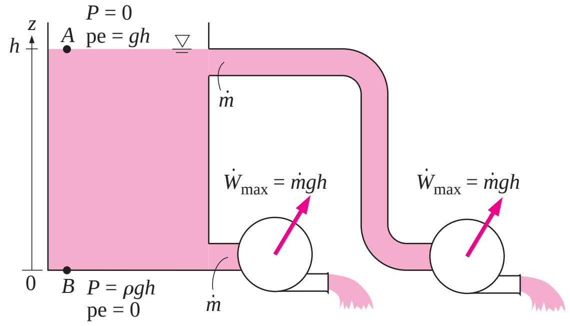


FIGURE 5–15
The mechanical energy of water at the bottom of a container is equal to the mechanical energy at any depth including the free surface of the container.

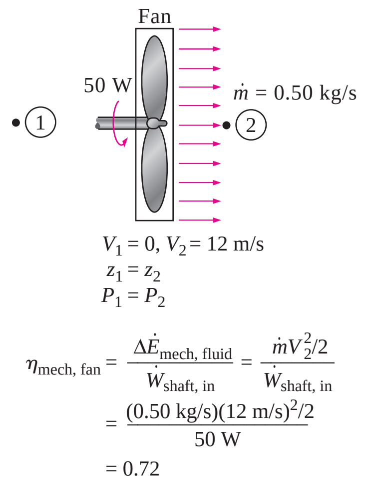


FIGURE 5–16
The mechanical efficiency of a fan is the ratio of the kinetic energy of air at the fan exit to the mechanical power input.

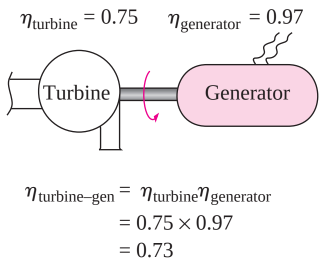


FIGURE 5–17
The overall efficiency of a turbine–generator is the product of the efficiency of the turbine and the efficiency of the generator, and represents the fraction of the mechanical energy of the fluid converted to electric energy.

mechanical efficiency of 97 percent indicates that 3 percent of the mechanical energy input is converted to thermal energy as a result of frictional heating, and this will manifest itself as a slight rise in the temperature of the fluid.

In fluid systems, we are usually interested in increasing the pressure, velocity, and/or elevation of a fluid. This is done by *supplying mechanical energy* to the fluid by a pump, a fan, or a compressor (we will refer to all of them as pumps). Or we are interested in the reverse process of *extracting mechanical energy* from a fluid by a turbine and producing mechanical power in the form of a rotating shaft that can drive a generator or any other rotary device. The degree of perfection of the conversion process between the mechanical work supplied or extracted and the mechanical energy of the fluid is expressed by the **pump efficiency** and **turbine efficiency,** defined as

$$\eta_{\text{pump}} = \frac{\text{Mechanical energy increase of the fluid}}{\text{Mechanical energy input}} = \frac{\Delta \dot{E}_{\text{mech, fluid}}}{\dot{W}_{\text{shaft, in}}} = \frac{\dot{W}_{\text{pump, }u}}{\dot{W}_{\text{pump}}} \quad \textbf{(5–26)}$$

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

$$\eta_{\text{turbine}} = \frac{\text{Mechanical energy output}}{\text{Mechanical energy decrease of the fluid}} = \frac{\dot{W}_{\text{shaft, out}}}{|\Delta \dot{E}_{\text{mech, fluid}}|} = \frac{\dot{W}_{\text{turbine}}}{\dot{W}_{\text{turbine, }e}} \quad \textbf{(5–27)}$$

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

The mechanical efficiency should not be confused with the **motor efficiency** and the **generator efficiency,** which are defined as

Motor:
$$\eta_{\text{motor}} = \frac{\text{Mechanical power output}}{\text{Electric power input}} = \frac{\dot{W}_{\text{shaft, out}}}{\dot{W}_{\text{elect, in}}} \quad \textbf{(5–28)}$$

and

Generator:
$$\eta_{\text{generator}} = \frac{\text{Electric power output}}{\text{Mechanical power input}} = \frac{\dot{W}_{\text{elect, out}}}{\dot{W}_{\text{shaft, in}}} \quad \textbf{(5–29)}$$

A pump is usually packaged together with its motor, and a turbine with its generator. Therefore, we are usually interested in the **combined** or **overall efficiency** of pump–motor and turbine–generator combinations (Fig. 5–17), which are defined as

$$\eta_{\text{pump–motor}} = \eta_{\text{pump}}\,\eta_{\text{motor}} = \frac{\dot{W}_{\text{pump, }u}}{\dot{W}_{\text{elect, in}}} = \frac{\Delta \dot{E}_{\text{mech, fluid}}}{\dot{W}_{\text{elect, in}}} \quad \textbf{(5–30)}$$

and

$$\eta_{\text{turbine–gen}} = \eta_{\text{turbine}}\,\eta_{\text{generator}} = \frac{\dot{W}_{\text{elect, out}}}{\dot{W}_{\text{turbine, }e}} = \frac{\dot{W}_{\text{elect, out}}}{|\Delta \dot{E}_{\text{mech, fluid}}|} \quad \textbf{(5–31)}$$

All the efficiencies just defined range between 0 and 100 percent. The lower limit of 0 percent corresponds to the conversion of the entire mechanical or electric energy input to thermal energy, and the device in this case functions like a resistance heater. The upper limit of 100 percent corresponds to the case of perfect conversion with no friction or other irreversibilities, and thus no conversion of mechanical or electric energy to thermal energy.

EXAMPLE 5–3 Performance of a Hydraulic Turbine–Generator

The water in a large lake is to be used to generate electricity by the installation of a hydraulic turbine–generator at a location where the depth of the water is 50 m (Fig. 5–18). Water is to be supplied at a rate of 5000 kg/s. If the electric power generated is measured to be 1862 kW and the generator efficiency is 95 percent, determine (*a*) the overall efficiency of the turbine–generator, (*b*) the mechanical efficiency of the turbine, and (*c*) the shaft power supplied by the turbine to the generator.

SOLUTION A hydraulic turbine–generator is to generate electricity from the water of a lake. The overall efficiency, the turbine efficiency, and the shaft power are to be determined.

Assumptions **1** The elevation of the lake remains constant. **2** The mechanical energy of water at the turbine exit is negligible.

Properties The density of water can be taken to be $\rho = 1000$ kg/m^3.

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

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

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

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

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

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

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

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

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

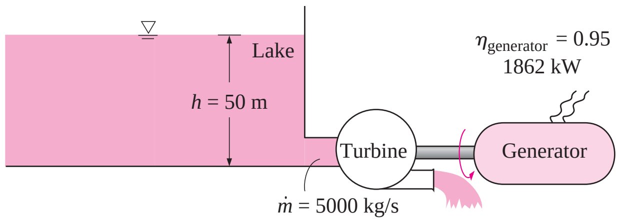


FIGURE 5–18
Schematic for Example 5–3.

Discussion Note that the lake supplies 2455 kW of mechanical energy to the turbine, which converts 1964 kW of it to shaft work that drives the generator, which generates 1862 kW of electric power. There are irreversible losses through each component.

EXAMPLE 5–4 Conservation of Energy for an Oscillating Steel Ball

The motion of a steel ball in a hemispherical bowl of radius h shown in Fig. 5–19 is to be analyzed. The ball is initially held at the highest location at point A, and then it is released. Obtain relations for the conservation of energy of the ball for the cases of frictionless and actual motions.

SOLUTION A steel ball is released in a bowl. Relations for the energy balance are to be obtained.

Assumptions The motion is frictionless, and thus friction between the ball, the bowl, and the air is negligible.

Analysis When the ball is released, it accelerates under the influence of gravity, reaches a maximum velocity (and minimum elevation) at point B at the bottom of the bowl, and moves up toward point C on the opposite side. In the ideal case of frictionless motion, the ball will oscillate between points A and C. The actual motion involves the conversion of the kinetic and potential energies of the ball to each other, together with overcoming resistance to motion due to friction (doing frictional work). The general energy balance for any system undergoing any process is

$$\underbrace{E_{in} - E_{out}}_{\substack{\text{Net energy transfer} \\ \text{by heat, work, and mass}}} = \underbrace{\Delta E_{system}}_{\substack{\text{Change in internal, kinetic,} \\ \text{potential, etc., energies}}}$$

Then the energy balance for the ball for a process from point 1 to point 2 becomes

$$-w_{friction} = (ke_2 + pe_2) - (ke_1 + pe_1)$$

or

$$\frac{V_1^2}{2} + gz_1 = \frac{V_2^2}{2} + gz_2 + w_{friction}$$

since there is no energy transfer by heat or mass and no change in the internal energy of the ball (the heat generated by frictional heating is dissipated to

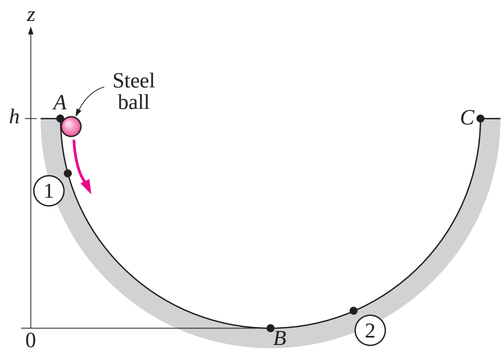


FIGURE 5–19
Schematic for Example 5–4.

the surrounding air). The frictional work term w_{friction} is often expressed as e_{loss} to represent the loss (conversion) of mechanical energy into thermal energy.

For the idealized case of frictionless motion, the last relation reduces to

$$\frac{V_1^2}{2} + gz_1 = \frac{V_2^2}{2} + gz_2 \quad \text{or} \quad \frac{V^2}{2} + gz = C = \text{constant}$$

where the value of the constant is $C = gh$. That is, *when the frictional effects are negligible, the sum of the kinetic and potential energies of the ball remains constant.*

Discussion This is certainly a more intuitive and convenient form of the conservation of energy equation for this and other similar processes such as the swinging motion of the pendulum of a wall clock. The relation obtained is analogous to the Bernoulli equation derived in Section 5–4.

Most processes encountered in practice involve only certain forms of energy, and in such cases it is more convenient to work with the simplified versions of the energy balance. For systems that involve only *mechanical forms of energy* and its transfer as *shaft work*, the conservation of energy principle can be expressed conveniently as

$$E_{\text{mech, in}} - E_{\text{mech, out}} = \Delta E_{\text{mech, system}} + E_{\text{mech, loss}} \qquad \textbf{(5–32)}$$

where $E_{\text{mech, loss}}$ represents the conversion of mechanical energy to thermal energy due to irreversibilities such as friction. For a system in steady operation, the mechanical energy balance becomes $\dot{E}_{\text{mech, in}} = \dot{E}_{\text{mech, out}} + \dot{E}_{\text{mech, loss}}$ (Fig. 5–20).

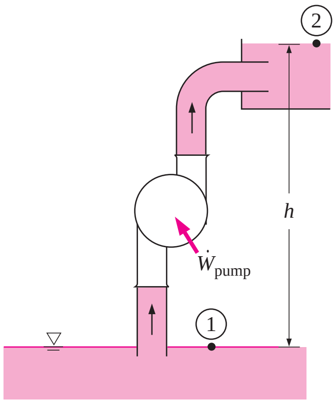


FIGURE 5–20
Most fluid flow problems involve mechanical forms of energy only, and such problems are conveniently solved by using a *mechanical energy* balance.

5–4 ▪ THE BERNOULLI EQUATION

The **Bernoulli equation** is *an approximate relation between pressure, velocity, and elevation*, and is valid in *regions of steady, incompressible flow where net frictional forces are negligible* (Fig. 5–21). Despite its simplicity, it has proven to be a very powerful tool in fluid mechanics. In this section, we derive the Bernoulli equation by applying the *conservation of linear momentum principle*, and we demonstrate both its usefulness and its limitations.

The key approximation in the derivation of the Bernoulli equation is that *viscous effects are negligibly small compared to inertial, gravitational, and pressure effects*. Since all fluids have viscosity (there is no such thing as an "inviscid fluid"), this approximation cannot be valid for an entire flow field of practical interest. In other words, we cannot apply the Bernoulli equation *everywhere* in a flow, no matter how small the fluid's viscosity. However, it turns out that the approximation *is* reasonable in certain *regions* of many practical flows. We refer to such regions as *inviscid regions of flow*, and we stress that they are *not* regions where the fluid itself is inviscid or frictionless, but rather they are regions where net viscous or frictional forces are negligibly small compared to other forces acting on fluid particles.

Care must be exercised when applying the Bernoulli equation since it is an approximation that applies only to inviscid regions of flow. In general, frictional effects are always important very close to solid walls (*boundary layers*) and directly downstream of bodies (*wakes*). Thus, the Bernoulli

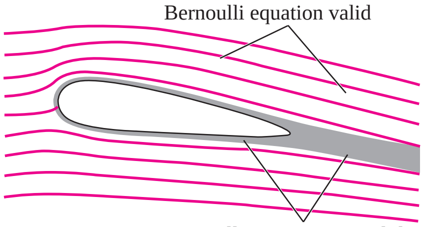


FIGURE 5–21
The *Bernoulli equation* is an *approximate* equation that is valid only in *inviscid regions of flow* where net viscous forces are negligibly small compared to inertial, gravitational, or pressure forces. Such regions occur outside of *boundary layers* and *wakes*.

approximation is typically useful in flow regions outside of boundary layers and wakes, where the fluid motion is governed by the combined effects of pressure and gravity forces.

The motion of a particle and the path it follows are described by the *velocity vector* as a function of time and space coordinates and the initial position of the particle. When the flow is *steady* (no change with time at a specified location), all particles that pass through the same point follow the same path (which is the *streamline*), and the velocity vectors remain tangent to the path at every point.

Acceleration of a Fluid Particle

Often it is convenient to describe the motion of a particle in terms of its distance s along a streamline together with the radius of curvature along the streamline. The velocity of the particle is related to the distance by $V = ds/dt$, which may vary along the streamline. In two-dimensional flow, the acceleration can be decomposed into two components: *streamwise acceleration* a_s along the streamline and *normal acceleration* a_n in the direction normal to the streamline, which is given as $a_n = V^2/R$. Note that streamwise acceleration is due to a change in speed along a streamline, and normal acceleration is due to a change in direction. For particles that move along a *straight path*, $a_n = 0$ since the radius of curvature is infinity and thus there is no change in direction. The Bernoulli equation results from a force balance along a streamline.

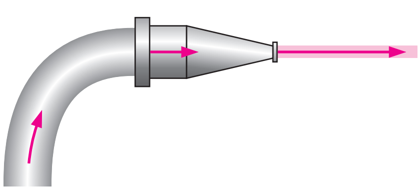


FIGURE 5–22
During steady flow, a fluid may not accelerate in time at a fixed point, but it may accelerate in space.

One may be tempted to think that acceleration is zero in steady flow since acceleration is the rate of change of velocity with time, and in steady flow there is no change with time. Well, a garden hose nozzle tells us that this understanding is not correct. Even in steady flow and thus constant mass flow rate, water accelerates through the nozzle (Fig. 5–22 as discussed in Chap. 4). *Steady* simply means *no change with time at a specified location*, but the value of a quantity may change from one location to another. In the case of a nozzle, the velocity of water remains constant at a specified point, but it changes from the inlet to the exit (water accelerates along the nozzle).

Mathematically, this can be expressed as follows: We take the velocity V of a fluid particle to be a function of s and t. Taking the total differential of $V(s, t)$ and dividing both sides by dt give

$$dV = \frac{\partial V}{\partial s} ds + \frac{\partial V}{\partial t} dt \qquad \text{and} \qquad \frac{dV}{dt} = \frac{\partial V}{\partial s}\frac{ds}{dt} + \frac{\partial V}{\partial t} \tag{5–33}$$

In steady flow $\partial V/\partial t = 0$ and thus $V = V(s)$, and the acceleration in the s-direction becomes

$$a_s = \frac{dV}{dt} = \frac{\partial V}{\partial s}\frac{ds}{dt} = \frac{\partial V}{\partial s} V = V\frac{dV}{ds} \tag{5–34}$$

where $V = ds/dt$ if we are following a fluid particle as it moves along a streamline. Therefore, acceleration in steady flow is due to the change of velocity with position.

Derivation of the Bernoulli Equation

Consider the motion of a fluid particle in a flow field in steady flow described in detail in Chap. 4. Applying Newton's second law (which is

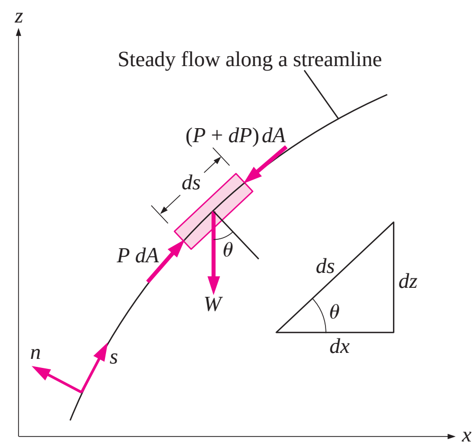


FIGURE 5–23
The forces acting on a fluid particle along a streamline.

referred to as the *conservation of linear momentum* relation in fluid mechanics) in the *s*-direction on a particle moving along a streamline gives

$$\sum F_s = ma_s \tag{5–35}$$

In regions of flow where net frictional forces are negligible, the significant forces acting in the *s*-direction are the pressure (acting on both sides) and the component of the weight of the particle in the *s*-direction (Fig. 5–23). Therefore, Eq. 5–35 becomes

$$P\,dA - (P + dP)\,dA - W \sin\theta = mV\frac{dV}{ds} \tag{5–36}$$

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

$$-dP\,dA - \rho g\,dA\,ds\frac{dz}{ds} = \rho\,dA\,ds\,V\frac{dV}{ds} \tag{5–37}$$

Canceling dA from each term and simplifying,

$$-dP - \rho g\,dz = \rho V\,dV \tag{5–38}$$

Noting that $V\,dV = \frac{1}{2}\,d(V^2)$ and dividing each term by ρ gives

$$\frac{dP}{\rho} + \frac{1}{2}\,d(V^2) + g\,dz = 0 \tag{5–39}$$

Integrating (Fig. 5–24),

Steady flow: $$\int\frac{dP}{\rho} + \frac{V^2}{2} + gz = \text{constant (along a streamline)} \tag{5–40}$$

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

Steady, incompressible flow: $$\frac{P}{\rho} + \frac{V^2}{2} + gz = \text{constant (along a streamline)} \tag{5–41}$$

This is the famous **Bernoulli equation,** which is commonly used in fluid mechanics for steady, incompressible flow along a streamline in inviscid

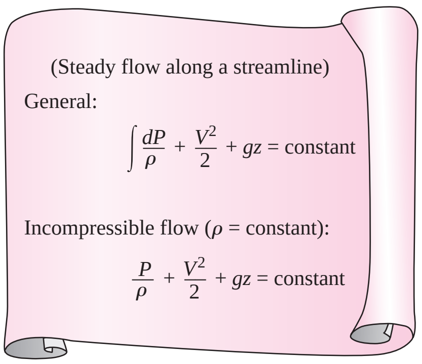


FIGURE 5–24
The Bernoulli equation is derived assuming incompressible flow, and thus it should not be used for flows with significant compressibility effects.

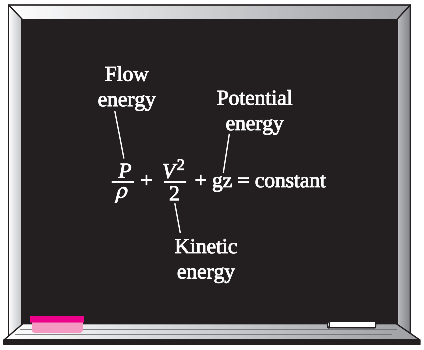


FIGURE 5–25
The Bernoulli equation states that the sum of the kinetic, potential, and flow energies of a fluid particle is constant along a streamline during steady flow.

regions of flow. The value of the constant can be evaluated at any point on the streamline where the pressure, density, velocity, and elevation are known. The Bernoulli equation can also be written between any two points on the same streamline as

Steady, incompressible flow:
$$\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2 \tag{5–42}$$

The Bernoulli equation is obtained from the conservation of momentum for a fluid particle moving along a streamline. It can also be obtained from the *first law of thermodynamics* applied to a steady-flow system, as shown in Section 5–7.

The Bernoulli equation was first stated in words by the Swiss mathematician Daniel Bernoulli (1700–1782) in a text written in 1738 when he was working in St. Petersburg, Russia. It was later derived in equation form by his associate Leonhard Euler in 1755. We recognize $V^2/2$ as *kinetic energy,* gz as *potential energy,* and P/ρ as *flow energy,* all per unit mass. Therefore, the Bernoulli equation can be viewed as an expression of *mechanical energy balance* and can be stated as follows (Fig. 5–25):

> The sum of the kinetic, potential, and flow energies of a fluid particle is constant along a streamline during steady flow when the compressibility and frictional effects are negligible.

The kinetic, potential, and flow energies are the mechanical forms of energy, as discussed in Section 5–3, and the Bernoulli equation can be viewed as the "conservation of mechanical energy principle." This is equivalent to the general conservation of energy principle for systems that do not involve any conversion of mechanical energy and thermal energy to each other, and thus the mechanical energy and thermal energy are conserved separately. The Bernoulli equation states that during steady, incompressible flow with negligible friction, the various forms of mechanical energy are converted to each other, but their sum remains constant. In other words, there is no dissipation of mechanical energy during such flows since there is no friction that converts mechanical energy to sensible thermal (internal) energy.

Recall that energy is transferred to a system as work when a force is applied to a system through a distance. In the light of Newton's second law of motion, the Bernoulli equation can also be viewed as: *The work done by the pressure and gravity forces on the fluid particle is equal to the increase in the kinetic energy of the particle.*

Despite the highly restrictive approximations used in its derivation, the Bernoulli equation is commonly used in practice since a variety of practical fluid flow problems can be analyzed to reasonable accuracy with it. This is because many flows of practical engineering interest are steady (or at least steady in the mean), compressibility effects are relatively small, and net frictional forces are negligible in regions of interest in the flow.

Force Balance across Streamlines

It is left as an exercise to show that a force balance in the direction n normal to the streamline yields the following relation applicable *across* the streamlines for steady, incompressible flow:

$$\frac{P}{\rho} + \int \frac{V^2}{R}\,dn + gz = \text{constant} \qquad \text{(across streamlines)} \tag{5–43}$$

For flow along a straight line, $R \rightarrow \infty$ and thus relation (Eq. 5–44) reduces to $P/\rho + gz =$ constant or $P = -\rho gz +$ constant, which is an expression for the variation of hydrostatic pressure with vertical distance for a stationary fluid body. Therefore, the variation of pressure with elevation in steady, incompressible flow along a straight line is the same as that in the stationary fluid (Fig. 5–26).

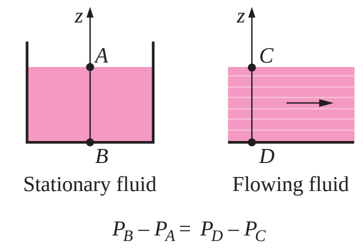


FIGURE 5–26
The variation of pressure with elevation in steady, incompressible flow along a straight line is the same as that in the stationary fluid (but this is not the case for a curved flow section).

Unsteady, Compressible Flow

Similarly, using both terms in the acceleration expression (Eq. 5–33), it can be shown that the Bernoulli equation for *unsteady, compressible flow* is

Unsteady, compressible flow: $$\int \frac{dP}{\rho} + \int \frac{\partial V}{\partial t}\, ds + \frac{V^2}{2} + gz = \text{constant} \qquad \textbf{(5–44)}$$

Static, Dynamic, and Stagnation Pressures

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

$$P + \rho \frac{V^2}{2} + \rho gz = \text{constant (along a streamline)} \qquad \textbf{(5–45)}$$

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

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

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

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

$$P_{\text{stag}} = P + \rho \frac{V^2}{2} \qquad \text{(kPa)} \qquad \textbf{(5–46)}$$

The stagnation pressure represents the pressure at a point where the fluid is brought to a complete stop isentropically. The static, dynamic, and stagnation pressures are shown in Fig. 5–27. When static and stagnation pressures are measured at a specified location, the fluid velocity at that location can be calculated from

$$V = \sqrt{\frac{2(P_{\text{stag}} - P)}{\rho}} \qquad \textbf{(5–47)}$$

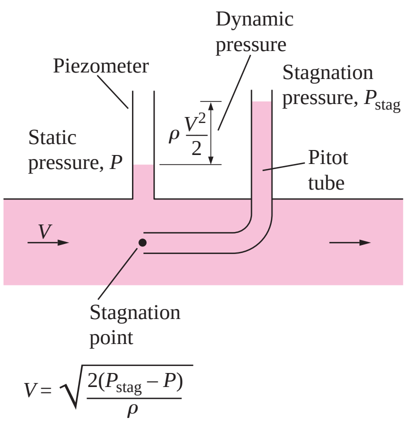


FIGURE 5–27
The static, dynamic, and stagnation pressures.

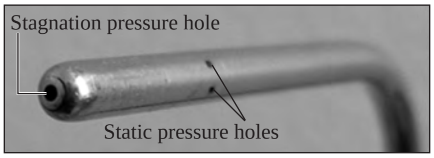


FIGURE 5–28
Close-up of a Pitot-static probe, showing the stagnation pressure hole and two of the five static circumferential pressure holes.
Photo by Po-Ya Abel Chuang. Used by permission.

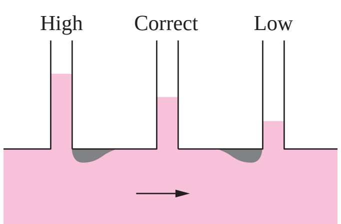


FIGURE 5–29
Careless drilling of the static pressure tap may result in an erroneous reading of the static pressure.

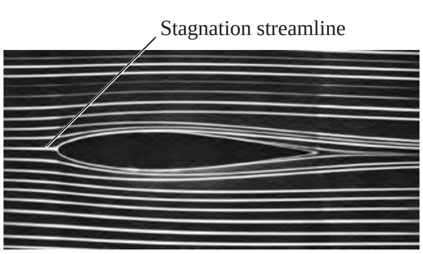


FIGURE 5–30
Streaklines produced by colored fluid introduced upstream of an airfoil; since the flow is steady, the streaklines are the same as streamlines and pathlines. The stagnation streamline is marked.
Courtesy ONERA. Photograph by Werlé.

Equation 5–47 is useful in the measurement of flow velocity when a combination of a static pressure tap and a Pitot tube is used, as illustrated in Fig. 5–27. A **static pressure tap** is simply a small hole drilled into a wall such that the plane of the hole is parallel to the flow direction. It measures the static pressure. A **Pitot tube** is a small tube with its open end aligned *into* the flow so as to sense the full impact pressure of the flowing fluid. It measures the stagnation pressure. In situations in which the static and stagnation pressure of a flowing *liquid* are greater than atmospheric pressure, a vertical transparent tube called a **piezometer tube** (or simply a **piezometer**) can be attached to the pressure tap and to the Pitot tube, as sketched in Fig. 5–27. The liquid rises in the piezometer tube to a column height (*head*) that is proportional to the pressure being measured. If the pressures to be measured are below atmospheric, or if measuring pressures in *gases*, piezometer tubes do not work. However, the static pressure tap and Pitot tube can still be used, but they must be connected to some other kind of pressure measurement device such as a U-tube manometer or a pressure transducer (Chap. 3). Sometimes it is convenient to integrate static pressure holes on a Pitot probe. The result is a **Pitot-static probe,** as shown in Fig. 5–28 and discussed in more detail in Chap. 8. A Pitot-static probe connected to a pressure transducer or a manometer measures the dynamic pressure (and thus fluid velocity) directly.

When the static pressure is measured by drilling a hole in the tube wall, care must be exercised to ensure that the opening of the hole is flush with the wall surface, with no extrusions before or after the hole (Fig. 5–29). Otherwise the reading will incorporate some dynamic effects, and thus it will be in error.

When a stationary body is immersed in a flowing stream, the fluid is brought to a stop at the nose of the body (the **stagnation point**). The flow streamline that extends from far upstream to the stagnation point is called the **stagnation streamline** (Fig. 5–30). For a two-dimensional flow in the *xy*-plane, the stagnation point is actually a *line* parallel the *z*-axis, and the stagnation streamline is actually a *surface* that separates fluid that flows *over* the body from fluid that flows *under* the body. In an incompressible flow, the fluid decelerates nearly isentropically from its free-stream value to zero at the stagnation point, and the pressure at the stagnation point is thus the stagnation pressure.

Limitations on the Use of the Bernoulli Equation

The Bernoulli equation (Eq. 5–41) is one of the most frequently used and misused equations in fluid mechanics. Its versatility, simplicity, and ease of use make it a very valuable tool for use in analysis, but the same attributes also make it very tempting to misuse. Therefore, it is important to understand the restrictions on its applicability and observe the limitations on its use, as explained here:

1. **Steady flow** The first limitation on the Bernoulli equation is that it is applicable to *steady flow*. Therefore, it should not be used during the transient start-up and shut-down periods, or during periods of change in the flow conditions. Note that there is an unsteady form of the Bernoulli equation (Eq. 5–44), discussion of which is beyond the scope of the present text (see Panton, 1996).

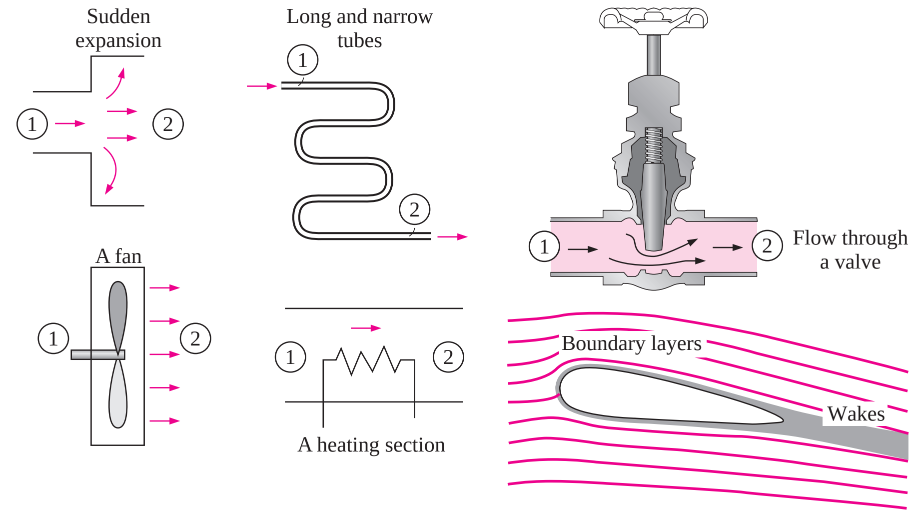
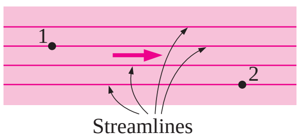


FIGURE 5–31
Frictional effects and components that disturb the streamlined structure of flow in a flow section make the Bernoulli equation invalid.

2. **Frictionless flow** Every flow involves some friction, no matter how small, and *frictional effects* may or may not be negligible. The situation is complicated even more by the amount of error that can be tolerated. In general, frictional effects are negligible for short flow sections with large cross sections, especially at low flow velocities. Frictional effects are usually significant in long and narrow flow passages, in the wake region downstream of an object, and in *diverging flow sections* such as diffusers because of the increased possibility of the fluid separating from the walls in such geometries. Frictional effects are also significant near solid surfaces, and thus the Bernoulli equation is usually applicable along a streamline in the core region of the flow, but not along a streamline close to the surface (Fig. 5–31).

   A component that disturbs the streamlined structure of flow and thus causes considerable mixing and backflow such as a sharp entrance of a tube or a partially closed valve in a flow section can make the Bernoulli equation inapplicable.
3. **No shaft work** The Bernoulli equation was derived from a force balance on a particle moving along a streamline. Therefore, the Bernoulli equation is not applicable in a flow section that involves a pump, turbine, fan, or any other machine or impeller since such devices destroy the streamlines and carry out energy interactions with the fluid particles. When the flow section considered involves any of these devices, the energy equation should be used instead to account for the shaft work input or output. However, the Bernoulli equation can still be applied to a flow section prior to or past a machine (assuming, of course, that the other restrictions on its use are satisfied). In such cases, the Bernoulli constant changes from upstream to downstream of the device.
4. **Incompressible flow** One of the assumptions used in the derivation of the Bernoulli equation is that ρ = constant and thus the flow is incompressible. This condition is satisfied by liquids and also by gases at Mach numbers less than about 0.3 since compressibility effects and thus density variations of gases are negligible at such relatively low



$$\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

FIGURE 5–32
When the flow is irrotational, the Bernoulli equation becomes applicable between any two points along the flow (not just on the same streamline).

velocities. Note that there is a compressible form of the Bernoulli equation (Eqs. 5–40 and 5–44).

5. **No heat transfer** The density of a gas is inversely proportional to temperature, and thus the Bernoulli equation should not be used for flow sections that involve significant temperature change such as heating or cooling sections.
6. **Flow along a streamline** Strictly speaking, the Bernoulli equation $P/\rho + V^2/2 + gz = C$ is applicable along a streamline, and the value of the constant C, in general, is different for different streamlines. But when a region of the flow is *irrotational*, and thus there is no *vorticity* in the flow field, the value of the constant C remains the same for all streamlines, and, therefore, the Bernoulli equation becomes applicable *across* streamlines as well (Fig. 5–32). Therefore, we do not need to be concerned about the streamlines when the flow is irrotational, and we can apply the Bernoulli equation between any two points in the irrotational region of the flow (Chap. 10).

We derived the Bernoulli equation by considering two-dimensional flow in the *xz*-plane for simplicity, but the equation is valid for general three-dimensional flow as well, as long as it is applied along the same streamline. We should always keep in mind the assumptions used in the derivation of the Bernoulli equation and make sure that they are not violated.

Hydraulic Grade Line (HGL) and Energy Grade Line (EGL)

It is often convenient to represent the level of mechanical energy graphically using *heights* to facilitate visualization of the various terms of the Bernoulli equation. This is done by dividing each term of the Bernoulli equation by g to give

$$\frac{P}{\rho g} + \frac{V^2}{2g} + z = H = \text{constant} \qquad \text{(along a streamline)} \tag{5–48}$$

Each term in this equation has the dimension of length and represents some kind of "head" of a flowing fluid as follows:

- $P/\rho g$ is the **pressure head;** it represents the height of a fluid column that produces the static pressure P.
- $V^2/2g$ is the **velocity head;** it represents the elevation needed for a fluid to reach the velocity V during frictionless free fall.
- z is the **elevation head;** it represents the potential energy of the fluid.

Also, H is the **total head** for the flow. Therefore, the Bernoulli equation can be expressed in terms of heads as: *The sum of the pressure, velocity, and elevation heads along a streamline is constant during steady flow when the compressibility and frictional effects are negligible* (Fig. 5–33).

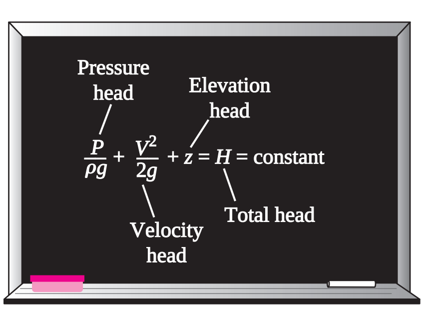


FIGURE 5–33
An alternative form of the Bernoulli equation is expressed in terms of heads as: *The sum of the pressure, velocity, and elevation heads is constant along a streamline.*

If a piezometer (measures static pressure) is tapped into a pipe, as shown in Fig. 5–34, the liquid would rise to a height of $P/\rho g$ above the pipe center. The *hydraulic grade line* (HGL) is obtained by doing this at several locations along the pipe and drawing a line through the liquid levels in the piezometers. The vertical distance above the pipe center is a measure of pressure within the pipe. Similarly, if a Pitot tube (measures static +

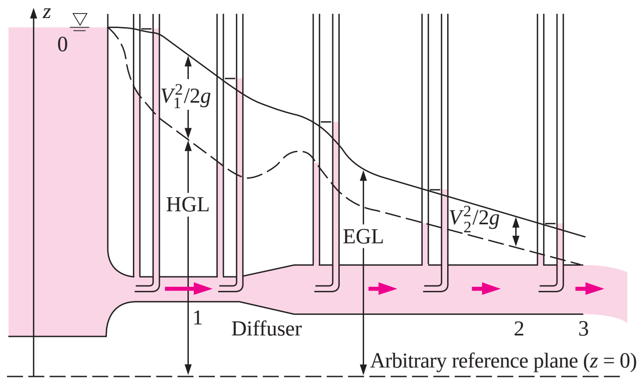


FIGURE 5–34
The *hydraulic grade line* (HGL) and the *energy grade line* (EGL) for free discharge from a reservoir through a horizontal pipe with a diffuser.

dynamic pressure) is tapped into a pipe, the liquid would rise to a height of $P/\rho g + V^2/2g$ above the pipe center, or a distance of $V^2/2g$ above the HGL. The *energy grade line* (EGL) is obtained by doing this at several locations along the pipe and drawing a line through the liquid levels in the Pitot tubes.

Noting that the fluid also has elevation head z (unless the reference level is taken to be the centerline of the pipe), the HGL and EGL can be defined as follows: The line that represents the sum of the static pressure and the elevation heads, $P/\rho g + z$, is called the **hydraulic grade line.** The line that represents the total head of the fluid, $P/\rho g + V^2/2g + z$, is called the **energy grade line.** The difference between the heights of EGL and HGL is equal to the dynamic head, $V^2/2g$. We note the following about the HGL and EGL:

- For *stationary bodies* such as reservoirs or lakes, the EGL and HGL coincide with the free surface of the liquid. The elevation of the free surface z in such cases represents both the EGL and the HGL since the velocity is zero and the static pressure (gage) is zero.
- The EGL is always a distance $V^2/2g$ above the HGL. These two lines approach each other as the velocity decreases, and they diverge as the velocity increases. The height of the HGL decreases as the velocity increases, and vice versa.
- In an *idealized Bernoulli-type flow,* EGL is horizontal and its height remains constant. This would also be the case for HGL when the flow velocity is constant (Fig. 5–35).
- For *open-channel flow,* the HGL coincides with the free surface of the liquid, and the EGL is a distance $V^2/2g$ above the free surface.
- At a *pipe exit,* the pressure head is zero (atmospheric pressure) and thus the HGL coincides with the pipe outlet (location 3 on Fig. 5–34).
- The *mechanical energy loss* due to frictional effects (conversion to thermal energy) causes the EGL and HGL to slope downward in the direction of flow. The slope is a measure of the head loss in the pipe (discussed in detail in Chap. 8). A component that generates significant frictional effects such as a valve causes a sudden drop in both EGL and HGL at that location.
- A *steep jump* occurs in EGL and HGL whenever mechanical energy is added to the fluid (by a pump, for example). Likewise, a *steep drop*

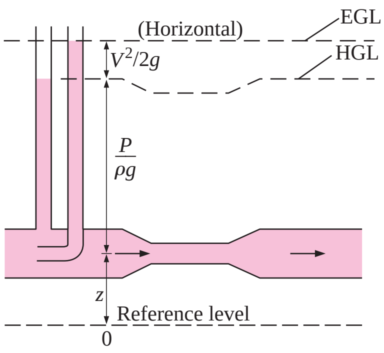


FIGURE 5–35
In an idealized Bernoulli-type flow, EGL is horizontal and its height remains constant. But this is not the case for HGL when the flow velocity varies along the flow.

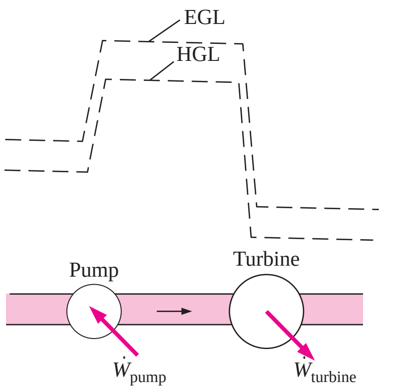


FIGURE 5–36
A *steep jump* occurs in EGL and HGL whenever mechanical energy is added to the fluid by a pump, and a *steep drop* occurs whenever mechanical energy is removed from the fluid by a turbine.

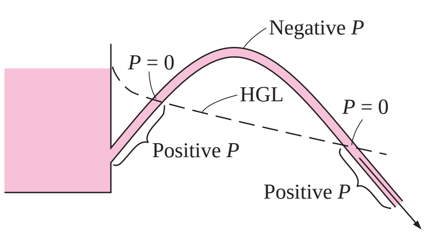


FIGURE 5–37
The pressure (gage) of a fluid is zero at locations where the HGL *intersects* the fluid, and the pressure is negative (vacuum) in a flow section that lies above the HGL.

occurs in EGL and HGL whenever mechanical energy is removed from the fluid (by a turbine, for example), as shown in Fig. 5–36.

- The pressure (gage) of a fluid is zero at locations where the HGL *intersects* the fluid. The pressure in a flow section that lies above the HGL is negative, and the pressure in a section that lies below the HGL is positive (Fig. 5–37). Therefore, an accurate drawing of a piping system and the HGL can be used to determine the regions where the pressure in the pipe is negative (below the atmospheric pressure).

The last remark enables us to avoid situations in which the pressure drops below the vapor pressure of the liquid (which causes *cavitation*, as discussed in Chap. 2). Proper consideration is necessary in the placement of a liquid pump to ensure that the suction side pressure does not fall too low, especially at elevated temperatures where vapor pressure is higher than it is at low temperatures.

Now we examine Fig. 5–34 more closely. At point 0 (at the liquid surface), EGL and HGL are even with the liquid surface since there is no flow there. HGL decreases rapidly as the liquid accelerates into the pipe; however, EGL decreases very slowly through the well-rounded pipe inlet. EGL declines continually along the flow direction due to friction and other irreversible losses in the flow. EGL cannot increase in the flow direction unless energy is supplied to the fluid. HGL can rise or fall in the flow direction, but can never exceed EGL. HGL rises in the diffuser section as the velocity decreases, and the static pressure recovers somewhat; the total pressure does *not* recover, however, and EGL decreases through the diffuser. The difference between EGL and HGL is $V_1^2/2g$ at point 1, and $V_2^2/2g$ at point 2. Since $V_1 > V_2$, the difference between the two grade lines is larger at point 1 than at point 2. The downward slope of both grade lines is larger for the smaller diameter section of pipe since the frictional head loss is greater. Finally, HGL decays to the liquid surface at the outlet since the pressure there is atmospheric. However, EGL is still higher than HGL by the amount $V_2^2/2g$ since $V_3 = V_2$ at the outlet.

5–5 ▪ APPLICATIONS OF THE BERNOULLI EQUATION

In Section 5–4, we discussed the fundamental aspects of the Bernoulli equation. In this section, we demonstrate its use in a wide range of applications through examples.

EXAMPLE 5–5 Spraying Water into the Air

Water is flowing from a hose attached to a water main at 400 kPa gage (Fig. 5–38). A child places his thumb to cover most of the hose outlet, causing a thin jet of high-speed water to emerge. If the hose is held upward, what is the maximum height that the jet could achieve?

SOLUTION Water from a hose attached to the water main is sprayed into the air. The maximum height the water jet can rise is to be determined.
Assumptions **1** The flow exiting into the air is steady, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The water pressure

in the hose near the outlet is equal to the water main pressure. **3** The surface tension effects are negligible. **4** The friction between the water and air is negligible. **5** The irreversibilities that may occur at the outlet of the hose due to abrupt expansion are negligible.

Properties We take the density of water to be 1000 kg/m^3.

Analysis This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. The water height will be maximum under the stated assumptions. The velocity inside the hose is relatively low ($V_1 \cong 0$) and we take the hose outlet as the reference level ($z_1 = 0$). At the top of the water trajectory $V_2 = 0$, and atmospheric pressure pertains. Then the Bernoulli equation simplifies to

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

Solving for z_2 and substituting,

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

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

Therefore, the water jet can rise as high as 40.8 m into the sky in this case.

Discussion The result obtained by the Bernoulli equation represents the upper limit and should be interpreted accordingly. It tells us that the water cannot possibly rise more than 40.8 m, and, in all likelihood, the rise will be much less than 40.8 m due to irreversible losses that we neglected.

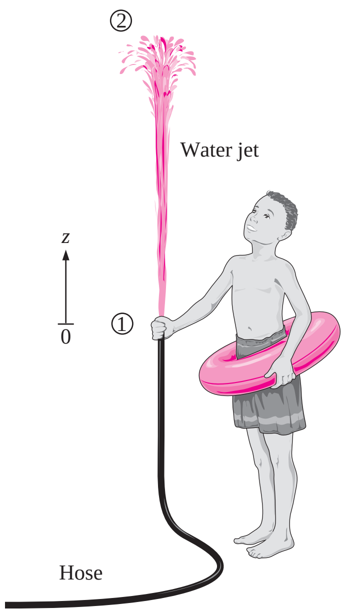


FIGURE 5–38
Schematic for Example 5–5.

EXAMPLE 5–6 Water Discharge from a Large Tank

A large tank open to the atmosphere is filled with water to a height of 5 m from the outlet tap (Fig. 5–39). A tap near the bottom of the tank is now opened, and water flows out from the smooth and rounded outlet. Determine the water velocity at the outlet.

SOLUTION A tap near the bottom of a tank is opened. The exit velocity of water from the tank is to be determined.

Assumptions **1** The flow is incompressible and irrotational (except very close to the walls). **2** The water drains slowly enough that the flow can be approximated as steady (actually quasi-steady when the tank begins to drain).

Analysis This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. We take point 1 to be at the free surface of water so that $P_1 = P_{atm}$ (open to the atmosphere), $V_1 \cong 0$ (the tank is large relative to the outlet), and $z_1 = 5$ m and $z_2 = 0$ (we take the reference level at the center of the outlet). Also, $P_2 = P_{atm}$ (water discharges into the atmosphere). Then the Bernoulli equation simplifies to

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

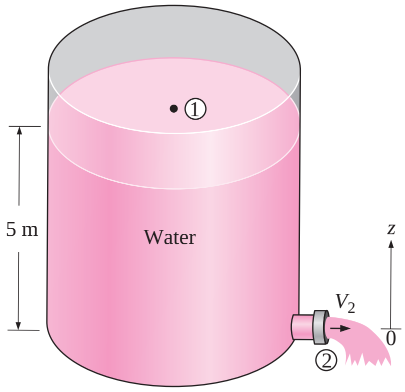


FIGURE 5–39
Schematic for Example 5–6.

Solving for V_2 and substituting,

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

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

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

Discussion If the orifice were sharp-edged instead of rounded, then the flow would be disturbed, and the velocity would be less than 9.9 m/s, especially near the edges. Care must be exercised when attempting to apply the Bernoulli equation to situations where abrupt expansions or contractions occur since the friction and flow disturbance in such cases may not be negligible.

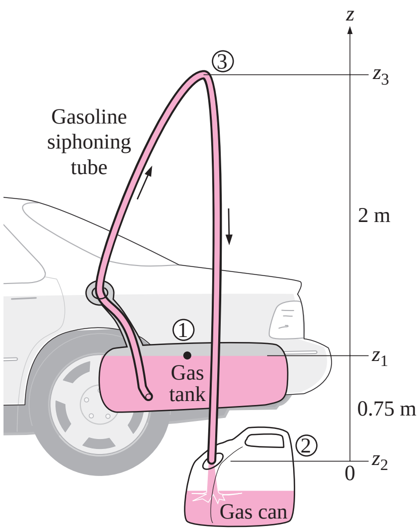


FIGURE 5–40
Schematic for Example 5–7.

EXAMPLE 5–7 Siphoning Out Gasoline from a Fuel Tank

During a trip to the beach (P_{atm} = 1 atm = 101.3 kPa), a car runs out of gasoline, and it becomes necessary to siphon gas out of the car of a Good Samaritan (Fig. 5–40). The siphon is a small-diameter hose, and to start the siphon it is necessary to insert one siphon end in the full gas tank, fill the hose with gasoline via suction, and then place the other end in a gas can below the level of the gas tank. The difference in pressure between point 1 (at the free surface of the gasoline in the tank) and point 2 (at the outlet of the tube) causes the liquid to flow from the higher to the lower elevation. Point 2 is located 0.75 m below point 1 in this case, and point 3 is located 2 m above point 1. The siphon diameter is 4 mm, and frictional losses in the siphon are to be disregarded. Determine (*a*) the minimum time to withdraw 4 L of gasoline from the tank to the can and (*b*) the pressure at point 3. The density of gasoline is 750 kg/m^3.

SOLUTION Gasoline is to be siphoned from a tank. The minimum time it takes to withdraw 4 L of gasoline and the pressure at the highest point in the system are to be determined.

Assumptions **1** The flow is steady and incompressible. **2** Even though the Bernoulli equation is not valid through the pipe because of frictional losses, we employ the Bernoulli equation anyway in order to obtain a *best-case estimate*. **3** The change in the gasoline surface level inside the tank is negligible compared to elevations z_1 and z_2 during the siphoning period.

Properties The density of gasoline is given to be 750 kg/m^3.

Analysis (*a*) We take point 1 to be at the free surface of gasoline in the tank so that $P_1 = P_{atm}$ (open to the atmosphere), $V_1 \cong 0$ (the tank is large relative to the tube diameter), and $z_2 = 0$ (point 2 is taken as the reference level). Also, $P_2 = P_{atm}$ (gasoline discharges into the atmosphere). Then the Bernoulli equation simplifies to

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

Solving for V_2 and substituting,

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

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

$$A = \pi D^2/4 = \pi(5 \times 10^{-3}\text{ m})^2/4 = 1.96 \times 10^{-5}\text{ m}^2$$

$$\dot{V} = V_2 A = (3.84\text{ m/s})(1.96 \times 10^{-5}\text{ m}^2) = 7.53 \times 10^{-5}\text{ m}^3\text{/s} = 0.0753\text{ L/s}$$

Then the time needed to siphon 4 L of gasoline becomes

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

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

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

Solving for P_3 and substituting,

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

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

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

Discussion The siphoning time is determined by neglecting frictional effects, and thus this is the *minimum time* required. In reality, the time will be longer than 53.1 s because of friction between the gasoline and the tube surface. Also, the pressure at point 3 is below the atmospheric pressure. If the elevation difference between points 1 and 3 is too high, the pressure at point 3 may drop below the vapor pressure of gasoline at the gasoline temperature, and some gasoline may evaporate (cavitate). The vapor then may form a pocket at the top and halt the flow of gasoline.

EXAMPLE 5–8 Velocity Measurement by a Pitot Tube

A piezometer and a Pitot tube are tapped into a horizontal water pipe, as shown in Fig. 5–41, to measure static and stagnation (static + dynamic) pressures. For the indicated water column heights, determine the velocity at the center of the pipe.

SOLUTION The static and stagnation pressures in a horizontal pipe are measured. The velocity at the center of the pipe is to be determined.
Assumptions **1** The flow is steady and incompressible. **2** Points 1 and 2 are close enough together that the irreversible energy loss between these two points is negligible, and thus we can use the Bernoulli equation.
Analysis We take points 1 and 2 along the centerline of the pipe, with point 1 directly under the piezometer and point 2 at the tip of the Pitot tube. This is a steady flow with straight and parallel streamlines, and the gage pressures at points 1 and 2 can be expressed as

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

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

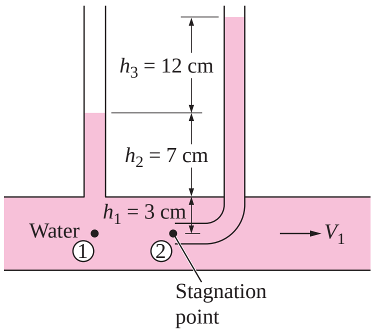


FIGURE 5–41
Schematic for Example 5–8.

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

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

Substituting the P_1 and P_2 expressions gives

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

Solving for V_1 and substituting,

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

Discussion Note that to determine the flow velocity, all we need is to measure the height of the excess fluid column in the Pitot tube.

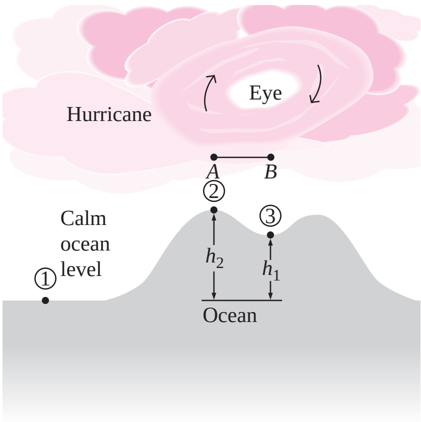


FIGURE 5–42
Schematic for Example 5–9. The vertical scale is greatly exaggerated.

EXAMPLE 5–9 The Rise of the Ocean Due to a Hurricane

A hurricane is a tropical storm formed over the ocean by low atmospheric pressures. As a hurricane approaches land, inordinate ocean swells (very high tides) accompany the hurricane. A Class-5 hurricane features winds in excess of 155 mph, although the wind velocity at the center "eye" is very low.

Figure 5–42 depicts a hurricane hovering over the ocean swell below. The atmospheric pressure 200 mi from the eye is 30.0 in Hg (at point 1, generally normal for the ocean) and the winds are calm. The hurricane atmospheric pressure at the eye of the storm is 22.0 in Hg. Estimate the ocean swell at (*a*) the eye of the hurricane at point 3 and (*b*) point 2, where the wind velocity is 155 mph. Take the density of seawater and mercury to be 64 lbm/ft^3 and 848 lbm/ft^3, respectively, and the density of air at normal sea-level temperature and pressure to be 0.076 lbm/ft^3.

SOLUTION A hurricane is moving over the ocean. The amount of ocean swell at the eye and at active regions of the hurricane are to be determined.
Assumptions **1** The airflow within the hurricane is steady, incompressible, and irrotational (so that the Bernoulli equation is applicable). (This is certainly a very questionable assumption for a highly turbulent flow, but it is justified in the solution.) **2** The effect of water drifted into the air is negligible.
Properties The densities of air at normal conditions, seawater, and mercury are given to be 0.076 lbm/ft^3, 64 lbm/ft^3, and 848 lbm/ft^3, respectively.
Analysis (*a*) Reduced atmospheric pressure over the water causes the water to rise. Thus, decreased pressure at point 2 relative to point 1 causes the ocean water to rise at point 2. The same is true at point 3, where the storm air velocity is negligible. The pressure difference given in terms of the mercury column height can be expressed in terms of the seawater column height by

$$\Delta P = (\rho g h)_{\text{Hg}} = (\rho g h)_{\text{sw}} \rightarrow h_{\text{sw}} = \frac{\rho_{\text{Hg}}}{\rho_{\text{sw}}} h_{\text{Hg}}$$

Then the pressure difference between points 1 and 3 in terms of the seawater column height becomes

$$h_1 = \frac{\rho_{Hg}}{\rho_{sw}} h_{Hg} = \left(\frac{848 \text{ lbm/ft}^3}{64 \text{ lbm/ft}^3}\right)[(30 - 22) \text{ in Hg}]\left(\frac{1 \text{ ft}}{12 \text{ in}}\right) = \mathbf{8.83 \text{ ft}}$$

which is equivalent to the storm surge at the *eye of the hurricane* since the wind velocity there is negligible and there are no dynamic effects.

(*b*) To determine the additional rise of ocean water at point 2 due to the high winds at that point, we write the Bernoulli equation between points *A* and *B*, which are on top of the points 2 and 3, respectively. Noting that $V_B \cong 0$ (the eye region of the hurricane is relatively calm) and $z_A = z_B$ (both points are on the same horizontal line), the Bernoulli equation simplifies to

$$\frac{P_A}{\rho g} + \frac{V_A^2}{2g} + \cancel{z_A} = \frac{P_B}{\rho g} + \frac{V_B^2}{2g}\nearrow^0 + \cancel{z_B} \quad \rightarrow \quad \frac{P_B - P_A}{\rho g} = \frac{V_A^2}{2g}$$

Substituting,

$$\frac{P_B - P_A}{\rho g} = \frac{V_A^2}{2g} = \frac{(155 \text{ mph})^2}{2(32.2 \text{ ft/s}^2)}\left(\frac{1.4667 \text{ ft/s}}{1 \text{ mph}}\right)^2 = 803 \text{ ft}$$

where ρ is the density of air in the hurricane. Noting that the density of an ideal gas at constant temperature is proportional to absolute pressure and the density of air at the normal atmospheric pressure of 14.7 psia $\cong$ 30 in Hg is 0.076 lbm/ft^3, the density of air in the hurricane is

$$\rho_{air} = \frac{P_{air}}{P_{atm\ air}} \rho_{atm\ air} = \left(\frac{22 \text{ in Hg}}{30 \text{ in Hg}}\right)(0.076 \text{ lbm/ft}^3) = 0.056 \text{ lbm/ft}^3$$

Using the relation developed above in part (*a*), the seawater column height equivalent to 803 ft of air column height is determined to be

$$h_{dynamic} = \frac{\rho_{air}}{\rho_{sw}} h_{air} = \left(\frac{0.056 \text{ lbm/ft}^3}{64 \text{ lbm/ft}^3}\right)(803 \text{ ft}) = 0.70 \text{ ft}$$

Therefore, the pressure at point 2 is 0.70 ft seawater column lower than the pressure at point 3 due to the high wind velocities, causing the ocean to rise an additional 0.70 ft. Then the total storm surge at point 2 becomes

$$h_2 = h_1 + h_{dynamic} = 8.83 + 0.70 = \mathbf{9.53 \text{ ft}}$$

Discussion This problem involves highly turbulent flow and the intense breakdown of the streamlines, and thus the applicability of the Bernoulli equation in part (*b*) is questionable. Furthermore, the flow in the eye of the storm is *not* irrotational, and the Bernoulli equation constant changes across streamlines (see Chap. 10). The Bernoulli analysis can be thought of as the limiting, ideal case, and shows that the rise of seawater due to high-velocity winds cannot be more than 0.70 ft.

The wind power of hurricanes is not the only cause of damage to coastal areas. Ocean flooding and erosion from excessive tides is just as serious, as are high waves generated by the storm turbulence and energy.

CHAPTER

6

MOMENTUM ANALYSIS OF FLOW SYSTEMS

When dealing with engineering problems, it is desirable to obtain fast and accurate solutions at minimal cost. Most engineering problems, including those associated with fluid flow, can be analyzed using one of three basic approaches: differential, experimental, and control volume. In *differential approaches*, the problem is formulated accurately using differential quantities, but the solution of the resulting differential equations is difficult, usually requiring the use of numerical methods with extensive computer codes. *Experimental approaches* complemented with dimensional analysis are highly accurate, but they are typically time-consuming and expensive. The *finite control volume approach* described in this chapter is remarkably fast and simple and usually gives answers that are sufficiently accurate for most engineering purposes. Therefore, despite the approximations involved, the basic finite control volume analysis performed with a paper and pencil has always been an indispensable tool for engineers.

In Chap. 5, the control volume mass and energy analysis of fluid flow systems was presented. In this chapter, we present the finite control volume momentum analysis of fluid flow problems. First we give an overview of Newton's laws and the conservation relations for linear and angular momentum. Then using the Reynolds transport theorem, we develop the linear momentum and angular momentum equations for control volumes and use them to determine the forces and torques associated with fluid flow.

OBJECTIVES

When you finish reading this chapter, you should be able to

- Identify the various kinds of forces and moments acting on a control volume
- Use control volume analysis to determine the forces associated with fluid flow
- Use control volume analysis to determine the moments caused by fluid flow and the torque transmitted

6–1 ▪ NEWTON'S LAWS AND CONSERVATION OF MOMENTUM

Newton's laws are relations between motions of bodies and the forces acting on them. Newton's first law states that *a body at rest remains at rest, and a body in motion remains in motion at the same velocity in a straight path when the net force acting on it is zero*. Therefore, a body tends to preserve its state of inertia. Newton's second law states that *the acceleration of a body is proportional to the net force acting on it and is inversely proportional to its mass.* Newton's third law states that *when a body exerts a force on a second body, the second body exerts an equal and opposite force on the first*. Therefore, the direction of an exposed reaction force depends on the body taken as the system.

For a rigid body of mass m, Newton's second law is expressed as

Newton's second law:

$$\vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d(m\vec{V})}{dt} \tag{6–1}$$

where $\vec{F}$ is the net force acting on the body and $\vec{a}$ is the acceleration of the body under the influence of $\vec{F}$.

The product of the mass and the velocity of a body is called the *linear momentum* or just the *momentum* of the body. The momentum of a rigid body of mass m moving with a velocity $\vec{V}$ is $m\vec{V}$ (Fig. 6–1). Then Newton's second law expressed in Eq. 6–1 can also be stated as *the rate of change of the momentum of a body is equal to the net force acting on the body* (Fig. 6–2). This statement is more in line with Newton's original statement of the second law, and it is more appropriate for use in fluid mechanics when studying the forces generated as a result of velocity changes of fluid streams. Therefore, in fluid mechanics, Newton's second law is usually referred to as the *linear momentum equation*.

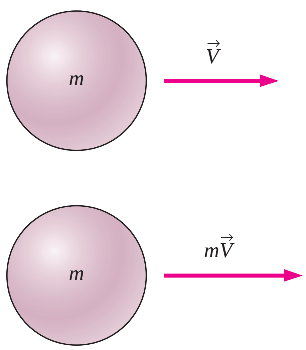


FIGURE 6–1
Linear momentum is the product of mass and velocity, and its direction is the direction of velocity.

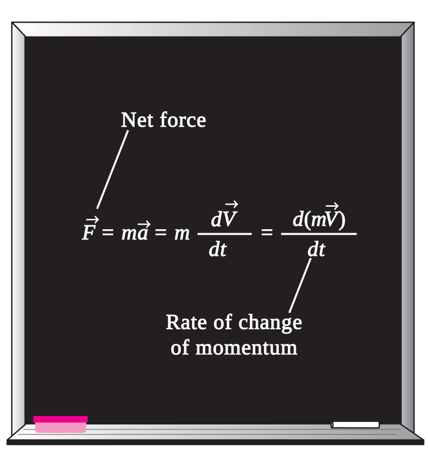


FIGURE 6–2
Newton's second law is also expressed as *the rate of change of the momentum of a body is equal to the net force acting on it.*

The momentum of a system remains constant when the net force acting on it is zero, and thus the momentum of such systems is conserved. This is known as the *conservation of momentum principle*. This principle has proven to be a very useful tool when analyzing collisions such as those between balls; between balls and rackets, bats, or clubs; and between atoms or subatomic particles; and explosions such as those that occur in rockets, missiles, and guns. The momentum of a loaded rifle, for example, must be zero after shooting since it is zero before shooting, and thus the rifle must have a momentum equal to that of the bullet in the opposite direction so that the vector sum of the two is zero.

Note that force, acceleration, velocity, and momentum are vector quantities, and as such they have direction as well as magnitude. Also, momentum is a constant multiple of velocity, and thus the direction of momentum is the direction of velocity. Any vector equation can be written in scalar form for a specified direction using magnitudes, e.g., $F_x = ma_x = d(mV_x)/dt$ in the x-direction.

The counterpart of Newton's second law for rotating rigid bodies is expressed as $\vec{M} = I\vec{\alpha}$, where $\vec{M}$ is the net moment or torque applied on the body, I is the moment of inertia of the body about the axis of rotation, and $\vec{\alpha}$ is the angular acceleration. It can also be expressed in terms of the rate of change of angular momentum $d\vec{H}/dt$ as

Angular momentum equation: $$\vec{M} = I\vec{\alpha} = I\frac{d\vec{\omega}}{dt} = \frac{d(I\vec{\omega})}{dt} = \frac{d\vec{H}}{dt} \quad \textbf{(6–2)}$$

where $\vec{\omega}$ is the angular velocity. For a rigid body rotating about a fixed x-axis, the angular momentum equation can be written in scalar form as

Angular momentum about x-axis: $$M_x = I_x\frac{d\omega_x}{dt} = \frac{dH_x}{dt} \quad \textbf{(6–3)}$$

The angular momentum equation can be stated as *the rate of change of the angular momentum of a body is equal to the net torque acting on it* (Fig. 6–3).

The total angular momentum of a rotating body remains constant when the net torque acting on it is zero, and thus the angular momentum of such systems is conserved. This is known as the *conservation of angular momentum principle* and is expressed as $I\omega$ = constant. Many interesting phenomena such as ice skaters spinning faster when they bring their arms close to their bodies and divers rotating faster when they curl after the jump can be explained easily with the help of the conservation of angular momentum principle (in both cases, the moment of inertia I is decreased and thus the angular velocity ω is increased as the outer parts of the body are brought closer to the axis of rotation).

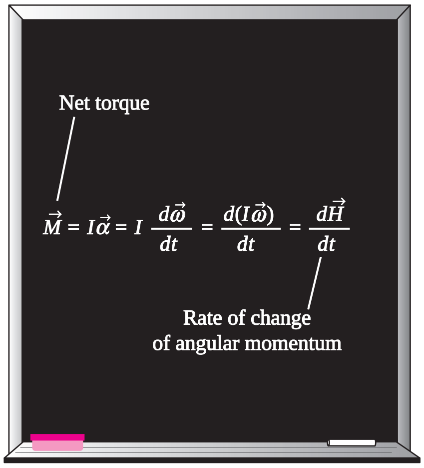


FIGURE 6–3
The rate of change of the angular momentum of a body is equal to the net torque acting on it.

6–2 ▪ CHOOSING A CONTROL VOLUME

We now briefly discuss how to *wisely* select a control volume. A control volume can be selected as any arbitrary region in space through which fluid flows, and its bounding control surface can be fixed, moving, and even deforming during flow. The application of a basic conservation law is simply a systematic procedure for bookkeeping or accounting of the quantity under consideration, and thus it is extremely important that the boundaries of the control volume are well defined during an analysis. Also, the flow rate of any quantity into or out of a control volume depends on the flow velocity *relative to the control surface*, and thus it is essential to know if the control volume remains at rest during flow or if it moves.

Many flow systems involve stationary hardware firmly fixed to a stationary surface, and such systems are best analyzed using *fixed* control volumes. When determining the reaction force acting on a tripod holding the nozzle of a hose, for example, a natural choice for the control volume is one that passes perpendicularly through the nozzle exit flow and through the bottom of the tripod legs (Fig. 6–4*a*). This is a fixed control volume, and the water velocity relative to a fixed point on the ground is the same as the water velocity relative to the nozzle exit plane.

When analyzing flow systems that are moving or deforming, it is usually more convenient to allow the control volume to *move* or *deform*. When determining the thrust developed by the jet engine of an airplane cruising at constant velocity, for example, a wise choice of control volume is one that encloses the airplane and cuts through the nozzle exit plane (Fig. 6–4*b*). The control volume in this case moves with velocity $\vec{V}_{CV}$, which is identical to the cruising velocity of the airplane relative to a fixed point on earth. When determining the flow rate of exhaust gases leaving the nozzle, the proper

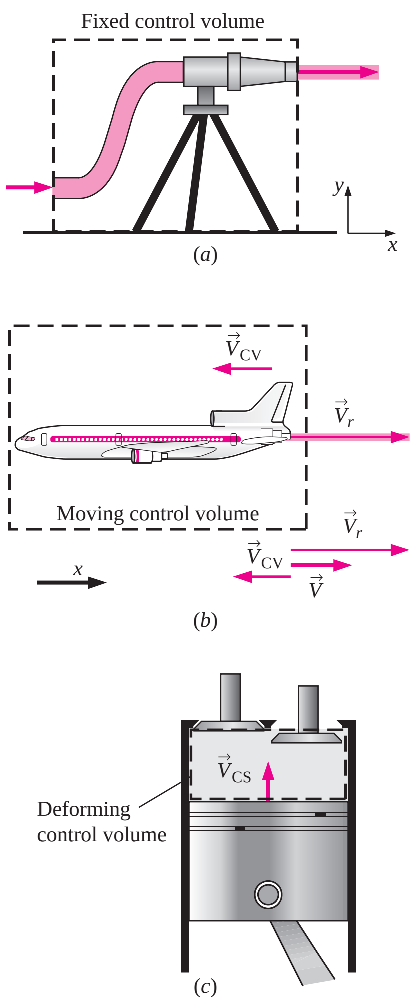


FIGURE 6–4
Examples of (*a*) fixed, (*b*) moving, and (*c*) deforming control volumes.

velocity to use is the velocity of the exhaust gases relative to the nozzle exit plane, that is, the *relative velocity* $\vec{V}_r$. Since the entire control volume moves at velocity $\vec{V}_{CV}$, the relative velocity becomes $\vec{V}_r = \vec{V} - \vec{V}_{CV}$, where $\vec{V}$ is the *absolute velocity* of the exhaust gases, i.e., the velocity relative to a fixed point on earth. Note that $\vec{V}_r$ is the fluid velocity expressed relative to a coordinate system moving *with* the control volume. Also, this is a vector equation, and velocities in opposite directions have opposite signs. For example, if the airplane is cruising at 500 km/h to the left, and the velocity of the exhaust gases is 800 km/h to the right relative to the ground, the velocity of the exhaust gases relative to the nozzle exit is

$$\vec{V}_r = \vec{V} - \vec{V}_{CV} = 800\vec{i} - (-500\vec{i}) = 1300\vec{i} \text{ km/h}$$

That is, the exhaust gases leave the nozzle at 1300 km/h to the right relative to the nozzle exit (in the direction opposite to that of the airplane); this is the velocity that should be used when evaluating the outflow of exhaust gases through the control surface (Fig. 6–4*b*). Note that the exhaust gases would appear motionless to an observer on the ground if the relative velocity were equal in magnitude to the airplane velocity.

When analyzing the purging of exhaust gases from a reciprocating internal combustion engine, a wise choice for the control volume is one that comprises the space between the top of the piston and the cylinder head (Fig. 6–4*c*). This is a *deforming* control volume, since part of the control surface moves relative to other parts. The relative velocity for an inlet or outlet on the deforming part of a control surface (there are no such inlets or outlets in Fig. 6–4*c*) is then given by $\vec{V}_r = \vec{V} - \vec{V}_{CS}$ where $\vec{V}$ is the absolute fluid velocity and $\vec{V}_{CS}$ is the control surface velocity, both relative to a fixed point outside the control volume. Note that $\vec{V}_{CS} = \vec{V}_{CV}$ for moving but nondeforming control volumes, and $\vec{V}_{CS} = \vec{V}_{CV} = 0$ for fixed ones.

6–3 ▪ FORCES ACTING ON A CONTROL VOLUME

The forces acting on a control volume consist of **body forces** that act throughout the entire body of the control volume (such as gravity, electric, and magnetic forces) and **surface forces** that act on the control surface (such as pressure and viscous forces and reaction forces at points of contact).

In control volume analysis, the sum of all forces acting on the control volume at a particular instant in time is represented by $\sum \vec{F}$ and is expressed as

Total force acting on control volume:
$$\sum \vec{F} = \sum \vec{F}_{body} + \sum \vec{F}_{surface} \quad \textbf{(6–4)}$$

Body forces act on each volumetric portion of the control volume. The body force acting on a differential element of fluid of volume dV within the control volume is shown in Fig. 6–5, and we must perform a volume integral to account for the net body force on the entire control volume. *Surface forces* act on each portion of the control surface. A differential surface element of area dA and unit outward normal $\vec{n}$ on the control surface is shown in Fig. 6–5, along with the surface force acting on it. We must perform an area integral to obtain the net surface force acting on the entire control surface. As sketched, the surface force may act in a direction independent of that of the outward normal vector.

The most common body force is that of **gravity,** which exerts a downward force on every differential element of the control volume. While other body forces, such as electric and magnetic forces, may be important in some analyses, we consider only gravitational forces here.

The differential body force $d\vec{F}_{\text{body}} = d\vec{F}_{\text{gravity}}$ acting on the small fluid element shown in Fig. 6–6 is simply its weight,

Gravitational force acting on a fluid element: $$d\vec{F}_{\text{gravity}} = \rho\vec{g}\, dV \quad \textbf{(6–5)}$$

where ρ is the average density of the element and $\vec{g}$ is the gravitational vector. In Cartesian coordinates we adopt the convention that $\vec{g}$ acts in the negative *z*-direction, as in Fig. 6–6, so that

Gravitational vector in Cartesian coordinates: $$\vec{g} = -g\vec{k} \quad \textbf{(6–6)}$$

Note that the coordinate axes in Fig. 6–6 have been rotated from their usual orientation so that the gravity vector acts *downward* in the $-z$-direction. On earth at sea level, the gravitational constant g is equal to 9.807 m/s^2. Since gravity is the only body force being considered, integration of Eq. 6–5 yields

Total body force acting on control volume: $$\sum \vec{F}_{\text{body}} = \int_{\text{CV}} \rho\vec{g}\, dV = m_{\text{CV}}\vec{g} \quad \textbf{(6–7)}$$

Surface forces are not as simple to analyze since they consist of both *normal* and *tangential* components. Furthermore, while the physical force acting on a surface is independent of orientation of the coordinate axes, the *description* of the force in terms of its coordinate components changes with orientation (Fig. 6–7). In addition, we are rarely fortunate enough to have each of the control surfaces aligned with one of the coordinate axes. While not desiring to delve too deeply into tensor algebra, we are forced to define a **second-order tensor** called the **stress tensor** σ_{ij} in order to adequately describe the surface stresses at a point in the flow,

Stress tensor in Cartesian coordinates: $$\sigma_{ij} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix} \quad \textbf{(6–8)}$$

The diagonal components of the stress tensor, σ_{xx}, σ_{yy}, and σ_{zz}, are called **normal stresses;** they are composed of pressure (which always acts inwardly normal) and viscous stresses. Viscous stresses are discussed in more detail in Chap. 9. The off-diagonal components, σ_{xy}, σ_{zx}, etc., are called **shear stresses;** since pressure can act only normal to a surface, shear stresses are composed entirely of viscous stresses.

When the face is not parallel to one of the coordinate axes, mathematical laws for axes rotation and tensors can be used to calculate the normal and tangential components acting at the face. In addition, an alternate notation called **tensor notation** is convenient when working with tensors but is usually reserved for graduate studies. (For a more in-depth analysis of tensors and tensor notation see, for example, Kundu, 1990.)

In Eq. 6–8, σ_{ij} is defined as the stress (force per unit area) in the *j*-direction acting on a face whose normal is in the *i*-direction. Note that *i* and *j* are merely *indices* of the tensor and are not the same as unit vectors $\vec{i}$ and $\vec{j}$. For example, σ_{xy} is defined as positive for the stress pointing in the *y*-direction

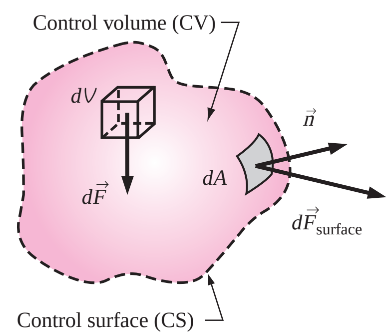


FIGURE 6–5
The total force acting on a control volume is composed of body forces and surface forces; body force is shown on a differential volume element, and surface force is shown on a differential surface element.

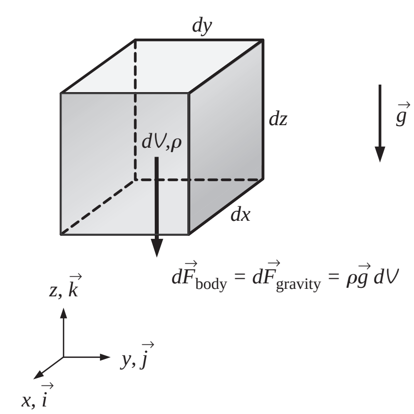


FIGURE 6–6
The gravitational force acting on a differential volume element of fluid is equal to its weight; the axes have been rotated so that the gravity vector acts *downward* in the negative *z*-direction.

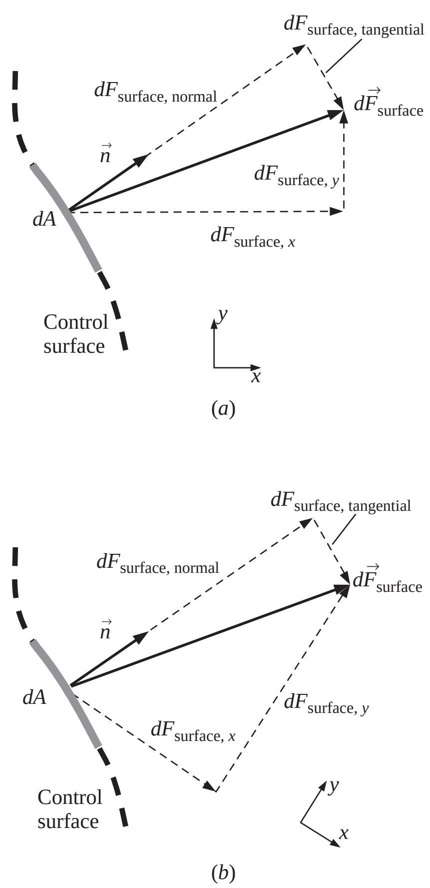


FIGURE 6–7
When coordinate axes are rotated (*a*) to (*b*), the components of the surface force change, even though the force itself remains the same; only two dimensions are shown here.

on a face whose outward normal is in the x-direction. This component of the stress tensor, along with the other eight components, is shown in Fig. 6–8 for the case of a differential fluid element aligned with the axes in Cartesian coordinates. All the components in Fig. 6–8 are shown on positive faces (right, top, and front) and in their positive orientation by definition. Positive stress components on the *opposing* faces of the fluid element (not shown) point in exactly opposite directions.

The dot product of a second-order tensor and a vector yields a second vector; this operation is often called the **contracted product** or the **inner product** of a tensor and a vector. In our case, it turns out that the inner product of the stress tensor σ_{ij} and the unit outward normal vector $\vec{n}$ of a differential surface element yields a vector whose magnitude is the force per unit area acting on the surface element and whose direction is the direction of the surface force itself. Mathematically we write

Surface force acting on a differential surface element: $$d\vec{F}_{\text{surface}} = \sigma_{ij} \cdot \vec{n}\, dA \qquad \textbf{(6–9)}$$

Finally, we integrate Eq. 6–9 over the entire control surface,

Total surface force acting on control surface: $$\sum \vec{F}_{\text{surface}} = \int_{\text{CS}} \sigma_{ij} \cdot \vec{n}\, dA \qquad \textbf{(6–10)}$$

Substitution of Eqs. 6–7 and 6–10 into Eq. 6–4 yields

$$\sum \vec{F} = \sum \vec{F}_{\text{body}} + \sum \vec{F}_{\text{surface}} = \int_{\text{CV}} \rho \vec{g}\, dV + \int_{\text{CS}} \sigma_{ij} \cdot \vec{n}\, dA \qquad \textbf{(6–11)}$$

This equation turns out to be quite useful in the derivation of the differential form of conservation of linear momentum, as discussed in Chap. 9. For practical control volume analysis, however, it is rare that we need to use Eq. 6–11, especially the cumbersome surface integral that it contains.

A careful selection of the control volume enables us to write the total force acting on the control volume, $\Sigma \vec{F}$, as the sum of more readily available quantities like weight, pressure, and reaction forces. We recommend the following for control volume analysis:

Total force: $$\underbrace{\sum \vec{F}}_{\text{total force}} = \underbrace{\sum \vec{F}_{\text{gravity}}}_{\text{body force}} + \underbrace{\sum \vec{F}_{\text{pressure}} + \sum \vec{F}_{\text{viscous}} + \sum \vec{F}_{\text{other}}}_{\text{surface forces}} \qquad \textbf{(6–12)}$$

The first term on the right-hand side of Eq. 6–12 is the body force *weight*, since gravity is the only body force we are considering. The other three terms combine to form the net surface force; they are pressure forces, viscous forces, and "other" forces acting on the control surface. $\Sigma \vec{F}_{\text{other}}$ is composed of reaction forces required to turn the flow; forces at bolts, cables, struts, or walls through which the control surface cuts; etc.

All these surface forces arise as the control volume is isolated from its surroundings for analysis, and the effect of any detached object is accounted for by a force at that location. This is similar to drawing a free-body diagram in your statics and dynamics classes. We should choose the control volume such that forces that are not of interest remain internal, and thus they do not complicate the analysis. A well-chosen control volume exposes only the forces that are to be determined (such as reaction forces) and a minimum number of other forces.

Only external forces are considered in the analysis. The internal forces (such as the pressure force between a fluid and the inner surfaces of the flow section) are not considered in a control volume analysis unless they are exposed by passing the control surface through that area.

A common simplication in the application of Newton's laws of motion is to subtract the *atmospheric pressure* and work with gage pressures. This is because atmospheric pressure acts in all directions, and its effect cancels out in every direction (Fig. 6–9). This means we can also ignore the pressure forces at outlet sections where the fluid is discharged to the atmosphere since the discharge pressures in such cases will be very near atmospheric pressure at subsonic velocities.

As an example of how to wisely choose a control volume, consider control volume analysis of water flowing steadily through a faucet with a partially closed gate valve spigot (Fig. 6–10). It is desired to calculate the net force on the flange to ensure that the flange bolts are strong enough. There are many possible choices for the control volume. Some engineers restrict their control volumes to the fluid itself, as indicated by CV A (the colored control volume). With this control volume, there are pressure forces that vary along the control surface, there are viscous forces along the pipe wall and at locations inside the valve, and there is a body force, namely, the weight of the water in the control volume. Fortunately, to calculate the net force on the flange, we do *not* need to integrate the pressure and viscous stresses all along the control surface. Instead, we can lump the unknown pressure and viscous forces together into one reaction force, representing the net force of the walls on the water. This force, plus the weight of the faucet and the water, is equal to the net force on the flange. (We must be very careful with our signs, of course.)

When choosing a control volume, you are not limited to the fluid alone. Often it is more convenient to slice the control surface *through* solid objects such as walls, struts, or bolts as illustrated by CV B (the gray control volume) in Fig. 6–10. A control volume may even surround an entire object, like the one shown here. Control volume B is a wise choice because we are not concerned with any details of the flow or even the geometry inside the control volume. For the case of CV B, we assign a net reaction force acting at the portions of the control surface that slice through the flange. Then, the only other things we need to know are the gage pressure of the water at the flange (the inlet to the control volume) and the weights of the water and the faucet assembly. The pressure everywhere else along the control surface is atmospheric (zero gage pressure) and cancels out. This problem is revisited in Section 6–4, Example 6–7.

6–4 ▪ THE LINEAR MOMENTUM EQUATION

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

$$\sum \vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d}{dt}(m\vec{V}) \tag{6–13}$$

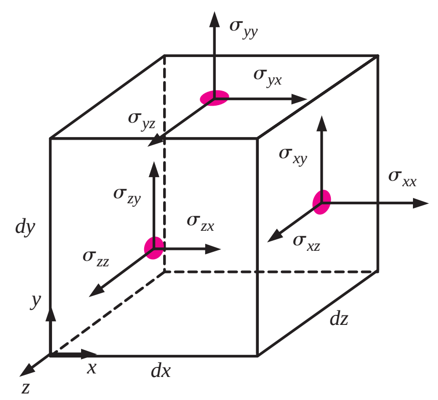


FIGURE 6–8
Components of the stress tensor in Cartesian coordinates on the right, top, and front faces.

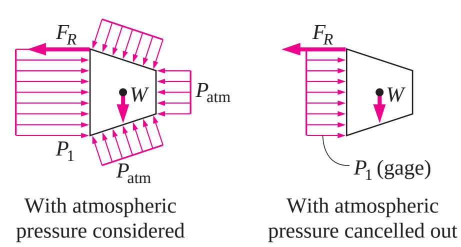


FIGURE 6–9
Atmospheric pressure acts in all directions, and thus it can be ignored when performing force balances since its effect cancels out in every direction.

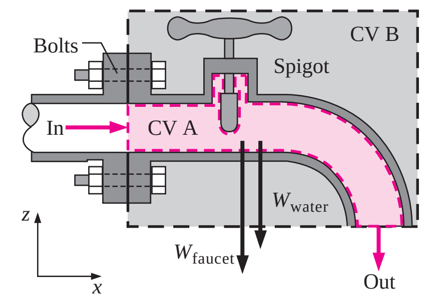


FIGURE 6–10
Cross section through a faucet assembly, illustrating the importance of choosing a control volume wisely; CV B is much easier to work with than CV A.

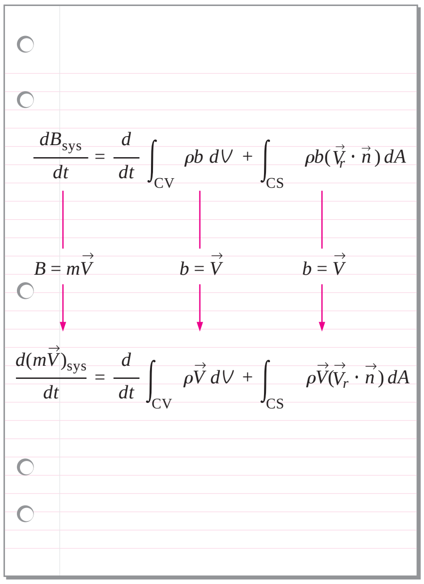


FIGURE 6–11
The linear momentum equation is obtained by replacing B in the Reynolds transport theorem by the momentum $m\vec{V}$, and b by the momentum per unit mass $\vec{V}$.

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

$$\sum \vec{F} = \frac{d}{dt}\int_{sys} \rho\vec{V}\, dV \tag{6–14}$$

where $\delta m = \rho dV$ is the mass of a differential volume element dV, and $\rho\vec{V}\, dV$ is its momentum. Therefore, Newton's second law can be stated as *the sum of all external forces acting on a system is equal to the time rate of change of linear momentum of the system*. This statement is valid for a coordinate system that is at rest or moves with a constant velocity, called an *inertial coordinate system* or *inertial reference frame*. Accelerating systems such as aircraft during takeoff are best analyzed using noninertial (or accelerating) coordinate systems fixed to the aircraft. Note that Eq. 6–14 is a vector relation, and thus the quantities $\vec{F}$ and $\vec{V}$ have direction as well as magnitude.

Equation 6–14 is for a given mass of a solid or fluid and is of limited use in fluid mechanics since most flow systems are analyzed using control volumes. The *Reynolds transport theorem* developed in Section 4–5 provides the necessary tools to shift from the system formulation to the control volume formulation. Setting $b = \vec{V}$ and thus $B = m\vec{V}$, the Reynolds transport theorem can be expressed for linear momentum as (Fig. 6–11)

$$\frac{d(m\vec{V})_{sys}}{dt} = \frac{d}{dt}\int_{CV} \rho\vec{V}\, dV + \int_{CS} \rho\vec{V}\,(\vec{V}_r \cdot \vec{n})\, dA \tag{6–15}$$

But the left-hand side of this equation is, from Eq. 6–13, equal to $\Sigma\vec{F}$. Substituting, the general form of the linear momentum equation that applies to fixed, moving, or deforming control volumes is obtained to be

General:
$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho\vec{V}\, dV + \int_{CS} \rho\vec{V}(\vec{V}_r \cdot \vec{n})\, dA \tag{6–16}$$

which can be stated as

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

Here $\vec{V}_r = \vec{V} - \vec{V}_{CS}$ is the fluid velocity relative to the control surface (for use in mass flow rate calculations at all locations where the fluid crosses the control surface), and $\vec{V}$ is the fluid velocity as viewed from an inertial reference frame. The product $\rho(\vec{V}_r \cdot \vec{n})\, dA$ represents the mass flow rate through area element dA into or out of the control volume.

For a fixed control volume (no motion or deformation of control volume), $\vec{V}_r = \vec{V}$ and the linear momentum equation becomes

Fixed CV:
$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho\vec{V}\, dV + \int_{CS} \rho\vec{V}(\vec{V} \cdot \vec{n})\, dA \tag{6–17}$$

Note that the momentum equation is a *vector equation*, and thus each term should be treated as a vector. Also, the components of this equation can be resolved along orthogonal coordinates (such as x, y, and z in the Cartesian coordinate system) for convenience. The force $\vec{F}$ in most cases consists of weights, pressure forces, and reaction forces (Fig. 6–12). The momentum equation is commonly used to calculate the forces (usually on support systems or connectors) induced by the flow.

Special Cases

During *steady flow*, the amount of momentum within the control volume remains constant, and thus the time rate of change of linear momentum of the contents of the control volume (the second term of Eq. 6–16) is zero. It gives

Steady flow:
$$\sum \vec{F} = \int_{CS} \rho \vec{V}\,(\vec{V}_r \cdot \vec{n})\, dA \tag{6–18}$$

Most momentum problems considered in this text are steady.

While Eq. 6–17 is exact for fixed control volumes, it is not always convenient when solving practical engineering problems because of the integrals. Instead, as we did for conservation of mass, we would like to rewrite Eq. 6–17 in terms of average velocities and mass flow rates through inlets and outlets. In other words, our desire is to rewrite the equation in *algebraic* rather than *integral* form. In many practical applications, fluid crosses the boundaries of the control volume at one or more inlets and one or more outlets, and carries with it some momentum into or out of the control volume. For simplicity, we always draw our control surface such that it slices normal to the inflow or outflow velocity at each such inlet or outlet (Fig. 6–13).

The mass flow rate $\dot{m}$ into or out of the control volume across an inlet or outlet at which ρ is nearly constant is

Mass flow rate across an inlet or outlet:
$$\dot{m} = \int_{A_c} \rho(\vec{V} \cdot \vec{n})\, dA_c = \rho V_{avg} A_c \tag{6–19}$$

Comparing Eq. 6–19 to Eq. 6–17, we notice an extra velocity in the control surface integral of Eq. 6–17. If $\vec{V}$ were uniform ($\vec{V} = \vec{V}_{avg}$) across the inlet or outlet, we could simply take it outside the integral. Then we could write the rate of inflow or outflow of momentum through the inlet or outlet in simple algebraic form,

Momentum flow rate across a uniform inlet or outlet:
$$\int_{A_c} \rho \vec{V}(\vec{V} \cdot \vec{n})\, dA_c = \rho V_{avg} A_c \vec{V}_{avg} = \dot{m}\vec{V}_{avg} \tag{6–20}$$

The uniform flow approximation is reasonable at some inlets and outlets, e.g., the well-rounded entrance to a pipe, the flow at the entrance to a wind tunnel test section, and a slice through a water jet moving at nearly uniform speed through air (Fig. 6–14). At each such inlet or outlet, Eq. 6–20 can be applied directly.

Momentum-Flux Correction Factor, β

Unfortunately, the velocity across most inlets and outlets of practical engineering interest is *not* uniform. Nevertheless, it turns out that we can still convert the control surface integral of Eq. 6–17 into algebraic form, but a

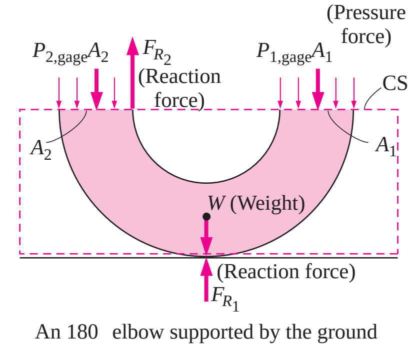


FIGURE 6–12
In most flow systems, the force $\vec{F}$ consists of weights, pressure forces, and reaction forces. Gage pressures are used here since atmospheric pressure cancels out on all sides of the control surface.

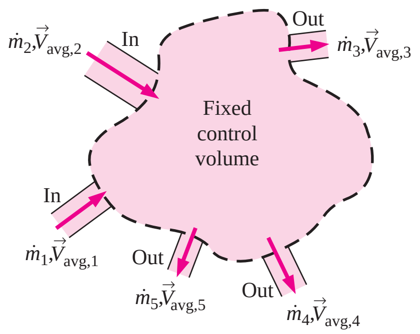


FIGURE 6–13
In a typical engineering problem, the control volume may contain many inlets and outlets; at each inlet or outlet we define the mass flow rate $\dot{m}$ and the average velocity $\vec{V}_{avg}$.